MIKE+

User Guide

Model Manager

PLEASE NOTE

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.

1 Welcome to MIKE+

MIKE+ is a flexible system for modelling and design of water distribution networks, collection systems for waste water and storm water and river networks.

MIKE+ is based on a database for storing network as well as hydraulic modelling data. This database is based upon the SQLite and/or PostGIS. The SQLite database is a C-language library that implements a small, fast, self-contained SQL Database engine. It is the preferred solution for an easy installation and for individual usage, as it doesn't require any other installation and configuration than MIKE+. PostGIS is a spatial database extender for PostgreSQL object-relational database. It adds support for geographic objects allowing location queries to be run in SQL. The use of a PostGIS database requires that a PostgreSQL installation is already available on a server, and is relevant for collaborative work in companies. The installation and initial configuration of the PostgreSQL database is not controlled by MIKE+.

MIKE+ allows quick Integration to ArcGIS Pro for a quick built of a personal GeoDatabase in a native GIS data storage format. Hence operation directly by standard GIS applications is still possible.

With MIKE+ you have:

- GIS-based model building and management
- Powerful hydraulic simulation engines
- Integrated water quality, fire flow, and real time control simulation (water distribution)
- Integrated water quality, sediment transport, control rules for structures, and long-term statistics (collection systems)
- Integrated water quality (River networks and 2D overland)
- Scenario management
- Full undo and redo capability in all editors
- Thematic mapping and integrated dynamic result visualization
- Open data models - easy integration with other applications
- Worldwide support and training
- Integrates directly with online and real time control systems

MIKE+ has a modular structure, to fit to all applications' needs. The list of modules is presented in the following chapters.

1.1 Model Manager

The Model Manager is the main module of MIKE+ and includes a common data module for all types of applications. Input editors for all simulation engines are included irrespective of the installed simulation engines.

1.2 Water Distribution

MIKE+ for Water Distribution comes with the following modules:

- WD-Basic. For modelling water distribution networks using EPANET engine
- WD-Tools. Allows for modelling advanced features including fire flow, network vulnerability, cost analysis, shutdown planning, flushing, pressure dependent demands, variable speed pumps, real-time control and optimization.

1.2.1 WD-Basic

Allows standard modelling of water distributing networks using the EPANET engine including water age/quality.

1.2.2 WD-Tools

With MIKE+ Water Distribution Tools you get the following capabilities:

Fire Flow Analysis

Calculating water availability for fire protection requirements is one of the most frequent modelling tasks of water utility. The Fire Flow Analysis module allows you to calculate the available flow for the design pressure or to calculate the residual pressure for the design flow.

There are two basic ways to model a fire flow:

- Specify a design fire flow rate and compute the available fire flow pressure
- Specify a design fire flow pressure and compute the available fire flow rate

Network vulnerability

Network vulnerability is used to predict the water distribution system response to pipe break situations, planned reconstructions, and other scenarios of limited water supply. Network vulnerability also allows the development of a pipe ranking based on the importance for the water supply, such importance can be then considered into a pipe rehabilitation and construction plan.

Cost Analysis
Cost analysis allows you to review the energy consumption results on more details, create tabular outputs, and great graphs of pump utilization, average power consumption, and costs.

Shutdown Planning
The Shutdown Planning analysis allows to determine the impact of pipe maintenance work on the water supply conditions. It helps the user to define the shutdown, find isolation valves, run hydraulic simulations and evaluate simulation results.

Management of shutdowns contains the following tools:

- Planning shutdown
- Close pipes for selection isolation valves
- Analyse shutdown
- Compare results before and during the shutdown

Flushing Analysis
Flushing of pipelines is a practice done since the early days of municipal water systems. The conventional way to flush pipelines was to simply open selected fire hydrants letting them flow until the running water showed a clean appearance. This is still an effective strategy in many cases. However, these days many water utilities do unidirectional flushing (UDF), which is a more engineered and effective way to flush pipelines. UDF involves closing or opening selected valves to direct flow through target pipes in order to achieve higher velocities for the same hydrant flows. The set of valves that need to be operated and hydrant that is opened is called a flushing sequence. A UDF design consists of a series of flushing sequences that are run in a particular order so that water is always being drawn from clean parts of the system.

Extended Rule-Based Controls
Rule-Based Controls allow link status and settings to be based on a combination of conditions that might exist in the network over an extended period simulation. In order to allow for modelling of chains of pumps or valves chains in efficient way, the EPANET Rule-based control syntax was extended by adding LIKE keyword, for example.

Optimization
This functionality supports scheduling of pumps and operation of control valves. The optimization is based on optimization algorithms that can run with any extended period analysis model.

Online analysis
This functionality enables mapping real-time data available online to the model elements described in the WD model network. This functionality is for use with WD Online.

Multi-species analysis

The multi-species analysis allows modelling water quality for any system of multiple, interacting chemical species. This module is based on the EPANET MSX engine.

Autocalibration

This functionality is based on optimization algorithms, and is meant to calibrate a network model against a number of provided measured time series of pressure, flow, head or water depth. It can be used for optimizing the friction in selections of pipes, the water demands, leakage and/or the Open/Closed status of pipes and valves.

1.3 Collection Systems

The main module for Collection Systems is CS-Pipeflow, which includes DHI's MIKE 1D engine. With MIKE+ CS-Pipeflow you get access to:

- Hydrodynamic simulation of networks
- Long term statistics

Hydrodynamic Simulation

The MIKE 1D Hydrodynamic Pipe Flow Model solves the complete St. Venant (dynamic flow) equations throughout the drainage network (looped and dendritic), which allows for modelling of backwater effects, flow reversal, surcharging in manholes, free-surface and pressure flow, tidal outfalls and storage basins. The MIKE 1D hydrodynamic engine has been designed to handle any type of pipe network system with alternating free surface and pressurized flows as well as open channel network and pipes of any shape. Virtually any construction can be described including pumps, weirs, orifices, inverted siphons, etc.

The computational scheme uses an implicit, finite-difference numerical solution of the St. Venant flow equations. The numerical algorithm uses a self-adapting time-step, which provides efficient and accurate solutions in multiple connected branched and looped pipe networks. This computational scheme is applicable to unsteady flow conditions that occur in pipes ranging from small-profile collectors for detailed urban drainage, to low-lying, often pressurized, sewer mains affected by varying outlet water levels. Both sub-critical and supercritical flows are treated by means of the same computational scheme that adapts to the local flow conditions. In addition, flow phenomena, such as backwater effects and surcharges, are precisely simulated.

Long Term Statistics

MIKE 1D Long Term Statistics (LTS) allows that a collection system network with intermittent hydrological inputs can be setup for a long-term simulation, covering a continuous historical period, possibly over several years. The system automatically combines dynamic pipe flow simulations during wet weather and simple hydrological simulation during dry weather periods, which results in accurate computation of wastewater treatment plant loads, CSOs and other system outputs, while preserving rationality in use of computational resources. The results are presented both in the form of time series and a range of statistical parameters for selected variables. By running simulations with the current system configuration and the planned upgrade, impacts of the planned investments (e.g. new sewers, retention tanks, RTC schemes) on the system performance can be tested. This allows the user to develop the optimal rehabilitation / upgrade strategy, e.g. for achieving the consent with the environmental regulators' requirements.

1.4 Rivers

The main module for river networks is CS-Rivers, which includes DHI's MIKE 1D engine for hydrodynamic modelling.

MIKE 1D's hydrodynamic module solves the complete St. Venant (dynamic flow) equations throughout the river network (looped and dendritic), which allows for modelling of backwater effects, flow reversal, surcharging in closed sections, free-surface flows, tidal outlets and storage in reservoirs.

The computational scheme uses an implicit, finite-difference numerical solution of the St. Venant flow equations. The numerical algorithm may use a self-adapting time-step, which provides efficient and accurate solutions. Both subcritical and supercritical flows are treated by means of the same computational scheme that adapts to the local flow conditions. In addition, flow phenomena, such as backwater effects and surcharges, are precisely simulated.

1.5 2D overland

MIKE+ 2D overland module uses DHI's 2D engine MIKE 21 FM. This engine solves the two-dimensional St. Venant (dynamic flow) equations, using a cell-centered finite volume method. The time integration is performed using an explicit scheme and the numerical solution uses a self-adapting time step for optimizing stability and simulation times. The spatial discretisation can either be done through a rectangular grid or a flexible mesh.

The 2D overland module can be used to simulate free-surface flows to describe detailed flows in channels or describe surface floods from e.g. surcharging collection system networks, rivers, or sea surges.

1.6 Cross-domain modules

The following modules may be used in combination with the Collection Systems, Rivers, and/or 2D overland modules.

1.6.1 CS-Rainfall-Runoff

MIKE+ CS-Rainfall-Runoff modules may be used with both the Collection Systems and Rivers modules. It includes the MIKE 1D engine for rainfall-runoff modelling. With MIKE+ CS-Rainfall-Runoff you get access to:

- Several types of surface runoff models
- Rainfall dependent infiltration (RDI)

Surface Runoff

MIKE 1D Surface Runoff includes several types of surface runoff computation for the description of the urban catchment surfaces. This means that the surface runoff computations can be adjusted according to the amount of available information. The models run with well proven default hydrological parameters, which can be adjusted for better accuracy. The computed hydrographs are used as input to the MIKE 1D Pipe Flow model.

Rainfall Dependent Infiltration

MIKE 1D Rainfall Dependent Infiltration provides detailed, continuous modelling of the complete land phase of the hydrological cycle, providing support for urban, rural and mixed catchments analyses. Precipitation is routed through four different types of storage: snow, surface, root zone and ground water, resulting in more accurate hydrographs. Instead of performing hydrological load analysis of the sewer system only for short periods of high intensity rainstorms, a continuous, long-term analysis can be used to look at periods of both wet and dry weather, as well as inflows and infiltration to the sewer network. This provides a more accurate picture of actual loads on treatment plants and combined sewer overflows.

1.6.2 CS-Control

MIKE+ CS-Control module can be used in combination with the Collection Systems module. It allows real-time control devices to be included in defining the urban drainage sewer network model. A selection of controllable devices is provided, along with a fully generic specification of control rules for any simple or complex global control scheme. The system allows the application of setting or set point (PID controller) based control functions, selected on the basis of logical evaluation of the actual system states (reactive control) or after the specified time series.

1.6.3 Transport

Under the name Transport, the MIKE 1D engine provides several modules for the simulation of sediment transport and water quality for both catchments surfaces and networks. Since pollutants are carried by sediment, sediment transport processes and water quality in sewer systems are closely interconnected. This is important for understanding phenomena like the first flush effect, which can only be simulated with a description of the temporal and spatial distribution of sediment deposits on the catchment surface and in the sewer system.

MIKE 1D can model these complex mechanisms using its Surface Water Quality (SWQ), Advection-Dispersion (AD), Water Quality (MIKE ECO Lab) and Sediment Transport (ST) modules.

In MIKE+ Transport the following can be modelled:

- Stormwater Runoff Quality (SWQ)
- Catchment Discharge Water Quality
- Advection-Dispersion (AD)
- Water Quality (MIKE ECO Lab)
- Sediment Transport (ST)

Stormwater Runoff Quality

The Storm-water Runoff Quality (SWQ) is for use with the Rainfall-Runoff module only. The primary role of the Storm-water Runoff Quality (SWQ) module is to provide a physically-based description of the relevant processes associated with sediments and pollutants due to surface runoff, and then provide surface runoff sediment and pollutant data for the other pipe sewer network sediment transport and water quality modules. The following processes can be accounted for:

- Build-up and wash-off of sediment particles on the catchment
- Surface transport of pollutants attached to the sediment particles
- Build-up and washout of dissolved pollutants in potholes and stilling basins

Sediment deposits can greatly reduce the hydraulic capacity of sewer pipes by restricting their flow area and increasing the bed friction resistance. The Pipe Sediment Transport functionality included in Pollution Transport can account for these problems, by simulating pipe sewer network sediment transport-including deposition and erosion from non-uniform (graded) sediments. Contributions from rainstorm wash-off and dry-weather wastewater flow can be included. The Sediment Transport feature runs in conjunction with the dynamic flow routing, thereby simulating dynamic deposition of sediment and providing feedback due to the change in pipe area and resistance caused by sediment deposition. The following issues can be addressed:

- Prediction of sediment deposit locations and associated pollutants and metals in the sewer system
- Prediction of reduction in hydraulic capacity due to observed and simulated sediment deposits
- Analysis of the sewer system due to modified regulation strategies

Advection-Dispersion (AD)

The Advection-Dispersion (AD) module may be used with all modules: Collection Systems, Rivers, and 2D overland. It simulates the transport of dissolved substances and suspended fine sediments in pipe and river networks as well as on the surface. Conservative materials as well as those that are subject to a linear decay can be simulated. The computed discharges, water levels, and cross-sectional flow areas are used in the AD module computation. The solution of the advection-dispersion equation is obtained using an implicit, finite-difference scheme which has negligible numerical dispersion. Concentration profiles with very steep fronts can be accurately modelled. The computed results can be displayed as longitudinal concentration profiles and pollutant graphs, which could be used at the inflow to a sewage treatment plant or an overflow structure.

The AD module can be linked to the Long Term Statistics module to provide long-term simulations of pollutant transport.

Water Quality (MIKE ECO Lab)

Water Quality module with MIKE ECO Lab can be used with the Collection Systems module, the Rivers module and/or the 2D overland module. Different biological processes can be modelled by means of MIKE ECO Lab working in conjunction with the Advection-Dispersion part of MIKE+ Water Quality. It provides many options for describing the reaction processes of multi-compound systems, including degradation of organic matter, bacterial fate, exchange of oxygen with the atmosphere and oxygen demand from eroded sewer sediments. This allows realistic analysis of complex phenomena related to water quality in sewer systems.

The module includes diurnal variation of foul flow discharges and user-specified concentrations of foul flow components. The sediment types are foul flow organic sediments, and fine and course mineral in-pipe sediments originating from catchment runoff, potholes and stilling basins. The following can be accounted for with this module:

- Decay of BOD/COD in bio-film and water phase
- Hydrolysis of suspended matter
- Growth of suspended biomass
- Oxygen consumption from decay of BOD/COD, bio-film and erosion of sediment
- Re-aeration
- Bacterial fate
- Interaction with sediments for nutrients and metals

Sediment Transport

This module is used for sediment transport modelling in pipes and river networks. It comes in a basic version tailored primarily for application to pipe networks, and an advanced version giving full access to all its modelling capabilities suited for e.g. long term assessment of river morphology changes. It can include various model types (e.g. van Rijn, Meyer-Peter & Muller, Engelund-Hansen, Engelund-Fredsoe, Yang, or user-defined empirical formulas). Graded or mixed sediment descriptions can be applied by defining a number of different sediment fractions, which are treated separately by the sediment transport module. Sediment transport is computed from hydrodynamic conditions, and dynamic changes in the river morphology can in return affect the hydrodynamic conditions.

1.7 SWMM collection systems

MIKE+ includes the SWMM (EPA's Storm Water Management Model) engine for storm water modelling. SWMM allows for the hydrodynamic simulation of flows and water levels in 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. The simulations can be carried out for single events.

1.8 Demo limitations

Creating and editing model setups requires a valid license for the relevant modules of MIKE+. Without a valid license, MIKE+ will run in demo mode and will allow creating model setups with the following limitations:

- 'Rivers, collection system and overland flows' mode:
  - 15 nodes
  - 10 pipes and canals (40 grid points)
  - 1 river (40 grid points)
  - 1 structure (expect Dambreak structure which is not allowed)
  - 1 control rule
  - 10 catchments
  - 2 inflow boundary conditions on the network
  - 2 Q/h boundary conditions on the network
  - 2000 wet nodes in 2D domain
- SWMM collection systems:
  - 15 nodes
  - 10 conduits
  - 2000 wet nodes in 2D domain
- Water distribution network:
  - 15 junctions
  - 10 pipes.

Demo mode is valid for hydrodynamic simulations only. Additional modules (water quality, sediment transport, etc.) are not allowed.

Results viewing is allowed with no limitation.

2 Getting Started

2.1 How to Start MIKE+

During the installation of MIKE+, a program shortcut is placed in the Programs' section of the Windows 'Start' menu (found under MIKE+ 20XX). Pin to the taskbar and/or the start menu for quick access. You can also choose to create a MIKE+ icon on the Desktop and launch MIKE+ from this by creating a shortcut to DHI.MIKEPlus.Shell.exe (found in the bin-directory of your MIKE+ installation).

Open the program and explore the MIKE+ user interface. An example is shown in Figure 2.1 below. Note that it is possible to have multiple instances of MIKE+ opened in one session.

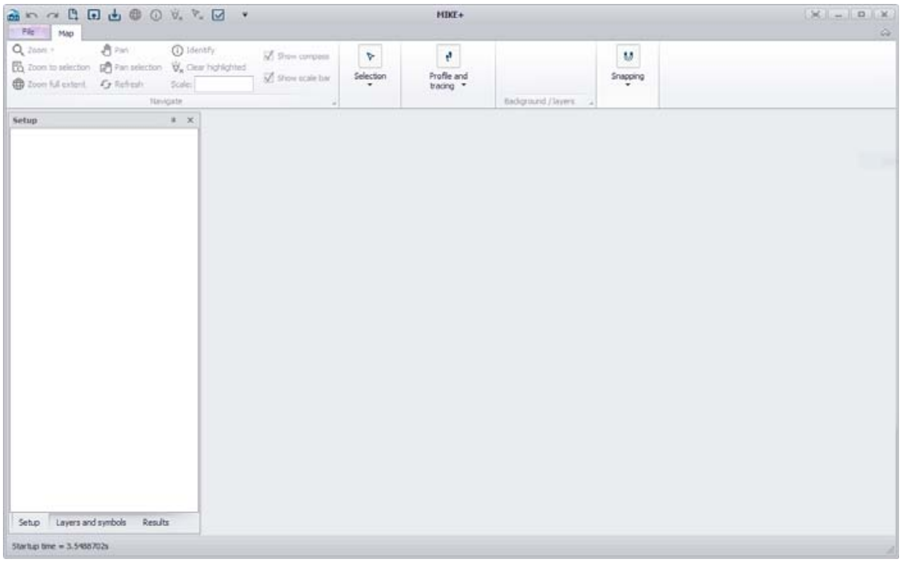


Figure 2.1 MIKE+ desktop

Once MIKE+ is opened, you can create a new project following the steps below:

1. Go to File | New (or click the 'New' icon in the Quick Access Toolbar at the top of the window)
2. In the dialog that appears, select the expected model type. Note that this type is used to control the list of editors and tools that will be visible, and this model type can be changed from the 'Model type' editor once the model database has been created.

3. Select the desired unit system. This unit system can also be modified later from the 'Model type' editor. Refer to 3.1.2 Customizing Unit Environment *(p. 128)* for more information.
4. Select a database type (SQLite or PostGIS), file path for the database and file path for the *.mupp project file.
5. In the second tab 'Coordinate System', select a coordinate system (refer to 2.12 Selecting a Coordinate System *(p. 115)*).
6. Once a coordinate system has been selected, an extra tab will appear to optionally add a background layer (Open street map, Google map, WMS server or country/coastline boundaries). Select the wished option.
7. In the 'Description' tab, add a title and description for the project.
8. Click 'OK' to create the database with the specified settings.

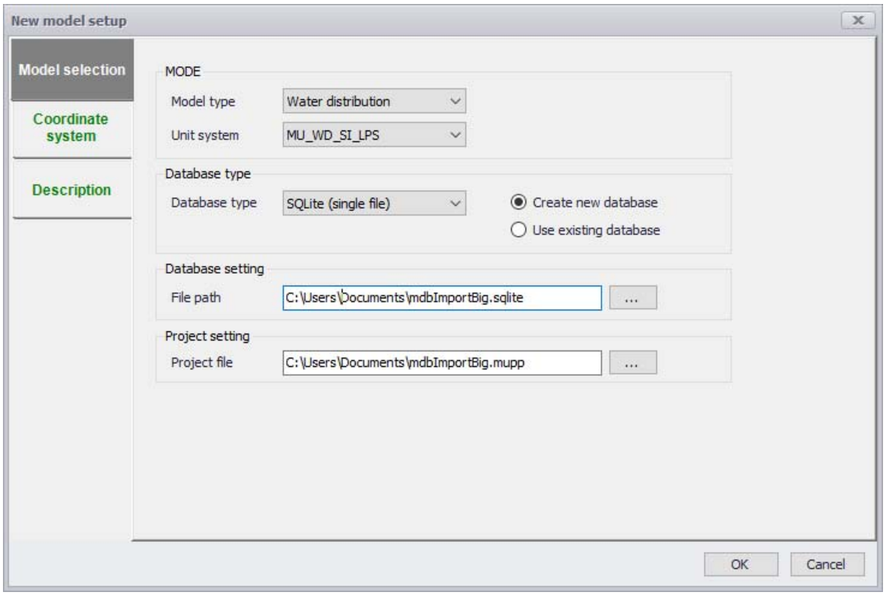


Figure 2.2 Creating a new model database

You are now ready to start entering data into your model. This can be done by typing data manually into the various editors, importing data by connecting to an external data storage, graphically digitizing data on the map, or a combination of all three methods.

All changes will be saved directly (automatically) to the database, but with an unlimited number of 'undo' and 'redo', within the current session (up to when the MIKE+ model was opened). 'Undo' enables changes to be 'undone' in the order they were entered. 'Redo' will redo the changes in the order they were 'undone'. 'Undo' and 'Redo' are available in various tabs in the ribbon, or using the shortcuts Ctrl + Z (undo) and Ctrl + Y (redo).

2.2 Map Window

The main Map window displays a layout plot of the pipe network system. The individual model elements (i.e. nodes, pipes, pumps, etc.) are displayed. The Map window also allows the user to graphically layout the pipe network system.

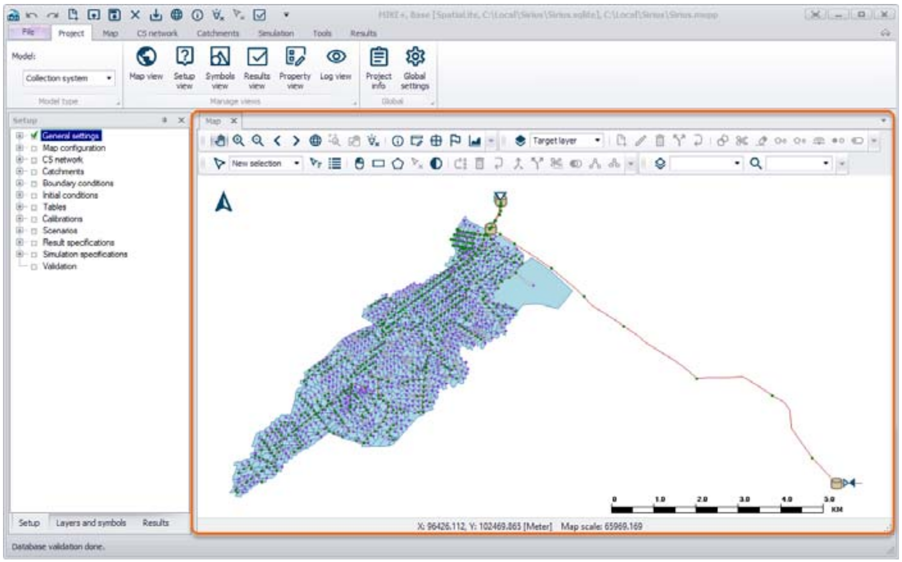


Figure 2.3 Map view of the model with default symbology

The map is per default “docked” but can be “floated” (right “click” on the tab heading + Float). The map can be brought into view by clicking on the “Map” tab or the “Map View” button on the Project menu ribbon.

2.3 Editors

Model setup editors are accessed from the Setup tree view to the left of the main window. Clicking on a setup item opens a new tab with an editor related to the item.

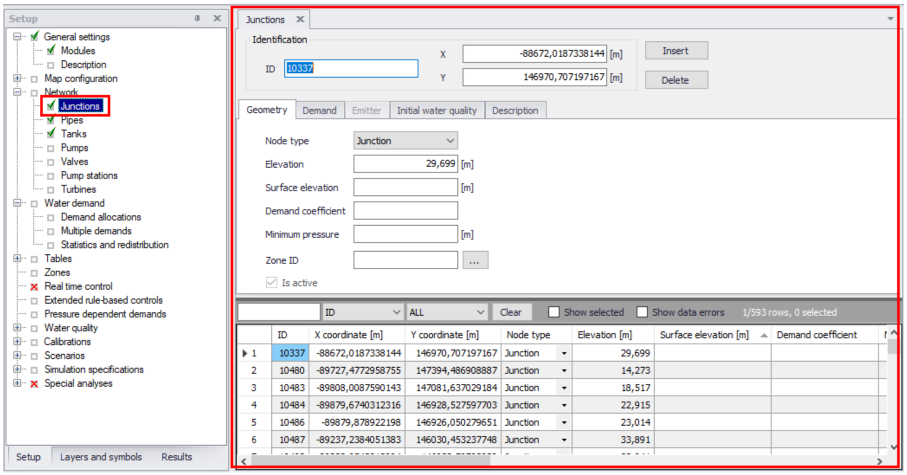


Figure 2.4 Example Editor (for WD model Junctions) shown on the main window

Editors in MIKE+ may be "docked" or "floated":

- When "Floated", editors are displayed in a stack, with the active editor on top. This is shown in Figure 2.5.
- “Docked" editors are displayed one at a time, or side-by-side. Any editor can be brought to the front by clicking on its tab. This is shown in Figure 2.6 below.

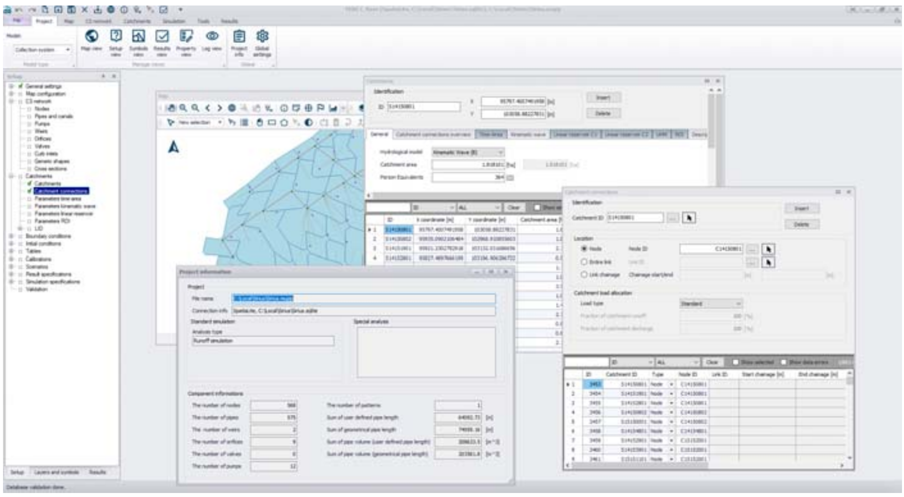


Figure 2.5 MIKE+ user interface with "floating" editors

Figure 2.6 shows the MIKE+ user interface with all editors "docked". In this case only the active editor is visible.

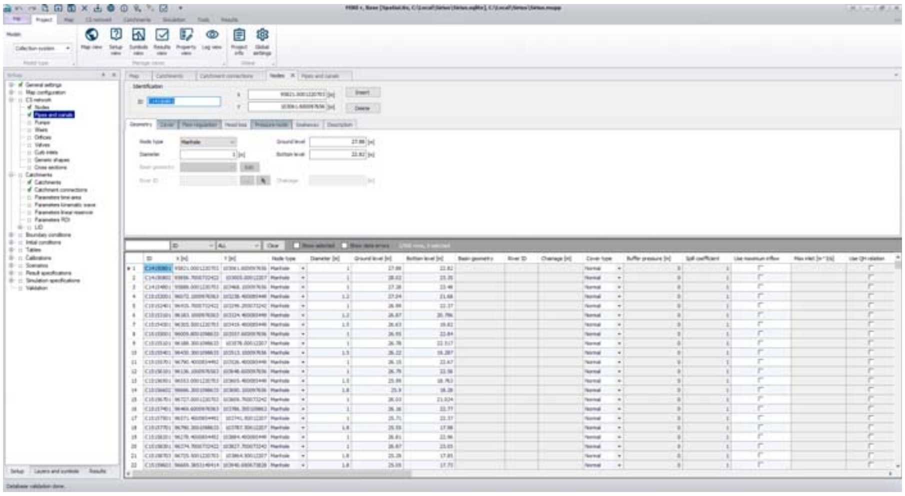


Figure 2.6 MIKE+ user interface with all editors "docked"

Most of the editors contain an overview table at the bottom, which offers a number of functionalities as described below:

- A search field above the table allows to filter the table, in order to show only the relevant items in the list (e.g. show only the nodes with a specific type, or show a specific ID, etc.). Type a text to show only the records with the selected field (e.g. ID) starting with this text. Type * followed by a text to show only the records with the selected field ending with this text. Type a text between two * to show all records containing this text.
- Check boxes above the table allows to show only the selected records, or show only the records with validation errors
- Double-clicking a row number will zoom to the corresponding item / record on the map. Pressing the Ctrl key while double-clicking will pan to the corresponding item / record on the map (keeping the map scale unchanged).
- Right-clicking in the header of a column, it is possible to either select the entire column (to later copy its content), start the ‘Field calculator’ or start the ‘Select by attributes' tool
- Right-clicking within the table offers several options to:
  - Copy and paste data
  - Manage selections
  - Add user defined columns
  - Clone (duplicate) selected rows
  - Show in the table the columns from the active tab only
  - Show the table only, hiding the part of the editor above the table.

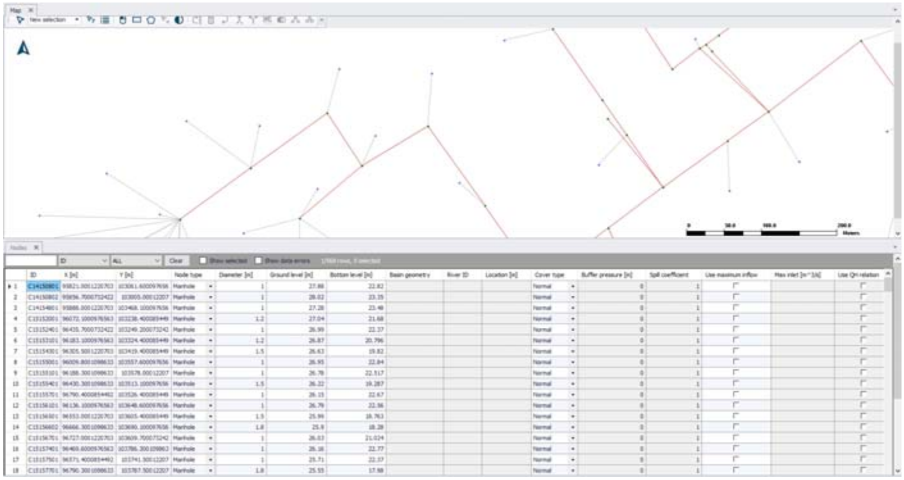


Figure 2.7 An editor with option 'Show grid only' active, and docked below the map

2.4 Status Line and Tooltips

As you move your cursor on the map, the status line will display the coordinates and map scale.

Also, hovering your cursor over an edit field in an Editor will display a tooltip with the name of the table and attribute name where data for that field is saved (Figure 2.8).

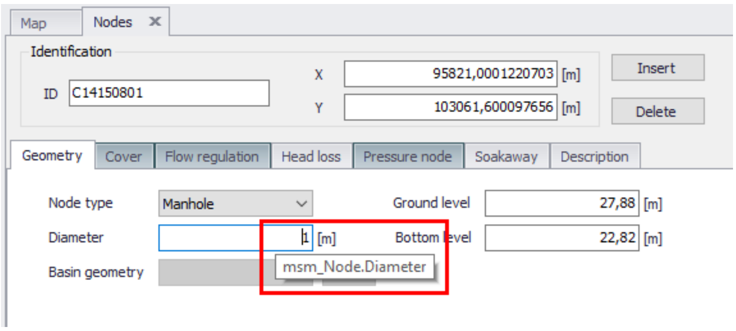


Figure 2.8 Example tooltip shown for the Node Diameter, indicating that in the database, the data is contained in the 'msm_Node' table under the attribute 'Diameter'

2.5 Identify

Use the Identify tool to view information about a feature displayed on the Map. It is accessed from the Map ribbon, the Map view toolbar, or the result map toolbar.

The Identify tool allows you to see the attributes of your data. Clicking the Identify tool on a location inside a data frame will display the attributes of the element at that location. The Identify tool is the easiest way to learn about something on a map.

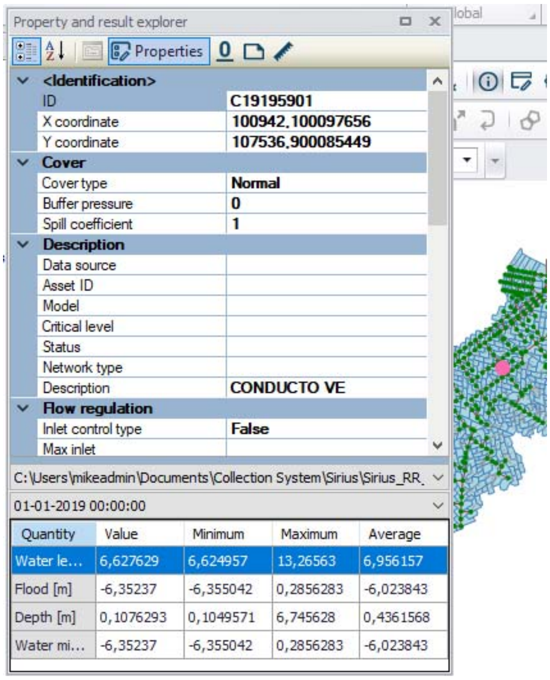


Figure 2.9 The identify tool displays information on the element chosen on the map

2.6 Online Help F1

Online help is available for MIKE+. The MIKE+ help system utilizes the Microsoft help technology known as HTML Help.

The Help system can be accessed by pressing F1 from any location on the interface. The relevant online help page will be displayed in the active window.

MIKE+ context help is viewed in the HTML Help Viewer (see Figure 2.10) which consists of:

- Topic Pane: Where the help topics are viewed.
- Navigation Pane: Where you can navigate through the Help file. Index and search for instances of e.g. a word are possible.

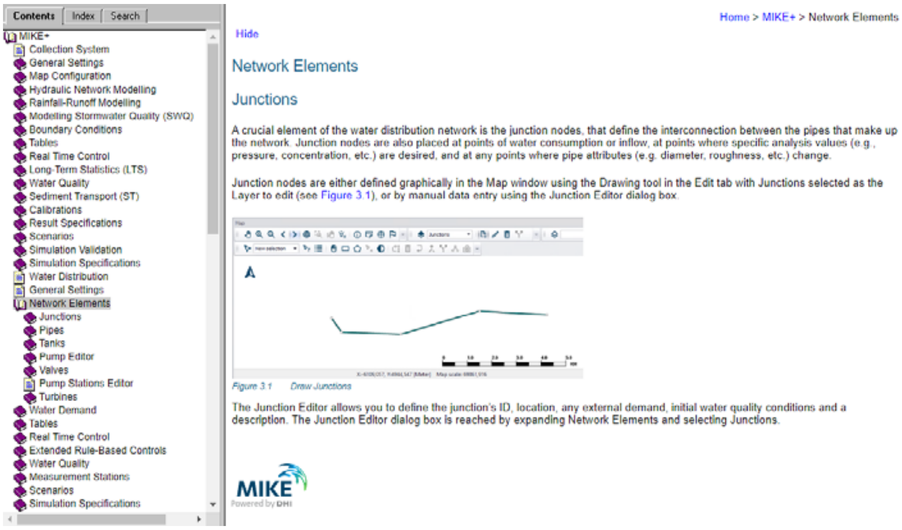


Figure 2.10 The online context-sensitive Help function in MIKE+

2.7 MIKE+ Examples

MIKE+ includes several examples demonstrating how to use the various modules. These are initially placed in the Program Files directory.

The examples can be installed/copied from here to a user-defined location by going to File| Install Examples, see Figure 2.11. It is possible to choose which examples to install/copy.

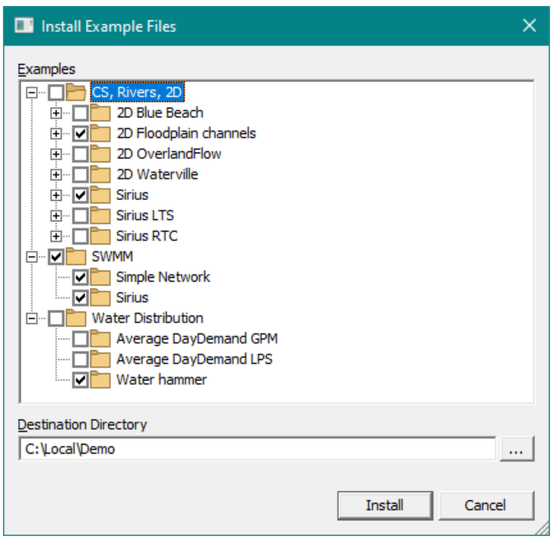


Figure 2.11 Installing/copying the examples to a different folder

2.8 View Panels

MIKE+ includes four main panels by default to the left of the window:

- **Setup View**. Tree structure with access to the data editors, where all data can be edited in forms or in tables. This tree view provides an overview of data validation to the model components by showing a green tick or a red cross next to each item. A red cross indicates that some records contain errors: open the corresponding editor to get more details on the error.
- **Layers and Symbols View**. Lists the symbols and layers used in the Map. Allows you to configure graphics and model components symbols.
- **Results View**. Lists all loaded result files in the project. Used for result presentation. The following buttons at the top are available to manage the files:
  - Add file: this button allows to select new files to load in the project. While loading a result file, a window allows to select which result items from this file are to be loaded in memory, and which ones are to be shown as a layer on the map. Note that a result layer can be added to the map only if the corresponding result item is loaded in memory. If a result item is loaded but not added to the map at the same time, it can be added to the map at a later stage from the 'Layers and symbols' view.
  - Add folder: this button inserts a folder to the tree structure to organize the list of result files.
  - Remove: this button removes all the selected result files. Multiple files can be selected from the list using the Ctrl key, and selected files are highlighted in blue. An alternative option to remove all result files is available in the context menu (right-click on the list of files).
  - Refresh: this button re-loads the result file.
  - Properties: this button shows the result items from the result files which are currently loaded into the project, and allows to change the list of loaded items.
  - Compare: this button compares two result files obtained from two different simulations but on the same network. Refer to Result Comparison (p. 482) for more information.
- **Plots View.** Allows saving and organizing all types of result presentation windows, like time series plots, profile plots, results tables, result maps, etc. Refer to Plots Management (p. 487) for more information.

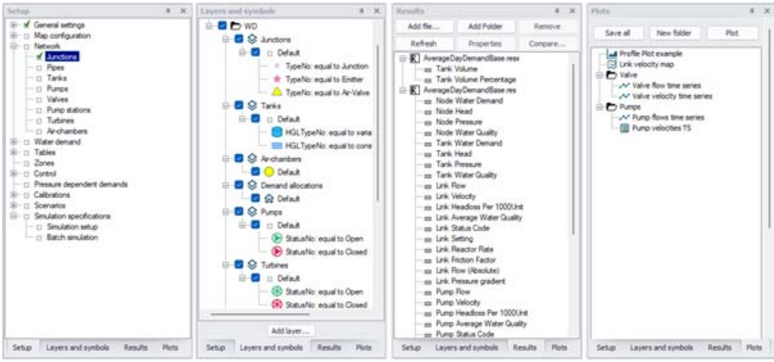


Figure 2.12 Setup, Layers and Symbols, and Results panels in MIKE+

A 'Property view' can also be shown, especially to display the properties of an item selected on the map with the 'Identify' button. It is by default opened on the right-end side of the screen. It can display properties from model data and from other layers added to the map, like elevation from a DEM at the selected location, attributes from a selected item from a feature layer, or result values. It contains a 'Layer' list at the top to control which layer to identify, using the following options:

- Top-most layers: this option displays the properties of the item at the selected location from the first visible layer on the map, according to the order defined in the 'Layers and symbols' tree.
- Visible model layers: this option displays properties only from the model data.
- Or by selecting a specific additional layer (DEM, result file, or feature layer) from the list.

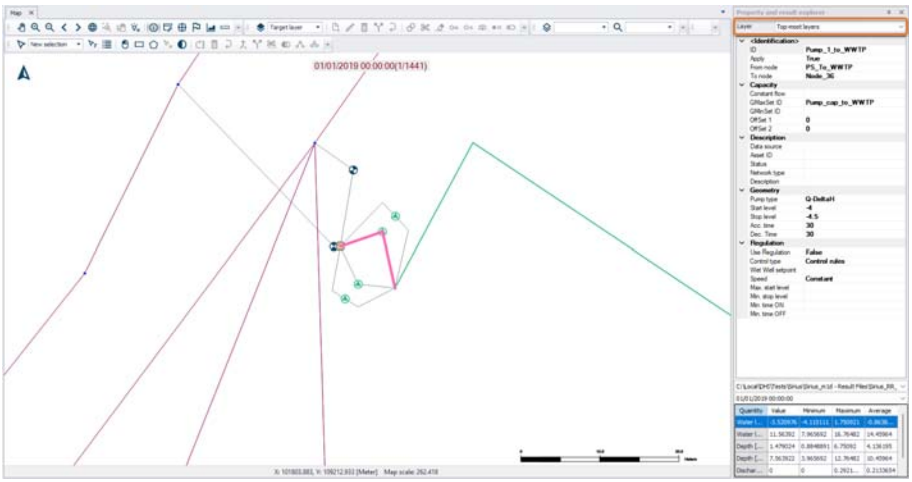


Figure 2.13 Controlling the source layer for the Identify tool

When displaying properties of the model layers, the Property view can be an alternative way to edit the data (instead of the main editor opened from the Setup view), which can easily be used side-by-side with the Map view. When 1D result files are loaded, this Property view also shows the results for the selected item.

When selecting the option 'Visible model layers', additional properties are available, depending on the selected button:

- **Properties**. This shows the main information from the Property view, i.e. it shows the various data for the item picked on the map using the 'Identify' button, e.g. geometrical and hydraulic properties.
- **Default values**. When this button is selected, the Property view shows the default values (used for future items to be created) for the item type which has been selected. For example, while editing nodes, this view will show the default values that will be applied when new nodes will later be created. These 'Default values' are common for all data in a given table, i.e. the values displayed in that mode are not specific to the selected item.
- **Status**. When this button is selected, the Property view shows the attribute's status. Each attribute (property) for each record (item) can store a status information. This can e.g. be used to keep track of updates or to qualify the data.
- **Enum info**. When this button is selected, the Property view shows the unit type (e.g. Water level) and the unit (e.g. [m]) used for each attribute in the edited table. This is especially useful if a different unit should be applied, in order to identify which unit type should be edited in the 'units customisation' dialog. These 'Enum info' are common for all data in a given table, i.e. the information displayed in that mode is not specific to the selected item.

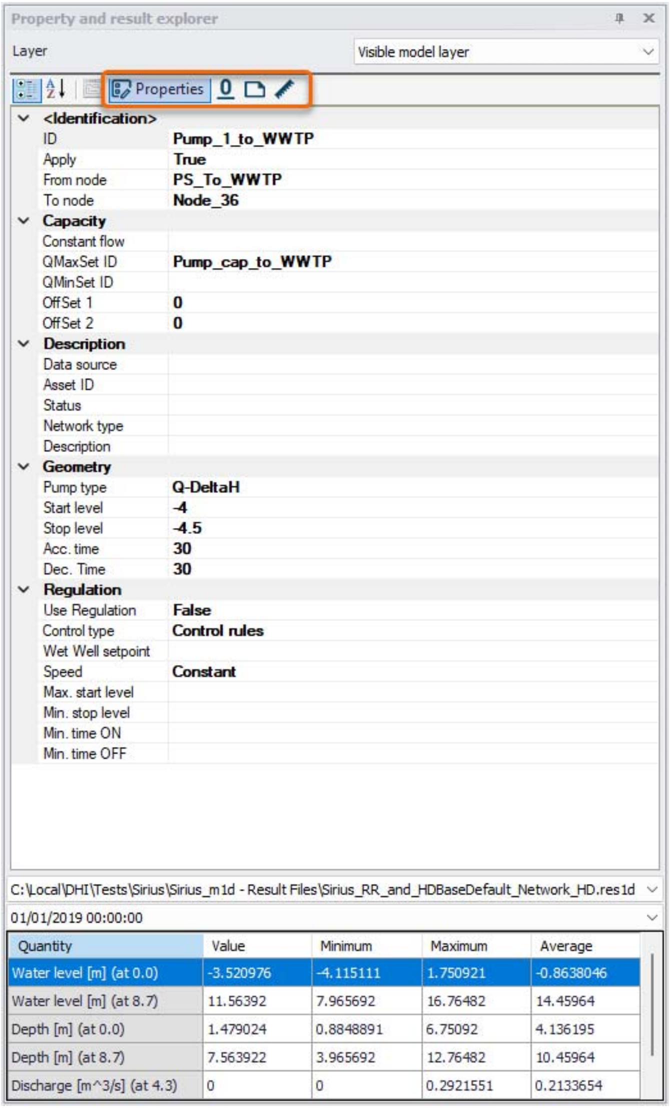
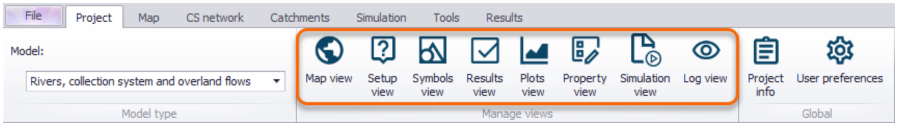


Figure 2.14 Model data in the Property view. Buttons in the rectangle control the displayed information.

A 'Simulation view' can be used to display the various messages reported by the simulation engines while executing a simulation. This view is automatically opened when starting a new simulation.

A 'Log view' can be used to display various warnings and errors reported by MIKE+, for instance while using one of the predefined import and export routines, or when attempting to load invalid data on the map.

All these various Views can be opened via the ‘Manage Views’ toolbox on the Project tab in the ribbon, after they have been closed.

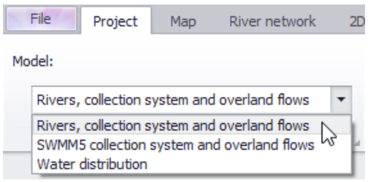


2.8.1 Working Modes

MIKE+ can operate in three different modes:

- Rivers, collection system and overland flows
- SWMM5 collection system and overland flows
- Water Distribution

The working mode can be selected from the 'Project' tab in the ribbon, or from the 'Model type' menu in the Setup panel.



Choosing a specific working mode affects the visible layers on the Map view. However, regardless of the selected working mode, any layer contained in the database can be displayed by ticking appropriate group and layer check boxes in the Layers and Symbols View.

Alternatively, use the button 'View CS network' in the WD network tab in the ribbon, or the 'View WD network' in the CS network tab, which will also make the corresponding data layers visible on the map.

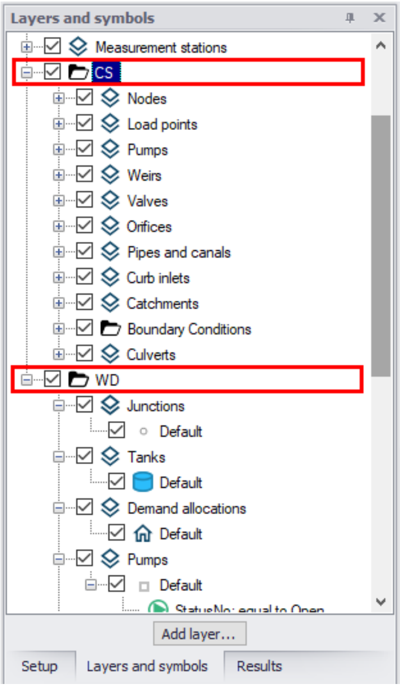


Figure 2.15 Working modes in the Layers and Symbols tree view

2.8.2 Map Layers

Various types of data layers can be shown on the Map view:

- Background map: a background overlay, mainly from online sources, can be selected from the 'Background map' editor. Refer to the 'Background Map' editor description for more information.
- Model data layers: these layers show the model data, which are added in the various editors from the Setup View. The list of model data layers shown on the map is controlled by the working mode and the active features of the project, as defined in the 'Model type' editor.
- Result layers: when a result file is available in the Results View, its results can also be shown on the map using dedicated layers. Result layers are by default added automatically on the map at the end of a simulation. They can also be manually added using the 'Add layer…' button in the Layers and Symbols View or in the Map tab of the ribbon.
- External layers: additional layers from files containing GIS data can also be added to the Map view. They can be added using the 'Add layer…' button in the Layers and Symbols View or in the Map tab of the ribbon.

Note: automatic addition of result layers at the end of simulations can be disabled from the 'User preferences' dialog.

The following types of external layers can be added to the map:

- Shape files (*.shp): a file containing either points, lines or polygons. Select the 'Feature layer' type in the 'Add layer' window, to select this type of file.
- XYZ files (*.xyz): a text file containing scatter points, which can be used as input topography information for interpolating elevation on a 2D overland domain file. Select the 'Feature layer' type in the 'Add layer' window, to select this type of file.
- CAD files (*.dwg, *.dxf): a file containing various types of drawings. Select the 'Feature layer' type in the 'Add layer' window, to select this type of file.
- Geodatabase (*.gdb): a database containing feature layers. After selecting the database, the list of feature layers from the database to be displayed on the map needs to be selected. Select the 'Feature layer' type in the 'Add layer' window, to select this type of file.
- BIM files (*.ifc): a Building Information Modelling file containing buildings drawings. After selecting the file, the list of feature layers from the file to be displayed on the map needs to be selected. A number of settings must also be specified to properly locate the file on the map: Coordinate system, Reference point location (identification of a reference point in the file), Reference point projected coordinates (location on the map of the identified reference point, expressed in the selected coordinate system), Rotation from true North (measured anticlockwise, from the reference point), Scale factor (e.g. used when the unit in the file is millimeters instead of meters). The specified settings are automatically saved to a *.xml file with the same name as the *.ifc file, and will be automatically reloaded the next time the *.ifc file is added to MIKE+. Select the 'Feature layer' type in the 'Add layer' window, to select this type of file.
- dfs2 files (*.dfs2): DHI proprietary file format for raster files, containing e.g. input topography or results from a 2D overland model. Select the 'Raster layer' type in the 'Add layer' window, to select this type of file.
- Raster text files (*.txt, *.asc): text file format for raster files, typically containing DEM data. Select the 'Raster layer' type in the 'Add layer' window, to select this type of file.
- Arc/Info Binary Grid: file format typically containing DEM data. Select the 'Raster layer' type in the 'Add layer' window, to select this type of file.
- TIFF files (*.tif, *.tiff): file format used for regular images or raster data (typically DEM data). To add a DEM or another type of raster file, select the 'Raster layer' type in the 'Add layer' window. Such raster layers must also store geographical coordinates information as metadata in the file (also referred to as GeoTIFF file). To add a regular image file, select the 'Image layer' type in the 'Add layer' window. Geographical coordinates from TIF images can either be read from the file's metadata (GeoTIFF file), defined manually or read from a world file (see page 117).

- Mesh files (*.dfsu, *.mesh): DHI proprietary file format for flexible mesh files, containing input topography (*.mesh files) or results (*.dfsu files) from a 2D overland model. Select the 'Mesh layer' type in the 'Add layer' window, to select this type of file.
- Bitmap files (*.bmp): image file. Select the 'Image layer' type in the 'Add layer' window, to select this type of file. Geographical coordinates from BMP images can either be defined manually or read from a world file (see page 117).
- JPEG files (*.jpg, *.jpeg): image file. Select the 'Image layer' type in the 'Add layer' window, to select this type of file. Geographical coordinates from JPEG images can either be defined manually or read from a world file (see page 117).
- PNG files (*.png): image file. Select the 'Image layer' type in the 'Add layer' window, to select this type of file. Geographical coordinates from PNG images can either be defined manually or read from a world file (see page 117).

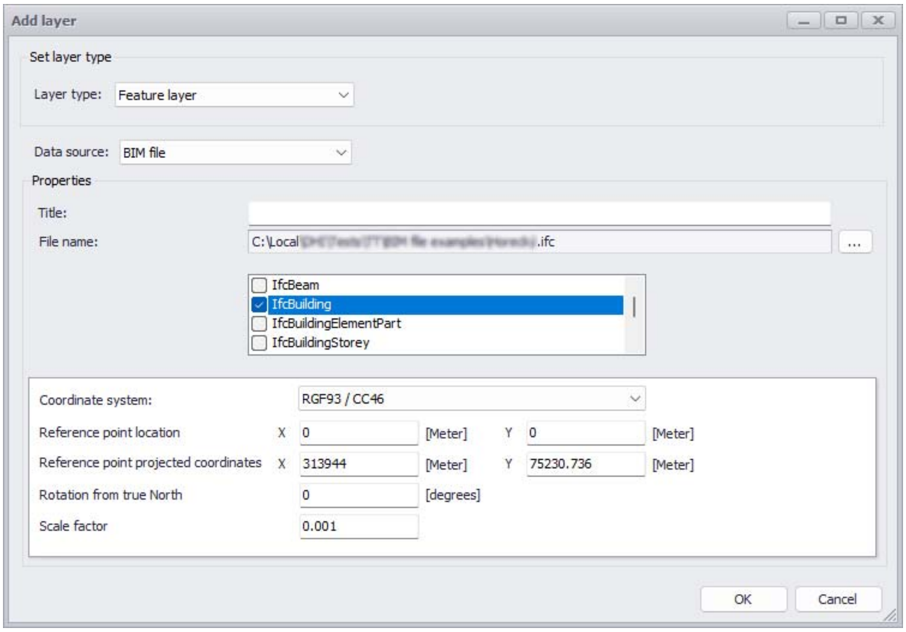


Figure 2.16 Providing settings to add a BIM file layer on the map

2.8.3 Boundary Conditions Displayed on the Map

Boundary conditions are per default displayed on the Map. To be displayed, boundary conditions must be applied and contain at least one 'Boundary Item'.

To ensure the Map view reflects all recent boundary condition changes, access the Map local context menu (i.e. right-click) and select the 'Refresh boundary visualization' option.

Collection System

The different boundary conditions that can be visualized for collection system networks are seen in Figure 2.17.

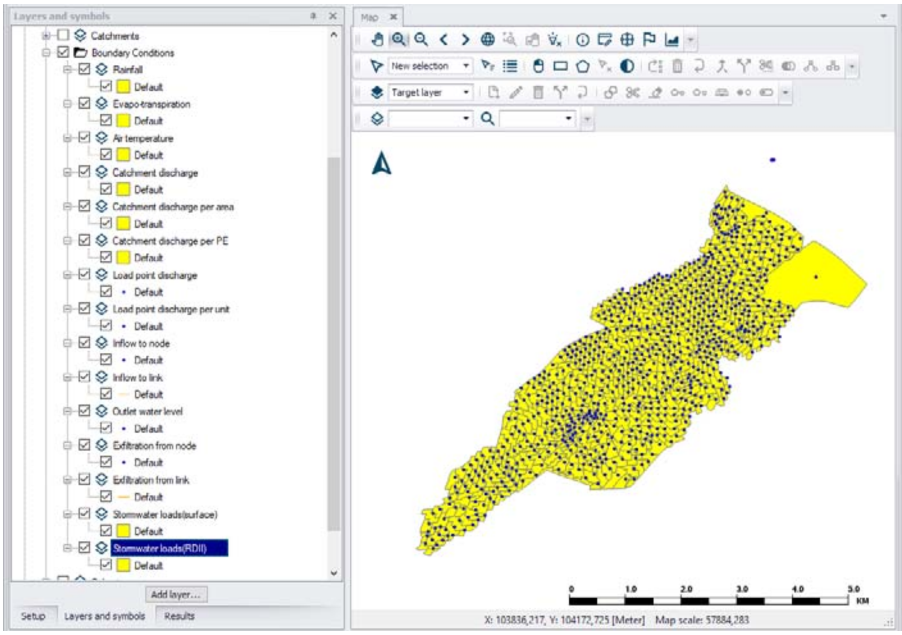


Figure 2.17 CS network boundaries

For further information on Collection System Boundary Conditions, please refer to the relevant chapter in the MIKE+ Collection System User Guide.

Water Distribution

Node demands can be displayed by ticking the Water Node Demands layer. Per default, the different demand categories will be differentiated when displayed.

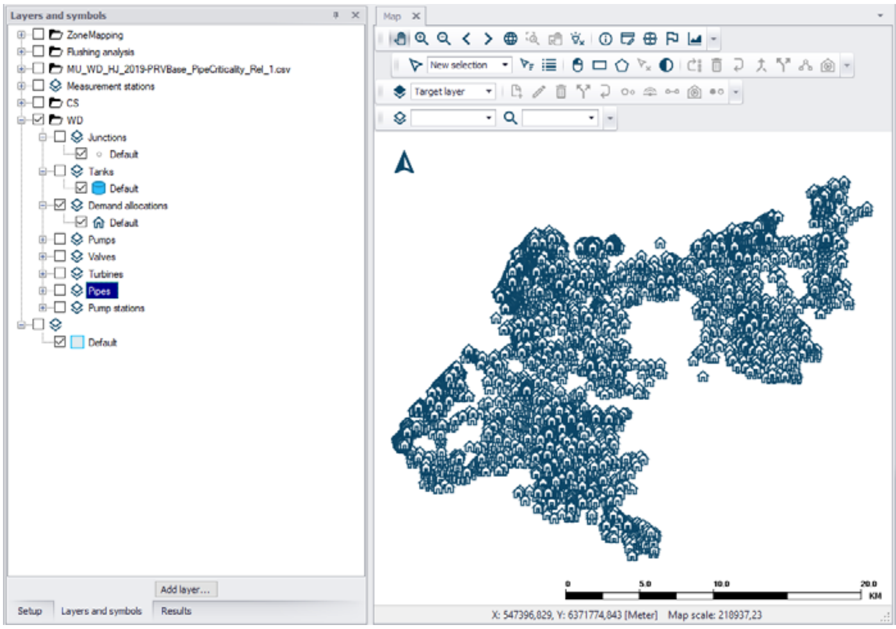


Figure 2.18 WD demand allocation points

2.8.4 Symbology settings

Labelling and symbology for layers on the main Map View may be customized via the Symbology Settings dialog.

On the Symbols and Layers panel, click on a layer item. This will open the Symbology Settings editor (Figure 2.19). Note that this editor functions similarly as the Edit style dialog from the result map window.

Set parameters inside the Symbology and Label tabs to customize the appearance of the layer on the main Map.

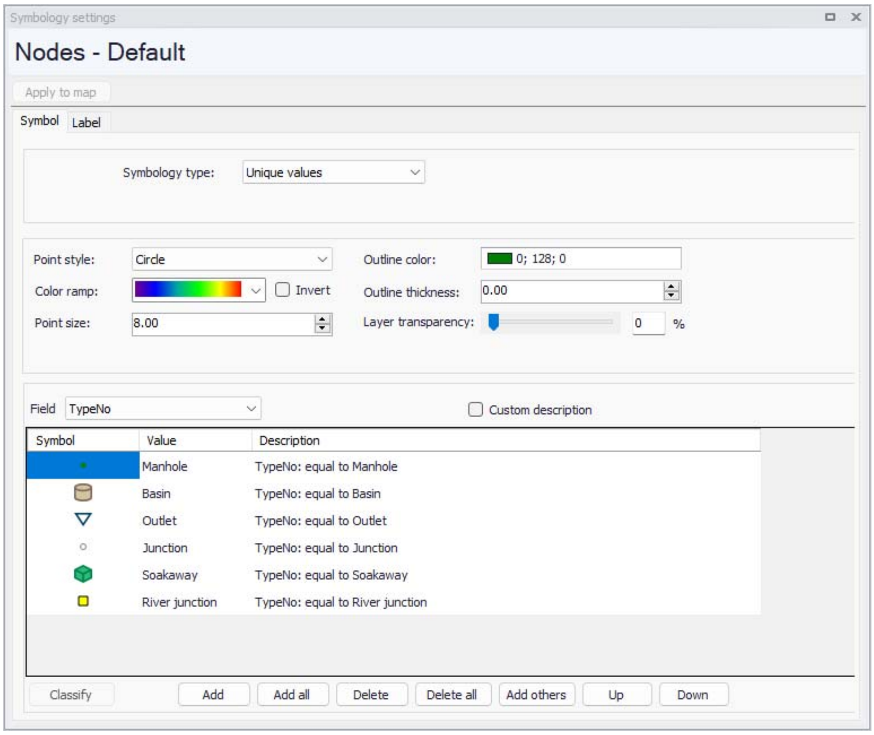


Figure 2.19 Symbology Settings dialog from the Symbols and Layers view panel

Table 2.1 Options on the Symbols tab

Item	Description	Usage
Visible	Checkbox for showing or hiding the layer on the plot	Yes
Symbology type	Dropdown menu for selecting symbology type: - Single symbol - Graduated color - Graduated size - Unique values - Range values	Yes
Point style	Dropdown menu for selecting point symbol type	If point layer
Fill color	Color for point symbol	If point result layer and Symbology type = Graduated size
Symbols size from _ to _	Minimum and maximum range of symbol size to use	If Symbology type = Graduate size

Table 2.1 Options on the Symbols tab

Item	Description	Usage
Line style	Dropdown menu for selecting line symbol style	If line layer
Color ramp	Dropdown menu for selection of color ranges to use in symbolizing values	If Symbology type = Graduated color
Invert	Checkbox for inverting the application of the color ramp to the range of values	Yes
Point size	Point symbol size	If point layer
Line thickness	Line symbol thickness	If line layer
Outline color	Symbol outline color	Yes
Outline thickness	Symbol outline thickness	Yes
Draw direction arrow	Checkbox option for showing direction arrows. For pipes or rivers, this will show the direction of the link. For result layers, it can show the flow direction.	Yes
Position of arrow	Position of the arrow along the link geometry: - Mid vertex - End vertex	If Draw direction arrow = Active
Layer transparency	Slider for controlling the transparency of the layer on the map	Yes
Custom description	Checkbox for allowing customization of the symbology descriptions	Yes

The dialog allows for changing symbol size, color, and value classes. E.g. one may wish to have links displayed by color, applying four different colors over the range of values.

Use the 'Classify' button in the dialog to define the number of classes to use and the break values for each class.

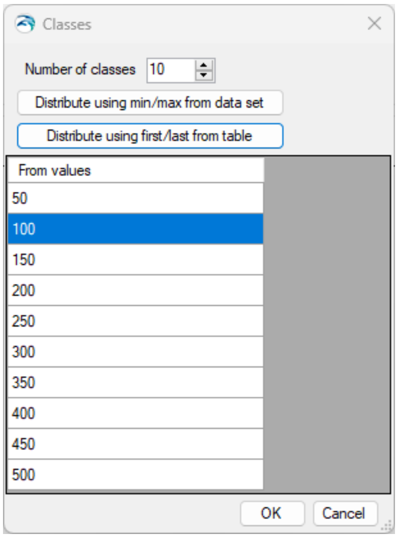


Figure 2.20 Customizing value classes for the symbology

In this classification window, the 'Distribute using min/max from data set' button will distribute equidistantly the specified number of classes using the minimum and maximum values of the feature layer being edited, for the field being classified. The 'Distribute using first/last from table' button will distribute equidistantly the specified number of classes using the first and last custom values specified in the table in this classification window.

Also see Chapter 20.9 Labelling and Symbology *(p. 393)* to edit symbology settings from result maps.

Symbol

Change layer symbology settings on the Symbol tab.

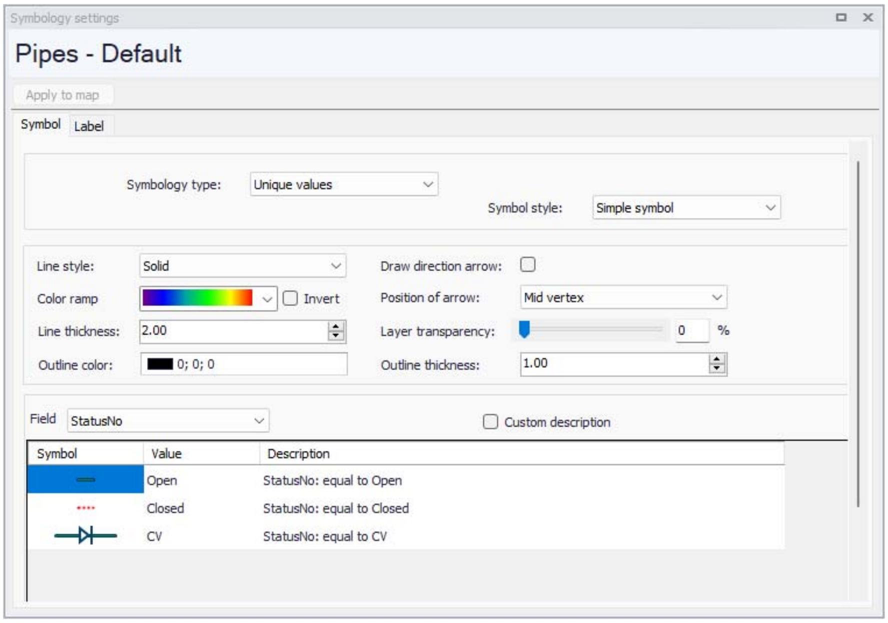


Figure 2.21 The Symbol tab

Label

Add labels to map result plots through the Label tab on the Edit Style dialog. The position of the label, font label, number of decimals displayed and when to display labels can be set.

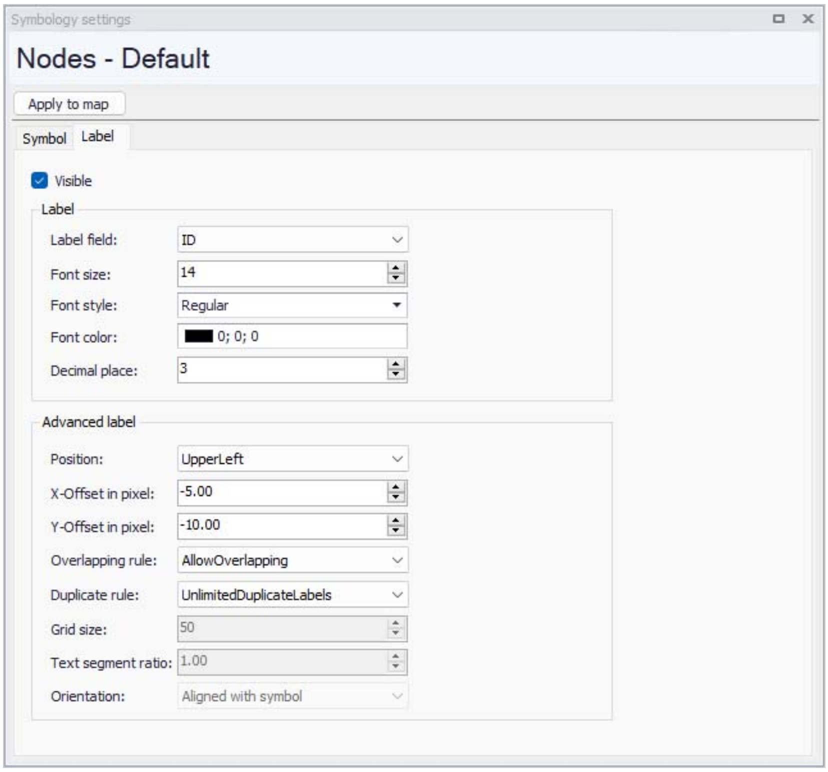


Figure 2.22 The Label tab

The following settings can be used to control the labels on the map:

- Visible: Check box for showing or hiding the labels on the map
- Label field: Parameter's values to display in the labels. For a result layer, the only available field is the time series value.
- Font size
- Font style
- Font color
- Decimal place
- Position: The location of the labels, relative to the location of the corresponding features on the map.
- X-Offset in pixel: Offset of the labels along the X-axis, relative to the specified 'Position'. A positive value will move the labels on the right, and a negative value on the left.
- Y-Offset in pixel: Offset of the labels along the Y-axis, relative to the specified 'Position'. A positive value will move the labels upward, and a negative value downward.
- Overlapping rule: Controls whether overlapping labels are allowed or not.

- Duplicate rule: There are three options to handle duplicate labels:
  - No duplicate labels: this will remove all duplicates, i.e. only one label will be kept.
  - One duplicate label per quadrant: this will remove duplicate labels only if they are in the same quarter of the screen. The screen will be divided into four quadrants, and when two duplicate labels are in different quadrants, they will both be kept.
  - Unlimited duplicate labels: this will keep all duplicates.
- Grid size: The grid size determines how many labels may be shown on the map. The smaller the grid size, the higher the density of labels.
- Text segment ratio: This allows removing labels where the label length would greatly exceed the line length. It is a maximum ratio between the label length and the line length, above which the label is not shown. For example, when the ratio is set to 1, then the label will be suppressed if it is longer than the line. If the ratio is lower, then the label will be shown only if it is shorter than the line. If higher, then the label may be shown even if it is longer than the line.
- Orientation: For line layers, the labels are by default aligned with (i.e. parallel to) the feature line on the map. This option can be used to force all labels to be horizontal.

In order to apply a custom label, first create a user-defined column in the layer's editor (e.g. an expression column, which can be a function of other fields) and then select this user-defined column as label field.

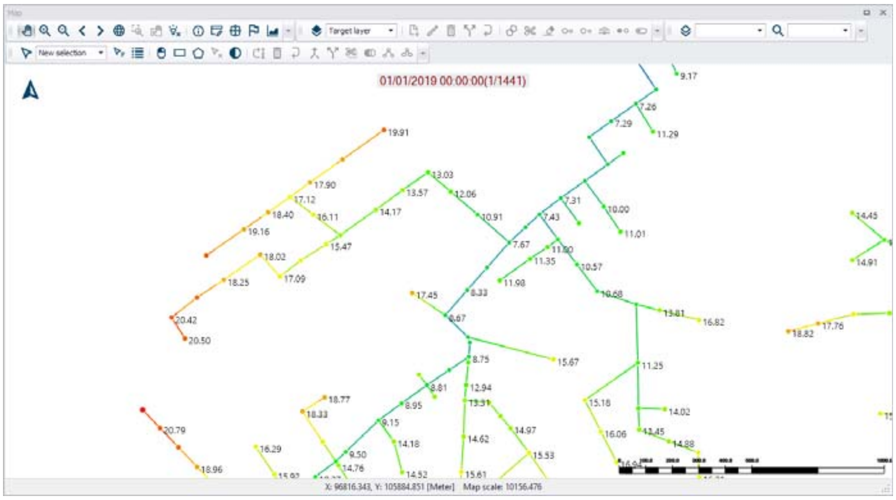


Figure 2.23 Example label configuration showing max.water level at nodes

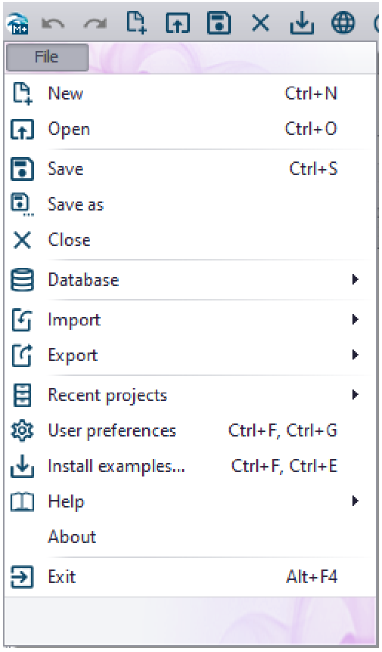
Also see Chapter 20.9 Labelling and Symbology *(p. 393)* to edit symbology settings from result maps.

2.9 Main Ribbon Menus

MIKE+ offers several menus and tools to ease the workflow and simplify the user interface.

2.9.1 File Menu

On the File menu it is possible to create, open, save and close MIKE+ projects. The menu provides access to import and export functionality of multiple model formats, as well as access to recently opened projects and more.



New

This allows you to create a new project/database. It is required to name the project, choose the preferred directory, model type, database type and coordinate system, as shown in Figure 2.24.

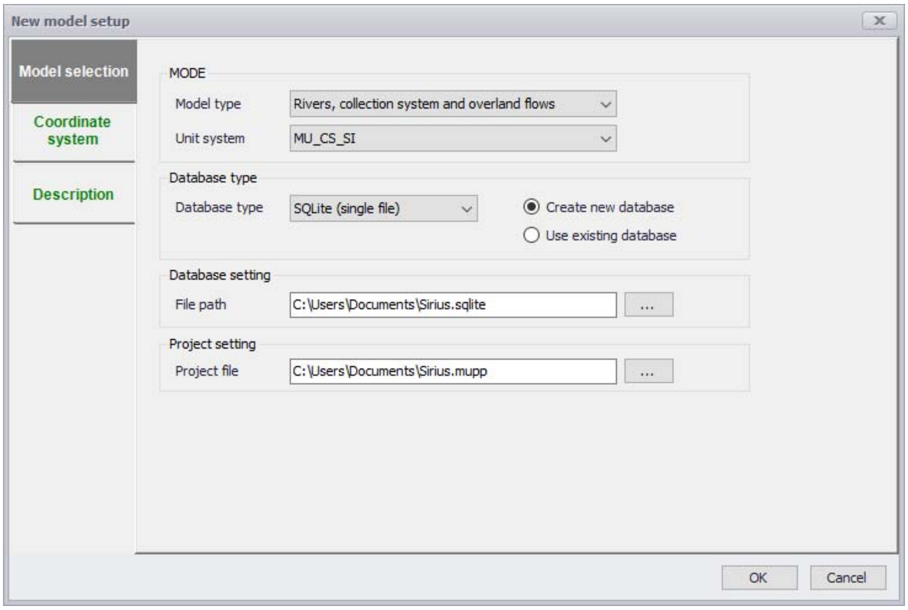


Figure 2.24 New model setup

The option 'Create new database' will create both a database, containing the model data, and the related *.mupp file, which stores information about the layout (location of the various opened windows) and the map (link to additional background layers, symbology). The option 'Use existing database' will only create a new *.mupp file, for use with an existing database: this is e.g. required in order to open a model database if the *.mupp file is missing.

Open

Opens an existing MIKE+ project file (*.mupp) and its related database. It is also possible to select a database file (*.sqlite) directly: if a *.mupp file with the same name exists in the same folder, it will be used to open the project and load all its layout settings, and if no *.mupp file with the same name is found a new one will automatically be created.

Save

Saves MIKE+ project file (*.mupp) only. Note that changes applied to the model data are automatically and continuously saved to the database, so the Save button is only used to save the windows layout, the list of background layers added to the map, or the symbology applied to the map.

Save As

Save a copy of the model as a new copy

Close

Close the model without closing the MIKE+ application

Database | Clone Database

The option to clone a database is used to copy a source database into a new target database. It can therefore be used to change the database format (SQLite to PostGIS database type, or vice versa).

It can also be used to recover from damaged tables in the database. While cloning, corrupt table(s) will be skipped and the cloned database may be usable again in MIKE+. However, the data from the skipped table(s) will need to be re-imported afterwards. If the skipped table is a "system" table, which is strictly required by MIKE+, then the cloned database may not open properly. System tables e.g. include:

- MIKE 1D engine configuration
- Fields' status
- Default values
- User-defined column information
- Status codes
- Bookmarks
- Model type settings
- Custom units

More information on how to use the tool can be found in chapter 6.5 Cloning the MIKE+ Database *(p. 193)*.

Cloning a database doesn't create a corresponding *.mupp file which is necessary in MIKE+. Therefore, in order to open a cloned database in MIKE+, it is required to create a new project using the cloned database.

Import

MIKE+ offers options for importing different model databases and setups into a MIKE+ project database. Those are:

- Import MU classic model (*.mdb). Imports all data from a MIKE URBAN database in *.mdb format.
- Import MU classic model (*.gdb). Imports all data from a MIKE URBAN database in *.gdb format.
- Import MIKE HYDRO River model (mhydro). Imports river model data from a MIKE HYDRO River file. Some functionalities and options from MIKE HYDRO River are not supported in MIKE+ and cannot be imported.
- Import MIKE 11 model (sim11). Imports river model data from a MIKE 11 file. Some functionalities and options from MIKE 11 are not supported in MIKE+ and cannot be imported.
- Import EPANET model. Imports a water distribution model data from an *.inp file.

- Import full MIKE FLOOD model setup (*.couple). Imports all data from a MIKE FLOOD setup file. All data files used in the selected MIKE FLOOD setup will be imported to the MIKE+ database: MIKE URBAN classic, MIKE 21, MIKE HYDRO River, MIKE 11, couplings. Some functionalities and options from MIKE HYDRO River and MIKE 11 are not supported in MIKE+ and cannot be imported. If the MIKE HYDRO River data must not be altered, use the coupling to MIKE HYDRO River instead of this import option.
- Import MIKE 21 model setup (*.M21, *.M21FST, *.M21FM).
- Import MIKE FLOOD couplings (*.couple). Imports only the couplings from a MIKE FLOOD file. Related river, urban and/or 2D data files are not imported.

Before using the predefined import for a MIKE URBAN Classic model, it is necessary to update any old models to MIKE URBAN Classic Release 2020 Update 1 so that the *.MDB or *.GDB source database is in the correct format. Before importing any other MIKE file, the file should also be updated to the latest version to be in the correct format.

Also see Chapter 6.4 Predefined Import and Export Routines *(p. 177)*.

Export

- Export EPANET model. Export WD model to EPANET *.INP file.
- Export to M1DX file. Export CS model to a MIKE 1D engine *.M1DX file.
- Export to MIKE 21 FM setup file. Export MIKE+ 2D model setup to a *.M21FM file.
- Export to MIKE FLOOD couple file. Export MIKE+ flood model setup to a *.COUPLE file.

Also see Chapter 6.4 Predefined Import and Export Routines *(p. 177)*.

Recent projects

To access recent projects which were opened recently at MIKE+, click on File | Recent projects, and choose the desired project

User preferences

User preferences include general options relating to the MIKE+ installation. All these settings apply to the MIKE+ installation on the computer and therefore apply to all model setups opened on this installation, unless stated otherwise. This includes:

- Language selection. You can choose one of the languages available in the software. Note that the selected Language needs to have been installed during program installation.

- Preferred unit system. You can select between SI and US unit system. The selected unit system will control the unit system proposed per default when creating a new project. It will also control in which unit system results are shown, when no model database is opened :
  - MIKE 1D results will be displayed in a unit solely controlled by this preferred unit system
  - Water Distribution and SWMM results will be displayed in the unit system in which they were created if it is consistent with the preferred unit system, and will be displayed in the preferred unit system otherwise.
- Use of single editor style: when this style is active, only one editor can be shown besides the map: clicking a different menu in the Setup tree view will simply open this editor in the same tab. When this is inactive, an unlimited number of editors can be added: clicking a different menu in the Setup tree view will open this editor in a new tab.
- Show warning on undo buffer clear
- Auto-load result files after simulation finished: automatically adds result files to the Results tree view after the computation is completed, and loads them for visualization in the various results views. This only applies to single runs (for batch simulations, result files are never loaded).
- Auto-add result layers after simulation finished: automatically adds default result layers to the model map after the computation is completed. This only applies to single runs and if 'Auto-load result files after simulation finished' is also active.
- Auto restore the project layout when re-opening a project
- Retain exported 2D setup files for simulation: when running a coupled simulation, multiple files are created to execute the simulations and are automatically deleted at the end of the run. Selecting this option will keep the files at the end of the simulation. This setting is saved in the model database, and must therefore be adjusted as necessary for each model setup individually.
- Auto-refresh changes made by others (PostGIS only): when this option is active, MIKE+ continuously checks if changes have been made to the PostGIS database by other users, and refresh the content shown on the screen as necessary. This is only relevant when connecting to a PostGIS database and is irrelevant when working with a .sqlite database. When multiple users work simultaneously on the same PostGIS database, it is therefore recommended that ALL users enable this option in order to see changes made by others, otherwise data shown in the editors may deviate from the actual data saved in the database. If only one user at a time connects to the PostGIS database, it is instead recommended to leave this option unticked to reduce data traffic over the network and improve the performance of the application. See PostGIS database specifics, p. 145, to read more about multi-user access with PostGIS databases.

- Specifies the number of significant digits in editor and max. row count per table preview

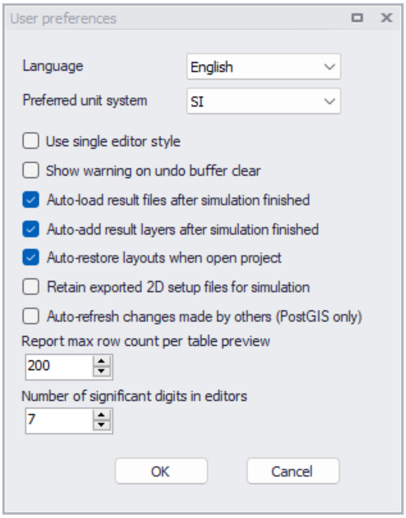


Figure 2.25 Global settings editor in MIKE+

Install Examples

As a new user it would be useful to load examples and practice testing various items in the example model. Examples can be loaded in your local directory, with a *.PDF explaining the content of the example and *.MUPP file to load into MIKE+.

Help

This menu gives access to various sources of help or information related to MIKE+:

- Documentation: opens the Documentation Index page, containing links to the various user guides and release note of MIKE+ available online, in PDF format.
- Online help: opens the help page, providing the same content as the user guides but in a more searchable format.
- Customer Care portal: opens DHI's Customer Care portal, with access to support contact, FAQs and software updates.
- YouTube videos: opens the YouTube channel including demonstration videos for MIKE products and their functionalities.
- Training portal opens DHI’s training portal, offering access to training courses, conferences and webinars.
- MIKE+Py: opens the download page for MIKE+Py, a Python interface for MIKE+.

About

Provides details about the software release, contact details and product license.

Exit

Exit the project and close MIKE+.

2.9.2 Project Menu

The Project menu offers additional quick access to the different Views as well as to general tools and functionalities grouped under toolboxes.

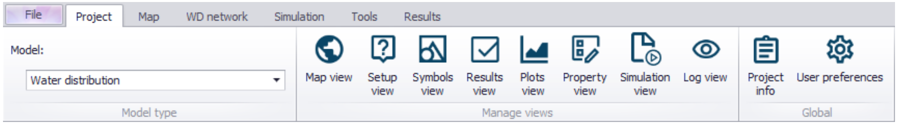


Figure 2.26 MIKE+ Project menu ribbon

Model Type

This toolbox allows you to switch from one model type/mode to another amongst the following list:

- Rivers, collection system and overland flows
- SWMM5 rivers and collection system flows
- Water Distribution

Manage Views

This toolbox manages the views of the model records and divides it into six different tabs and menus.

Map View

To view the main Map, click on Project | Map view. Access the local context menu from the Map to access options to:

- Recreate overview
- Run database validation
- Show validation items on map
- Show feature fly-by when interactive
- Refresh boundary visualization
- Clear flags and paths
- New selection list from map
- Add bookmark

- Show bookmarks
- Reset toolbars. Option for resetting the toolbars shown on the top border of the Map.

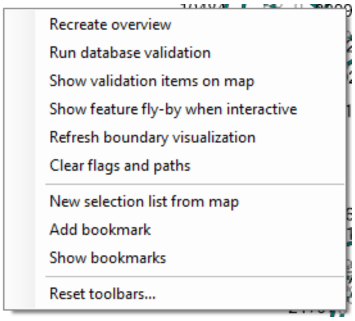


Figure 2.27 Access the main Map local context menu by right-clicking on the Map

Setup View

The setup view provides data validation that can allow you to quickly examine the model data as shown in Figure 2.28.

- Green ticks: all OK
- Red crosses: Some data is incomplete or incorrect

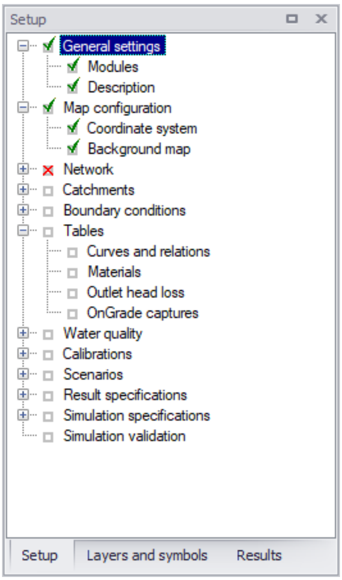
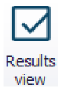


Figure 2.28 Example of data validation in the Setup view

Symbols View

This presents the Layers and Symbols panel where you can customise the model layers and visualise the different components as desired by colour coding each component. In this panel you can add a new layer i.e. shapefile or results layer to the Map, which will allow a better visualisation of the model results and the network overall.



Results View

Once a simulation is complete the result file will be automatically added into the Results manager, unless this automatic loading is disabled in the ‘User preferences’ dialog. This panel allows you to manage result files and visualise simulation results in various ways. See Results Presentation chapter (page 391) for more information on how to display results from this list of result files.

Plots View

This opens the 'Plots' panel where result views (e.g. time series plots, maps, etc.) may be saved and organized, to be re-opened at a later stage without having to re-configure the results selection and symbology customization. See Plots Management chapter (page 487) for more information.

Property View

This opens the Property and Result Explorer. When you want information about a feature displayed on the Map, you can use the ‘Identify’ tool and information about the selected feature on the Map is displayed in the Property and Result Explorer.

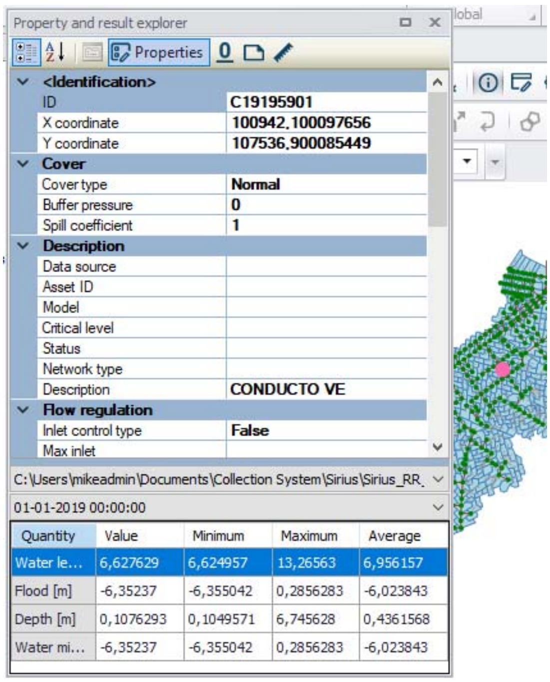


Figure 2.29 Property And Result Explorer showing properties of the feature highlighted (in pink) on the Map

Simulation View

A Simulation view shows a panel displaying the content of the simulation's log file as reported by the simulation engine. The same content is written to a file on the disk during the simulation, and can be retrieved even after closing the software.

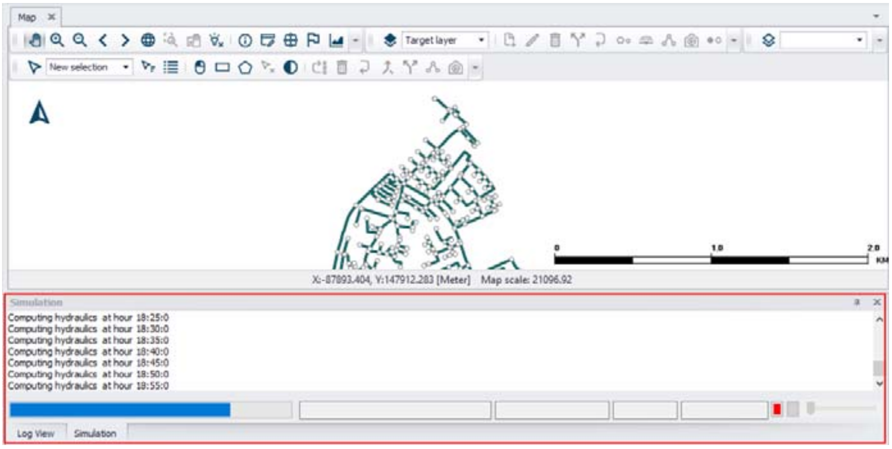


Figure 2.30 MIKE+ Simulation View shown at the bottom of the interface

Log View

A log view shows a panel displaying information on data processing, such as data import or error messages.

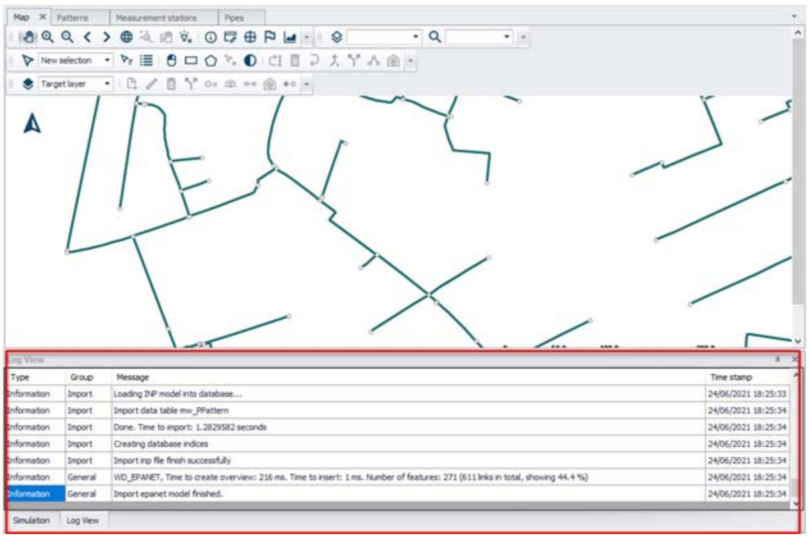
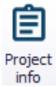


Figure 2.31 MIKE+ Log View shown at the bottom of the interface

Global



Project Info

This provides general information on the project and model components, such as number of nodes, number of pipes, project database type, etc.

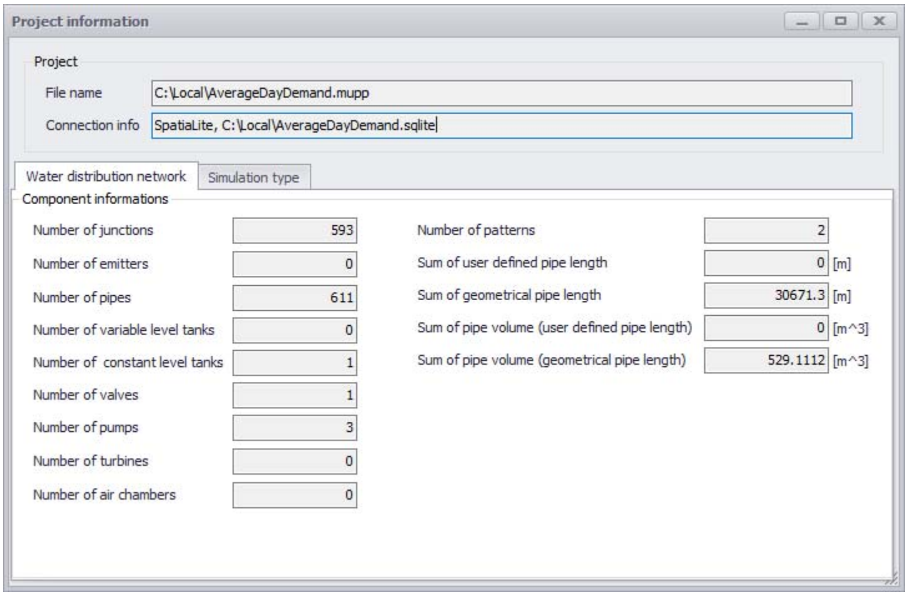


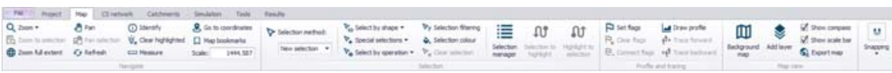
Figure 2.32 MIKE+ Project Information window

User preferences

Provides options for customizing general program behaviour, e.g. Automatic loading of results, language, etc. See “User preferences” on page 54.

2.9.3 Map Menu

The Map menu in MIKE+ provides tools and functionalities that can be used to modify and query model components on the Map.



Note that these tools are applicable only to the main Map (i.e. Map View) and not for result map plots.

Navigate

The Navigate Toolbox contains tools allowing easy navigation around the Map.

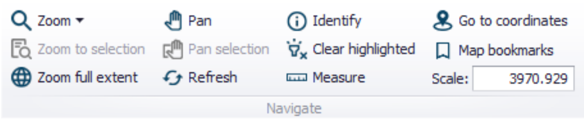


Figure 2.33 The Navigation Toolbox on the Map menu ribbon

Zoom

Zoom in, zoom out, zoom next and zoom to previous options on the Map.

Zoom to selection

Zooms to the maximum extent of selected features on the Map.

Zoom full extent

Shows the full extent of model data.

Pan

When this tool is active, click and drag the map, to move it without changing the zoom level.

Pan selection

Centres the map to the selection without changing the zoom level.

Refresh

Ensures applied edits are reflected on the map.

Identify

The identify tool allows you to see the attributes of your data. Clicking the identify tool on a location will display the attributes of element at that location via the Properties and Result Explorer.

Clear highlighted

This clears any highlighted items (i.e. items highlighted when using the "identify" tool).

Go to coordinates

Opens a window to specify the X and Y coordinates at which the map should zoom.

Map bookmarks

You can switch from one bookmark to the other using map bookmarks, or you can save a specific model extent.

Scale

Input box displaying the map scale corresponding to the current zoom level on the Map. Specifying the map scale adjusts the zoom level accordingly.

Measure

Tool to measure distances on the map. The tool shows the total distance of the digitized polyline as well as the length of the last segment. It also shows a polygon's area, when digitizing a polyline and closing it by double-clicking the first point of the polyline.

Please note: It is also possible to zoom to a specific item (pipe, pump, dike, etc.) by double-clicking on the corresponding item row in the editor. For example, to zoom to a pump, open the 'Pumps' editor, search for the pump of interest in the table, and double-click its row in the column containing the row number.

Selection

The Selection Toolbox offers various tools and functionalities related to the selection of model elements and features on the Map.

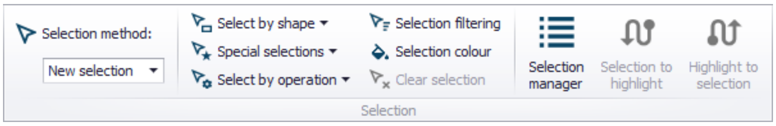
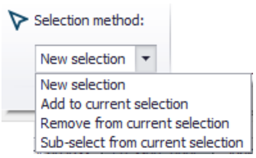


Figure 2.34 The Selection Toolbox on the Map menu ribbon

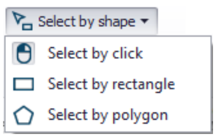
Selection method



Choose how the selection shall be considered:

- New selection
- Add to current selection
- Remove from current selection
- Sub-select from current selection. Select from currently-selected elements.

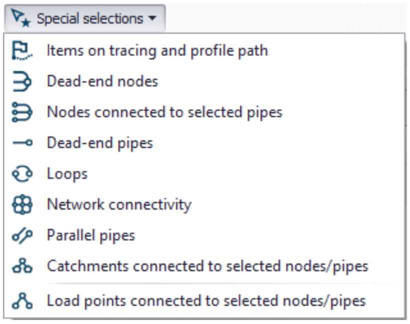
Select by shape



Options for how elements are selected on the Map:

- Select by click
- Select by rectangle
- Select by polygon. By drawing a free-form polygon on the Map.

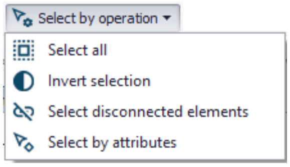
Special selections



Options for selecting from elements categorized according to characteristics/properties:

- Items on tracing and profile path
(selects network items shown with the green path on the map, either obtained from connected flags or using the tracing forward / backward tool)
- Dead-end nodes
- Pipes connected to selected nodes. In WD projects, this selects pipes connected to selected junctions, tanks and air-chambers.
- Nodes connected to selected pipes
- Dead-end pipes
- Loops
- Network connectivity
- Parallel pipes
- Catchments connected to selected nodes/links (CS and SWMM)
- Nodes/links connected to selected catchments (CS and SWMM)
- Load points connected to selected nodes/pipes (CS)
- Pump stations connected to selected pumps (WD)
- Demand allocation connected to selected nodes/links (WD)
- Network items coupled in selected 1D-2D couplings (2D overland)
- 1D-2D couplings of selected network items (2D overland)
- Network items for selected autocalibration controls (WD): selects network elements such as pipes and junctions, where parameters are calibrated or where targets are defined, in the Autocalibration special analysis.
- Network items for selected optimization controls (WD): selects network elements such as pipes and valves, where parameters are optimized or where targets are defined, in the Optimization special analysis.

Select by operation



General operation options for making selections:

- Select all
- Invert selection. Switch selecting to previously unselected elements.
- Select disconnected elements: This tool changes the current selection on the map, to select all nodes and links that are disconnected from the current selection. Some network elements should be selected before performing this operation. All structures are considered being connecting elements. Inactive pipes ('Enabled' box being unselected) are considered being disconnecting elements.
- Select by attributes. Select elements from data tables using operations based on attribute values.

The 'Select by attributes' tool will select records from the layer (table) selected at the top of the tool, using the operation defined by the expression specified in the text field at the bottom. The operation must be expressed using the SQL syntax, e.g. text strings should be written between quotes (for example, LinkID = 'Pipe1'). The expression can be typed either manually or using the items and buttons available in the tool:

- The 'Fields' group provides a list of attributes existing in the layer being selected. Double-click an item from this list will insert the item name in the expression field.
- The 'Get unique values' button will list all values currently applied in the database for the selected 'Field' in the upper list. Double-click a unique value from the list will insert this value in the expression field. You can find a record from the list of unique values by typing its name in the 'Go to' field.
- The other buttons will insert operators in the expression field.

The 'Method' list allows selecting among four selection methods:

- New selection: This will clear any selection being active before executing the tool, and will then select new items using the specified selection expression.
- Add to current selection: This will keep any selection being active before executing the tool, and then append new selected items using the specified selection expression.

- Sub-select from current selection: This will keep only the records fulfilling the specified selection expression, from the selection being active before executing the tool.
- Remove from current selection: This will remove from the selection (being active before executing the tool) the records fulfilling the specified selection expression.

It is also possible to use fields / attributes from other tables than the one being selected, using the general SQL syntax, using the "from" command to search in other tables. For example, links from table msm_Link which have their upstream level (attribute msm_Link.UpLevel) lower than the invert level of their upstream node (msm_Node.InvertLevel for the node selected in msm_Link.FromNodeID), can be selected using the following SQL expression:

UpLevel<(select InvertLevel from msm_node where msm_node.muid = msm_link.fromnodeid)

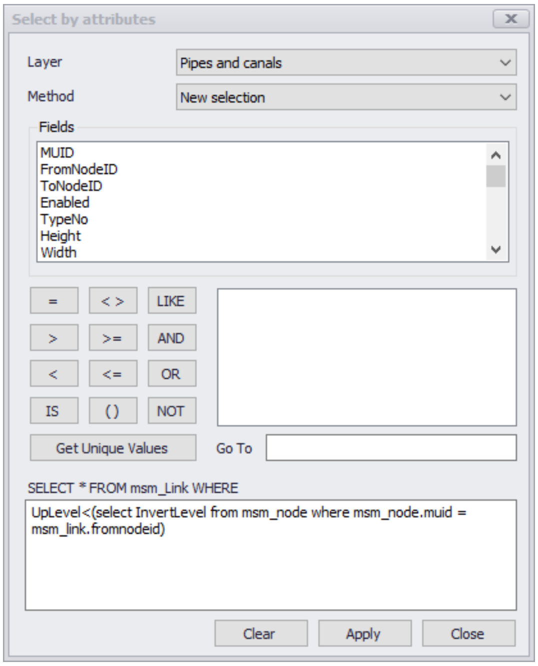


Figure 2.35 Selecting links from the 'Pipes and canals' layer, having their upstream level lower than the invert level of their upstream node

It is also possible to create a selection depending on another selection. To achieve this, the other selection must be saved to the database using the 'Selection manager'. For example, gates structures located on rivers selected in the selection called 'RiverSelection', can be selected using the following SQL expression:

RiverID = (select itemmuid from m_Selection where m_Selection.selectionid = 'RiverSelection' AND m_Selection.tablename = 'mrm_Branch')

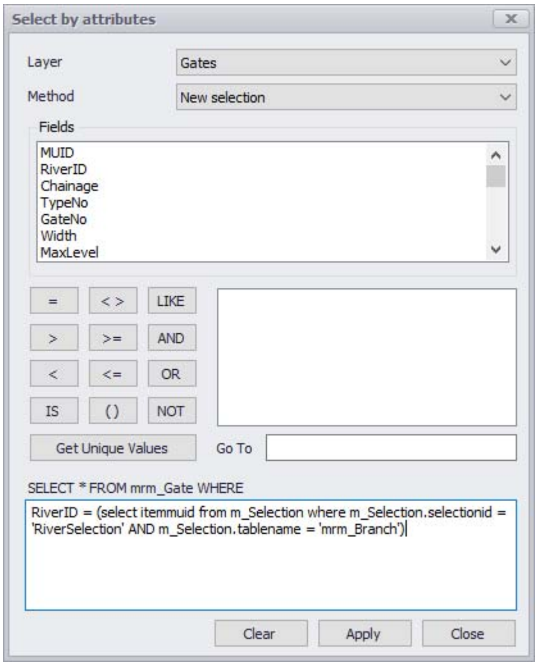


Figure 2.36 Selecting gates located on rivers selected in the selection called 'RiverSelection'

It is also possible to select some records based on the value of an attribute status (see Flagging, page 195). For example, nodes from table msm_Node with the status of the 'Diameter' attribute set to 'Verified' can be selected using the following SQL expression:

muid IN (select ItemMUID from m_Status where m_Status.TableName = 'msm_Node' AND m_Status.FieldName = 'Diameter' AND m_Status.StatusText = 'Verified')

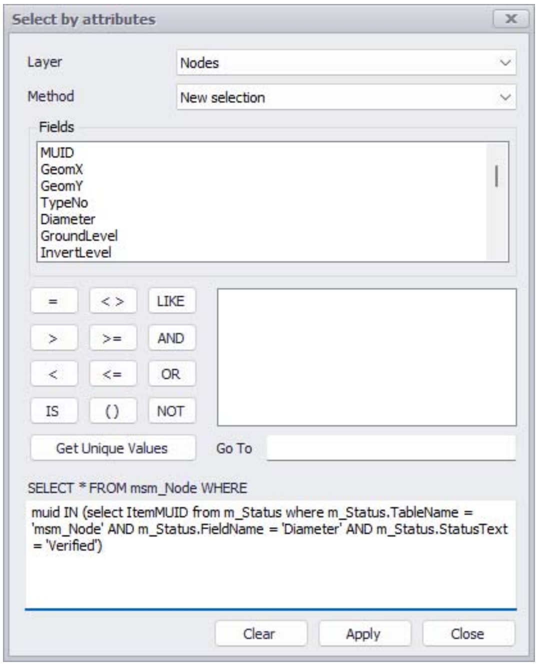


Figure 2.37 Selecting nodes with diameter's status set to 'Verified'

Selection colour

Option for defining colour to use on the Map to highlight selections.

Clear selection

Deselect all selected elements.

Selection filtering

Option for defining model elements from where selections can be made.

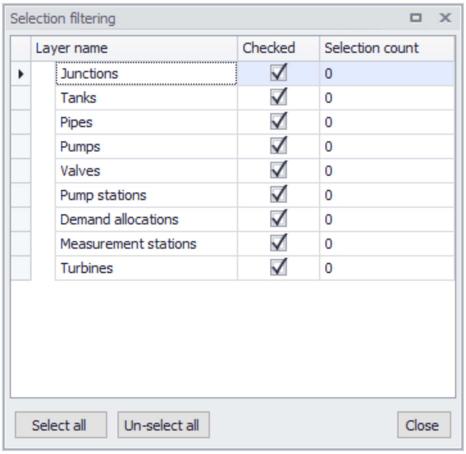


Figure 2.38 Selection filtering dialog for WD models

Selection manager

Dialog wherein user-defined selection lists may be specified for easy reuse in multiple functions and tools for the project and in the application.

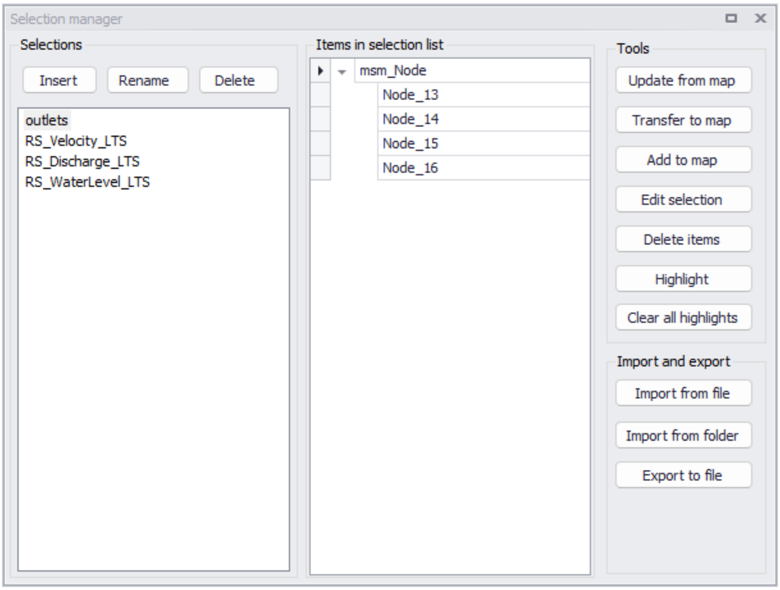


Figure 2.39 The Selection Manager in MIKE+

The dialog contains the following sections:

- Selections: this is the main list containing user-defined selections. Selections are inserted / deleted / renamed using the buttons above the list.

- Items in selection list: this is the content of the selection which is selected in the 'Selections' section, showing typically model features, boundary conditions, etc. This content is typically updated using the buttons in the 'Tools' section.
- Tools: the following buttons are used to edit the content of the selection.
  - Update from map: this action updates the current selection list, so that the list contains all items currently selected on the map and in the various editors. It is typically used after creating a new selection list, to save the current selection on the map.
  - Transfer to map: this action selects on the map and in the editors, all items included in the selected list. Items previously selected get unselected, if they are not part of the list.
  - Add to map: this action adds to the selection on the map and in the editors, all items included in the selected list. Items previously selected remain selected, even if they are not part of the list.
  - Edit selection: this action opens a window showing the full lists of items from the various database tables, to manually add or remove items from the selection.
  - Delete items: this action removes the selected items from the selection. It does not delete them from the project.
  - Highlight: this action highlights on the map all items included in the selected list.
  - Clear all highlights: this action clears all highlights from the map.
- Import and export: the following buttons are used to save or load the selection to external files.
  - Import from file: this action loads the content of the selection from a .mus text file.
  - Import from folder: this action loads the content from multiple .mus files located in the selected folder.
  - Export to file: this action exports the content of the selection to a .mus text file.

Selection to highlight

To highlight (i.e. in pink) selected elements on the Map. Elements are only subject to querying and not editing when highlighted (as opposed to selected).

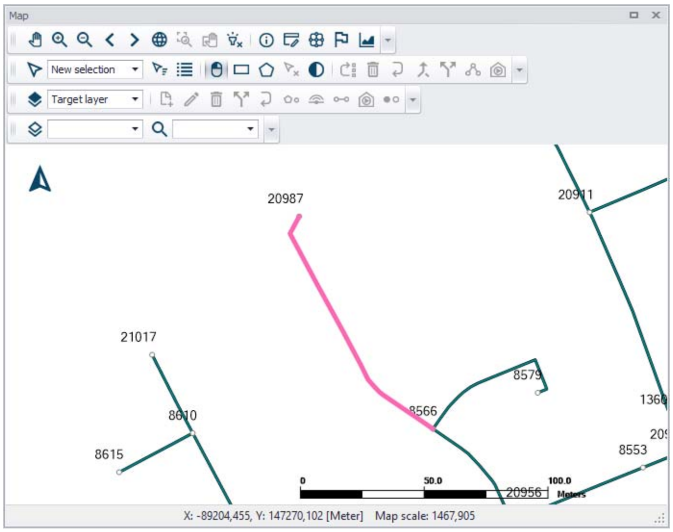


Figure 2.40 Pipe element highlighted on the Map

Highlight to selection

Highlighted elements (i.e. in pink) on the Map are selected. Elements are highlighted when e.g. the Identify tool is used to view its properties from the Map.

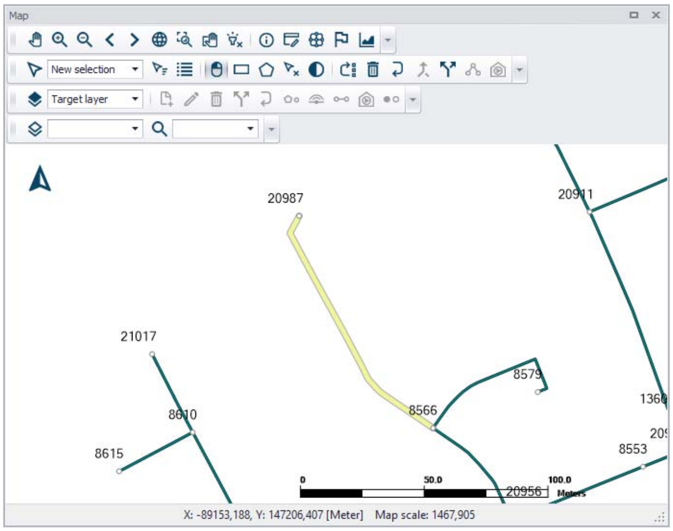
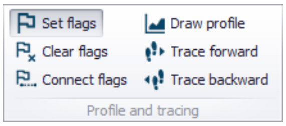


Figure 2.41 Pipe element selected from the Map

Profile and Tracing



The Profile and Tracing Toolbox contains tools for creating longitudinal profiles along model networks from the main Map. It also has tools for tracing and checking network connectivities.

Set flags

Tool for placing flags at node locations on the main Map view in preparation for creating longitudinal profile plots or analyzing network connectivity.

Clear flags

Removes all flags set on the main Map.

Connect flags

Identifies connections between the first and last set flags along the model network on the Map.

Draw profile

Window presenting generated longitudinal profile plots. Also used for creating new profile plots when flags are moved/re-set on the Map.

Tracing forward

Tool for tracing forward connections from a set flag point on the Map.

When no result layer is available on the main map, the tracing is based on the From and To Node definitions of link elements. The forward direction of a link is from its 'From' node to its 'To' node.

When a result layer is available on the main map, the 'Select target to trace' window will appear, offering two tracing methods:

- From model network connectivity: the tracing is based on the From and To Node definitions of link elements
- From flow direction results: this will use the actual flow direction computed during the simulation. For this option, it is therefore required to select the result file and the corresponding date and time of the results to be used. It is also required to specify a threshold: the tool won't trace results in links where the resulting value is smaller than the specified threshold.

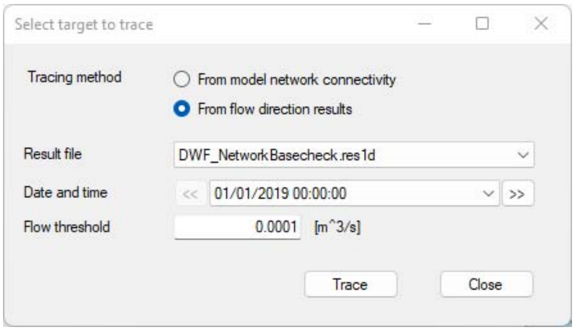


Figure 2.42 The dialog controlling the tracing settings

When tracing with results, the flow analysis is performed using the average flow on the link, and the link is either entirely included or entirely excluded from the tracing path. The flow tracing will therefore not stop at an intermediate grid point along the link where the discharge result will become smaller than the threshold.

Forward tracing is also available from extra result maps, by selecting 'Forward tracing' in the context menu on the map, after setting a flag on this result map.

Tracing backward

Tool for tracing connections backwards from a set flag point on the main Map.

The logic is the same as for the forward tracing, but tracing in the opposite direction. When tracing using the network connectivity, the backward direction of a link is from its 'To' node to its 'From' node.

Backward tracing is also available from extra result maps, by selecting 'Backward tracing' in the context menu on the map, after setting a flag on this result map.

Also see Chapter 20.6 Profile Plots *(p. 417)* for related information.

Context menu

When the 'Set flags' button is active in the ribbon, right-clicking on the map opens the context menu shown on Figure 2.43, which offers extra options to work with flags.

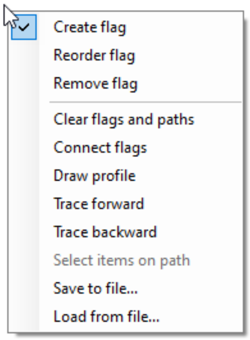


Figure 2.43 The context menu options to work with flags

This menu offers shortcuts for the actions also available in the ribbon (Draw profile, Trace forward, etc.) as well as options to edit flags (reorder or delete them). It is also possible to save the list and location of the current flags to a file (*.path), which allows reusing the same path at a later stage by loading the flags again from the file.

Map View

Background map

MIKE+ provides several background map options, available either as online resources as shown in Figure 2.44 below or installed on the local computer. The choice made during project creation can be modified at any time. Use the Background Map tool from the Map menu ribbon, or launch the Background Map editor from the Setup tree view. Select a background map from the available options as shown in the Figure 2.44 below.

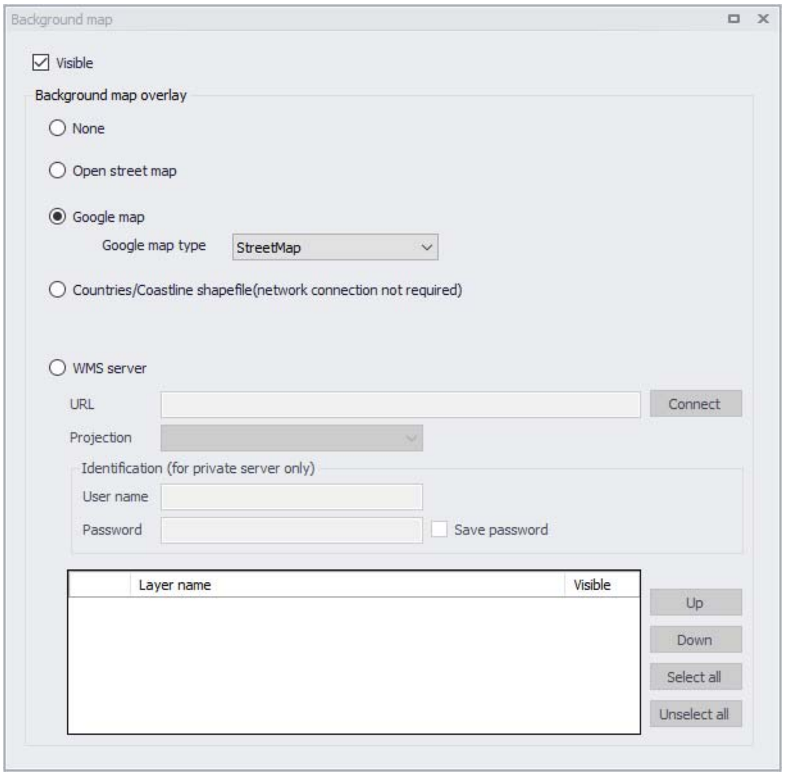


Figure 2.44 MIKE+ background map options

Add layer

Activate this tool to add data layers to visualize on the Map or use in the model via the Add Layer dialog. Added layers appear in the Layers and Symbols panel tree view.

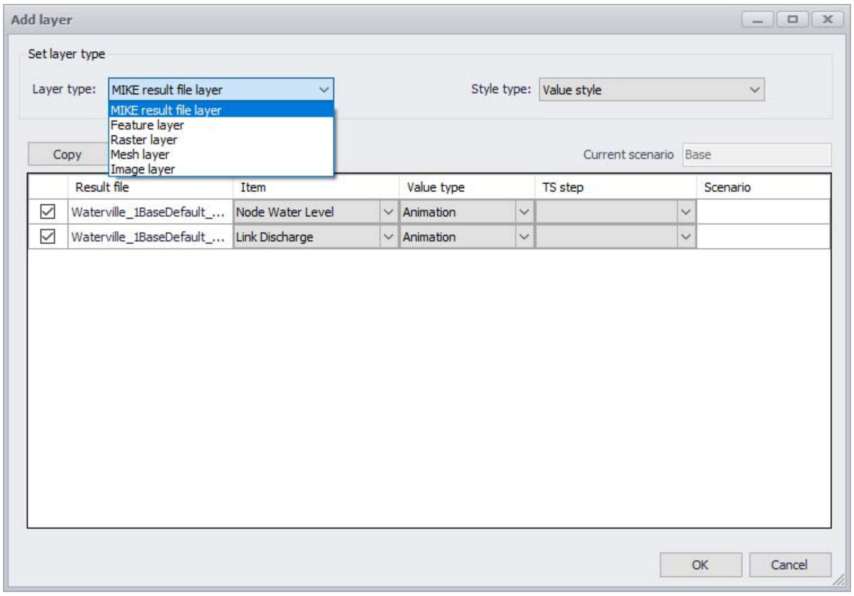


Figure 2.45 Add Layer dialog

Show compass

Show the compass symbol on the map.

Show scale bar

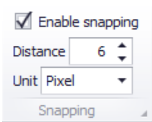
Show the map scale bar on the map.

Export map

Activate this tool to save the map view to an image file. The tool allows selecting the file type as well as the resolution (number of pixels) of the created file, which can be used to coarsen the picture to reduce the image size.

When active, the option to save the image coordinates to a world file will also create an extra text file holding the coordinates of the image. This file can later be detected and used by MIKE+ or other software products to display the image at the proper location on a map.

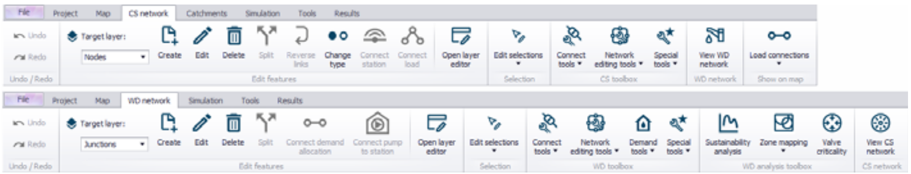
Snapping



Specifies the snapping tolerance used between features when doing graphical editing. The units can be selected from the options of either pixel or meter.

2.9.4 CS/WD Network

MIKE+ offers tools targeted for editing Water Distribution or Collection System networks through the WD Network or CS Network menu (depending on the working model type).



Undo/Redo

Offers Undo or Redo options during data editing.

Edit Features

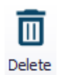
The Edit Features Toolbox contains tools that are used for interactively laying out the model network on the Map. The list of tools within the toolbox are listed below.

Create

This tool is used graphically add a component by selecting the target layer and clicking within the Map view. Double click to end the feature creation.

Edit

For editing features i.e. moving nodes, realigning polyline features, or reshaping polygons. Right click outside the feature being edited to end the editing.



Delete

Deletes the selected features.

Split

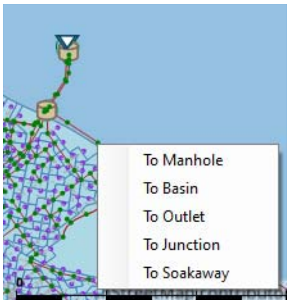
This tool is used to graphically split links on the Map.

Reverse links

This tool is used to swap the pipe orientation (i.e. From and To Nodes) for a selected pipe on the Map.

Change type

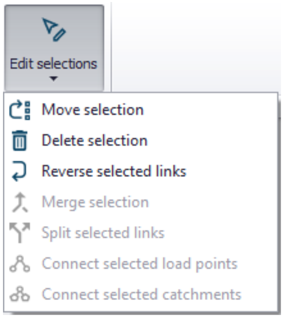
Option for CS Nodes. Option to quickly change the Node Type of a selected node on the Map (e.g. from Manhole to Outlet).



Open layer editor

Offers quick access to the Editor of the model feature selected from the Map. The editor is opened as a new tab document on the main window.

Selection



The 'Edit selections' list contains tools for editing/manipulating selected elements:

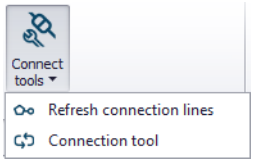
- Move selection
- Delete selection
- Reverse selected links: swap the From and To Nodes for links
- Merge selection: for merging selected elements.
- Split selected links: opens a dialog offering options for dividing the link geometry into more segments. This tool only applies to pipes (not to structure links).
- Connect selected demand allocations (WD)
- Connect selected pumps to pump station (WD)
- Connect selected load points (CS)
- Connect selected catchments (CS)

CS/WD Toolbox

This toolbox in MIKE+ includes specific tools for Water Distribution or Collection System models, which are used to connect, edit and simplify models. The available tools depend on the active model type.

Click on WD/CS network tab, then in the ‘WD/CS toolbox’, you will find Connect tools, Network editing tools and Special tools.

Connect Tools (WD)



Refresh connection lines

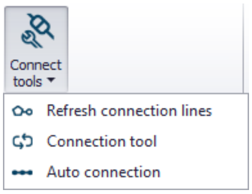
Apply this tool to refresh or recreate the connection lines (e.g. to measurement stations), in case they do not appear properly on the map.

Connection tool

Use the tool to configure automatic (bulk) connection of model features (demand allocations or measurement stations) to the WD network.

Also see chapter Connection Tool for related information.

Connect Tools (CS)



Refresh connection lines

Apply this tool to refresh or recreate the connection lines (e.g. to measurement stations or catchments), in case they do not appear properly on the map.

Connection tool

Use the tool to configure automatic (bulk) connection of model features (e.g. catchments, load points, measurement stations) to the CS network.

Also see Chapter 10 Connection Tool *(p. 253)* for related information.

Auto connection

Tool for making connections between 1D networks. Use the tool to configure the automatic (bulk) creation of connections between network layers, e.g. an overland flow network and underground sewer network (i.e. 1D/1D models).

Chapter 19.4 Auto Connection Tool *(p. 380)* has more details on the Auto Connection tool.

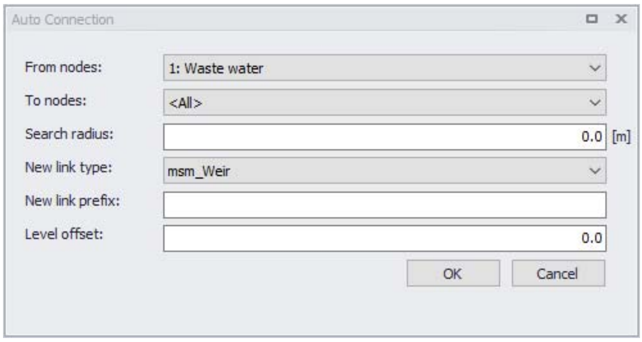
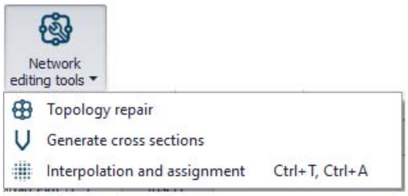
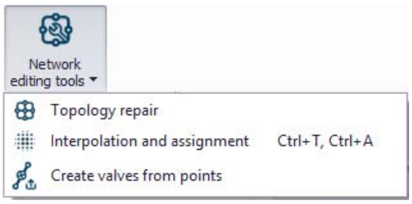


Figure 2.46 Auto connection tool dialog

Network Editing Tools





MIKE+ network editing tools provide automatic re-interpretation of features and attributes, when the imported model data does not compare to reality. The tools offered by MIKE+ allows a batch repair of the network at once, rather than a case by case scenario, by either using topology repair or by interpolating and assigning records accordingly.

Topology repair

Offers a way to detect and repair topology or network geometry issues in the model.

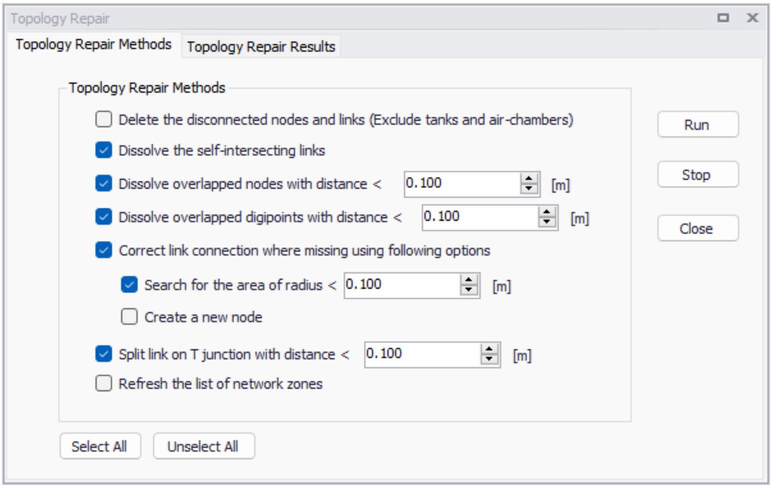


Figure 2.47 Topology repair in MIKE+

The following methods can be applied to the repair process:

- Delete disconnected nodes and links: isolated nodes (other than tanks) and links / pipes, disconnected from the rest of the network, will be removed.
- Dissolve self-intersecting links: when a link is self-intersecting (drawing a loop on the map, and crossing its own polyline), this operation will remove a vertex from the polyline. If the link is still self-intersecting after the operation, this operation must be repeated as necessary to remove more vertices. This operation also removes links which use the same node as 'From node' and as 'To node'.
- Dissolve overlapped nodes: when the distance between two nodes is smaller than the specified search radius, one of the nodes will be removed. Connected links are reconnected accordingly afterwards.
- Dissolve overlapped digipoints: when the distance between two digipoints (intermediate points defining the link's polyline) is smaller than the specified search radius, some digipoints will be removed.
- Correct link connection where missing: when a link's end is not connected to a node, this operation will connect it either to the closest existing node within the specified search radius, or to a new node. If the two options (search for existing nodes and create new ones) are enabled, the operation will first search for existing nodes, and will create a new one only if no existing node is found in the search radius.
- Split link on T-junction: when the end of a link overlaps a second link, this second link is split at the intersection and a node is inserted, and all the three resulting pipes are connected to this new node.

- Refresh the list of network zones: for Water Distribution projects, this operation refreshes the list of network zones in the 'Zones' editor, based on the information from the 'Zone ID' fields in the various network editors (pipes, junctions, etc.).

After running the tool, the 'Topology Repair Results' tab will list all the issues found, and the changes that have been applied. For items edited or inserted, double-clicking in the first column of this table will zoom to the corresponding item on the map.

Generate cross sections (For CS models)

Use the tool to derive CRS cross section data from terrain data. See Chapter 19.1 Cross Section Generation tool *(p. 376)* for details.

Interpolation and assignment

This tool allows you to derive (missing) model parameter values from other model or data layer information. More details on the tool are found in Chapter 12 Interpolation and Assignment Tool *(p. 267)*.

Create valves from points (For WD models)

This tool allows you to insert new valves in the network. It will split pipes when necessary, and insert the new valves at locations defined by points in a selected shape file. The main valves' properties can be read from the attributes of the shape file. More details on the tool are found in Chapter 13 Create Valves from Points Tool *(p. 277)*.

Special Tools

MIKE+ offers special tools for model simplification and feature editing. This includes the following:

Network simplification

This tool offers options for simplifying the model network through:

- Trimming: Removal of network inside an area of interest.
- Merging: Simplification of network by removal of interior nodes.

Chapter 14 Simplification Tool *(p. 279)* gives more details on Network Simplification in MIKE+.

Submodel manager

The Submodel Manager tool is used to create models where a specified area of interest is detailed, and the remainder of the model is simplified. See Chapter 16 Submodel Manager *(p. 335)* for details.

Spatial processing

This tool offers spatial processing tools for model features, such as clipping, erasing, merging, etc.

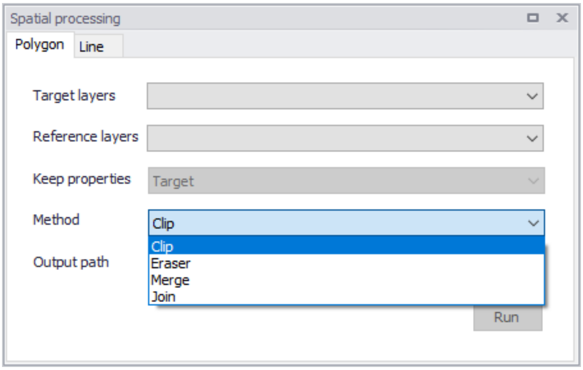


Figure 2.48 Spatial processing in MIKE+

Lateral snapping (For CS models)

The Lateral Snapping tool is used for automatically move nodes and snapping them laterally to the lowest DEM value along a lateral snap alignment.

See Chapter 19.3 Lateral Snapping Tool *(p. 378)* for more details on the tool.

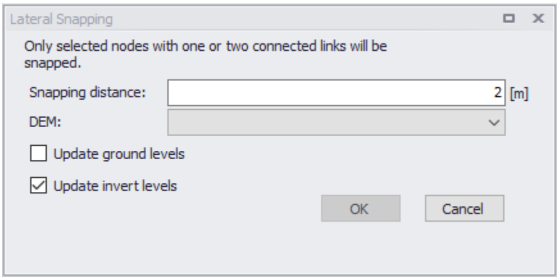


Figure 2.49 Lateral Snapping tool

Duplicate pipe parameters (For WD models)

Allows you to select pipes from the Map and duplicate their attributes to pipes with missing attributes, such as diameter, material, etc. The tool automatically duplicates the selected parameter to all pipes that are adjacent to the selected pipe(s) until a "T" or other complex junction exists.

Aggregation (For WD models)

The Aggregation tool allows to develop total junction demands based on demand connections.

Distributed demand (For WD models)

The Distributed demands tool allows you to distribute a portion of a total demand (to be specified by the user) to every pipe in the network.

Set pumps critical levels (For CS models)

The tool 'Set pumps critical levels' assigns a critical level at pumping stations' wet well nodes, computed from the geometry of the network upstream of the

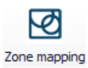
pumps.
See Chapter 52 Set Pumps Critical Levels Tool (p. 959) for more details on the tool.

WD Analysis Toolbox

Several special analysis tools are offered by MIKE+ for Water Distribution models.

Sustainability analysis

The tool helps understand WD simulation results and analyze them for possible problems, anomalies, critical areas, and similar.



Zone mapping

Zone Mapping offers two different ways to generate "zones" in the model, which can then be displayed on the map:

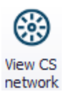
- Create zones from network separators: this first option creates zones based on the network topology and geometry, closed pipes, closed valves, and pumps.
- Create zones from GIS layer: this second option assign zones to the network elements using a layer of polygons providing the extent of the various zones.

Defining zones helps to visualise how different network parts are hydraulically interconnected and where the HGL line breaks. It helps understand the hydraulic behaviour of the network prior to running the hydraulic simulation, and also helps detect possible errors in the network connectivity.

Valve criticality

The Valve Criticality tool allows analysis of a valve from the valve layer to determine which valves need to be closed in order to replace the selected valve.

CS/WD Network



This functionality allows you to overlay and view another model type (e.g. CS) on top of the active model (e.g. WD). Note, this tools only allows you to view the other network model. To edit the other model network, change the Model Type under the Project menu.

Show on map

This group contains two tools to check connections of load points to the collection system network on the map.

This tool highlights all the load points connected to the currently active network.

Remove graphical highlights using the 'Clear highlighted' tool in the 'Map' tab.

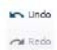
This tool highlights all the load points without connections to the currently active network.

Remove graphical highlights using the 'Clear highlighted' tool in the 'Map' tab.

2.9.5 River Network

MIKE+ offers tools targeted for editing river networks through the ‘River network’ menu.

Undo/Redo



Offers Undo or Redo options during data editing.

Edit Features

The Edit Features Toolbox contains tools that are used for interactively laying out the model network on the Map. The list of tools within the toolbox are listed below.

Create

This tool is used graphically add a component by selecting the target layer and clicking within the Map view. Double click to end the feature creation.

Edit

For editing features i.e. refining river line features, moving a structure along the river or reshaping polygons. Right click outside the feature being edited to end the editing.

Delete

Deletes the selected features.

Split

This tool is used to graphically split rivers on the Map.

Merge

This tool is used to merge rivers on the Map.

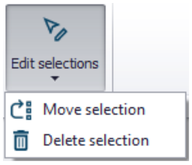
Connect station

This tool is used to connect a measurement station to the river network on the Map.

Open layer editor

Offers quick access to the Editor of the model feature selected from the Map. The editor is opened as a new tab document on the main window.

Selection



The 'Edit selections' list contains tools for editing/manipulating selected elements:

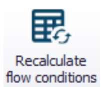
- Move selection
- Delete selection

Rivers Toolbox

This toolbox includes specific tools for river networks models, which are used to manage rivers, cross sections and storages.

See chapters River Tools and Tools and Plugins for details.

Structures

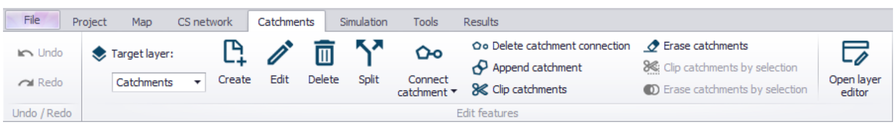


Recalculate Flow Conditions

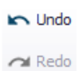
This tool is used for recalculating the Q/h relations of weirs and culverts when changes are made to the structures and/or to the up/downstream cross-sections.

2.9.6 Catchments Menu

The Catchments menu offers tools and functionalities related to creating catchment features for setting-up rainfall-runoff models for collection systems.

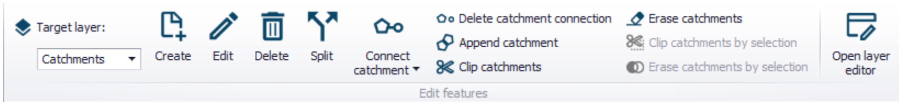


Undo/Redo



Offers Undo or Redo options during Catchments editing.

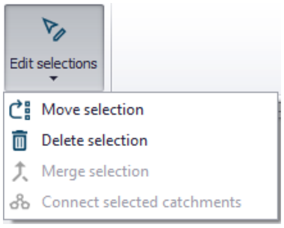
Edit Features



Tools for graphical editing of Catchment features on the Map, such as for creating, editing, and deleting catchment features.

Please refer to Chapter 9.2.2 Tools for Graphical Catchment Editing *(p. 216)* for more details on the various tools under the Edit Features toolbox on the Catchments ribbon.

Selection



The 'Edit selections' list contains tools for editing/manipulating selected elements:

- Move selection
- Delete selection
- Merge selection: for merging selected elements.
- Connect selected catchments

Catchment Toolbox

Please refer to Chapter 9.5 Automated Catchment Tools *(p. 229)* for more details on the various tools under the Catchment toolbox.

The toolbox includes tools for:

Catchment delineation

Tool for automatic catchment delineation as Thiessen polygons or derived from a digital elevation model (DEM).

Catchment processing

Tool to calculate imperviousness, time of concentration and other hydrological parameters for hydrological models.

Catchment slope and length

tool is an automated way to calculate the slope and length of a catchment based on a DEM. These parameters are used for MIKE 1D rainfall-runoff Kinematic Wave models.

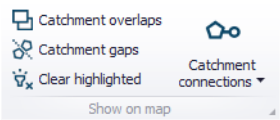
Connection tool

Tool for automatic connection of catchments to the network.

Special tools

Offers additional GIS spatial processing operations such as Merge and Join.

Show on Map

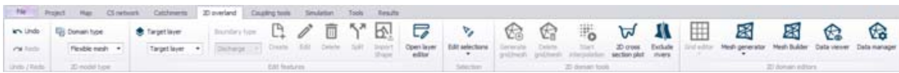


Contains tools and functionalities for validating catchment layer topology, as well as checking catchment and connections to the collection system network.

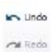
Please refer to Chapter 9.4 Graphical Tools for Connecting Catchments to Networks *(p. 226)* for more details on the various tools under the Show on Map toolbox.

2.9.7 2D Overland Menu

The 2D overland menu offers tools and functionalities related to creating and editing the 2D domain, 2D structures, or other 2D input data like spatial variations of surface roughness.



Undo/Redo



Offers Undo or Redo options during data editing.

Edit Features

The Edit Features Toolbox contains tools that are used for interactively laying out the 2D overland model on the Map. The list of tools within the toolbox are listed below.

Boundary type

This list is active only when the selected target layer is '2D boundary conditions'. The boundary type controls the type of boundary to be drawn on the map:

- For the point source type, the boundary conditions are represented by points on the map, which are located within the 2D domain.
- For the distributed source type, the boundary conditions are represented by polygons on the map, which are located within the 2D domain.
- For the other types, boundary conditions are represented by polylines along the borders of the 2D domain.

Create

This tool is used graphically add a component by selecting the target layer and clicking within the Map view. Double click to end the feature creation.

Edit

For editing features i.e. moving nodes, realigning polyline features, or reshaping polygons. Right click outside the feature being edited to end the editing.

Delete

Deletes the selected features.

Split

This tool is used to graphically split polygons on the Map (e.g. for 2D initial conditions or 2D surface roughness polygons).

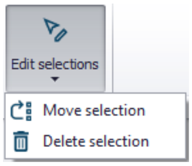
Import shape

This tool is used to import mesh arcs from a shape file.

Open layer editor

Offers quick access to the Editor of the model feature selected from the Map. The editor is opened as a new tab document on the main window.

Selection



The 'Edit selections' list contains tools for editing/manipulating selected elements:

- Move selection
- Delete selection

2D domain tools

This group contains tools that are used to create and edit the 2D domain.

Generate grid/mesh

When the 2D domain file is defined with the source type 'Domain file created from MIKE+ definition' in the '2D domain' editor, this button generates the grid or the mesh according to the grid and mesh definitions shown on the map. It is the same functionality as for the 'Generate grid' or 'Generate mesh' buttons in the '2D domain' editor.

Delete grid/mesh

When the 2D domain file is defined with the source type 'Domain file created from MIKE+ definition' in the '2D domain' editor, this button clears the 2D domain.

Start interpolation

When the 2D domain file is defined with the source type 'Domain file created from MIKE+ definition' in the '2D domain' editor, this button starts the interpolation of input elevation data on the 2D domain.

2D cross section plot

This opens a tool to draw cross sections from the 2D domain (possibly combined with river cross sections) and/or from a DEM.

Cross section plots are created from the main Map view, and the input files must therefore be loaded beforehand. From the 'Add cross section plot' window, it is possible to control the following:

- 2D domain and river cross sections: when this option is selected, the cross section of the 2D domain file will be shown for the selected location. This cross section can optionally include a river cross section, for coupled models.
- DEM layer: when this option is selected, the cross section of the selected DEM will be shown for the selected location.
- Maximum spacing between points: this controls the number of points to be plotted along the 2D cross sections. A different spacing can be specified for the 2D domain and for the DEM, in order to adapt to their respective resolutions.

Two types of locations can be used:

- Digitize location on map: when this option is selected, the location of the cross section is digitized on the map using a polyline. Double-click to stop digitizing and to show the cross section plot. While digitizing the polyline, it is possible to include a river cross section to obtain a common cross section plot of the coupled model, based on 1D and 2D input data: to do so, simply click at the intersection between the river and the cross section during the digitization process.

- Load from file: if the location of a previous cross section plot has been saved to a file, this location can be re-used by selecting this file, in order to create a new plot from the same location.

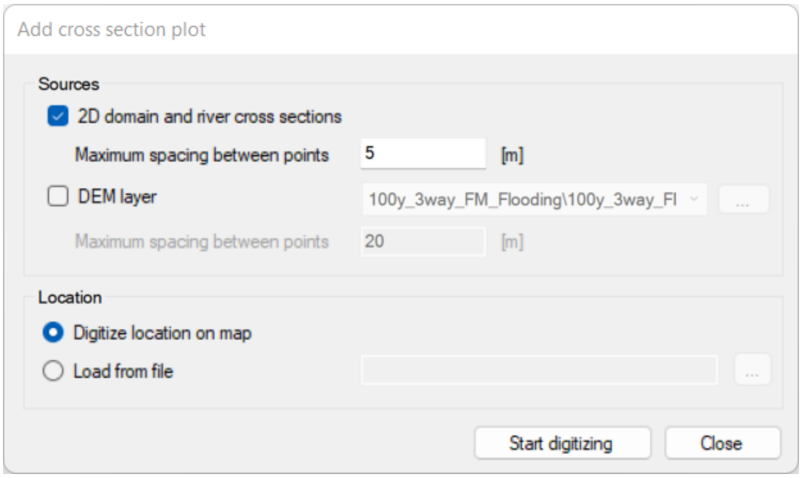


Figure 2.50 The tool used to create Cross section plots of 2D data

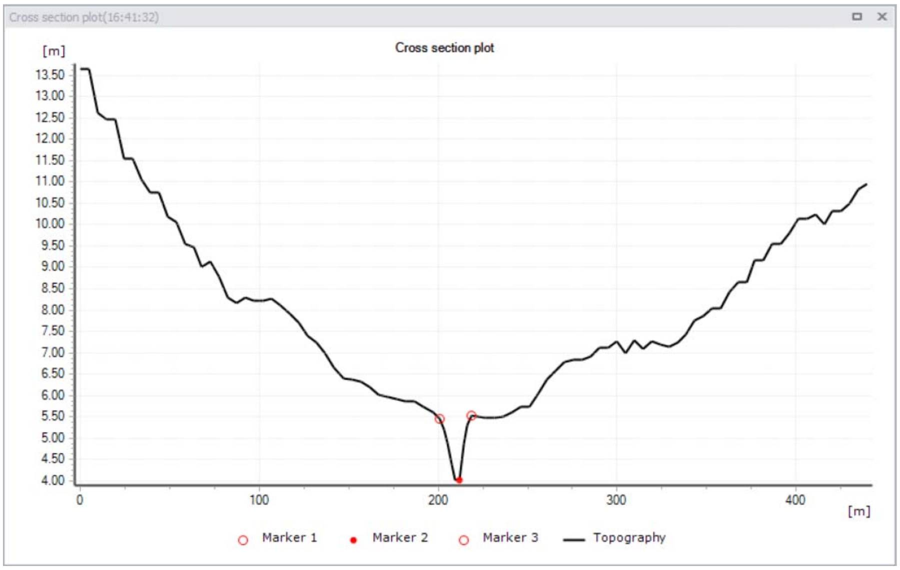


Figure 2.51 The Cross section plot window showing a combined river and 2D overland cross section

The context menu of the cross section plot offers the following options:

- Zoom to full extent, Zoom in, Next zoon, Previous zoom: Allows to zoom in and out on the plot. Zoom to full extent brings you back to the full view of visible cross section data on the plot. Panning is also enabled upon activation of zoom options, using the 'Shift' key.
- Copy to clipboard: Copies the cross section view displayed to the clipboard and allows it to be pasted into other applications.
- Save to image file: Saves the cross section view displayed to an image file on the disk, using various supported image formats.
- Export to text file: Saves the cross section's data to text file, to further process the obtained elevation data.
- Save location: Saves the location to a file, to later create new cross sections at the same location.
- Print: Prints the cross section view displayed to the clipboard.
- Properties: Activate this option to view the Cross section plot Properties dialog.

Note: Note: Closed river cross sections cannot be combined to the 2D domain's topography. They will be ignored if they are selected during the digitization of the cross section location.

Exclude rivers

This opens a tool which generates mesh arcs, defining polygons representing the river extent and being excluded from the mesh. When the 2D domain is defined with a rectangular grid, the tool creates polygons defining inactive areas. This is to be used when the 2D domain file is defined with the source type 'Domain file created from MIKE+ definition' in the '2D domain' editor, in order to ensure that the river extent is not modelled both in the 1D river network and in the 2D overland domain.

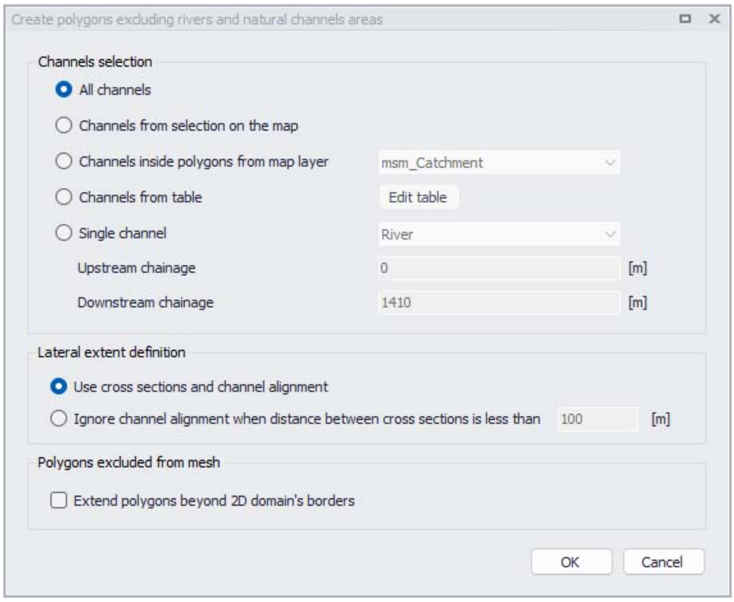


Figure 2.52 The tool used to exclude river extents from the 2D overland domain

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

- All channels: Creates polygons describing the area covered by all natural channels (specified in the CS network | Pipes and Canals editor) or rivers (specified in River network | Rivers).
- Channels from selection on the map: Select channels via the Map or network feature overview tables. The currently active selections will be excluded from the 2D domain.
- Channels inside polygons from map layer: Select channels to be excluded from a layer only (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) and tick on/off the channels to be excluded. 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 exclude one channel. In this case, select the ID of the channel to be excluded as well as which section of the channel to exclude (upstream and downstream chainage of the channel).

The polygons encapsulating the selected channels, and to be excluded from the 2D domain, are defined based on the extents of cross sections along channels. Two methods can be used:

- Use cross-sections and channel alignment: The location of the polygon's border 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 border at each cross section location, and at each change of direction (i.e. at each vertex) of channel. This method ensures that the polygon's border 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 polygon's border is solely based on the extent of the cross sections in the river reach. The polygon's border 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.

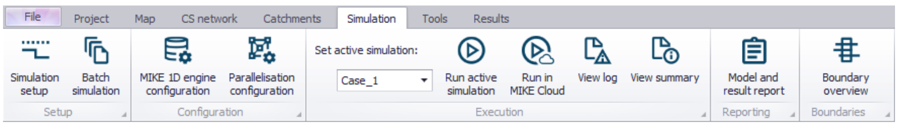
It is recommended to use the same method as the one used to create the 1D-2D couplings, in case they have been created with the 'Create couplings' tool, to ensure consistency between the borders of the 2D domain and the coupling locations.

By default, the generated polygons will be limited by the extent of the existing 2D domain. This will e.g. avoid creating arcs for a flexible mesh creation, outside of the expected area. This limitation can however be removed by activating the option 'Extend polygons beyond 2D domain's borders'.

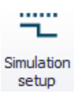
After running the tool to generate the polygons describing the excluded river extents, the 2D domain must be regenerated to take these polygons into account.

2.9.8 Simulation Menu

The Simulation menu offers functionalities related to simulation-running for both Collection System and Water Distribution models.



Setup



Simulation setup

Launches the Simulation Setup editor. Provides access to various options for a simulation setup. You can select the desired module from the General tab, specify the simulation period, and computation time steps, among other things.

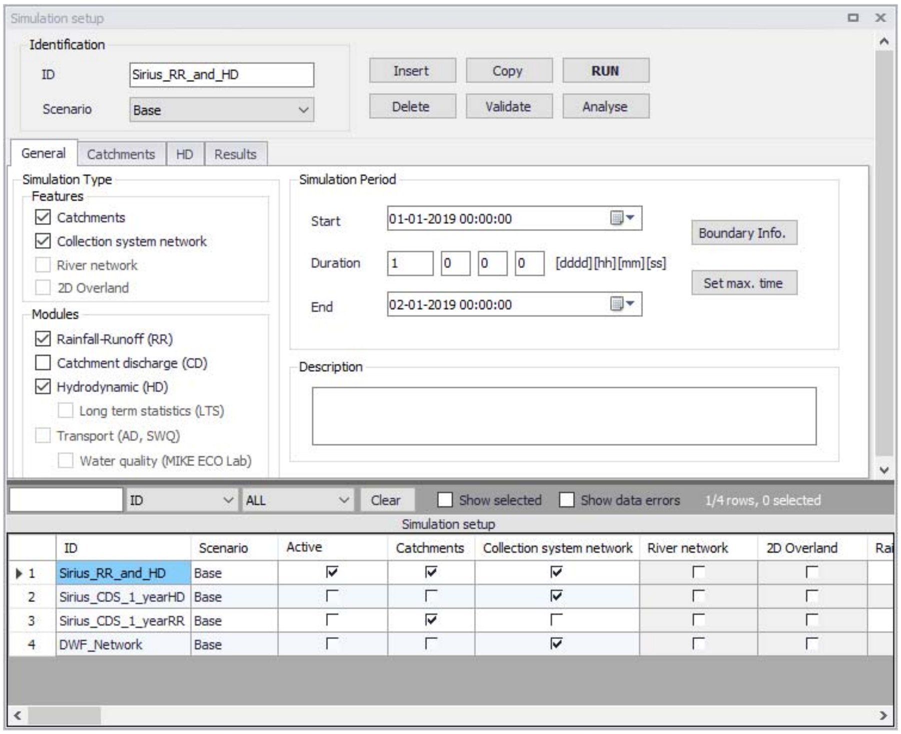


Figure 2.53 MIKE+ Simulations Setup editor

Batch simulation

Batch simulation

Launches the Batch Simulation editor, which offers the option of running simulation setups in batch (i.e. consecutively).

Configuration (‘Rivers, collection system and overland flows’ models)

MIKE 1D engine configuration

MIKE 1D engine configuration

This dialog offers options for specifying or defining special parameters for the MIKE 1D computation engine to use during simulations.

In the 'Predefined options' tab, it is possible to set a number of (selected) computation engine parameters. These parameters may activate different formulations for specific aspects in the computations, set the thresholds or default values for variables, etc.

A description of the parameter is shown in the lower right panel of the window. Note that the parameters for which user-specified values differ from default are highlighted in red in the window (see Figure 2.54 below).

Use the ‘Reset defaults’ button to revert to default values for all customizable parameters listed in this tab.

The 'Import from file' button can be used to import the equivalent parameters from a dhiapp.ini file used by MIKE URBAN Classic, in case this file was not imported with the MIKE URBAN Classic data into MIKE+.

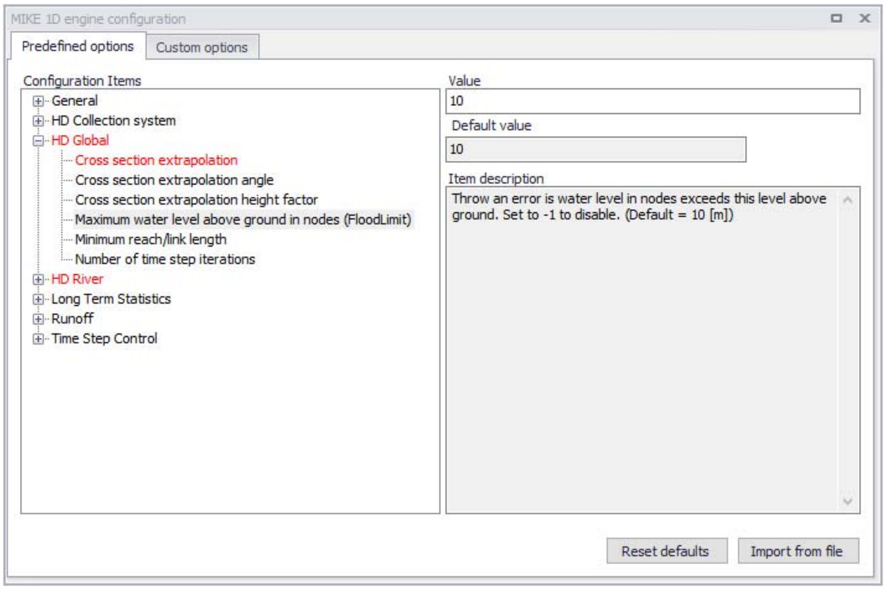


Figure 2.54 The predefined options in MIKE 1D Engine Configuration dialog

The 'Custom options' tab is used to apply more advanced options for the MIKE 1D simulation. These advanced options are in general not required, so this tab is used only for very specific applications.

Custom options can be additional parameters, as described in the "MIKE 1D additional parameters" chapter here:

https://docs.mikepoweredbydhi.com/engine_libraries/mike1d/mike1d_api/#mike-1d-additional-parameters

A general application example of this is to convert a specific error message into a warning, to prevent from stopping the simulation when the issue is met (and assuming that the simulation engine can continue, after ignoring the problem). This is achieved by inserting an option with the name 'MessageIdToWarning', with value type 'Text' and with the value matching the error name as reported in brackets in the simulation log file (e.g. 'ST_ERR_ReachStructureBottomLevelMismatch').

A custom option can also be used to enable the use of a script for the MIKE 1D simulation, as described here:

https://docs.mikepoweredbydhi.com/engine_libraries/mike1d/mike1d_scripts/

Custom options are added or removed from the list using the 'Insert' and 'Delete' buttons above the table. A custom option will only apply to simulations if the 'Apply option' box is ticked. Additionally, it is possible to filter the simulations to which the option applies, by ticking the box 'Apply only to following Simulation IDs' and by specifying the relevant simulation IDs in the text box underneath. Simulation IDs must be separated by a semicolon character.

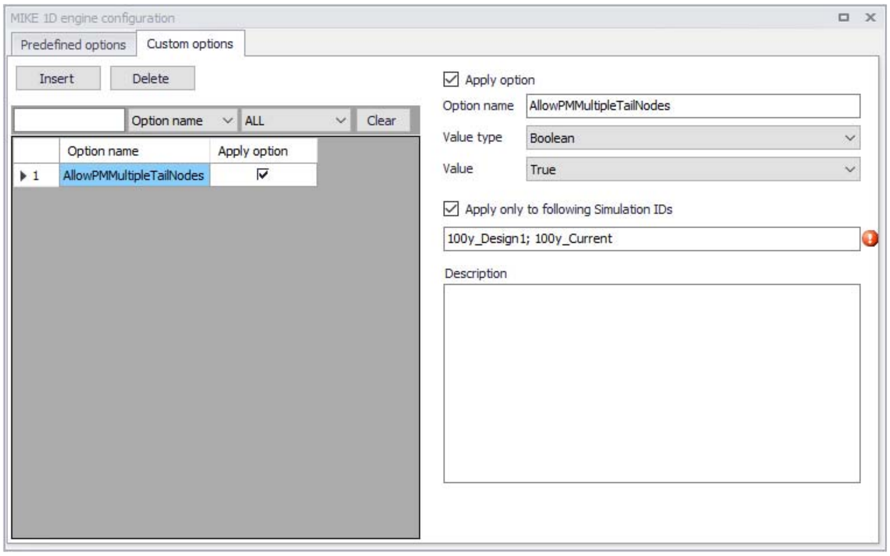


Figure 2.55 The custom options in the MIKE 1D Engine Configuration dialog

Parallelisation configuration

Allows customization of computation optimization options for ‘Rivers, collection system and overland flows’ model computations.

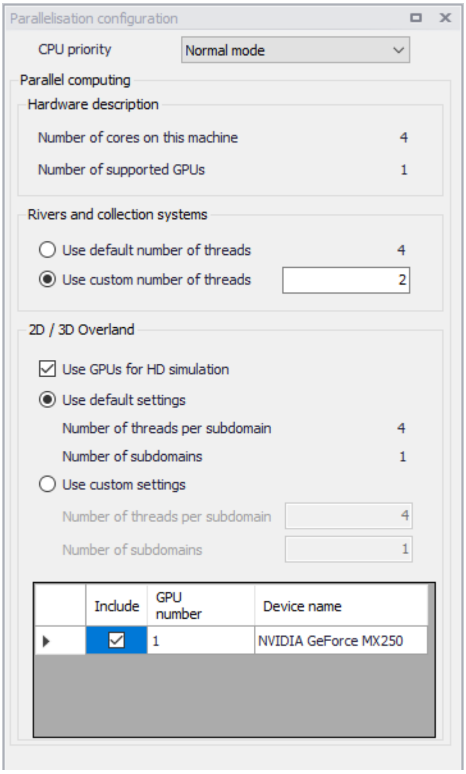


Figure 2.56 Parallelisation configuration dialog in MIKE+

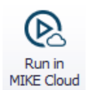
Execution

Set active simulation

Dropdown menu for selecting the Active simulation among the existing simulation setups in the project.

Run active simulation

For launching the Active simulation.

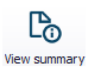


Run in MIKE Cloud

For launching simulations in MIKE Cloud. Read Chapter 2.13 Working with MIKE Cloud for more information.

View log

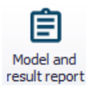
For viewing the simulation log file after a simulation.



View summary

For viewing the result summary file after a simulation.

Reporting



The Model Result and Report tool offers facilities for setting up reports based on information from model data as well as simulation results.

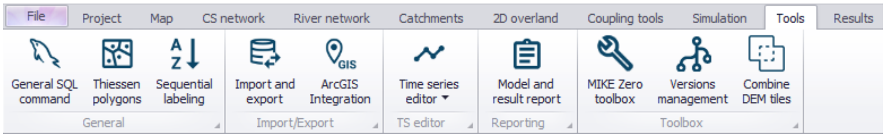
Also see Chapter 20.14 Reports *(p. 473)* for details in reports in MIKE+.

Boundaries (For CS models)

Launches the Boundary Overview window, which graphically displays the temporal extent of each boundary condition in the model setup.

2.9.9 Tools Menu

The Tools menu offers general data editing tools that are available for both Water Distribution and Collection System model types.



General

Under the General group, three main tools are available to assist you in building and updating models in an easier manner.

General SQL command

Allows you to add SQL commands to interrogate and edit model data. The ‘General SQL Command’ tool allows you to define, save, load, and execute an unlimited number of SQL commands.

The syntax you use to build SQL commands differs depending on the data source. This is because although SQL is a standard, not all database software implements the same dialect of SQL.

With the SQL commands you can for instance execute SQL UPDATE statements that will change the pipe diameter from the nominal (outside) diameter to the inside diameter or, you can define the pipe friction coefficient based on the pipe material and the pipe age.

General SQL commands can work on any MIKE+ geodatabase table such as "mw_*", "msm_*", and you can use standard SQL commands including UPDATE (to update table field values). Multiple SQL statements need to be separated by ";" (see Figure 2.57).

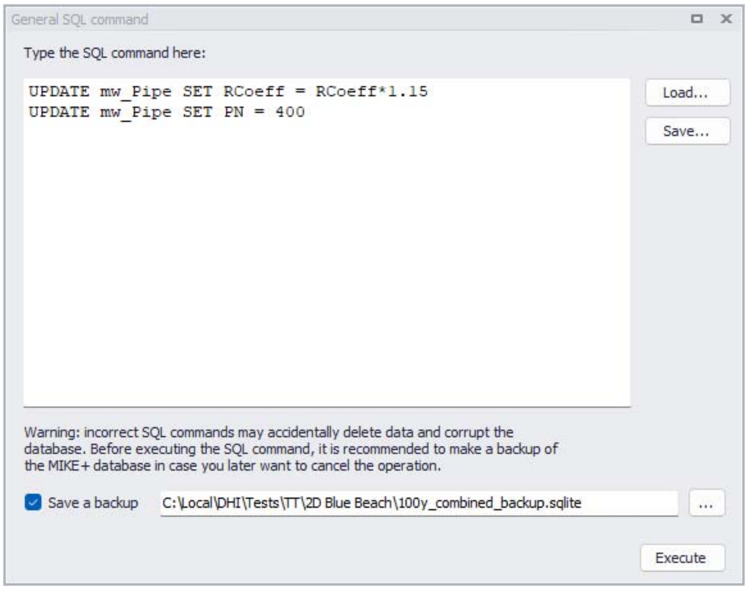


Figure 2.57 Example commands on the General SQL Command window

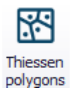
The functionality of the buttons to the right are given below:

- Execute: Executes the SQL statements
- Load: Loads a text file with previously saved SQL statements
- Save: Saves SQL statements to a text file for later reuse

Some more examples are provided in the table below. Please note that SQL commands are dependent on the type of database.

Table 2.2 SQL command examples

Command	Explanation
UPDATE mw_Pipe SET RCoeff=RCoeff*1.15	General update of table: rcoeff value in mw_pipe table is multiplied by 1.15
UPDATE msm_node SET diameter = 1.0 WHERE diameter = 0.99	Specific update of table: diameter in msm_node table is set to 1.0 all the places where the diameter is currently 0.99



Thiessen polygons

A tool that allows the delineation of polygons around point features, e.g. specific tanks, or selected list of nodes/ junctions. The generated Thiessen polygon layer could then be exported to a new polygon layer shapefile.

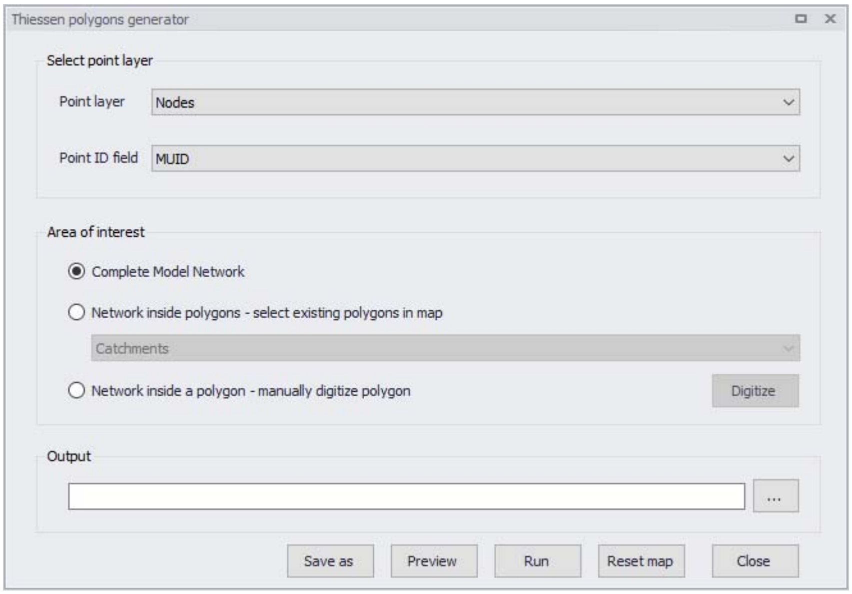


Figure 2.58 The Thiessen Polygons Generator dialog

Sequential labelling

A tool for automatic (bulk) assignment of element IDs(i.e. MUIDs) to selected model elements. See Chapter 19.5 Sequential Labelling Tool *(p. 383)* for details.

Import/Export

MIKE+ offers the ability to import model data from various data sources, such as databases, shapefiles, Excel, etc Similarly, MIKE+ allows you to export model components to various file types using the Import and Export tool, and the ArcGIS Integration option.

Import and Export

The Import and Export tool provides a versatile and flexible environment for exchanging data between various external repositories and the MIKE+ database. The data can be imported to and exported from the MIKE+ database.

See Chapter 6 Import and Export *(p. 147)* for more details on Import/Export functionalities in MIKE+.

ArcGIS Integration

Exports selected model components to a *.GDB file format and opens the model components (selected components) in ArcGIS Pro.

Also see Chapter 4 Linking to ArcGIS Pro *(p. 135)*.

TS Editor

The Time Series Editor tool allows creating and editing *.DFS0 time series files from the MIKE+ interface.

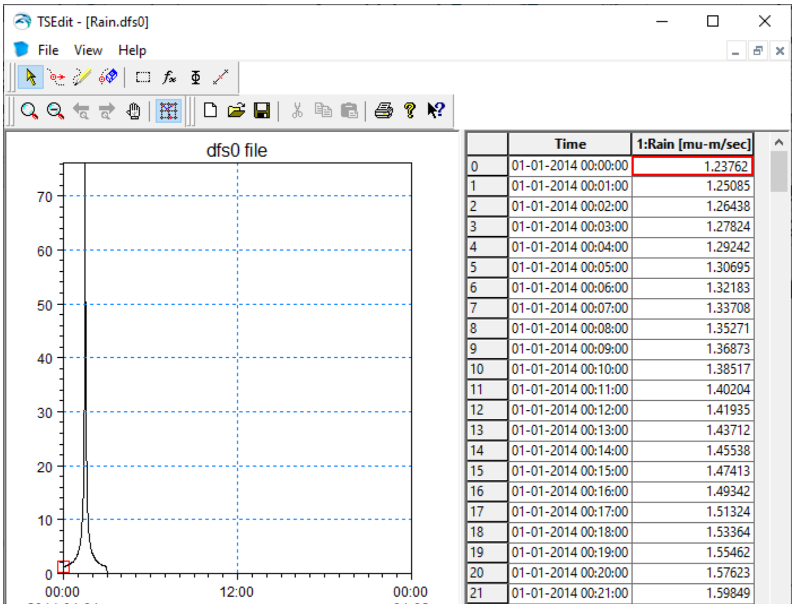


Figure 2.59 MIKE+ Time Series Editor window

To create a new time series, select 'Create new time series' from the ribbon, and then select the source of data to be imported into the time series.

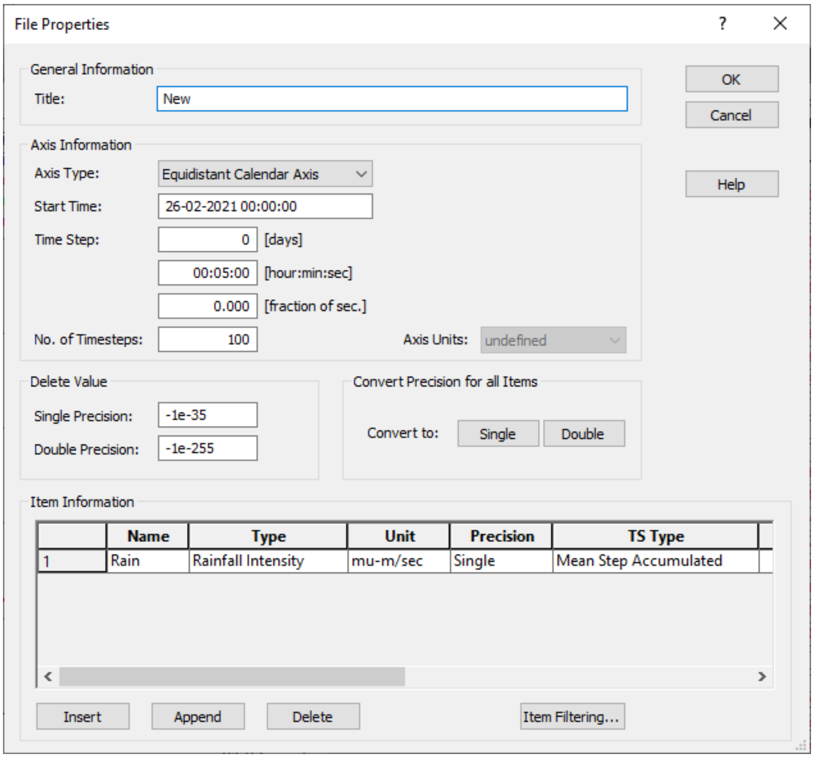
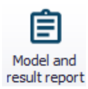


Figure 2.60 Example of TS editor properties setup for a new time series

Alternatively, when the time series editor is opened, go to File | Import in order to import data either from an Excel, ASCII or kmd/km2 file into a new time series file. Note that this operation creates a new time series file, but doesn't import into the active time series file. When you have multiple time series opened in the Time series editor, you can switch from one to another using the 'Window' menu in the upper bar.

Reporting



The Model Result and Report tool offers facilities for setting up reports based on information from model data as well as simulation results.

Also refer to Chapter 20.14 Reports *(p. 473)* for details on creating Reports in MIKE+.

Toolbox

This group offers various tools for data management.

MIKE Zero toolbox

The MIKE Zero toolbox contains a set of tools for e.g. data extraction or format conversion. The following tools are available:

- Profileseries from 2D files: extracts a profile series (.dfs1 file) along a line, from a 2D grid file (.dfs2)
- Timeseries from 2D files: extracts a time series (.dfs0 file) at point locations, from a 2D grid file (.dfs2)
- Mesh Converter: creates a flexible mesh file (.mesh) from other file formats (.dfsu or other formats)
- Mesh Manager: offers options for filtering and/or refining a mesh file.
- Georeferencing Image File: creates a World file for georeferencing a background image file.
- Grd2Mike: converts a grid file in Esri ASCII format (.asc or .txt) to DHI format (.dfs2)
- Mike2Grd: converts a grid file in DHI format (.dfs2) to Esri ASCII format (.asc or .txt)
- Mike2Shp: converts a 2D grid file (.dfs2) or flexible mesh file (.mesh or .dfsu) to a polygon shape file. When converting a .mesh file, a point shape file is also created with elevations at mesh nodes.
- Mike2Txt: converts a 2D grid file (.dfs2) to a text file containing a list of cells and their XY horizontal coordinates and Z elevation in three columns
- Shp2Xyz: converts a shape file to a text file containing a list of points, for use as input in the Mesh Generator
- TxStat: this tool computes statistics from time series (.dfs0 files), profile series (.dfs1 files) and rectangular grid series (.dfs2 files).
- Dfsu statistics: this tool computes statistical values for temporal data in .dfsu result files.
- Interpolate Time Series: creates a new time series file (.dfs0) with values interpolated on a fixed time step
- Preprocessing Temporal Data: generates time series (.dfs0 files) or spatially-varying time series (.dfs2 or .dfsu) of rainfall from measurement stations, using a spatial interpolation or the Thiessen polygons method
- Time Series Batch Conversion: automates the conversion of time series data between .dfs0 format and text or Excel files, using a customised configuration.

Please refer to the MIKE Zero Toolbox user guide for detailed information.

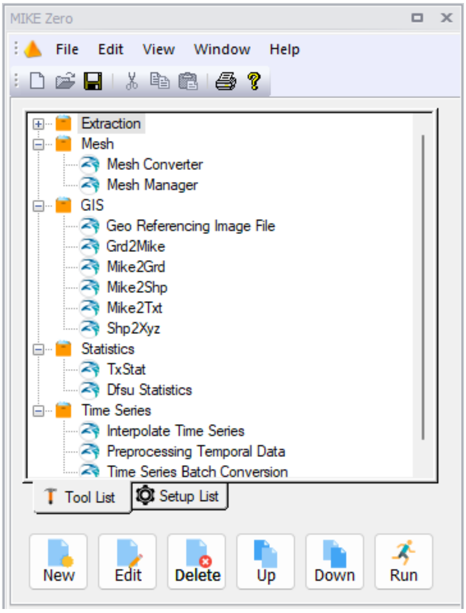


Figure 2.61 The MIKE Zero toolbox as available in MIKE+

Versions management

The Versions Management tool is designed to support a cost-effective model maintenance. The tool can identify, report and visualize differences between any two versions of a model setup, as well as it can automatically update any model with the identified differences. Additionally, the Versions Management tool facilitates the organization of various model versions into a tree-like dependency structure that reflects the actual models' mutual relations and evolution.

See Versions Management *(p. 339)* for more details on the tool.

Combine DEM tiles

The 'Combine DEM tiles' tool is used to generate a single DEM file by combining multiple tiles together. This is especially useful when the input DEM is provided in the form of tiles and should be used in MIKE+ to trace rivers or catchments, in which case the DEM data must be supplied in a single file.

Add or remove input tiles to be included in the final DEM file, using the 'Insert' and 'Delete' buttons at the top.

Supported formats are DHI grid files (.dfs2), Esri ASCII format (.asc, .txt) and .tiff raster files. All input tiles must be in the same format. For .asc and .txt formats, the coordinates of the tiles are assumed to be expressed in the same map projection as the one being defined for the MIKE+ project. For the other formats, it is obtained from the files' metadata.

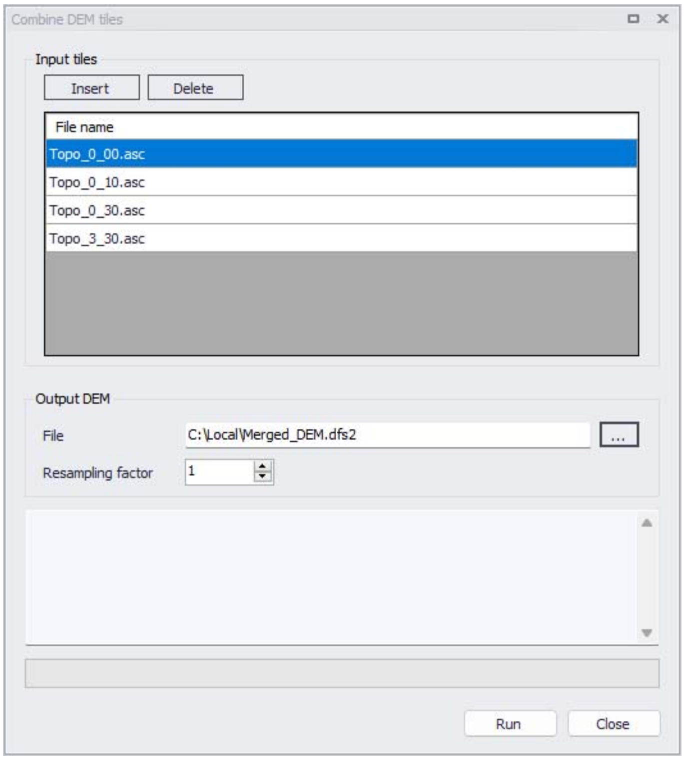


Figure 2.62 The Combine DEM tiles tool

It is mandatory that all input tiles have the same resolution (i.e. the same cell size).

Overlapping tiles are not supported. Gaps between tiles are however allowed, in which case empty cells will be returned in the output DEM.

The output DEM file supports .dfs2, .asc, .txt and .tiff raster file formats. This file format does not have to be the same as the input tiles' format.

The resampling factor may be used to coarsen the resolution of the output DEM. The cell size of the output DEM will be equal to the input tiles' cell size times this factor. So, with a factor of 1, the output DEM will use the same cell size as the input tiles, whereas it will use larger cells with a factor higher than 1. When the resampling factor is higher than 1, the average value from all underlying cells in the input tiles is used in the output DEM.

Simulation

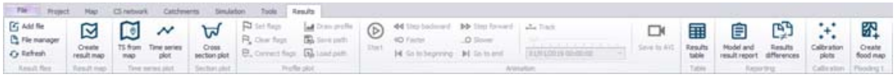
MIKE Cloud simulations

For monitoring simulations executed in MIKE Cloud, and also to start new simulations. See Working with MIKE Cloud *(p. 144)* for more information.

2.9.10 Results Menu

With MIKE+ you can present results in several ways. This includes map plots, time series plots, animations, profile plots, and more.

Chapter 20 Results Presentation *(p. 391)* provides details on results presentation in MIKE+.



Map Operations

Add file

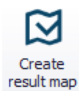
Option for adding/loading result files into the project.

File manager

Activates the Results View panel, wherein various result files loaded in the project are managed.

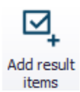
Refresh

Refreshes values for overwritten/modified result files.



Create result map

Launches the Result Items dialog, from where simulation results may be presented in result map plot.



Add result item

Option for adding a result item to an existing result map plot.

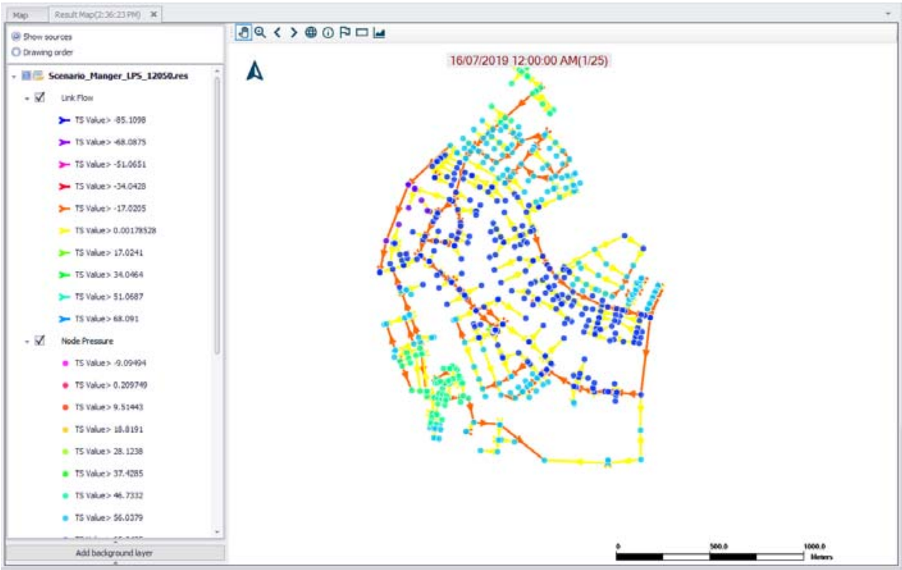
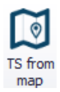


Figure 2.63 Example result map plot

Time Series Plot

Offers tools for creating time series plots of result file items. Also see Chapter 20.4 Time Series Plot *(p. 403)*.



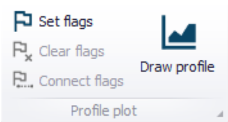
TS from map

Option for quickly creating a time series plot of simulation results by selecting features from the main Map.

Time series plot

Launches the Result Items dialog, from where simulation results may be presented in a time series plot. A tabular view of time series values is also available from the resulting TS Plot window.

Profile Plot



The Profile Plot toolbox on the Results menu ribbon contains tools and functionalities for creating longitudinal profile plots from **result maps**. I.e. they work for result map items, and **not** main Map items.

Set Flags

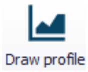
Tool for placing flags at nodes along a profile on a result map.

Clear Flags

Remove set flags on a result map.

Connect Flags

Identifies and highlights the path between set flags on the result map.



Draw Profile

Creates a longitudinal profile plot in a new widow. Use the 'Add result item' tool on the window to add result items to the profile plot as needed. Chapter 20.6 Profile Plots *(p. 417)* provides more details on the Profile Plot toolbox.

Figure 2.64 MIKE+ profile plot example

Save path

Saves the list and location of the current flags to a file (*.path).

Load path

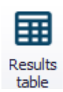
Re-uses flags previously saved to a *.path file.

Animation

This toolbox offers functionalities for animating dynamic simulation results on the result map plot. Various tools allow control of the animation.

See Chapter 20.13 Animations *(p. 471)* for more details.

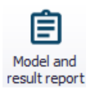
Table



The Results Table tool launches the Result Items dialog, from where simulation results may be presented in a table. The results table provides an overview of all or selected results in tabular form. Various information is available depending on the type of result file selected.

See Chapter 20.5 Results Table *(p. 410)* for more details on result tables.

Reporting



The Model Result and Report tool offers facilities for setting up reports based on information from model data as well as simulation results.

See Chapter 20.14 Reports *(p. 473)* for details on creating Reports in MIKE+.

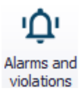
Calibration

Calibration is primarily focused on reproducing the observed hydraulics and water quality behaviour of the system in terms of flow depth/pressure, flow discharges and velocities. It involves comparisons between model simulation results and field measurements.

The Calibration Plots functionality offers options for setting-up comparison plots between simulated and observed data at various points in the model.

See Chapter 24 Calibration Plots *(p. 519)* for details on Calibration in MIKE+.

Alarms (For WD models)



The Alarms and Violations tool provides a way to impose user-defined checks for Water Distribution model results. It allows for quick examination of the performance of elements that are important to the WD system, or of particular interest to the user.

See Chapter 20.20.9 Alarms and Violations *(p. 487)* for details on Alarms and Violations in MIKE+.

2.10 The Toolbars

MIKE+ toolbars are located in menu item ribbons on top of the interface or at the borders of maps. Toolbars and tools on menu ribbons have been described in previous sections (2.9 Main Ribbon Menus *(p. 51)*)

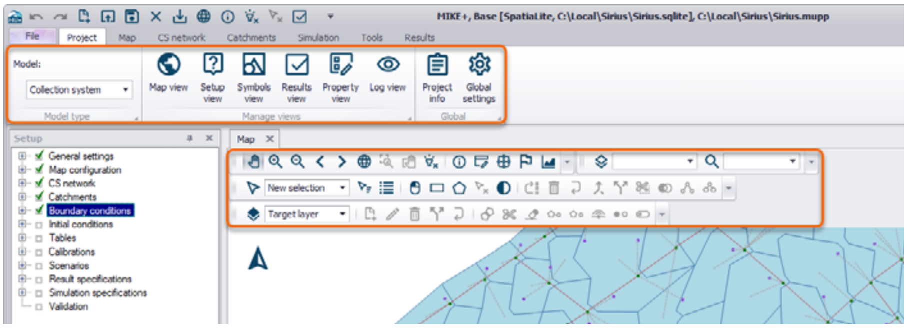
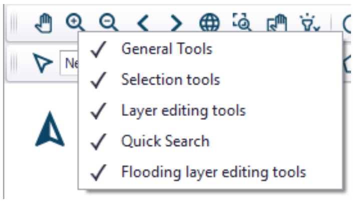


Figure 2.65   Toolbars are displayed in menu ribbons and map borders

2.10.1 Map Toolbars

Toolbars around map borders can be activated/deactivated and placed/floated where you prefer. The toolbars provide shortcuts for program functions.

- For any tool on a toolbar, the same functionality can always be found in a main menu ribbon.
- Individual toolbars can be switched on and off. MIKE+ saves the current toolbar combination for the next MIKE+ session.
- Activate a map toolbar by right-clicking on an active toolbar and selecting from the list of available map toolbars shown.



- Re-set the displayed map toolbars on the main Map via the ‘Reset toolbars’ option from the Map local context menu (i.e. right-click on the main Map).

The displayed toolbars get automatically activated or de-activated (greyed-out) according to the presently active graphical window or dialog.

For customizing, please see Chapter 3.4 Customizing the User Interface *(p. 131)*. Also see Chapter 8.2.1 Toolbars *(p. 202)* for details on tools and toolbars.

General Tools

Offers quick access to tools for navigating around the Map, querying model element properties, and creating profile plots from the main Map. The tools are described in Chapter 2.9.3 Map Menu *(p. 61)*.

- Pan
- Zoom in
- Zoom out
- Zoom to previous
- Zoom to next
- Zoom to full extent
- Zoom to selection
- Pan selection
- Clear highlighted
- Identify: Activate this option and click an element on the map to show its properties in the Property view. If a model element (e.g. pipe or node) exists at this location on the map, it will display this element's properties. If simulation results are available, then they will also be visible in this Property view. If no model element exists at the clicked location, or if the layer is not visible or not selectable, the Identify tool will select items from background feature layers
- Open layer editor
- Network connectivity
- Set flags
- Profile manager
- Measure

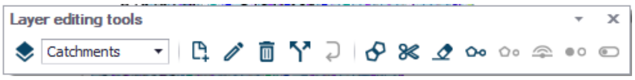
Selection Tools

Presents shortcuts to tools for selecting model elements from the main Map for e.g. editing or further processing. Also see the section on Selection *(p. 63)* for further information.

- Selection filtering

- Selection manager
- Select by click
- Select by rectangle
- Select by polygon
- Clear selection
- Invert selection: inverts the selection within a feature layer
- Move selection
- Delete selection
- Reverse selected links
- Merge selection
- Split selected links
- Connect selected demand allocations. For WD models.
- Connect selected pumps to pump station. For WD models.
- Clip catchments by selection. For CS models.
- Erase catchments by selection. For CS models.
- Connect selected load points. For CS models.
- Connect selected catchments. For CS models.

Layer Editing Tools

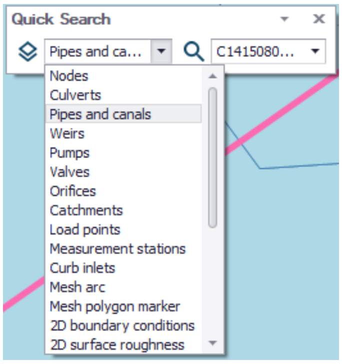


Provides easy access to data layer editing tools on the Map. Also refer to Chapters 2.9.4 CS/WD Network *(p. 77)* and 2.9.6 Catchments Menu *(p. 86)* for more details on the tools listed below.

- Create
- Edit
- Delete
- Split
- Reverse link
- Append catchment. For CS models.
- Clip catchment.For CS models.
- Erase catchments. For CS models.
- Connect catchment. For CS models.
- Connect demand allocation. For WD models.
- Connect pump station. For WD models
- Delete catchment connection
- Connect station
- Change element type

- Open/Close element. For CS models. This tool ticks/unticks the 'Enabled' parameter for Pipes and Canals. This option is useful for simulating, e.g. link blocking or removal scenarios.

Quick Search



Option for quickly finding model elements on the Map. The tool zooms into and highlights (in pink) the specified element on the main Map.

2.11 Languages

MIKE+ can be run in a number of languages. You can switch language in MIKE+ via the Global Settings functionality in the Project menu ribbon.

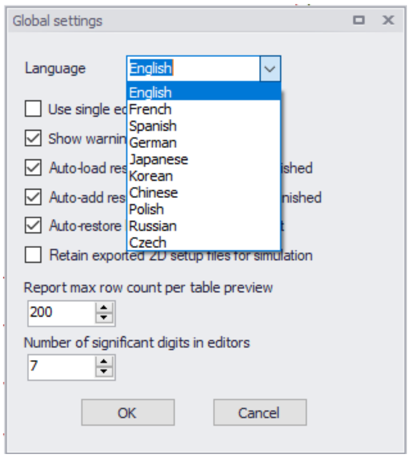


Figure 2.66 Language setting options in MIKE+

The application needs to be restarted when switching Language for changes to take effect.

Note that the corresponding Language should have been installed during installation of MIKE+. The application needs to be restarted when switching Language for changes to take effect.

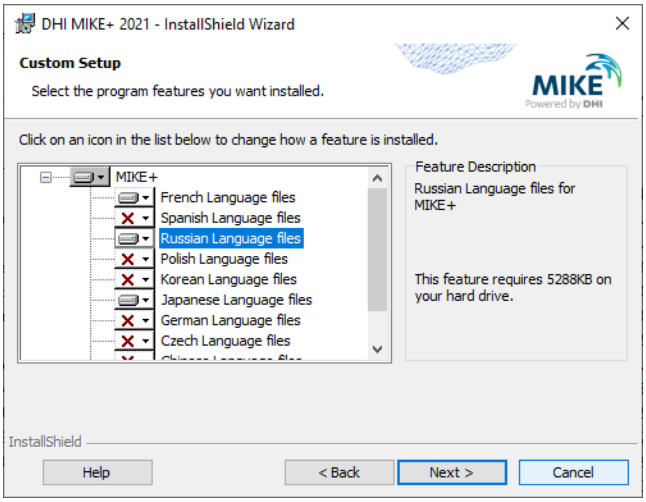


Figure 2.67 Language installation window during program installation

See also Chapter 3.2.1 Languages *(p. 128)*.

2.12 Selecting a Coordinate System

Choosing a coordinate system for your MIKE+ database may be an important part of setting up a MIKE+ database. The default coordinate system is 'Local Coordinates' and allows you to operate most of MIKE+ features. It is basically treated as a rectangular system in unknown units.

However, several features of MIKE+ will only work when the coordinate system is properly defined. So if any of the below points apply to you, you should consider setting up the coordinate system correctly:

- You would like to visualize your model against an online background map on the MIKE+ interface.
- You need to calculate pipe lengths and other geographic measures in a specific unit.
- You have a corporate GIS and will be using data from this in MIKE+.
- You receive data in different coordinate systems and projections and need to overlay them correctly.
- You want to use MIKE+ features that require the system to calculate the scale (e.g. maps to scale, switching on and off features/labels depending on scale, etc.)

These functions will only work properly if MIKE+ knows about the units and other definitions in your coordinate system. If you do not already know your coordinate system, your GIS department or GIS data vendor will normally be able to tell you.

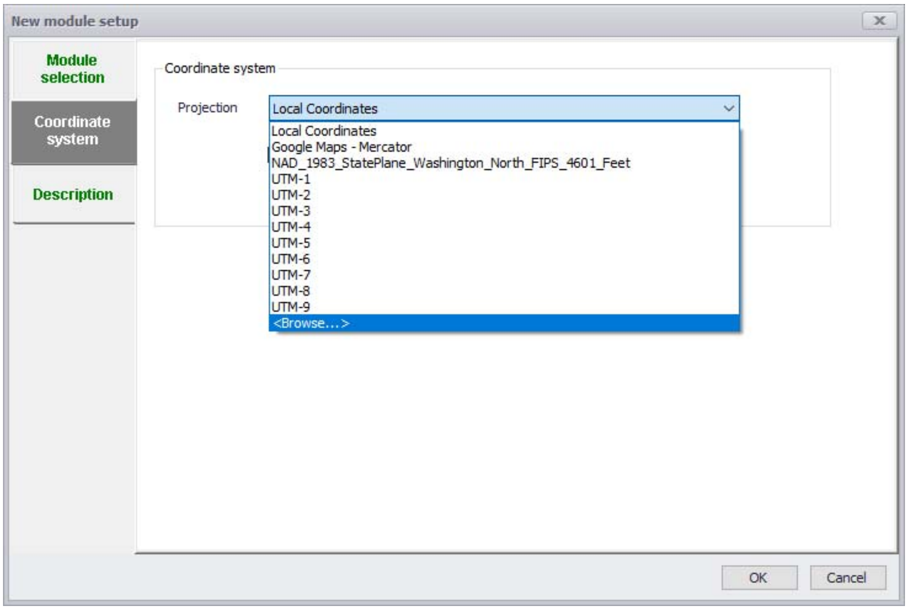


Figure 2.68   Defining Coordinate System for MIKE+ project

The New Module Setup dialog for creating a new MIKE+ project (Figure 2.68) handles the setting of Coordinate System.

In many cases you will simply select one of the predefined coordinate systems. Most often coordinate systems used in MIKE+ will be of the type 'Projected coordinate systems', and you simply browse through the available coordinate systems options and select the one matching your data.

Click OK and the coordinate system is defined, after selecting the system you should double-check the values in the domain as the defaults are changed when you select the coordinate system.

You can, however, also create a new one or by importing a coordinate system from e.g. a *.PRJ projection file. Choose the ‘<Browse...>’ option to access the Map Projection Editing dialog (Figure 2.69) for doing this.

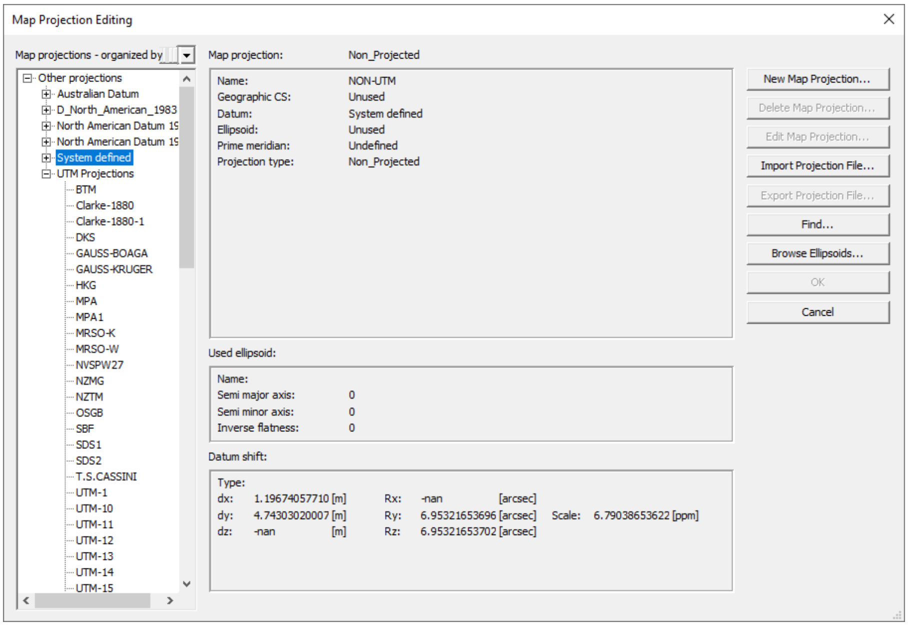


Figure 2.69 Defining a new or importing a coordinate system

Tip

If you do not know your coordinate but still want to utilize the GIS functionality for length and scale calculations then use a rectangular system (e.g. UTM) in the unit (typical meter or feet) that you are working in. This will provide scaling and calculations, but unexpected results may occur if you open the database together with other (correctly geo-located) data.

2.13 World Files for Background Images

Images may be added as background images for the Map in MIKE+.

Images are interpreted as raster data, where each cell in the image has a row and column number. In order to display images with GIS data, it is necessary to establish an image-to-world transformation that converts the image coordinates to real-world coordinates.

This transformation information is stored with the image.

Some image formats, such as GeoTIFF, and ESRI grids, store the georeferencing information in the header of the image file. MIKE+ uses this information if it is present.

However, other image formats store this information in a separate ASCII file. This file is generally referred to as the world file, since it contains the real-world transformation information used by the image. World files can be created with any editor.

World file naming conventions

It is easy to identify the world file which should accompany an image file: world files use the same name as the image, with a "w" appended. For example, the world file for the image file mytown.tif would be called mytown.tifw.

How georeferencing information is accessed

The image-to-world transformation is accessed each time an image is displayed (e.g., when you pan or zoom). The transformation is calculated from one of the following sources, listed in order of priority:

- The world file
- The header file (if the image type supports one)
- From the row/column information of the image (an identity transformation)

Because a world file has higher priority, you can override the header file transformation information by creating your own world file.

World file contents

The contents of the world file will look something like this:

20.17541308822119
0.00000000000000
0.00000000000000
20.17541308822119
424178.11472601280548
4313415.90726399607956

When this file is present, MIKE+ performs the image-to-world transformation. The image-to-world transformation is a six-parameter affine transformation in the form of:

x1 = Ax + By + C
y1 = Dx + Ey + F

where:

x1 = calculated x-coordinate of the pixel on the map
y1 = calculated y-coordinate of the pixel on the map
x = column number of a pixel in the image
y = row number of a pixel in the image
A = x-scale; dimension of a pixel in map units in x direction
B, D = rotation terms
C, F = translation terms; x,y map coordinates of the center of the upper-left pixel
E = negative of y-scale; dimension of a pixel in map units in y direction

NOTE: The y-scale (E) is negative because the origins of an image and a geographic coordinate system are different. The origin of an image is located in the upper-left corner, whereas the origin of the map coordinate system is located in the lower-left corner. Row values in the image increase from the origin downward, while y-coordinate values in the map increase from the origin upward.

The transformation parameters are stored in the world file in this order:

20.17541308822119 - A
0.00000000000000 - D
0.00000000000000 - B
-20.17541308822119 - E
424178.11472601280548 - C
4313415.90726399607956 - F

2.14 Keyboard and mouse shortcuts

Many functionalities can be accessed using shortcuts as described below. When a button is associated with a shortcut, this shortcut is presented in the fly-by text of the button.

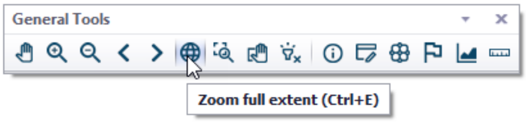


Figure 2.70 Example of shortcut presented in fly-by text

General functionalities

Create a new project (File | New): Ctrl+N.

Save the opened .mupp project file (File | Save): Ctrl+S.

Open a project file (File | Open): Ctrl+O.

Open online help (File | Help | Online help): F1.

Exit MIKE+ (File | Exit): Alt+F4.

Switch between open windows: Ctrl+Tab. This opens a window listing all opened editors and tools. Keep pressing the Ctrl key, and press the Tab key to move to a different window's name in the list. Release the keys to open the selected window. Undocked windows are shown in the left list of the pop-up window whereas docked windows are shown in the right list (use Left and Right keys to switch between the two lists).

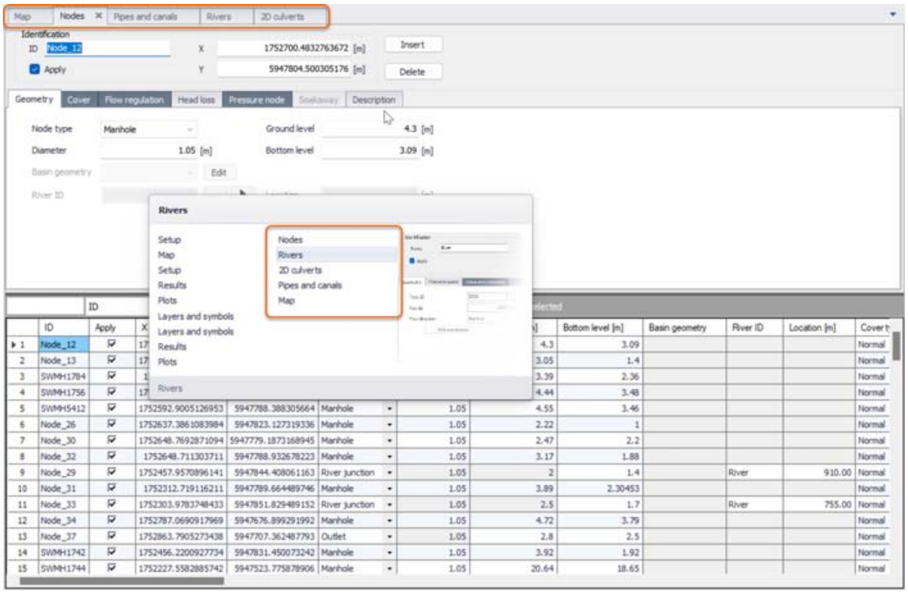


Figure 2.71 Switching between windows using Ctrl-Tab shortcut

Close current / active window: Ctrl+W. The active window is the one being edited and is shown with a slightly darker title. Note that moving the cursor above the map makes the map active even if no action has been made on the map.

Open the 'Setup' view: Shift+F1.

Open the 'Layers and symbols' view: Shift+F2.

Open the 'Results' view: Shift+F3.

Open the 'Plots' view: Shift+F4.

Run active simulation: Ctrl+R.

Edit active cell in a table: F2.

Delete records in a table: Delete. This deletes the selected records, if any, or the active row otherwise.

Copy selection: Ctrl+C.

Paste selection: Ctrl+V.

Map-related functionalities

Open the Map view: Ctrl+M.

Zoom to full extent: Ctrl+E.

Zoom to specific feature: double-click the row number in the overview table from the feature's editor, or search the feature in the Quick Search toolbar on the map.

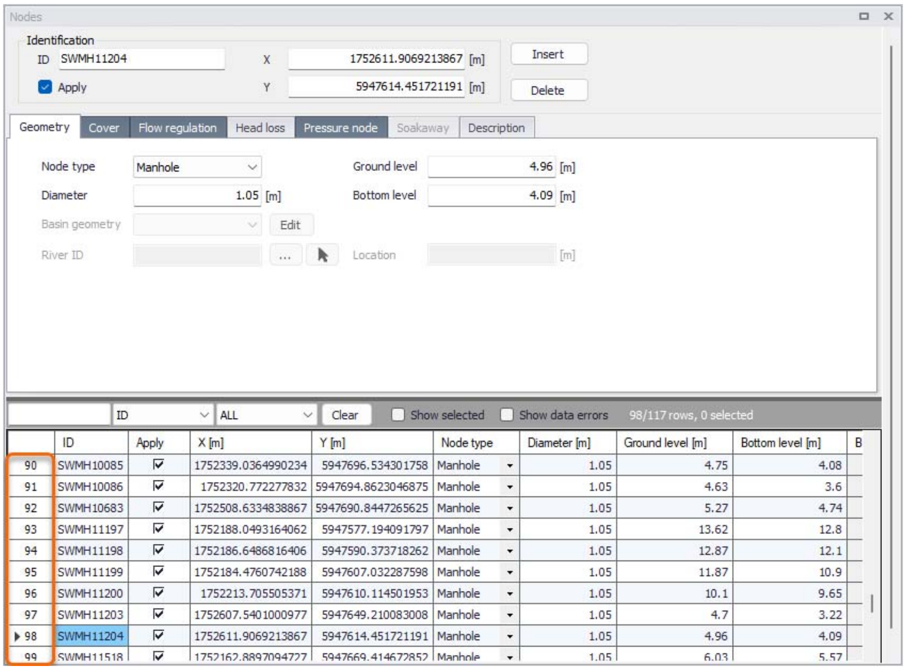


Figure 2.72 Double-click the row number column to zoom to corresponding record on the map

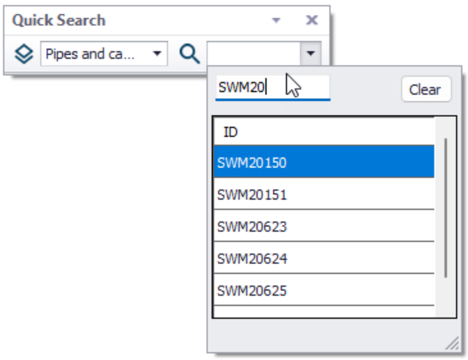


Figure 2.73 Searching and zooming to a feature using the Quick Search toolbar

Pan: various shortcuts may be used

- Shift + left mouse button
- Up / Down / Left / Right keys

- Pressing mouse wheel button.

Add to selection: Ctrl+Shift+click.

Clear selection: Ctrl+U.

Add new layer: Ctrl+L.

Open the Measure tool: Ctrl+D.

Open the Bookmark manager: Ctrl+Q.

Cross sections editor

Add a marker to a cross section's point: select the point on the cross section plot and press the marker number on the keyboard.

Delete a marker: select the point on the cross section plot which holds the marker, then press 'Ctrl' key and press the key with marker number to be removed.

Result windows

In profile plots, use Shift + left mouse button to enable 'Zoom to rectangle', or use the mouse wheel to zoom in or out.

In time series plots, use Shift + left mouse button to enable 'Zoom to rectangle', or use the mouse wheel to zoom in or out. Hold down the Ctrl key to pan.

3 Customizing MIKE+

The MIKE+ interface can be customised to suit visual preferences and input requirements, such as the:

- Choice of unit system and default values
- Choice of language
- Visual set up of the interface

3.1 Units, Default Values and Numeric Formats

MIKE+ is fully flexible concerning the applied units for numeric attributes, number of decimals and default values for any attribute in the database. The system allows for a number of predefined environments. By these means, any MIKE+ user can set up the current MIKE+ project to suit established corporate and/or national standards and conventions. The actual unit environment is valid for the entire MIKE+ project, i.e. both for water distribution and for rivers, collection system and/or 2D overland flows.

In addition to maintaining the database and presenting the computational results in required units, the system ensures appropriate unit conversions during the import of existing projects, without any interference by the user. Equally, the system takes into account the project data units and formats when submitted to the computational engines, which are automatically converted into the formats required by the computational engines.

3.1.1 Selecting an Appropriate Unit Environment

The term "unit environment" is a pre-defined set of definitions for units, default values and display formats. The unit environment can be in SI units (International System of units) or US units (United States customary units). The unit system controls in which units the various values are expressed in the editors, and also controls the units used for results presentation.

For rivers, collection systems and overland flows the unit system can be selected from the following options:

- SI units, CMS: SI environment, with flows in m^3/s
- US units, CFS: US environment, with flows in ft^3/s
- SI units, LPS: SI environment, with flows in l/s
- SI units, MLD: SI environment, with flows in Ml/day (million liters per day)
- SI units, LPM: SI environment, with flows in l/min
- SI units, CMH: SI environment, with flows in m^3/h
- SI units, CMD: SI environment, with flows in m^3/day

- US units, GPM: US environment, with flows in gal/min
- US units, MGD: US environment, with flows in Mgal/day (million gallons per day)
- US units, MIGD: US environment, with flows in Mgal/day (million imperial gallons per day)
- US units, AFD: US environment, with flows in ac-ft/day

For water distribution systems, there are ten pre-defined unit environments within the SI and US unit groups which differ from each other by the applied units for flows and volumes.

Within the SI group, the following unit environments are available:

- SI units, LPS: SI environment, with flows in L/s
- SI units, LPM: SI environment, with flows in L/min
- SI units, MLD: SI environment, with flows in ML/day
- SI units, CMH: SI environment, with flows in m^3/h
- SI units, CMD: SI environment, with flows in m^3/day

Within the US group, the following unit environments are available:

- US units, CFS: US environment, with flows in ft^3/s
- US units, GPM: US environment, with flows in gal/min
- US units, MGD: US environment, with flows in Mgal/day (million gallons per day)
- US units, MIGD: US environment, with flows in IMGD Mgal/day (million imperial gallons per day)
- US units, AFD: US environment, with flows in ac-ft/day

For SWMM5 collection system and overland flows, there are six pre-defined unit environments within the SI and US unit groups which differ from each other by the applied units for flows and volumes.

Within the SI group, the following unit environments are available:

- SI units, CMS: SI environment, with flows in m^3/s
- SI units, LPS: SI environment, with flows in L/s
- SI units, MLD: SI environment, with flows in ML/day (million liters per day)

Within the US group, the following unit environments are available:

- US units, CFS: US environment, with flows in ft^3/s
- US units, GPM: US environment, with flows in gal/min

- US units, MGD: US environment, with flows in Mgal/day (million gallons per day)

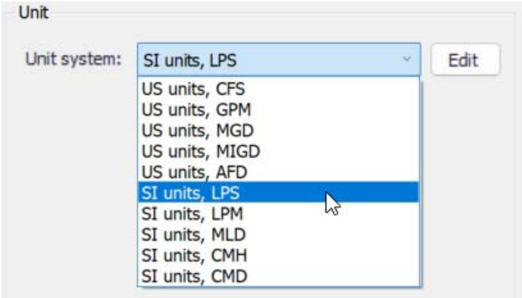


Figure 3.1 Selecting the unit environment

The unit environment will apply default units to all attributes (parameters) in the project. These units can then be customised with the 'Edit' button. The button opens the 'Units customisation' dialog, which contains a first table to select units for given attribute types. As an example, this table can be used to change the unit applied for 'Water level' type, and all attributes using this type will use the selected unit. Some types in this table use a "Default mixed unit", which means that different variables use per default a different unit, although they relate to the same type: this corresponds to a default setting and it is not possible to change the individual units for this type, unless using the second table described below.

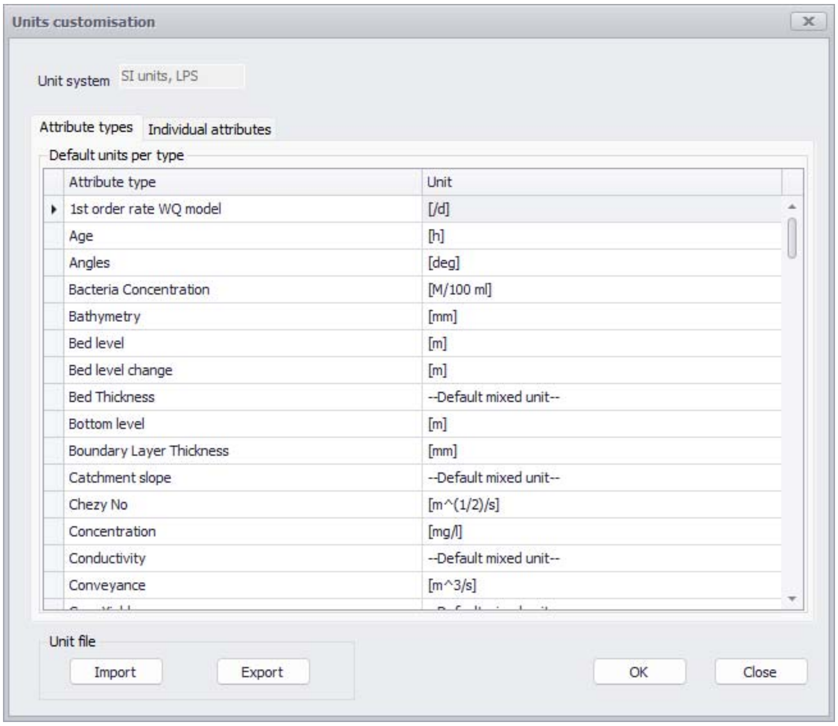


Figure 3.2 Customizing units

Prior to customising units for an attribute type, it may be useful to identify the attribute type used by the attributes which need to be changed. To achieve this, open the Property view (from the Project tab in the ribbon), and select the 'Eum info' button below. This will display a list of attributes available in the opened editor, where the attribute type is shown in the second column.

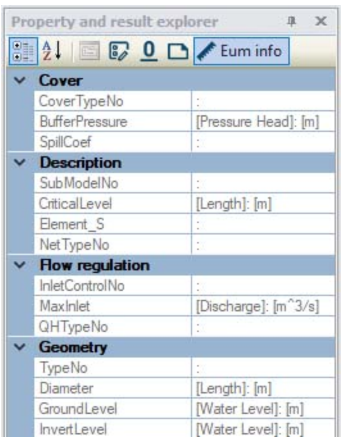


Figure 3.3 Editing units

The second table in the 'Units customization' dialog is used to select the unit for specific attributes. To control the unit for a specific attribute, press the 'Insert' button to add a new item to the table. Then select the table the attribute belongs to, select the attribute's name, and finally select its new unit.

The customized units will be saved in the database. If required, they can be exported to a configuration file by use of the 'Export' button, and then imported in another database with the 'Import' button.

The third tab in the 'Units customisation' dialog contains the units selection for control rules, for use in Collection system and River network simulations. This tab contains a first table for the various sensor types, where the selections control in which units the sensor values are considered in the expressions, for example in the 'Condition' expression of a control rule. It also contains a second table for the various action types, where the selections control in which units the action expressions (returning e.g. a weir crest level or a pump discharge) are expressed.

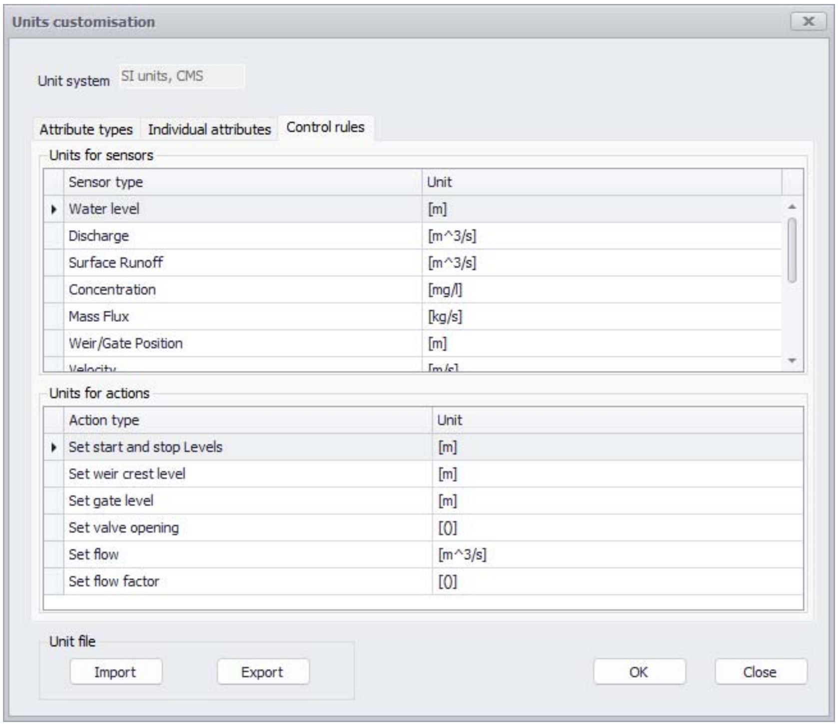


Figure 3.4 Selecting units for control rules

It is important to note that the selected control rules units are bound to the numerical values specified in the various expressions (e.g. equations) that may be used to define the condition for a control rule to apply, or to set the action. Changing these units may therefore require updating these expressions accordingly, if any. For this reason, changing the unit system doesn't update the control rules units, to keep consistency with possible existing expressions. If the control rules units are changed, updating existing expressions would have to be done manually.

3.1.2 Customizing Unit Environment

The unit environment is specified when the model is originally created but can be changed or modified at any stage. The units are automatically converted in the database. i.e. it is possible to change an existing database from one system to another.

Modification of the unit environment configuration in the current project is done in the Setup tab (in default View, this view is visible on the left side of the interface) which is also accessible via Project| Setup View, and then in the Setup View go to General Settings| Model type. On the top right of the dialog, a drop-down list of the unit systems is available, which can be further customised with the 'Edit' button.

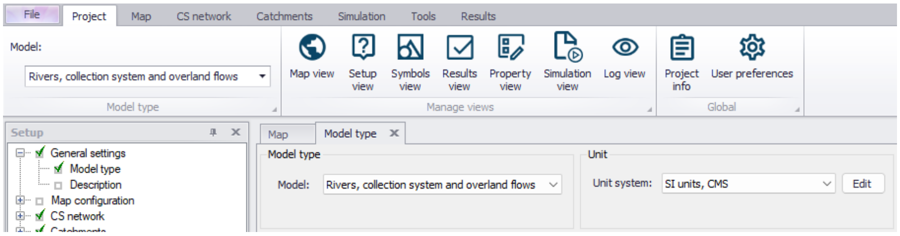


Figure 3.5 Custom Units dialog

3.2 User preferences

Through the 'Project | User preferences' dialog, it is possible to setup several general settings for the MIKE+ product installation.

See "User preferences" on page 54 for more detail.

3.2.1 Languages

You can switch language on the fly in your MIKE+ application by choosing Project | User preferences and then selecting the language of your choice.

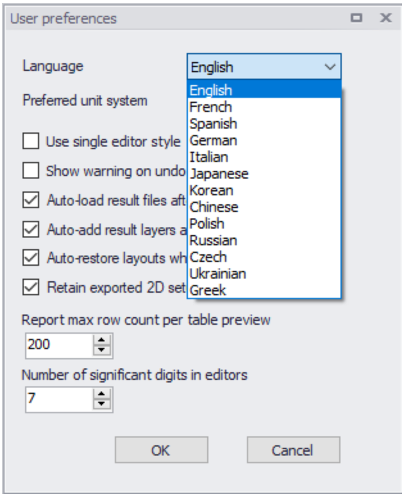


Figure 3.6 Switching language in MIKE+

Note that the corresponding Language should have been installed during installation of MIKE+.

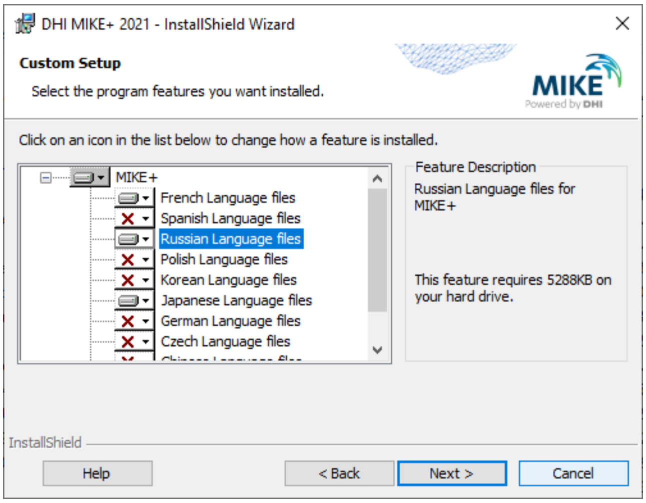


Figure 3.7 Language installation window during program installation

The application needs to be restarted when switching Language for changes to take effect.

3.3 General Settings

Through the Project| Setup View| General Settings menu it is possible to set general properties in the MIKE+ project.

Within the ‘Model type’ editor, the user can:

- Switch from one mode to the other i.e. Water Distribution or Collection systems
- Change the unit system as discussed in section 3.1.1 Selecting an Appropriate Unit Environment *(p. 123)*
- Select additional modules to be made visible in the Setup View for editing, display related tools in the interface ribbon and Map View, and run the model with these modules. E.g. for collection systems modelling this could include rainfall runoff, water quality, etc. Or for water distribution modelling this could include fire flow analysis, pipe criticality etc.

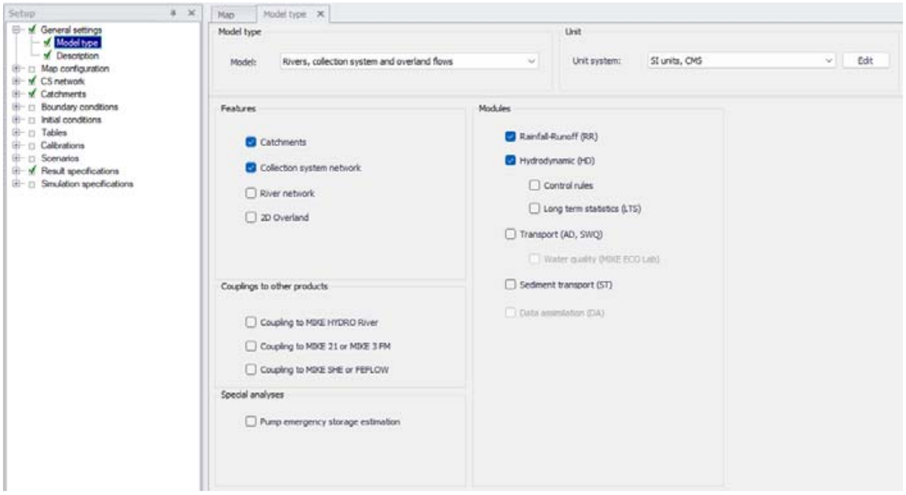


Figure 3.8 General Settings – Model type dialog in MIKE+

In general settings, it is also possible to add a description of the model details. The empty field can be used to provide a detailed description of the model including its purpose, how the model is schematised, limitations, etc.

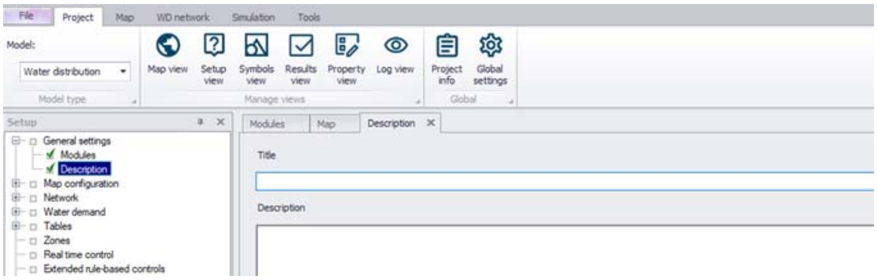


Figure 3.9 General Settings – Description dialog in MIKE+

3.4 Customizing the User Interface

A number of useful features exist in MIKE+ to customise how the interface is displayed, to suit a range of preferred working methods.

3.4.1 Minimise the Ribbon View

As a default, a ribbon is displayed along the top of the screen to enable easy access to views and tools. However, to enable more space on the screen for other information it is possible to unpin the ribbon by a right mouse click on the ribbon and selecting “minimise the ribbon”. This ensures that the ribbon is not always seen, but only visible when one of the options on the top menu is selected.

3.4.2 Quick Access Toolbar

The quick access toolbar with a few regularly used functions (undo, redo, save, close, etc) is available as a default above the ribbon. For a more traditional look, it is possible to place this toolbar below the ribbon by a right mouse click on the ribbon and selecting “Show Quick Access Toolbar below ribbon”.

Add regularly used tools to the toolbar by a simple mouse right click on the desired tool in the ribbon view and selecting “Add to Quick Acess Toolbar”.

To remove buttons from the quick access toolbar, mouse right click on the button on the toolbar to be removed and select “Remove from Quick Access Toolbar”.

The buttons visible on the quick access toolbar can also be customised by clicking on the drop-down arrow on the right hand side of the toolbar and ticking/unticking the desired tools to be displayed.

3.4.3 Customizing Windows

MIKE+ supports multi-screen use and the ability to customise views for efficient workflows. All tools and tables can be resized, maximised (double-click), shifted to another screen, docked on the main central part of the interface, or floated (double-click on the docked tab heading).

Right click on an open tab to dock/float (alternatively double-click on the tab), display horizontally/vertically (neatly display from the bottom or right of the screen), or close the current screen/all but the current screen. It is also possible to create multiple groups of tabs (especially useful when using multiple screens) by right clicking on a tab and selecting “move to next tab group.”

To toggle between open windows, click on the small triangle on the top right of each tab group and select/deselect tabs.

3.5 User defined columns

Most of the editors contain a table with a pre-defined list of columns, corresponding to the pre-defined attributes being edited in the editor. It is however possible to add extra columns containing custom data. This is achieved by right-clicking over the table and selecting the option 'Add user defined column'.

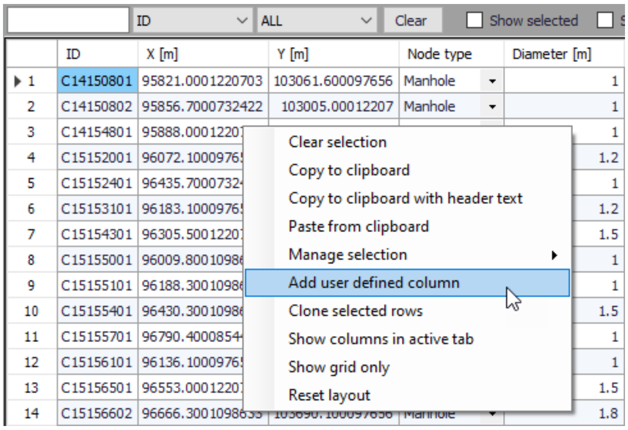


Figure 3.10 Adding a user defined column

Three types of columns can be created:

- A new column with values stored in the database: when this option is selected, the data type must be selected (double, string, integer, or date and time). Two names must also be specified: one which will be shown in the header of the table, and one for the column's name in the database. When the column is created, it will initially be empty, and it can then be populated with the same tools as for any other column. It can also be removed from the table by right-clicking on the header of the column and selecting 'Remove column', but the column will not be deleted in the database. Once this type of column has been removed, it can be restored in the table by adding again a user-defined column and selecting the option 'Restore removed column'.
- A column showing an expression, where the expression is a function of other attributes / columns. When this option is selected, the name to be shown in the header of the table must be specified. This type of column is not saved in the database. The expression is specified using the 'Edit' button which opens the Expression Editor. When the column is created, its content is updated dynamically as soon as the source attributes are updated.

- A column showing result values. When this option is selected, the name to be shown in the header of the table must be specified. This type of column is not saved in the database, but read from a result file. Note that this type of column is only available for pipe networks editors, and excluding the 'Air chambers' editor. Use the 'Select' button to select the type of result to be shown, using the following settings:
  - Result file: select the input result file to read results from. The result file must have been loaded in MIKE+ from the 'Results' tree before it can be selectable in this list.
  - Item: choose the result item among the list available in the selected result file (e.g. water level in nodes, discharge in links, etc.).
  - Value type: select which value to show for the selected result item. Possible options are 'Animation' (showing time-varying results, for the active time step selected in the ribbon), 'Minimum' (showing the minimum value throughout the simulation period), 'Maximum' (showing the maximum value throughout the simulation period), 'Average' (showing the average value computed from all time steps in the result file) or 'Time step' (showing values at a particular time step).
  - Time step: Select the date/time of the results to show, when the selected value type is 'Time step'.

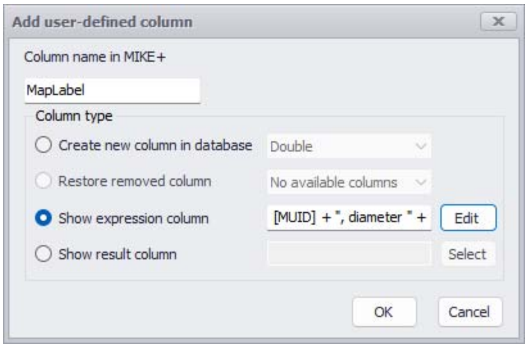


Figure 3.11 Specifying a user defined column

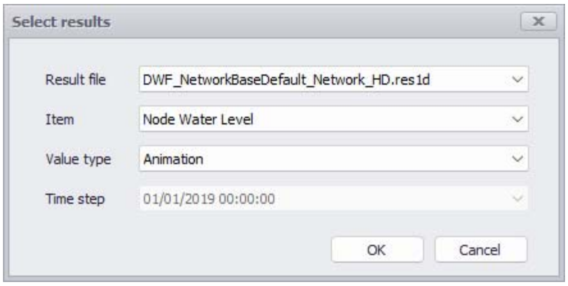


Figure 3.12 Defining results to show in a user defined column of results

4 Linking to ArcGIS Pro

ArcGIS Pro is the latest desktop GIS software from ESRI. ArcGIS Pro allows to explore, visualize and analyse data, create 2D maps and 3D scenes. Furthermore the work can be shared on ArcGIS Online or ArcGIS Enterprise portal.

Depending upon your license conditions you will have a number of possibilities of amending the functionality of MIKE+ with the more general functionality found in ArcGIS Pro. MIKE+ allows you to export selected model components to a geodatabase (*.GDB) file format and work with the (selected) model components in ArcGIS Pro.

MIKE+ operates on top of a SQLite| PostGIS database which can be quickly integrated into a personal geodatabase and stores all data in a designated data structure.

4.1 ArcGIS Integration Tool

Access the ArcGIS Integration tool from the Tools menu ribbon.

This launches the ArcGIS Integration dialog, from where:

- Select the elements which shall be linked to data in an ArcGIS Pro geodatabase. Choose from model- and result-related items, switching between the two types via the dropdown menu at the top.
- Define the file path and name of the geodatabase file to be created for the ArcGIS Pro project.
- Click on the ‘Link to ArcGIS Pro’ button to start the export of the MIKE+ (selected) data to the ArcGIS Pro database. The export can often take several minutes.

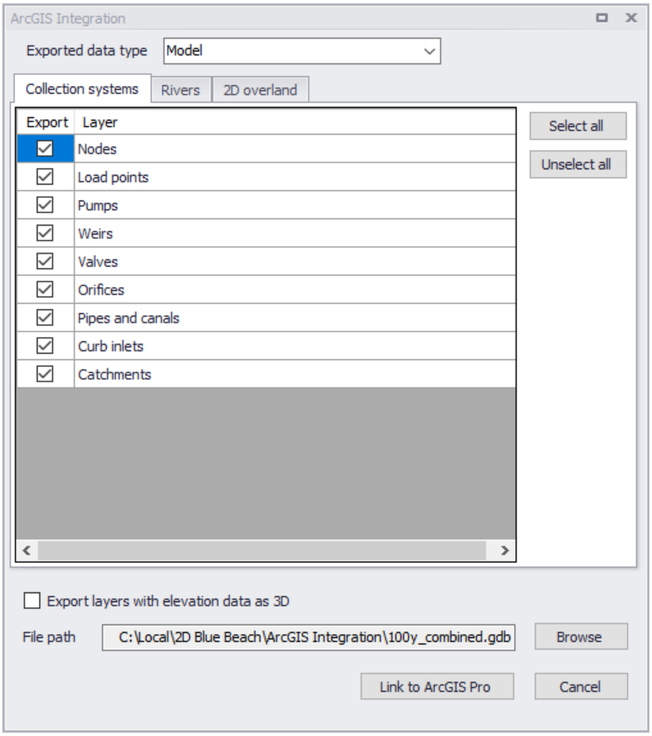


Figure 4.1 The ArcGIS Integration dialog

When selecting the 'Result' data type, the tool will list the result layers displayed on a map, and you can select which of these layers to export to the ArcGIS Pro database. Therefore, it is a prerequisite that the expected results layers are added to the main map or a result map in MIKE+, before they can be exported.

For 1D results, the exported time step of the results corresponds to the time step of the results shown on the map when running the tool. For 2D results, multiple time steps can be exported, following the settings in the 'Time steps selection' window, opened using the '…' button for each result file.

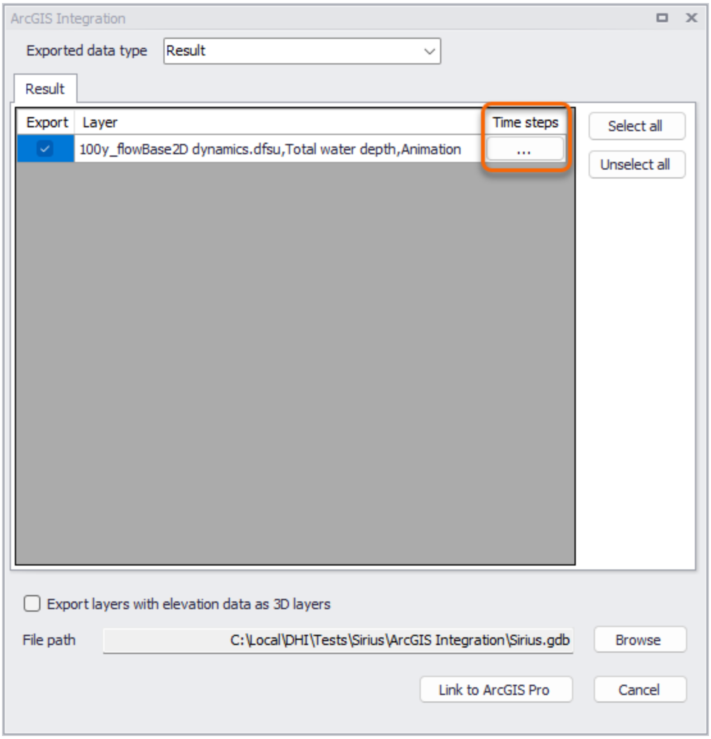


Figure 4.2 Accessing the time step settings before exporting a 2D result layer

In this window, it is possible to control the first and last time steps of the result file to export, as well as an interval to control the number of intermediate time steps. Note that the 'Last exported time step' is a maximum time step value, but the last time step actually exported to the ArcGIS Pro database may be smaller if the interval is such that the 'Last exported time step' is not selected for the export.

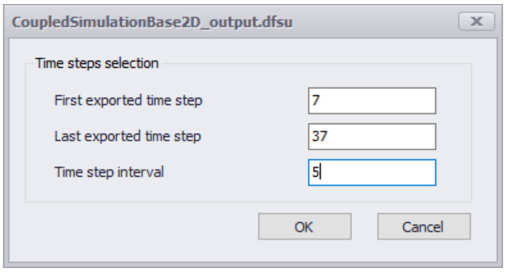


Figure 4.3 The 2D time steps selection window

If the option 'Export layers with elevation data as 3D layers' is selected, then the feature layers in the geodatabase will use the built-in 3D format when relevant, allowing for extra data processing in ArcGIS. As an example, pipes will be exported with their invert levels, allowing for nice 3D visualisations of the network in ArcGIS Pro. The following layers can be exported as 3D layers:

- Links (Collection system pipes and conduits - both from MIKE 1D and SWMM models - , Water distribution pipes)
- Nodes (Collection system - both from MIKE 1D and SWMM models - only)
- 2D overland domain.

The exported 3D layers are available in Map_3D (Local Scene) in ArcGIS Pro.

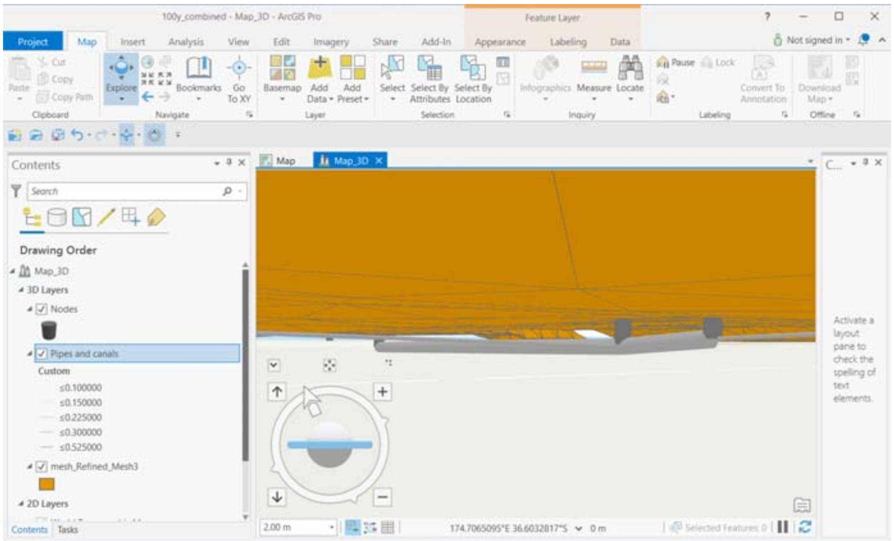


Figure 4.4 3D scene showing a collection system network and a 2D domain in ArcGIS Pro

For 3D visualisation of the networks, pipes can be drawn using 3D graduated symbol Tube. The layer property can be set to '*Display 3D symbols in real-world units*'. This provides proper zooming functionality. Because Collection and Water distribution systems are under ground, the Elevation surface \ Ground layer needs to be switched off. For more information about Ground elevation surface, please refer to ArcGIS Pro documentation.

The Planar navigation can be used to navigate in 3D mode.

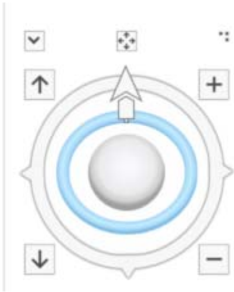


Figure 4.5 Use the planar navigation to move the 3D layers exported from MIKE+

For more details on how to use Local scenes, refer to the ArcGIS Pro documentation.

Layers are exported with a default symbology controlled by the *.lyrx files available in the MIKE+ installation folders (Collection System.lyrx - containing also Rivers and 2d Overland - , SWMM.lyrx, Water Distribution.lyrx, Domain3D.lyrx, Mesh_3D.lyrx, msm_Link3D.lyrx, msm_Node3D.lyrx, mss_Link3D.lyrx, mss_Node3D.lyrx, mw_Pipe3D.lyrx). These files can be modified if a different default symbology shall be used during the export.

For 2D overland models, it is possible to export both the 2D domain and the 2D results. When the domain type is a 'Flexible mesh', the 2D domain is exported as a polygon feature layer, containing elevations (optionally exported as 3D polygon layer). Flexible mesh results are exported as a polygon feature class containing attributes for all result items from the 2D results. The exported feature class also contains a Time attribute, providing the possibility to animate results. The result item to display on the map needs to be selected in the symbology settings, usually using graduated colors to display the results colors on the map.

For more details on how to use temporal data, refer to the ArcGIS Pro documentation.

Figure 4.6 Animating exported 2D results using the Time control

When the 2D domain type is a 'Rectangular grid', the domain is exported as a raster layer containing level values. Corresponding results are exported as one raster per combination of result item and time step. There is no time attribute, but the corresponding time of the raster is indicated in the raster name.

These exported raster data can be used to create a multidimensional raster dataset in ArcGIS Pro. Refer to the ArcGIS Pro documentation for more information about this: an-overview-of-multidimensional-raster-data, create-a-multidimensional-mosaic-dataset-from-a-set-of-time-series-images-in-arcgis-pro.

Note that MIKE+ supports rotated grids (i.e. with the Y-axis rotated from the North direction), which is not supported in ArcGIS Pro. Rotated domains and results are therefore re-sampled to create a non-rotated grid. For this reason, the geometry of the rasters and their cell values in ArcGIS Pro may differ from the original data in MIKE+.

Note that when displaying large 2D layers or making 3D visualisations in ArcGIS Pro, the performance depends on the computer's hardware and the visualisation can sometimes be slow. Better performance can be obtained on computers equipped with a powerful graphical card (GPU).

4.2 Working with MIKE+ Data in ArcGIS Pro

As ArcGIS Pro is a general GIS desktop application it is important to realize that ArcGIS Pro does not support the data consistency checks and other protection mechanisms normally performed by the MIKE+ program.

The following precautions should be recognized:

Data View

No restrictions. As long as ArcGIS Pro is not taken into 'edit mode' there is no danger of corrupting the database. Examples of tasks/objectives that can be accomplished in viewing mode are:

- Advanced GIS analysis
- Advanced presentations of model and saved data
- Complete layout tasks for inclusion in final reporting
- Ad-hoc analysis requiring the user to write scripts in VBA or similar
- Easy use of ArcPy, suitable to implement geographic data analysis, data conversion, data management, and map automation with Python

Data Editing

Uncritical editing of data in the MIKE+ database may easily corrupt the database and make it unusable in MIKE+. However, very powerful tools exist in ArcGIS Pro that may be utilized for editing. Provided that you take care to obey the constraints, editing from ArcGIS Pro can be very powerful. However the level of Integration with MIKE+ of ArcGIS Pro is limited to the exportation of the native personal geodatabase, hence modifying the personal geodatabase in ArcGIS Pro will not directly affect the data stored in the SQLite or PostGIS database until the data is reimported into MIKE+.

General recommendations when working with WD and CS network databases outside the MIKE+ environment:

Editing Geometry

In general the editing of geometry (pipe shapes, manhole placement, and catchments) can be done without danger of corruption to the data structure. This is probably also the area in which there is most benefit from working in ArcMap | ArcGIS Pro as the editing tools here are more advanced than in MIKE+.

Altering Attributes

In general this can be done to most attributes without danger, EXCEPT when the attribute is an ID field used to maintain consistency of the database. To avoid this you may consult the datamodel appendix and as a general do not edit any attribute with a name containing the alias 'ID'.

Inserting and Deleting Records

Should be avoided unless you have a very good knowledge about the MIKE+ data structure. Deleting or inserting in MIKE+ is often triggering a number of background operations that update other tables so the consequence of doing such operations outside MIKE+ may easily be a corrupt database.

4.3 Typical GIS Native Environment Tasks

Typical tasks that may be done in ArcGIS Pro are:

- Complex editing and data analysis requiring joining and relating several tables from various data sources
- Tasks involving complex spatial analysis, spatial joining of data
- Geoprocessing tasks like intersecting, overlaying etc. apart from the pre-defined tasks existing in MIKE+
- Working with layouts, plotting to scale and similar high-level reporting
- CAD-style editing
- Further some organizations have developed in-house tools in ArcGIS Pro to perform certain functions. If these functions need to be performed on MIKE+ data, you will need to run the scripts from ArcGIS Pro.

5 MIKE+ Data Model

5.1 MIKE+ Networks

MIKE+ Project database includes the data models for water distribution, collection system and/or networks.

A model network represents a water network in a form as expected by the computational model engine. As such, it is subject to very strict model-specific data requirements, definite and fixed catalogue of element types, simplifications, etc.

5.2 Data Model Structure

The MIKE+ data storage is divided into a number of data stores.

The main storage for the model data is SQLite (or PostGIS database, optionally). But a number of additional files and data stores define the MIKE+ functionality such as binary result files and configuration files.

MIKE+ is installed with SQLite/SpatialLite. If you want to use the alternative database option, PostgreSQL/PostGIS then you must install the two products found in the “Prerequisites\ PostgreSQL” and “Prerequisites\PostGIS” folders. For more details please check the MIKE+ Installation guide.

5.2.1 Terminology

Storage Database

This is a database with a structure similar to a GIS database. It is used to hold the model data such as physical network description and other physical data, input data to the various numerical engines as well as general setup information. A database may hold only one instance of a particular model (however one database may hold water distribution, collection system and river data). The format of the database is also the basic data format used by Spatial Lite SQLite database (alternatively, PostGIS).

The Project (MUPP) File

This file holds all information about the current user setup. A project file will thus hold the individual settings for the user such a symbology for the featured network elements, pointer to added background data, etc.

Configuration and Import Bridges

These files are normally not touched by the user but may in certain circumstances be changed to fit individual user setups.

Time Series and Result Data

Binary data in the form of input timeseries data as well as result data are stored externally in binary data files.

5.2.2 Storage Database Basics

The MIKE+ database uses a SQLite database. SQLite is an embedded SQL database engine. SQLite is a C-language library which implements a fast, self-contained and highly reliable SQL database engine.

MIKE+ also allows the use of PostGIS as storage database format. PostGIS is an extension to the PostgreSQL object-relational database system, this engine allows GIS objects to be stored in the database. The use of a PostGIS database requires that a PostgreSQL installation is already available on a server (consult your IT service for the initial configuration). For more information about PostGIS please visit http://postgis.net.

The structure of the MIKE+ database is very much like a normal database consisting of tables having columns (or fields). What makes it special are:

- The database contains a predefined datastructure needed for GIS to operate correctly on the database
- Some tables are called 'feature classes' because they contain a special binary formatted column defining the spatial geometry of the object (row)

You can operate the database through SQL commands, but please be very careful if you try to manipulate data with such tools; always have a backup of your database.

5.2.3 Scenario Management

The database may contain a variety of scenarios of each model. These scenarios are managed by MIKE+ through the Scenarios Editor. Scenarios are in general stored as difference tables - the use and documentation of these are beyond the scope of this documentation.

It must be noted however that when opened with any tool, the database will represent the current active MIKE+ scenario.

5.2.4 The MIKE+ Database Contents

The MIKE+ database will contain all model parameters for the active model(s). Data is stored in either:

- Feature classes. These are database tables with spatial contents (such as pipes, nodes, etc.)
- ‘Standard’ database tables i.e. tables without a spatial content.

Naming Convention

All feature classes and tables follow the same naming convention:

Table 5.1 Feature classes and tables - naming conventions

Table	Description
m_	Means that the table is a general MIKE+ table covering all parts of MIKE+ (typical configuration information)
ms_	Means that the table belongs to the collection systems part of MIKE+ i.e. common to any of the numerical engine models
mw_	Are tables belonging to the water distribution part of MIKE+
msm_	Are tables specific to the MIKE 1D model of collection systems
mrm_	Are tables specific to the MIKE 1D model for river networks

Information on the individual fields of the database may be found in the sections of the manual describing the individual parameters. Generally it is not recommended to use characters such as '?' and '"' in any unique names (MUIDs). Database fieldnames are also shown as tooltips in the individual data editors when hovering over the field.

5.3 PostGIS database specifics

Using a PostGIS database allows to save the project database on a central server. This e.g. eases sharing model data amongst users, since various users can successively connect to the database as long as they have access to the server, without having to move the project files.

With a PostGIS database, it is also possible to have multiple users connecting to the database simultaneously, allowing e.g. model building by multiple users. The primary goal of this functionality is to allow users to review and analyze the model data simultaneously, or to allow them to build / edit different parts of the model. The following applies while multiple users are accessing the same PostGIS database:

- Some settings or functionalities which have an impact on the entire project are disabled. For instance, importing other model setups into the database (from File \ Import menu) is not possible, and changing the unit system is disabled, in order to avoid severe conflicts between users which could otherwise happen. To enable such functionalities again, only one user must be connected.
- If multiple users are to edit data simultaneously, it is recommended that they enable the auto-refresh option, so that changes made by one user are visible to others as well (please see User preferences, p. 54, for details on the auto-refresh option).
- If multiple users are to edit data simultaneously, it is also recommended that they ensure to edit different data to avoid conflicts. For example, even if the auto-refresh option is active (see , a small delay will usually occur after a change is made by a user, before it appears for the other users. Therefore, if one user edits (or delete) a feature, the change should soon be visible to others but there is possibly a short period of time during which others may apply a conflicting change (and if the node was deleted by the first user, the changes made by others will simply be lost).
- If result files are loaded in the project, they may not be refreshed if another user later re-run a simulation which changes the results. As a consequence, the loaded results may not show the results of the latest simulation.
- The option to support multiple users only applies to the database. The MIKE+ project may however contain other types of files, storing e.g. time series data, 2D overland domain data or cross sections data, which can only be edited by one user at a time.

6 Import and Export

6.1 Introduction to MIKE+ Import/Export

Importing various data from external systems into a MIKE+ project is one of critical parts of the modelling work. Efficiency and versatility of the import workflow contributes significantly to the overall productivity. Exporting the MIKE+ project into various external formats is equally important. As a variety of formats are commonly used for storage and management of water systems data, very flexible and versatile import/export tools are required.

MIKE+ comes with some standard (automated) routines for import and export from and to commonly used formats. These include the following:

- Import of projects from MIKE URBAN Classic formats (*.MDB and *.GDB)
- Import of MIKE HYDRO River and MIKE 11 model files
- Import of MIKE 21 and MIKE FLOOD model files
- Import of EPANET model files
- Import of SWMM model files
- Export of CS and River model setups to MIKE1D engine input file (M1DX)
- Export of WD model setup to EPANET model file
- Export of SWMM collection system model setup to SWMM model file
- Backing up the MIKE+ database by cloning it in SQLite or PostGIS formats

In addition to this, the 'Import and export' tool facilitates configuration of custom imports and exports from and to various formats.

This chapter provides detailed information on the technical background and practical user guide for the 'Import and export' tool.

The Import/Export tool available in MIKE+ (Tools|Import and Export) provides a versatile and flexible environment for exchanging data between various external repositories and the MIKE+ database. The data can be imported to and exported from the MIKE+ database. The Import/Export tool features the following:

- Variety of supported data formats, both on target and source sides
- Multi-section and multi-job batch processing in user-controlled sequence
- User-specified variables
- Control of source strings format (decimal separator)

- Automatic creation of a network on basis of feature geometries
- Auto-mapping identity assignment
- Assignment expressions supported by various functions and operators, including a conditional clause
- Preview of source and target data
- Automatic unit conversion
- Automatic verification of import configuration
- Saving of import job configurations for later reuse

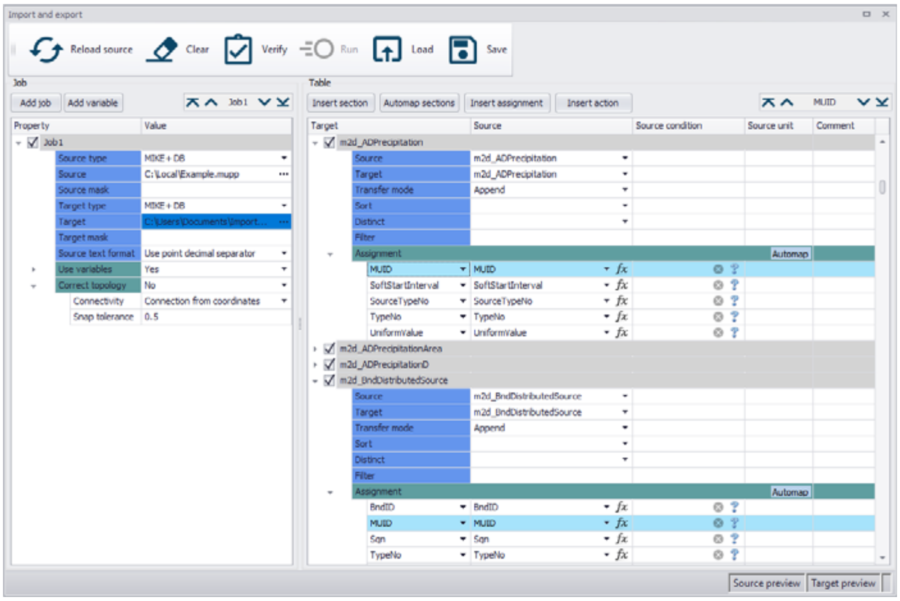


Figure 6.1 Import/Export GUI is divided in two boxes: in the left box, import jobs are specified, and in the right box import sections are specified. Additionally, there are several action buttons and a toolbar, including all functionalities for a full control over the import definition process

6.2 Technical Description of Import / Export Functionality

The common word used for an import/export procedure is 'job'. A job may consist of one or more 'table configurations'. An import/export job will normally consist of several table configurations (called 'sections'), making up a complete import/export. Each table configuration in the job relates to an individual table or feature class.

When importing data into a MIKE+ database from an external data source, the external data is referred to as the 'source' and the MIKE+ database is referred to as the 'target'. For the export jobs, the situation reverses - the

MIKE+ database is then referred to as the 'source' and the database or file that you wish to export the MIKE+ data to is referred to as the 'target'.

The procedure of importing/exporting data is done through a generic 'engine' as shown in Figure 6.2.

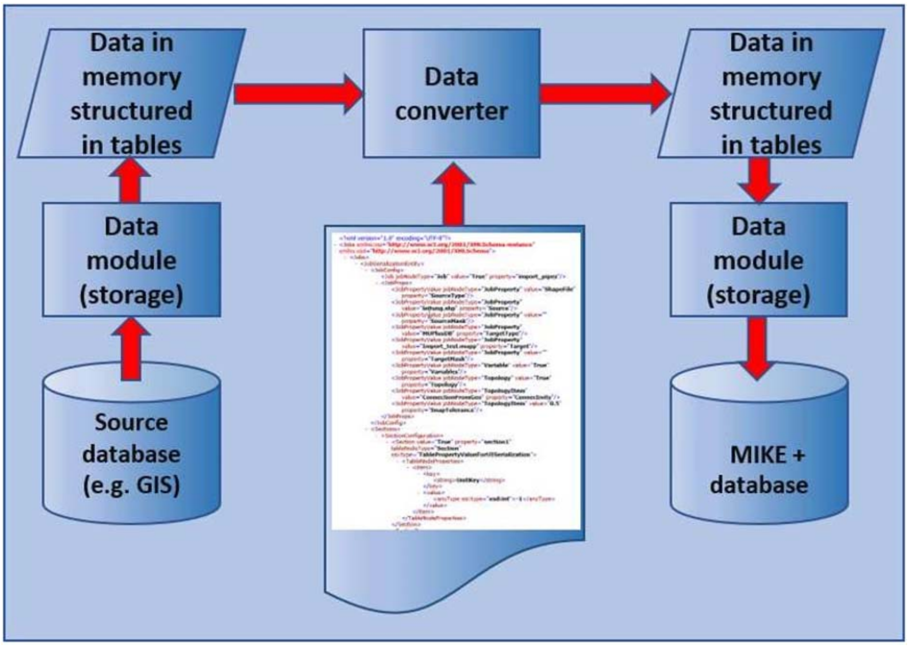


Figure 6.2 Generic presentation of Import/Export process

As the illustration suggests, import and export jobs are always executed following the same process scheme, involving storage drivers ('muStorages') - programs which read data from the source into the computer memory ('cache') and write data from the computer memory i.e. from 'cache' to the target, and a data converter ('muBridge') - a program which 'translates' the data from the source cache into the target cache.

6.2.1 Import/Export Job: Definition and Main Properties

An import job is a consistent set of instructions to the Import/Export engine, with a purpose of modifying data contents in the target by means of the source data, assignment expressions and underlying data processing. An Import/Export job consists of the general job definition and of at least one section containing assignment(s) for at least one target attribute.

An Import/Export job can be saved in an *.xml file using the ‘Save’ button for later reuse. It can also be opened from an *.xml file using the 'Load' button.

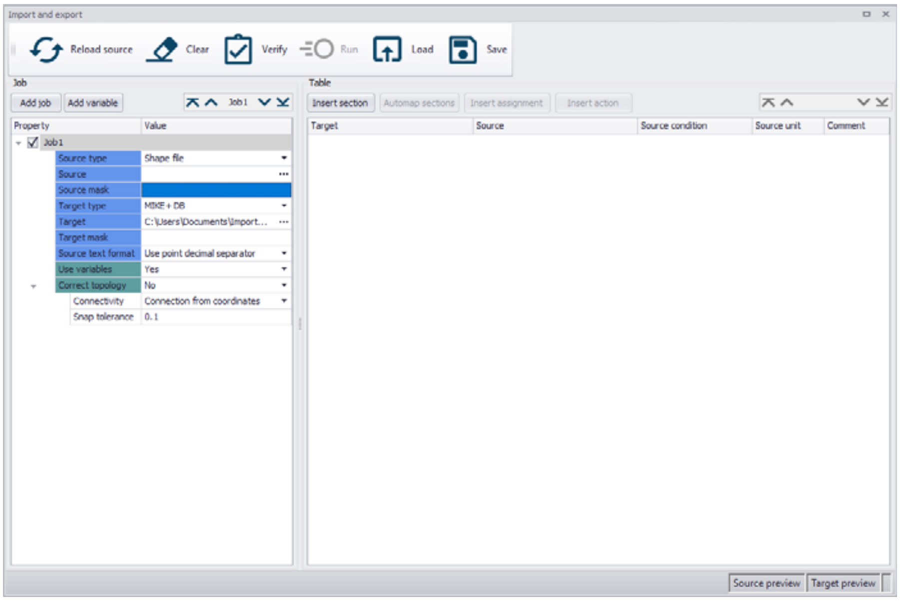


Figure 6.3 Creating a new job

To create a new job, click on "Add job" and define where the data will be imported from, location of this file, where the data will be imported to and customise details.

6.2.2 Job Properties

The following is a detailed description of the job definition parameters.

Job Name

When created, an import job gets a generic ID (name) Jobn, where "n" is a simple counter. By double-clicking the job ID turns editable, i.e. user-specified ID may be typed.

Job On/Off Toggle

A checkbox next to Job ID controls if the actual job will be included in the "Run" or not.

Source Type

This is a choice among available type of source files:

Shape file

The source can be a Shape file containing any feature type: point (containing data on e.g. network nodes), polyline (containing data on e.g. network links) or polygon (containing data on e.g. catchments).

As shape files contain one data table only, an import job with a Shape file as a source cannot include import of multiple tables.

Excel file

An EXCEL file may contain several data tables to be included in the job definition. Each column in a table must contain a text header in Row1 and the data in consecutive rows. All data in one column must be of the same type.

CAD file

AutoCAD proprietary format "DWG" is supported, as well as “DXF” format. Because it is not an open standard format, MIKE+ import supports it with some limitations. E.g. geometry and some simple attributes are supported, while labels are not supported.

Geodatabase

Import can access tables in the ESRI geodatabase

ODBC

Open Database Connectivity (ODBC) interface is a C programming language interface that makes it possible for applications to access data from a variety of database management systems (DBMSs). ODBC implementation in MIKE+ import supports the following sources:

- MS Access
- SQLite

Relevant drivers for reading these file formats are not installed with MIKE+. To import a Microsoft Access database, the user must therefore install the Microsoft Access Database Engine driver beforehand. To import a SQLite database, the user must install the SQLite ODBC driver.

The geometry for the features shown on the map must be defined in Well Known Text (WKT) format.

ISYBAU file

The ISYBAU exchange formats are used for the standardized exchange of data for the planning, construction and operation of wastewater facilities, originating from Germany. ISYBAU is a column-oriented exchange format in text form.

MIKE+ DB

MIKE+ database (SQLite or PostGIS) can be used as a source for data import, typically when the current project needs to be populated with data from another MIKE+ project.

Result Layer

This option is used for export of mapped 1D network results to external formats, e.g. to shape. 2D result layers cannot be exported from this tool (use instead the option to export to shape file, from the 'Layers and symbols' tree view).

SQL Server

This option is used to connect to a database hosted on a SQL Server.

The geometry for the features shown on the map must be defined in one of the following formats: SQL Geometry, Well Known Binary (WKB) format or Well Known Text (WKT) format.

Oracle spatial

This option is used to connect to an Oracle spatial database.

ICM (csv format)

This option is used to read and import individual .csv files previously exported from a ICM model setup (one file per ICM database table). The recommended workflow to import these data is as follows:

- Export the ICM model setup to individual .csv files, all saved in a common folder
- Create an import job in MIKE+, and select this folder as import source
- Click the 'Automap sections' buttons: this will load a predefined configuration, to import the collection system data (rivers and 2D model data not yet configured)
- Review the import configuration and adjust if necessary
- Run the import.

Note that this import option allows importing network data to MIKE+, but differences of numerical methods between the two different products imply that some data may not be imported or may need further manual adjustments after the import, especially for example for Rainfall-Runoff modelling. Also note that no automatic conversion of units nor coordinate system is performed during this import.

Source

This is a path and filename pointing to the file containing the data to be imported. When clicking on "…" button, browser opens with a filter including only file types available for the currently selected source type. The following file types are accessible:

Table 6.1 Import/Export job source types

Source Type	File type(s)
Shape file	*.SHP
Excel file	*.XLSX
CAD file	*.DWG, *.DXF
ODBC	*.MDB, *.XLSX, *.SQLite
ISYBAU file	*.XML
MIKE+ DB	SQLite, PostGIS
ICM	*.csv
Result layer	-

When importing data from shape files or from ICM, the source can either be a specific file or a folder. When selecting a folder as a source, the exact file to import is selected in the definition of the section, and each section within the same job can use a different file.

When connecting to a SQL Server database, the '…' button opens a dialog to supply the server name, the user name and password for authentication. The name of the database must also be selected among the list of databases hosted on the server.

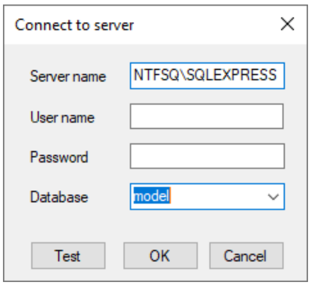


Figure 6.4 Connecting to a SQL Server database

When the SQL Server and MIKE+ are installed on the same machine and when the authentication for the SQL server uses 'Windows Authentication', the user name and password fields must remain empty.

When connecting to an Oracle Spatial database, the '…' button opens a dialog to supply the server name, the user name and password for authentication, as well as the service name. In order to connect to an Oracle Spatial database, the 64-bit version of "Oracle Data Provider for .NET" (ODP.Net) must be installed. It can be installed through ODAC installer or Oracle database client installer. So, when clicking the '…' button to connect to the server, you will first need to browse to the location of the installed file Oracle.DataAccess.dll.

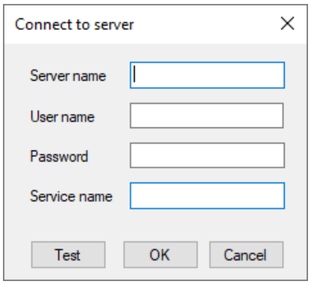


Figure 6.5 Connecting to an Oracle spatial database

One import job can only point to one source file.

Source Mask

This allows for including only those source tables which table name contains the specified string. E.g. typing "msm*" (without quotes) will exclude all the tables from a source (e.g.MU Classic *.MDB file) except these containing the string "msm", i.e. collection model model tables. This is useful in cases when a source file contains many tables, where reducing the display to only relevant tables facilitates a better overview and a more efficient work.

Source text format

This option relates to converting input strings to numbers, e.g. when numeric data (e.g. pipe diameters) in some data sources are defined as "string" / text type, instead of "real number" (e.g. Excel sheet). In that case it is necessary to specify which character is used as decimal separator.

User may select among two possible options:

1. **Use point decimal separator**: this will always look for point "." decimal separator. Records containing text using comma "," as a separator will therefore be invalid and won't be imported)
2. **Use computer’s separator**: this will use the format of the decimal separator selected in Windows' region settings, on the local computer. With this option, the separator selected in the Windows settings must match the separator used in the source file to be imported.

Target Type

This provides similar choices as Source Type.

Target Mask

The same functionality as "Source Mask", applied on the target.

Use variables

User-specified variables are often useful in creating assignments. User can assign a certain value to a variable, in a same way as for any target attribute, i.e. by creating an expression containing applicable source attributes, constants, functions and operators. A variable can be used in assignments for other target attributes in the current section, or in any other section within the actual import job.

Variables may be of the following types:

- String
- Double
- Int (i.e. integer)
- Bool (i.e. Boolean)
- DateTime

Use variables	Yes
Var1	double
Var2	bool

Figure 6.6 Example with two user-specified variables, "var1" and "var2".

Application of a variable in an assignment expression must be consistent with the variable type.

To use variables, change the option 'Use variables' to Yes, and add a variable to the import job using the button "Add variable". This creates a new line below "Use variables". A variable is defined by its name and type.

Note: Variables are not supported when exporting a result layer.

Correct topology

"Correct topology" refers to the processing of nodes and links data and creating a network. When activated, this option operates on the "Link" or Rivers table. For any other table, this option is of no relevance.

"Correct topology" works in two modes when creating the network connectivity:

- Connection from coordinates

  In this mode links geometry (i.e. coordinates) is defined, but the network connectivity (i.e. connections to node IDs) is undefined. When ‘Correct topology’ is activated with this mode, the network connectivity is established by geographical proximity of nodes and links' ending points.

If a node is within a specified snapping distance from a link's ending point, then a network connection for a link is established, either as "FromNode" (at the link's geometry start point) or "ToNode" (at the link's geometry end point). The established connection implies moving ("snapping") the links' end points to the nodes.

This process will create a network which may include orphan nodes (i.e. nodes not connected to any link), orphan links (i.e. links without any of "FromNode" or "ToNode"), very short links, etc. Resolving such incompleteness and/or anomalies of the network is supported by a dedicated tool "Topology repair", after completed import process.

For river networks, this option establishes the location of river structures on the river (i.e. assigns the River ID and chainage) according to the coordinates of the imported structures.

Additional parameter to this mode is "Snapping distance". It defines a search radius for nodes and river structures to be "snapped". Snapping distance is defined in actual map units.

- Coordinates from connection

  In this mode the network connectivity is defined by FROM and TO nodes, but the geometry (i.e. coordinates) of the links is not defined. In this case, the links geometry is defined as straight lines between FROM and TO nodes.

  For river networks, this option establishes the coordinates of the imported structures, according to the imported River ID and chainage.

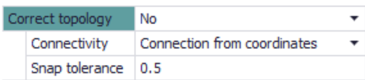


Figure 6.7 Example with "topology" activated. The network will be created by connecting nodes with the link-ends founds within snapping distance.

Dissolved lines

This option is available when the source file is a shape file containing dissolved features, for example multiple polylines or multiple polygons defining the same item in the shape file. In this case, two options are offered:

- Merge: with this option, the multiple objects defining a single item are merged. This option is especially relevant when a polyline has been digitized with multiple polylines describing the same feature, and in this case all the polylines defining a given feature / item are merged during the import. For example, if a river item is made of three dissolved polylines in the shape file, only one longer river will be created.
- Import separately: with this option, the multiple objects defining a single item are split, i.e. one item is created for each object. For example, if a river item is made of three dissolved polylines in the shape file, three rivers will be created.

6.2.3 Import Sections: Definition and Main Properties

An import section consists of a set of import properties related to one pair of source and target tables and at least one assignment in a target table. I.e. assignments in a section establish a relation between data from, primarily, one source table and one target table. By applying user-specified variables and LOOKUP function in assignment expressions, the data from other related source tables may be used.

An import job may include one or more sections. Sections within one job are executed sequentially. It is possible to re-order the sections, and also their assignments, using the buttons in the upper right corner.

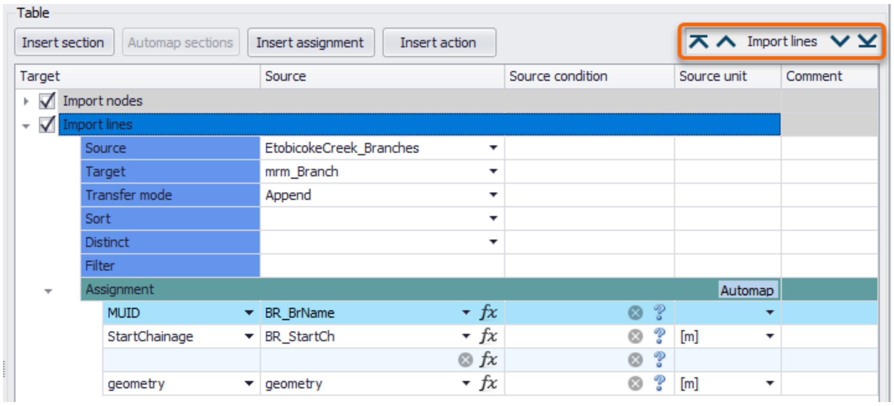


Figure 6.8 Example of a defined section to import nodes with assignments from the source data including conditional statements from the CoverTypeNo.

New sections are added using the 'Insert section' button at the top. Sections can be deleted by right-clicking on the section's top row.

When importing a MIKE+ database into another MIKE+ database, it is possible to create one section for each table by using the 'Automap sections' button. The button will only create sections for tables related to the active mode (e.g. 'Water Distribution'), and will populate the list of assignments for each section. It will not import "system" tables, which are e.g. used to store the following data:

- MIKE 1D engine configuration
- Fields' status
- Default values
- User-defined column information
- Status codes
- Bookmarks
- Model type settings
- Custom units

These data must therefore by defined manually after the import, if required.

When importing an ICM model setup in .csv files, this 'Automap sections' button will also load a predefined import configuration, providing a default mapping between collection system data in ICM files and in MIKE+ database.

6.2.4 Section Properties

The following is a detailed description of the section definition parameters.

Section Name

When created, a section gets a generic ID (name) Section*n*, where "*n*" is a simple counter. By double-clicking the section ID turns editable, i.e. user-specified ID may be typed.

Section On/Off Toggle

A checkbox next to Job ID controls if the actual job will be included in the "Run" or not.

Source

Source is ID (name) of the table in the data source specified for the current job. User selects the source table from the drop-down list containing all source tables. The full list of source tables may be reduced (and made easier to navigate) by setting up the Source mask (see above).

When the source file is a CAD file, each layer from the CAD file is divided into source layers per data types. For example, importing from the source layer "LayerA_Polyline" will import only the polylines from LayerA. This list of source tables is supplemented with extra options with format "Model_*ElementType*", e.g. *Model_Polyline or *Model_Polygon, Selecting one of these options, all source layers with the selected type will be imported. For this reason, only the common attributes (available in all layers with the selected type) can be applied in assignments. This is especially useful when the same type of data is saved in different layers in the CAD file.

Note: The same CAD data may be imported from multiple source layers. This is especially true for Blocks in the CAD file, which are defined by a reference point (location, associated to other attributes) and by a detailed geometry (symbol shown on the map, possibly created with multiple geometry types like polylines and polygons). For instance, blocks data may be imported from the following source layer types:

- Model_*ElementType*: Can import the parts of the detailed geometry (symbol) with the selected type. Only the common attributes from all layers with the selected type can be imported.
- *layerName_Elementype*: Can import all geometrical elements with the selected type, from the selected layer. All attributes from this layer can be imported.
- *Model_Point: Can import the reference point of the blocks. Only the common attributes from all point layers can be imported.
- *Model_Block: Can import all reference points from all layers. Only the common attributes from block layers can be imported.
- *layerName*_Insert: Can import the reference points from the selected layer. All attributes from this layer can be imported.

Target

Target is ID (name) of the table in the data target specified for the current job. User selects the target table from the drop-down list containing all target tables. The full list of target table may be reduced (and made easier to navigate) by setting up the Target mask (see above).

The geometry type of the selected target layer should normally match the geometry type of the source layer. For instance, source polygons cannot be imported to a point layer. It is however possible to import a source polyline layer to a polygon target layer, in which case each source polyline gets closed to create a polygon (the start and end points of the polyline don't have to be at the same location).

Filter

Purpose of filtering is to eliminate unwanted records from import. Filtering is applied to the source before executing the assignment.

Syntax of the filter is the same as SQL WHERE clause. User only needs to type the contents of the WHERE clause, e.g. CustomerID = 'abc', would include only records with specified CustomerID. All other source records will be neglected.

Sort

Sorting is applied to one of source fields (column) before executing the assignment.

The content can only be a source field name, selected from a combo-box.

Distinct

Distinction is applied to source data before executing the assignment. It means that only one (the first) instance of the original source field (column) value will be accepted and all other records containing the same value will be removed.

The content of "Distinction" can only be a source field name, selected from a combo-box.

Transfer Mode

The following transfer modes are available:

- Append
- Update
- Append & Skip existing
- Append & Update
- Overwrite
- Sync

In the following, the actual workings of transfer modes are described and illustrated with some examples.

Append

With this mode, the source data are appended (added) to the current content of the target table, i.e. it preserves original data in the target table.

If there is an assignment for the MUID, the program will ensure that there is no duplicate MUIDs after the import, to fulfill the requirement in MIKE+ that MUIDs must be unique in each table:

- If multiple records in the source have the same ID, only the last one will be imported.
- If an imported record in the source has the same ID as one of the original record in the target table, the source ID will be renamed.

This is illustrated with examples below. Note that the new or modified target content is shown in red fonts. Unchanged target contents are shown in black fonts.

Source

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	2	B
ID3	A	3	C
ID4	A	4	C

Target (before import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C

Target (after import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C
ID1_Renamed	A	1	B
ID3_Renamed	A	3	C
ID4	A	4	C

Figure 6.9 "Append" with "MUID" assignment, applied to the target with some initial contents. The existing data in the target are not changed. The first instance of the duplicate source record ID3 is ignored. Source records ID1 and ID3 are renamed.

Source

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	2	B
ID3	A	3	C
ID4	A	4	C

Target (before import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C

Target (after import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C
Node_3	A	1	B
Node_4	A	2	B
Node_5	A	3	C
Node_6	A	4	C

Figure 6.10 "Append" without "MUID" assignment, applied to the target with some initial contents. The existing data in the target are not changed. All source data are imported with a default MUID.

This mode is typically applied when building a model from several sources and/or when updating a target with newly added data from the same source.

Update

For each record in source data, the program looks for the matching records in the target, and for the found matching records it updates any mismatching attribute value. If no match is found in the target, the source record is not used.

If there are duplicated IDs in the source data and the same ID exists in the target, the last duplicate record in the source will be used to update the target.

Source

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	2	B
ID3	A	3	C
ID4	A	4	C

Target (before import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C

Target (after import)

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	3	C
ID5	A	3	C

Figure 6.11 "Update" applied to the target. Only the existing data in the target with matching ID in the source are updated, i.e. values of attributes set to their values from the source. Note that in case of double ID in the source (ID3), attribute values from the last instance will eventually be applied.

By default, matching records are found by comparing the MUID in the source data with the corresponding target attribute selected in the MUID assignment.

When another attribute than the MUID should be used to find matching records (e.g. the Asset ID), then an assignment must be created for this attribute, and this assignment can be used by right-clicking and selecting 'Select as matching key'. The assignment used to find matching records is shown in light blue.

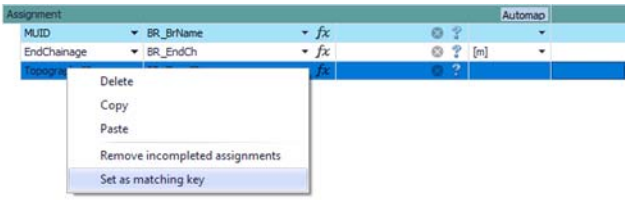


Figure 6.12 The assignment used to find matching records is shown in light blue. This matching key can be changed from the context menu

With initially empty target, "Update" operation would not do any change, i.e. the target would remain empty.

This mode is typically applied when maintaining the model data to fit with any modifications of imported data in the source.

Note that the tool cannot update the MUID attribute from existing records.

Append & Skip existing

For each record in source data, the program looks for a matching record in the target. If a matching record is found in the target, then this target record is kept unchanged (the import step is skipped). If not found already being in the target, the record will be appended into database. If there are duplicated IDs in source data, the last one will be used.

Source

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	2	B
ID3	A	3	C
ID4	A	4	C

Target (before import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C

Target (after import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C
ID4	A	4	C

Figure 6.13 "Append & Skip existing" applied to the target with some initial content. Only the missing records are imported whereas records already present in the target are kept unchanged.

This mode is typically applied when importing data from a new version of the source, without changing data imported earlier from a previous version of the source.

Append & Update

For each record in source data, the program looks for a matching record in the target, and for the found matching record it updates any mismatching attribute value. If not found already being in the target, the record will be appended into database as well. If there are duplicated IDs in source data, the last one will be used.

Source

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	2	B
ID3	A	3	C
ID4	A	4	C

Target (before import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C

Target (after import)

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	3	C
ID5	A	3	C
ID4	A	4	C

Figure 6.14 "Append & Update" applied to the target with some initial contents. Initially, the existing data in the target with matching ID in the source are updated, i.e. values of attributes set to their values from the source. Note that in case of double ID in the source (ID3), attribute values from the last instance will eventually be applied. Subsequently, any source record with non-matching ID in the target (ID4) will be appended to the target.

This mode is typically applied when updating data in the target with the source contents, both in terms of changes of already existing data and added new data in the source.

Note that the tool cannot update the MUID attribute from existing records.

Overwrite

Any data in the target table, before running the import/export job, get deleted. All source data are then appended (imported) to the empty target table. The behavior when appending data is the same as when using the 'Append' mode.

In this mode all source data are being imported.

This is illustrated with two examples below. Note that the new or modified target content is shown in red fonts. Unchanged target contents are shown in black fonts.

Source

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	2	B
ID3	A	3	C
ID4	A	4	C

Target (before import)

MUID	Attr1	Attr2	Attr3

Target (after import)

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	3	C
ID4	A	4	C

Figure 6.15 "Overwrite" applied to the initially empty target. The target table has the same contents as the source table, except that a duplicate record is renamed to ensure non-duplicate IDs in the target table.

Source

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	2	B
ID3	A	3	C
ID4	A	4	C

Target (before import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C

Target (after import)

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	2	B
ID4	A	4	C

Figure 6.16 "Overwrite" applied to the target with some initial content. All existing data in the target are deleted and the new content is imported. The target table has the same content as the source table, except that a duplicate record is renamed to ensure non-duplicate IDs in the target table.

With initially empty target, result of “Overwrite” mode would be identical as “Append”.

This mode is typically applied when started building a model, i.e. when populating target tables "from scratch".

Sync

The program will first do “Append & Update” mode process. When completed, it will also delete all the target records which are not existing in source data.

Source

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	2	B
ID3	A	3	C
ID4	A	4	C

Target (before import)

MUID	Attr1	Attr2	Attr3
ID1	A	2	C
ID3	A	2	B
ID5	A	3	C

Target (after import)

MUID	Attr1	Attr2	Attr3
ID1	A	1	B
ID3	A	3	C
ID4	A	4	C

Figure 6.17 Result of the "Sync" operation is identical as "Update & Append", except that the record ID5 is deleted from the target, as it has no matching ID in the source table.

This mode is typically applied when synchronizing data in the target with the source contents.

Note that the tool cannot update the MUID attribute from existing records.

Action

"Action" is a command which usually acts on the target records, not on the target values (except for the ‘Iterate’ action).

An action line is included in the current section by "Insert action" button and by selecting a wanted action from the drop-down list.

Action commands available are:

- DeleteRecord
- AggregateGeometry
- Log

- Iterate

DeleteRecord

"DeleteRecord", removes the current record from the target table.

To make it meaningful, "DeleteRecord" is always used with a condition. As such, "DeleteRecord" supports advanced filtering during import, i.e. it is an alternative to a (optional) filter specified in the section header.

Note that the record is first written to the target table, later to be removed if the condition associated with "DeleteRecord" is true.

Consider the following example, where source contains data on two types of pipes (ptype): "Mainline" and "Servicepipe".

ID	ptype		MUID
P1	Mainline		P1
P2	Servicepipe		P3
P3	Mainline		

Figure 6.18 Example of a source table and a target table after import including action assignment "DeleteRecord" for ptype = "Servicepipe"

If only pipe with ptype = "Mainline" are to be imported, such import can be achieved by "DeleteRecord" action, and condition ptype = "Servicepipe".

Another example is shown below where all empty cells in the source 'TypeNo' column will be deleted from the target table.

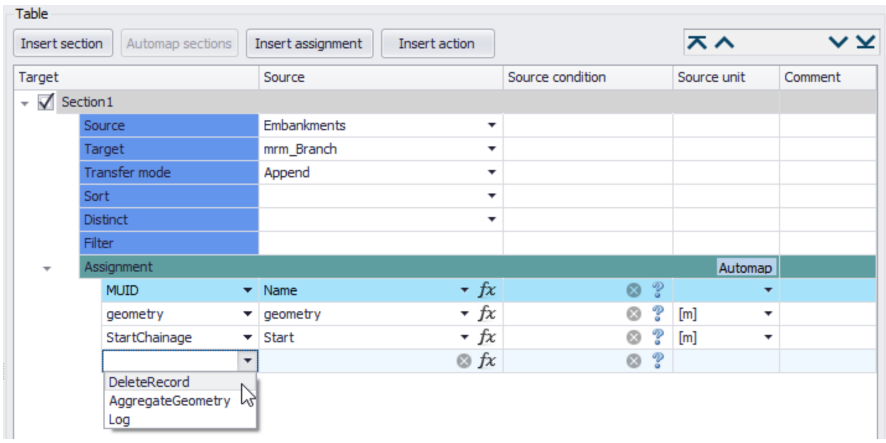


Figure 6.19 Add an assignment for a specific function (e.g. DeleteRecord)

AggregateGeometry

The geometry for lines is usually stored in a single line of text in MIKE+ (WKT format). The AggregateGeometry action can be used to import e.g. pipes coordinates from geometry tables, where the geometry is instead stored in columns (e.g. pipe ID, Vertices number, X coordinate, Y coordinate) for each vertice of the pipe.

The AggregateGeometry action must be used in combination with the PointFromXY function. After selecting the AggregateGeometry action, open the Expresssion editor, and apply the PointFromXY function which requires two attributes, corresponding to the source columns containing X and Y coordinates.

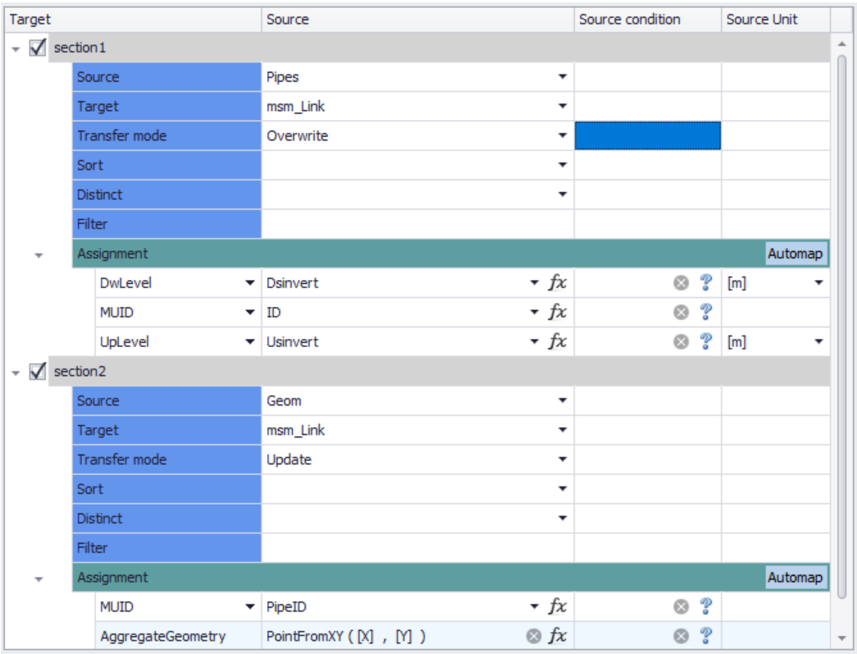


Figure 6.20 Using the AggregateGeometry action to import lines shapes from a geometry table

Log

The Log command writes to the MIKE+ log file. The log file can be found in the user folder under "\Appdata\Local\DHI\MIKE+\all-utf8.log".

The Log command is used when creating advanced assignments. When user-specified variables and LOOKUP function are used, it can be useful to see actual values of the variables to ensure that the data are correctly assigned. It is recommended to remove the Log command after completed testing of the assignment.

The Log command can also be used to create error messages when something unexpected occurs during execution of an import job.

The syntax is:

Log(Value)

The command parameter "Value" represents the value to be written to the log file. "Value" can either be an attribute in the assignment source data, a constant, a user-specified variable or a value returned by a specified function.

The Log command can be extended by a condition.

Iterate

Assignments usually import values from one field in the source to another corresponding field in the target. It may however happen that one field in the source combines multiple values in a string, from which the different values are distinguished with a specific separator. In this case, the 'Iterate' action allows to iterate through the different parts of the string, and apply the assignments using each part of the string successively.

The Iterate action must be used in combination with the Split function, which specifies the substring to be used as separator between the different parts. After selecting the Iterate action, open the Expression editor, and apply the Split function which requires two attributes, corresponding to the source string to be split and the substring used as separator.

Once the Iterate action is created, define the assignments to be applied to the resulting substrings. These resulting substrings are obtained from the variable 'Iterate'. Additionally, the variable 'IterateIndex' returns the number of the substring being processed within the current string.

The example below imports tabular data with multiple values for two variables, all stored in a single string, into the 'Curves and relations' editor.

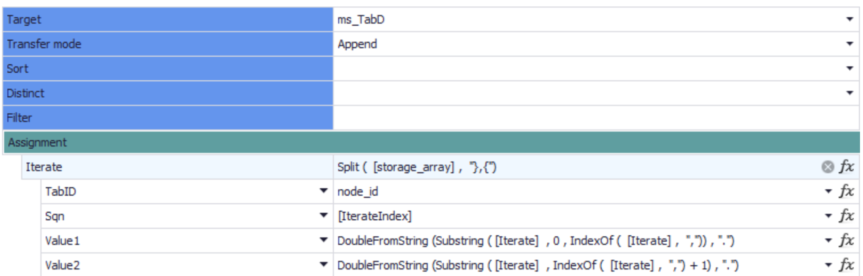


Figure 6.21 Using the Iterate action to import tabular data from a single string

Note that the 'Iterate' action cannot be combined with another action, i.e. it is not possible to add other actions within iterations.

6.2.5 Assignments

Assignment Structure

"Assignment" sets the value to one attribute in the target table. This implies that each assignment has a target attribute on the left side of "equals to" sign.

The right hand of the "equals to" sign may include one or more of the following:

- A simple constant
- Attribute value from a source table
- Attribute value from a LOOKUP table
- Value of a user-specified variable
- A value computed by an expression (including various functions and operators)
- A system-generated value
- A condition

Assignments are executed sequentially, as they appear in the editor. So, value of an attribute set in one assignment may be overwritten by another, subsequent assignment.

When importing data to a MIKE+ table, attributes without any assignment will be given the default (automatic) values from MIKE+. For example, when importing pipes geometries without assigning the pipes' MUIDs, each pipe will be given a default name like "Link_1".

Condition

Per default (i.e. with "condition" field empty), an assignment sets the target value unconditionally. Optionally, the assignment can be extended by a condition.

Creating Assignments

Creation of assignments is supported by the following:

- Automatic assignment (mapping)
- Add assignment and pick-up of target and source attributes from drop-down list
- Expression Editor, providing access to all source attributes, user-specified functions and operators

Auto Assignment

"Automap" button, located on the far right of the "Assignment" line, creates an assignment record for each target attribute name identical to source attribute name. I.e., if both source and target tables have identical structures, auto-assignment will create simple identity assignments for all attributes in these tables.

Repeated auto-assignment will re-write any existing assignments for the involved target attributes.

Insert Assignment

"Insert assignment" button inserts one empty assignment line above the currently active assignments in the current Section. In any case, user must select one target attribute from the drop-down list.

Depending on the situation, the next step may be the following:

- Select a source attribute from a dropdown list, thus creating a simple identity assignment, OR
- Open "Expression editor" (Press "fx" button in the "value" field on the current assignment line) and create an expression assignment

Optionally, use expression editor also to create a conditional clause.

Source Unit

Data in the source are often in different units than in the current MIKE+ project. Also, when exporting data from MIKE+ database, it is possible that the exported data should be in some different units than in MIKE+. For such cases, the tool supports automatic scaling of data.

When importing data to MIKE+, each assignment for a numerical "target" attribute, provides an information on unit for the attribute in the source table. Initially, this is set equal to the target (i.e. MIKE+) unit. In cases when the unit in MIKE+ and in the source are the same, no user action is needed: The value will be imported unchanged. If the source unit differs from MIKE+ unit, user must choose appropriate source unit from the drop-down list. If the actual source unit is not among the available units, the scaling factor for unit conversion must be specified directly as a multiplicator in the assignment expression.

Also, when the assignment statement involves two or more source attributes with different units, any unit inconsistency between the source and target must be handled explicitly by multiplication in the assignment statement.

Comment

This column can be used to add comments to any part of an import job, i.e. it is possible to comment a section and its main property lines, an assignment or an action. Comments will be saved with the import configuration, for later reuse.

Expression Editor

Expression editor supports creation of simple or complex assignment expressions, involving attributes, user-specified variables, functions and operators. Expression editor reduces the actual typing (hence the source of errors) to absolute minimum. Also, automatic expression validation is provided.

The left-hand side of the "equals to" sign of the expression is automatically provided. I.e. the user is expected to create only the right-hand side of the expression. This can be done either by direct typing, or by picking up the wanted variables, functions and operators from the respective drop-down lists. Typically, the process will involve both methods.

All variables in the expression should be embraced by square brackets ([]). This is a good practice, but not mandatory.

Strings should be embraced by double quotes ("").

The expression can be defined using the following controls.

"Variables" is a list including all attributes in the source table and any user-specified variable. A variable is included in the current expression by point & click. Square brackets are automatically provided.

"Functions" provides a list of available functions. A function is included in the current expression by point & click. Placeholders for the function's arguments are automatically provided.

"Operators" provide a list of available operators. An operator is included in the current expression by point & click.

The "Error list" reports "on-the-fly" any syntactic errors in the expression and provides advice on how to complete the expression.

"History" is a list of recently used expressions, available for reuse in the current assignment. Every new expression is automatically added to the history list. This allows for a very efficient reuse of similar assignments. "History" can be saved into a simple text file (*.TXT) and reloaded (Open) in a future import editing session.

The expression editor can be used in instances where the source data contains values in different formats to what is expected in MIKE+. For example, if the source data has node cover types specified in text format, while MIKE+ expects integers. The "fx" and "?" buttons in the assignment part of the "section" accesses the expression editor to state that if a condition is met ("?"), specify a value ("fx").

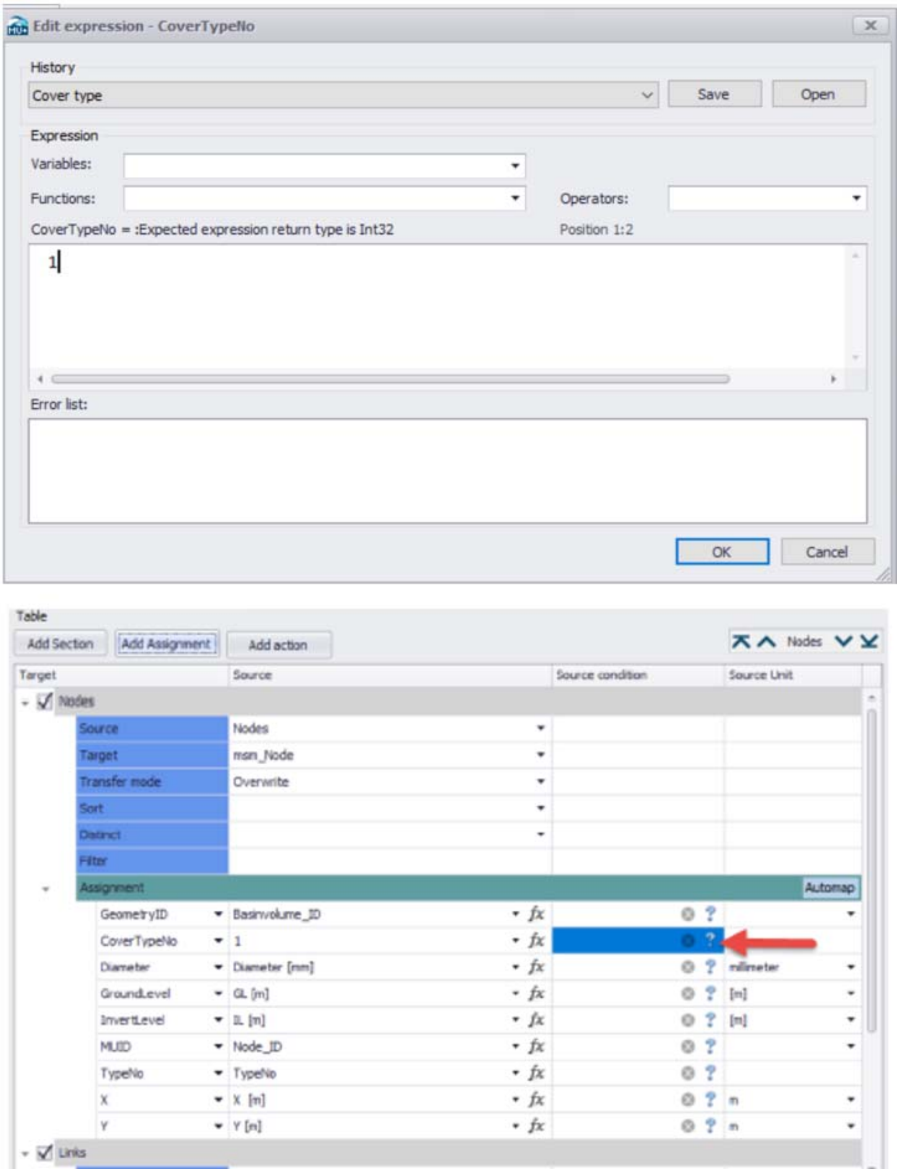


Figure 6.22 Expression editor is used for the easy creation of expressions with complex syntax. For example, if the "CoverTypeNo" is set as "Normal" in the source data, specify it as the number 1 in the target table.

A full reference on Expression editor's operators and functions is provided in Chapter 25 Expression Editor *(p. 533)*.

Assignments for CAD files

When MIKE+ data are exported to a CAD file, the following assignments can be used to control the properties of the CAD data:

- CAD_Text: text (label) for the exported feature. Only supported for point layers.
- CAD_LineTypeName: the line's name defined in the *.dwg template (MIKEPlusTemplate.dwg), e.g. Dash1.
- CAD_ColorIndex: the index of the color to be used in the CAD file. The index is an integer value. Can be used for point, text, polyline, polygon and block layers.
- CAD_ColorRGB: text string of RGB color representation. The divider is a comma (for example "128,0,255"). Can be used for point, text, polyline, polygon and block layers.
- CAD_LineWeight: line weight used by DWG files.
- CAD_Rotation: rotation of the element (typically used for text strings). The value is expressed in degrees.
- CAD_Height: height of the element (typically used for text strings). The value is expressed in the CAD "paper" unit.
- CAD_Block: name of the block. Some of the existing block names from the template file can be used. Block names available in the template are typically: Valve, Demand Allocation, Emitter, Check_valve, Load_point, Mouse_Basins, Mouse_Curb_Inlet, Mouse_Orifices, Mouse_Outlets, Mouse_Pumps, Mouse_Soakaway, Mouse_Valves, Mouse_Weirs, Reservoir, Swmm_Outlet, Tank, Turbine, Circle, Rectangle, Triangle.
- CAD_Block_ScaleX: scale of the block in the X direction.
- CAD_Block_ScaleY: scale of the block in the Y direction.
- CAD_Block_Height: block height. The value is expressed in the CAD “paper" unit.
- CAD_Block_Rotation: block rotation angle.
- CAD_FillColorIndex: index of the fill color, used by polygon layers.
- CAD_FillColorRGB: text string of RGB fill color representation, used by polygon layers.

These assignments must always be used in combination with a 'Geometry' assignment, used to export the shape on the map (point, polyline and polygon].

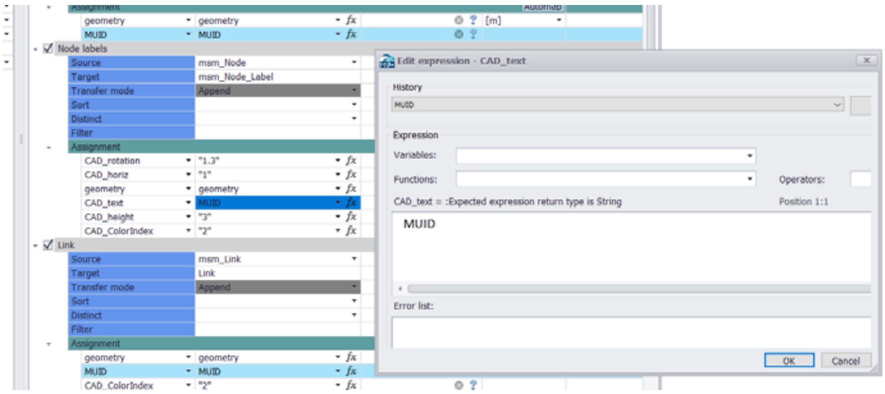


Figure 6.23 Creating assignments for CAD attributes

The source of these assignments must always be a text string. The text can be provided directly (to be specified in quotes, e.g. "1.25" where the decimal separator must be a point) or may be exported from a string attribute (e.g. MUID) or using a function 'ToString' to convert other input attributes to text.

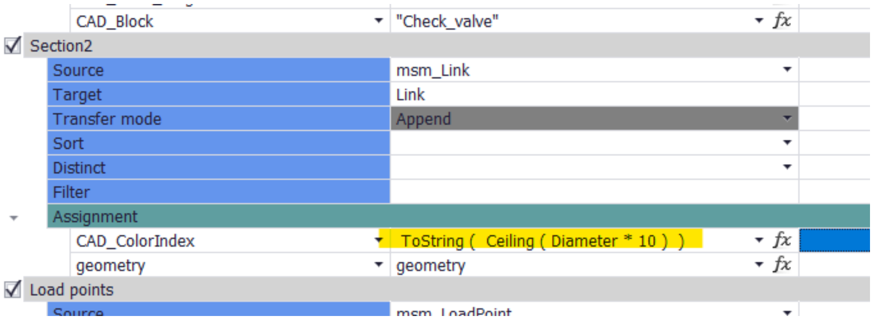


Figure 6.24 Creating an assignment controlling the color as a function of the diameter

6.2.6 Import/Export Toolbar

Several vital operations with Import/Export are accessible through Import/Export toolbar.

Figure 6.25 Toolbar

Reload Source: Updates the contents of the source storage cache

This functionality secures that the specified Import/Export configuration operates on the latest source data available.

Clear: Remove any configuration from the Import/Export tool

This functionality clears any configuration and cache data contents for the Import/Export tool.

Save: Save the import configuration for reuse

The configuration may be saved to a file (XML) for later reuse (by loading the saved XML file). This is very useful for supporting repetitive import/export operations.

The import/export configuration files include all jobs present in the tool at the time of file saving. I.e. individual jobs cannot be saved separately. The tool does not support consecutive loading of multiple configuration files. i.e. when a configuration file is loaded, any current contents in the tool is cleared.

Combining job configurations from two or more configuration files in one file (e.g. to achieve automatic sequential execution) can be done by editing (copy/paste) the configuration file in a simple text editor (e.g. Notepad). All information related to one job is found between the keywords <JobSerializationEntity> (beginning of one job configuration) and </JobSerializationEntity> (end of one job configuration).

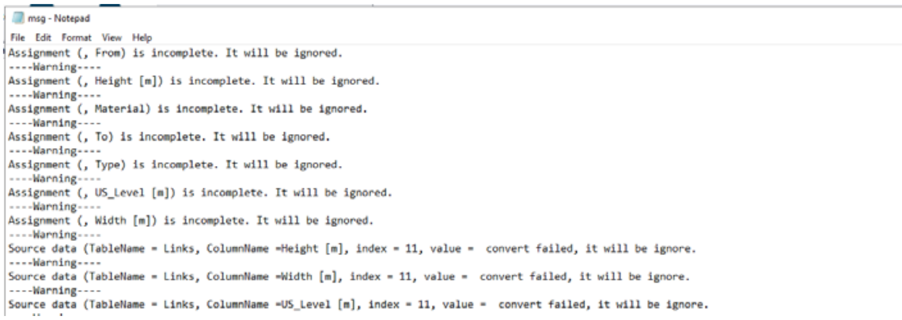
Verify: Check the configuration for errors and warning

The "Verify" function checks for syntax errors, missing data, etc. The configuration will only run if no errors are reported by the verification function.

However, syntactic correctness is only a necessary condition, but is not enough to ensure that the result of the specified import will be correct.

"Verify" may issue several warnings. Warnings are typically related to missing data. The presence of warning does not affect the execution of the import configuration.

By clicking on the 'Warning log' or 'Error log' message a text file is opened and enumerates all warning or error issues.


msg - Notepad

File Edit Format View Help

```
Assignment (, From) is incomplete. It will be ignored.
----Warning----
Assignment (, Height [m]) is incomplete. It will be ignored.
----Warning----
Assignment (, Material) is incomplete. It will be ignored.
----Warning----
Assignment (, To) is incomplete. It will be ignored.
----Warning----
Assignment (, Type) is incomplete. It will be ignored.
----Warning----
Assignment (, US_Level [m]) is incomplete. It will be ignored.
----Warning----
Assignment (, Width [m]) is incomplete. It will be ignored.
----Warning----
Source data (TableName = Links, ColumnName =Height [m], index = 11, value =  convert failed, it will be ignore.
----Warning----
Source data (TableName = Links, ColumnName =Width [m], index = 11, value =  convert failed, it will be ignore.
----Warning----
Source data (TableName = Links, ColumnName =US_Level [m], index = 11, value =  convert failed, it will be ignore.
```

Figure 6.26 Warning log or Error log message in the import/export configuration

Run: Execute the Import/Export setting

Upon verifying the specified configuration, the Import/Export job is executed by pressing the "Run" button.

6.3 Import/Export Workflows

6.3.1 Creating and executing new Import/Export configuration

1. Create the job(s) and sections by adding them with the respective 'Add Job' and 'Add Section' buttons.
2. Set up the job and sections.

   The import/export tool may include several jobs, each of these containing several sections. Jobs and sections can be excluded/included in the actual execution by toggling the ON/OFF checkbox in the left side of each job or section header line.

3. Select the job(s) and section(s) to be applied by toggling the checkbox ON/OFF for those to be executed.
4. Click on the 'Verify' in the top ribbon. Once there are no errors the 'Run' button is activated and the user can click on it.
5. The import commences and the progress is be visualised in the Log View window.

6.3.2 Reloading and executing existing Import/Export configuration

1. Create an Import/Export configuration as described in the previous chapter.
2. Save the configuration using the 'Save' button in the upper toolbar, to save the Import/Export configuration to a *.xml file.

This configuration file can later be re-used in any other MIKE+ project, following these steps:

1. Open the Import and Export tool in the new MIKE+ project.
2. Load the *.xml configuration file using the 'Load' button in the upper toolbar.
3. When the configuration is loaded, press the 'Run' button. The import commences and the progress is visualised in the Log window.

6.3.3 Executing an Import/Export configuration from command lines

When setting up numerical models, you often utilize the MIKE+ editor to access all the tools to define the model data, including the 'Import and export' tool. However, there are times when it is required to import or export files in an automated way without going through the related editors.

The MIKE+ executables enable you to execute some tools without opening the editor, through command lines. It is possible to run the 'Import and export' tool in this manner, assuming you have prepared the Import/Export configuration file beforehand in MIKE+.

Start by locating the MIKE+ executable named DHI.MIKEPlus.ToolShell.exe in the installation folder. From a command prompt, type the command below to access the list of supported tools, replacing the … characters by the actual path to the file:

"C:\...\DHI.MIKEPlus.ToolShell.exe" -h

The format of the command for executing an Import/Export configuration is:

"C:\...\DHI.MIKEPlus.ToolShell.exe" ImportTool -f [Configuration file] [Option]

Where [Configuration file] is the path to the *.xml configuration file.

The only option available is: -p [Path to Oracle.DataAccess.dll]. This option is mandatory when the source database is an Oracle Spatial database, and must supply the path to the file Oracle.DataAccess.dll which must be installed on the machine prior to the import.

Note: the selected *.xml configuration file must contain the path to the MIKE+ database to import to / from. This configuration file must be saved from the MIKE+ interface, and when the import job imports to the currently opened MIKE+ database, the configuration file is saved with an empty path and file name (this is designed to make the configuration file applicable to any database in which is it loaded). So, before using such a configuration file from a command line, it is necessary to update it and provide the path to the MIKE+ database (typically *.sqlite file). This is a path relative to the location of the configuration file, for example if the *.sqlite file is located in the same folder as the configuration file, the path should look like this:

<*JobPropertyValue property*=”Target” *value*=".\DatabaseName.sqlite" *jobNodeType*=”Jobproperty" />

6.4 Predefined Import and Export Routines

MIKE+ comes with some standard (automated) routines for import and export from and to commonly used formats. These include the following:

- Import of projects from MIKE URBAN formats (*.MDB and *.GDB)
- Import of MIKE 21 model setup
- Import of MIKE HYDRO River model setup.
- Import of MIKE 11 model setup. Some functionalities and options from MIKE 11 are not supported in MIKE+ and cannot be imported.
- Import of MIKE FLOOD model setup. Two options are available:
  - Import full MIKE FLOOD model setup: imports all data from a MIKE FLOOD setup file. All data files used in the selected MIKE FLOOD setup will be imported to the MIKE+ database: MIKE URBAN classic, MIKE 21, MIKE HYDRO River, MIKE 11, couplings. Some functionalities and options from MIKE 11 are not supported in MIKE+ and cannot be imported.
  - Import MIKE FLOOD couplings: imports only the couplings from a MIKE FLOOD file. Related river, urban and/or 2D data files are not imported. A related MIKE HYDRO River file can be reused in MIKE+ using the coupling to MIKE HYDRO River, instead of importing its data into the MIKE+ database.
- Import of SWMM model setup
- Import of EPANET model files
- Export of CS and River model setup to MIKE1D engine input file (M1DX)
- Export of 2D overland setup (MIKE 21 model)
- Export of MIKE FLOOD couplings
- Export of SWMM collection system model setup to SWMM model file
- Export of WD model setup to EPANET model file

The predefined Import/Export jobs are accessed directly through the main "File" menu option, rather than going to the Import/Export tool.

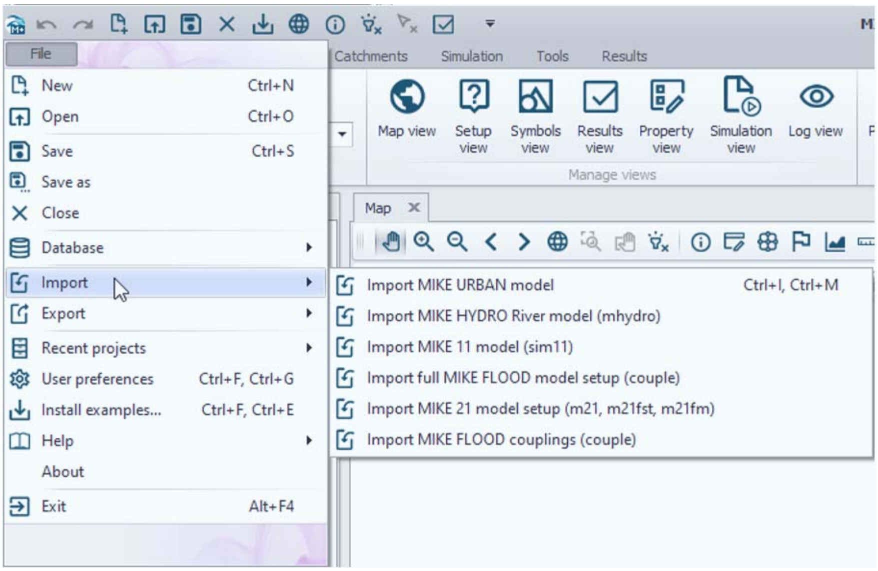


Figure 6.27 Example: Pre-defined Import jobs for MIKE+ CS and Rivers, accessed from "File" menu

Any pre-defined import works in "Overwrite" mode, i.e. any content in the current MIKE+ Database will be deleted and replaced by the new contents, except the import from MIKE HYDRO River and from MIKE 11 which will append the imported data to the existing data in the database.

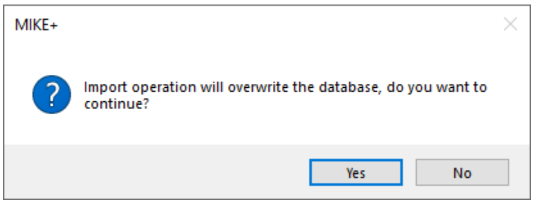


Figure 6.28 Before executing any the pre-defined import, user gets prompted to accept overwrite of any contents which might exist in the current target database.

Any warning or error occurring during a predefined import task will be reported in the log view in MIKE+. In some cases, this log view can report actions to undertake after the import, and should therefore be reviewed carefully. These reported messages are also saved to a log file saved in the database's folder, so that they can be reviewed at a later stage.

6.4.1 Import from a MIKE URBAN Model

Importing a MIKE URBAN model to MIKE+ requires that MIKE URBAN (Release 2020 Update 1) is installed on the computer. This is because the import routine uses some software components associated with MIKE URBAN Classic.

Before using the predefined import for a MIKE URBAN model, it is necessary to update any old models to MIKE URBAN Release 2020 Update 1 so that the *.MDB or .*.GDB source database is in the correct format. Ensure that all related files (including any selection files, customized dhiapp.ini files, time series files, etc.) are all correctly selected in the MIKE URBAN project so that the paths to these files can be properly imported. Finally, for Collection System models, ensure that your simulation(s) run with the MIKE 1D engine, and correct input data as necessary if any error is reported.

Once the MIKE URBAN model is prepared, the following steps will import the model into MIKE+:

- Open a new MIKE+ window
- Click on File|Import

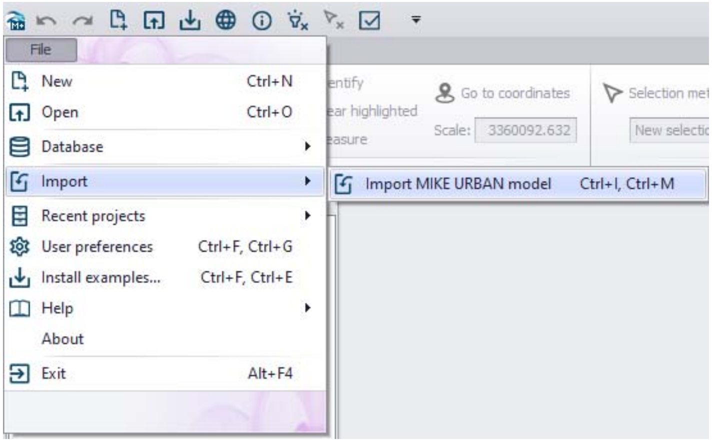


Figure 6.29 Accessing and activating the pre-defined import of a MIKE URBAN Classic project.

- Choose the source database type ("*.MDB" or "*.GDB")
- Browse the source database
- Specify the target MIKE+ database's type and location

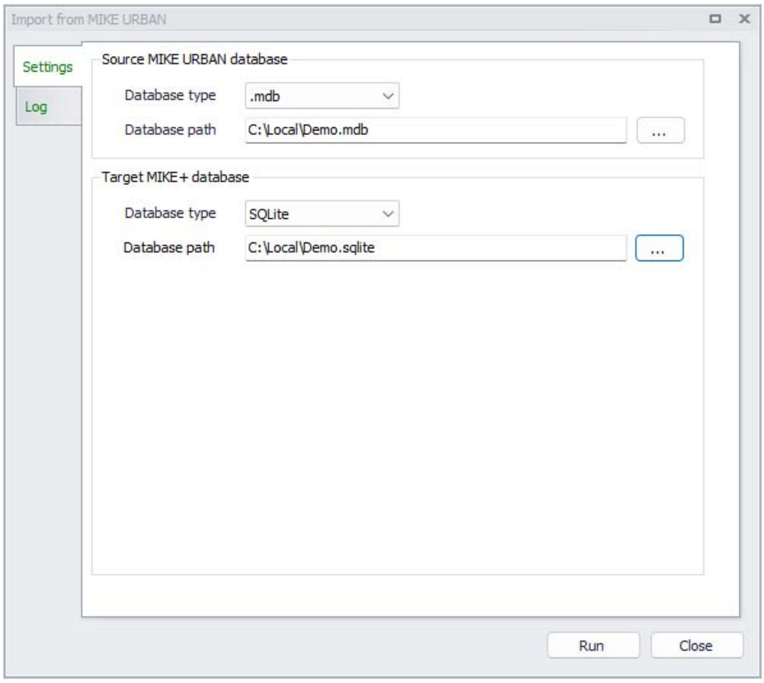


Figure 6.30 The 'Import from MIKE URBAN' window.

- Press the 'Run' button, which will initiate the import process
- The progress and success (or failure) of the import can be followed in the Log tab

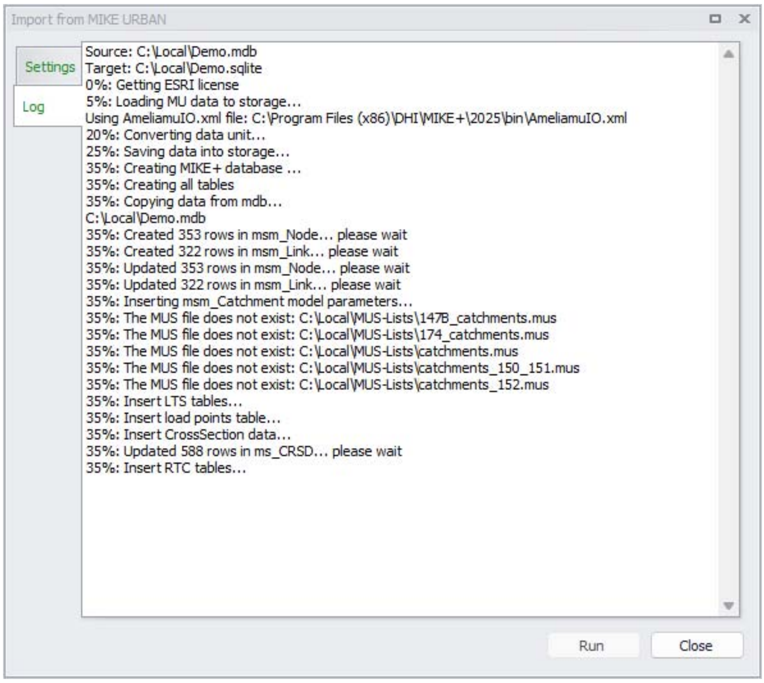


Figure 6.31 Log View reports on import progress, and issues warnings and error messages. Upon completed import, the log should be carefully reviewed and in case of any anomaly reported, an appropriate action should be taken.

If a MIKE+ database is opened before opening the 'Import from MIKE URBAN' window, the MIKE URBAN project will automatically be imported into the opened MIKE+ database. This MIKE+ database will be overwritten during the import, i.e. all data previously defined in the database will be lost and replaced by data and settings imported from MIKE URBAN.

Import limitations of MIKE URBAN models

The data below are not imported when importing a Collection System project from MIKE URBAN Classic:

- WQ Process model data (N/A in MIKE+)
- SRQ data (New concepts are implemented in MIKE+, see documentation)
- SWQ Surface Runoff pollutants (New concepts are implemented in MIKE+, see documentation)
- SWQ local treatments (N/A in MIKE+)
- ST data
- LTS statistics specifications (included as part of MIKE+ output definition)
- Emptying storage nodes (N/A in MIKE+)

6.4.2 Import from a MIKE HYDRO River model

Before importing a MIKE HYDRO River model to MIKE+, it is necessary to save the MIKE HYDRO River file in its latest software version, i.e. with MIKE Zero 2023, so that the file is in the correct format. Ensure that the MIKE HYDRO River simulation runs successfully before importing, in order to avoid errors during the import.

Once the MIKE HYDRO River model is prepared, the following steps will import the model into MIKE+:

- Select the model type 'Rivers, collection system and overland flows' in the 'Model type' editor
- Click on File|Import
- Choose 'Import MIKE HYDRO River model (mhydro)

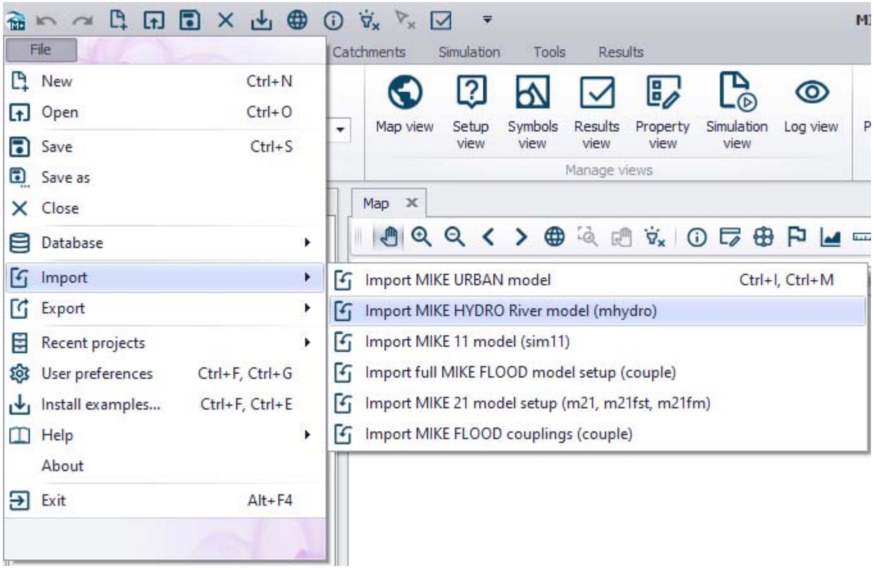


Figure 6.32 Accessing and activating the pre-defined import of a MIKE HYDRO River model.

- Browse the source file
- Press the 'OPEN' button, which will initiate the import process
- The progress and success (or failure) of the import can be followed in the Log View

During the import, if the map projections defined respectively in the MIKE HYDRO River and the MIKE+ models differ, the MIKE HYDRO River data will be reprojected in the map projection used in MIKE+.

6.4.3 Import from a MIKE 11 model

Before importing a MIKE 11 model to MIKE+, it is necessary to save the MIKE 11 files with their latest software version, i.e. with MIKE Zero 2023, so that the files are in the correct format. Ensure that the MIKE 11 simulation runs successfully with the MIKE 1D engine before importing, in order to avoid errors during the import.

Once the MIKE 11 model is prepared, the following steps will import the model into MIKE+:

- Select the model type 'Rivers, collection system and overland flows' in the 'Model type' editor
- Click on File|Import
- Choose 'Import MIKE 11 model (sim11)
- Browse the source *.sim11 file
- Press the 'OPEN' button, which will initiate the import process
- The progress and success (or failure) of the import can be followed in the Log View

During the import, if the map projections defined respectively in the MIKE 11 and the MIKE+ models differ, the MIKE 11 data will be reprojected in the map projection used in MIKE+.

Some functionalities and options from MIKE 11 are not supported in MIKE+ and cannot be imported. This especially applies to the Sediment Transport module, which has been fully revisited compared to its MIKE11 version. The functionalities and the scientific theory used in MIKE+ are therefore different: some MIKE 11 functionalities may not be supported anymore in MIKE+, whereas new functionalities are added. When converting a MIKE11 model including Sediment Transport to MIKE+, it is therefore expected that significant differences of sediment transport results are encountered, which may require that you adjust the model definition if needed.

6.4.4 Import of 2D Overland Setup Files

- Select the model type 'Rivers, collection system and overland flows' in the 'Model type' editor
- Click on File|Import
- Choose one among three available options for overland model sources (complete MIKE FLOOD setup, MIKE 21 model or MIKE FLOOD couplings)

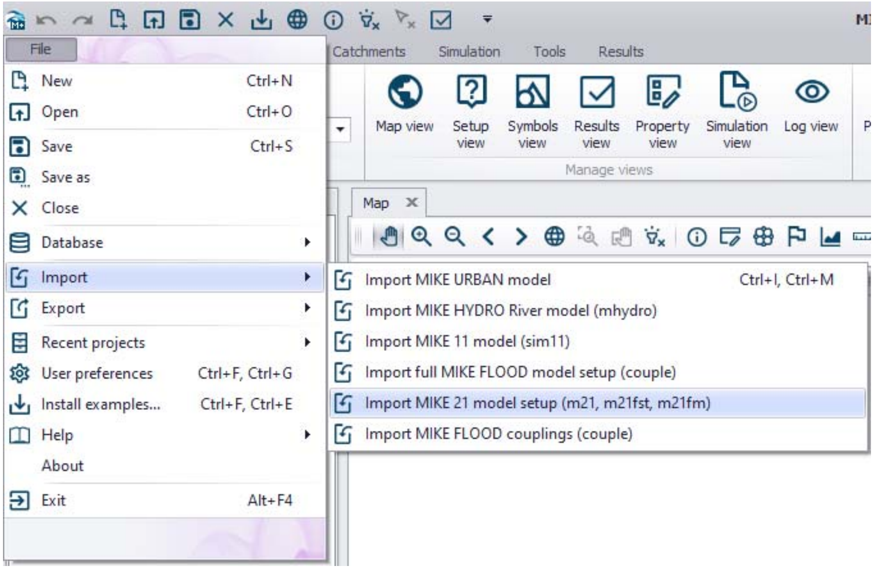


Figure 6.33 Accessing and activating the pre-defined import of a MIKE 21 model setup.

- Browse the source file

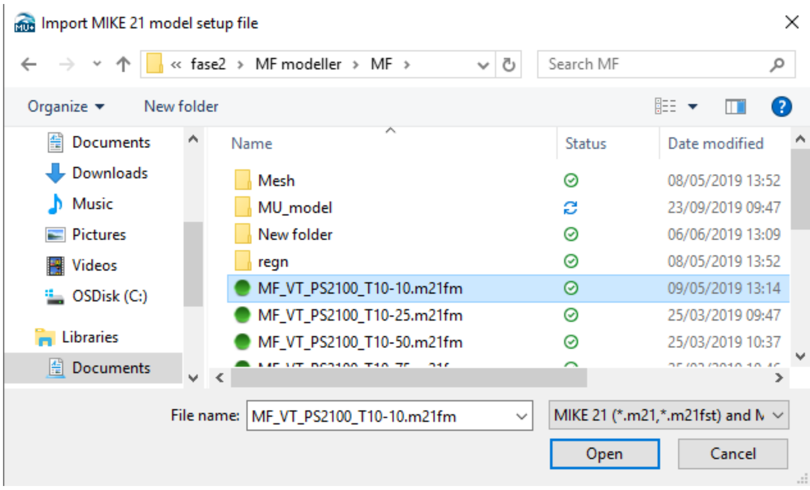


Figure 6.34 File-browser points to the wanted source file (*.M21, *.M21FST or *.M21FM). Alternatively, only coupling data or complete MF setup can be imported by pointing to a MF source file (*.couple).

- Press the 'OPEN' button, which will initiate the import process
- The progress and success (or failure) of the import can be followed in the Log View as well as a progress bar

Information	General	Import is about to start	23/09/2019 10:19:22
Information	General	Importing from C:\Users\bet\OneDrive - DHI\Documents\Gåsebækre…	23/09/2019 10:19:22
Warning	Import	MIKE 21 FM result file MF_VT_PS2100_T10-10_Hmax didn't save res…	23/09/2019 10:21:19
Warning	Import	MIKE 21 FM result file MF_VT_PS2100_T10-10_T10 didn't save result…	23/09/2019 10:21:19
Information	General	Import Flow Model FM (.m21fm) finished	23/09/2019 10:21:20

Figure 6.35 Log View reports on import progress, and issues warnings and error messages. Upon completed import, the log should be carefully reviewed and in case of any anomaly reported, an appropriate action should be taken.

6.4.5 Import of SWMM File

- Select the model type 'SWMM5 collection system and overland flows' in the 'Model type' editor
- Click on File|Import|Import SWMM model

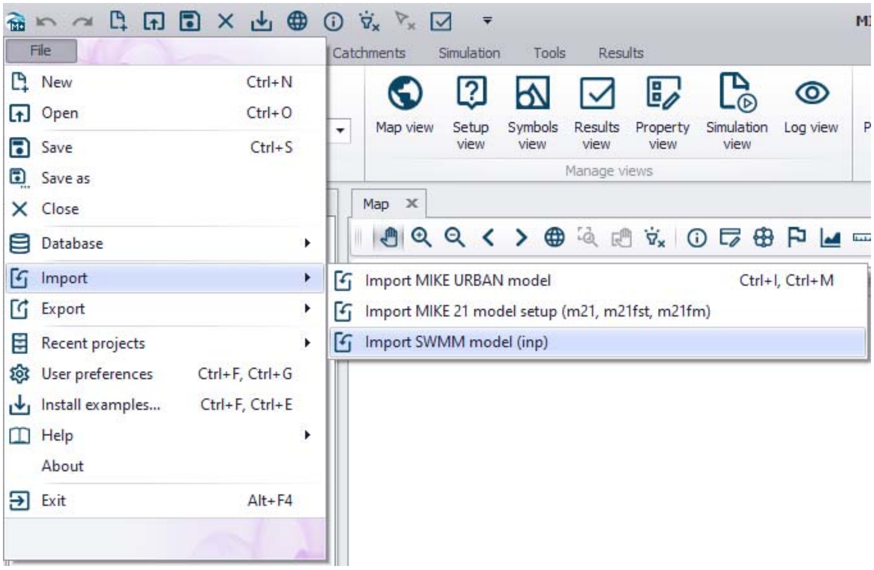


Figure 6.36 Accessing and activating the pre-defined import of SWMM model file (*.INP).

- Browse the source file (*.INP)

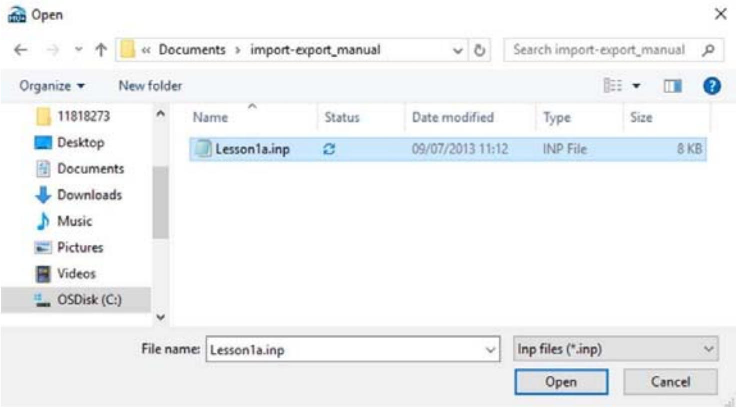


Figure 6.37 File-browser points to the wanted SWMM file (*.INP).

- Press the 'OPEN' button, which will initiate the import process
- The progress and success (or failure) of the import can be followed in the Log View as well as a progress bar

6.4.6 Import of EPANET File

- Select the model type 'Water distribution' in the 'Model type' editor
- Click on File|Import|Import EPANET model

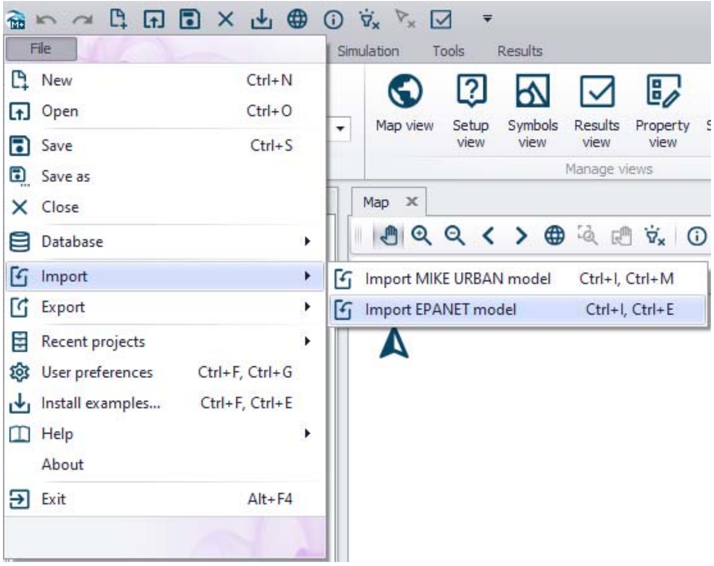


Figure 6.38 Accessing and activating the pre-defined import of EPANET model file (*.INP).

- Browse the source file (*.INP)

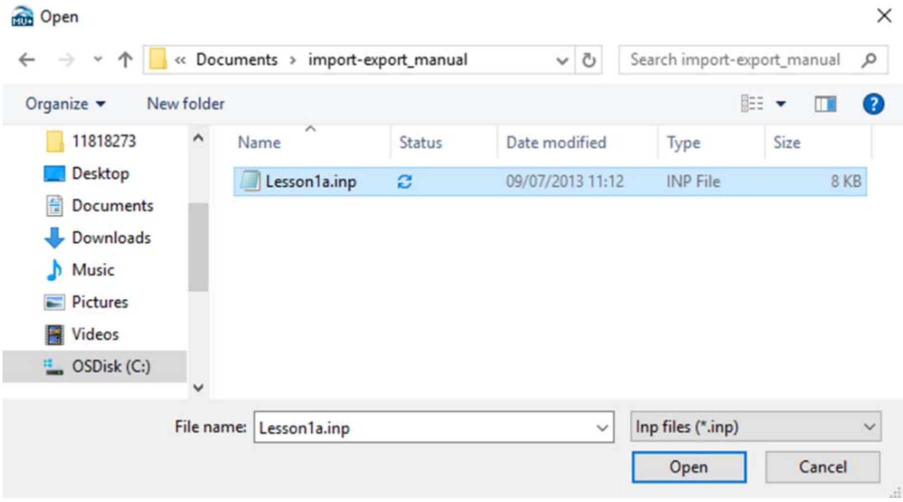
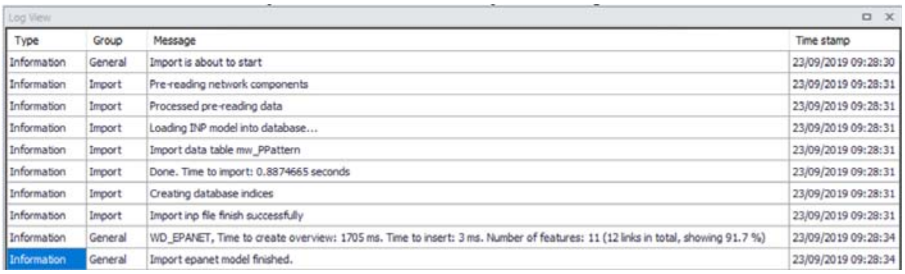


Figure 6.39 File-browser points to the wanted EPANET file (*.INP).

- Press the 'OPEN' button, which will initiate the import process
- The progress and success (or failure) of the import can be followed in the Log View as well as a progress bar



Type	Group	Message	Time stamp
Information	General	Import is about to start	23/09/2019 09:28:30
Information	Import	Pre-reading network components	23/09/2019 09:28:31
Information	Import	Processed pre-reading data	23/09/2019 09:28:31
Information	Import	Loading INP model into database...	23/09/2019 09:28:31
Information	Import	Import data table mw_PPattern	23/09/2019 09:28:31
Information	Import	Done. Time to import: 0.8874665 seconds	23/09/2019 09:28:31
Information	Import	Creating database indices	23/09/2019 09:28:31
Information	Import	Import inp file finish successfully	23/09/2019 09:28:31
Information	General	WD_EPANET, Time to create overview: 1705 ms. Time to insert: 3 ms. Number of features: 11 (12 links in total, showing 91.7 %)	23/09/2019 09:28:34
Information	General	Import epanet model finished.	23/09/2019 09:28:34

Figure 6.40 Log View reports on import progress, and issues warnings and error messages. Upon completed import, the log should be carefully reviewed and in case of any anomaly reported, an appropriate action should be taken

6.4.7 Export to M1DX File

To export the model to an M1DX file:

- Select the model type 'Rivers, collection system and overland flows' in the 'Model type' editor
- Choose File|Export
- Click on 'Export to M1DX file'

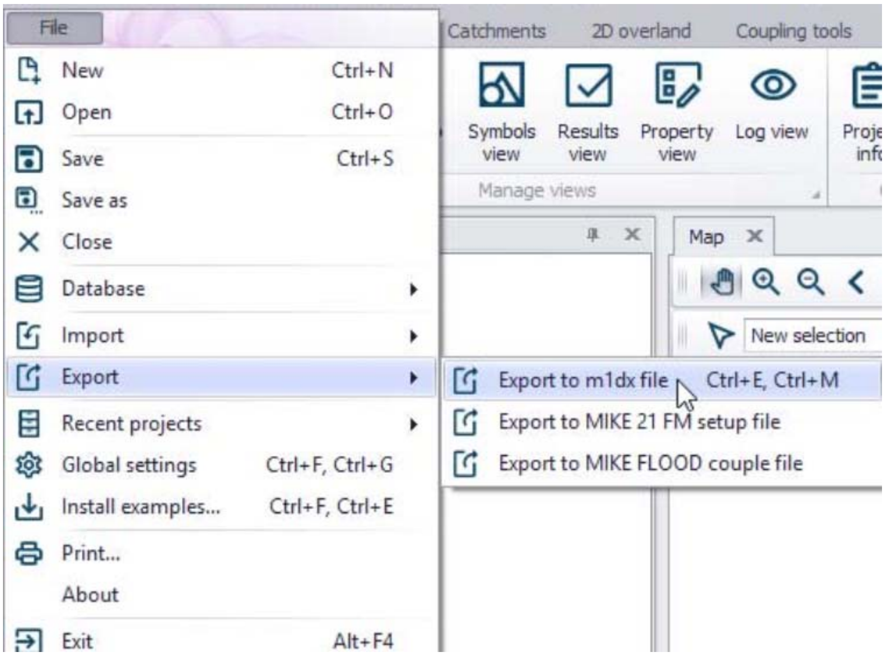


Figure 6.41 Accessing and activating the pre-defined export of MIKE+ CS project to MIKE1D input file (*.M1DX).

- Browse the location to export the new MIKE 1D file
- Optionally change the name of the new export file which has by default the name of the current project

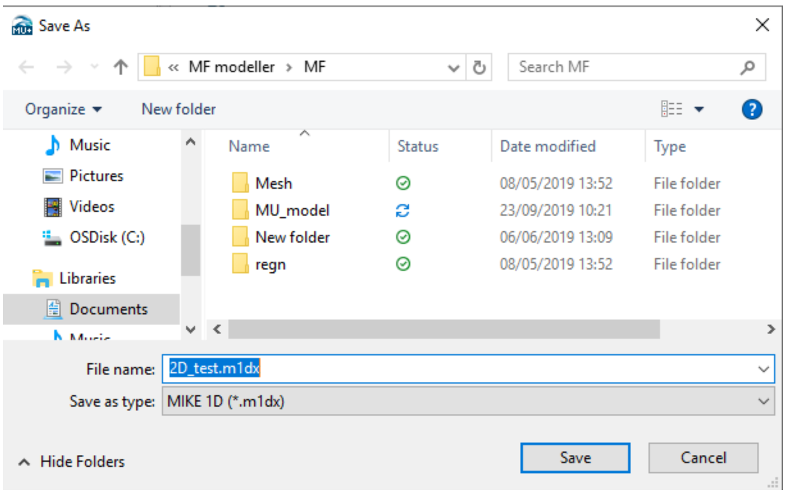


Figure 6.42 The file is saved at a location specified by the user. Per default, the file-name is given as the name of the MIKE+ source database but can be modified by the user.

- Press the 'Save' button

After the export has completed, the dialog will be closed automatically. Two files are created (CS network): MIKE Zero Cross Sections (*.XNS11) and an M1DX File (*.M1DX).

6.4.8 Export to MIKE 21 FM Setup File

To export the model to an *.M21FM file:

- Select the model type 'Rivers, collection system and overland flows' in the 'Model type' editor
- Choose File|Export
- Click on 'Export to MIKE 21 FM setup file'

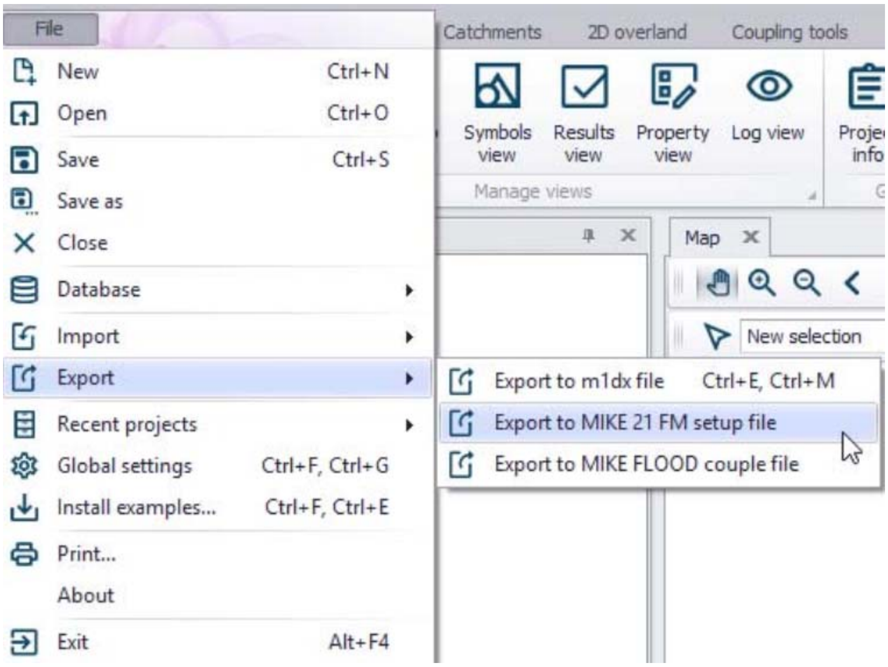


Figure 6.43 Accessing and activating the pre-defined export of MIKE+ 2D overland data MIKE 21 FM model file (*.M21FM).

- Browse the location to export the new MIKE 21 FM file
- Optionally change the name of the new export file which has by default the name of the current project

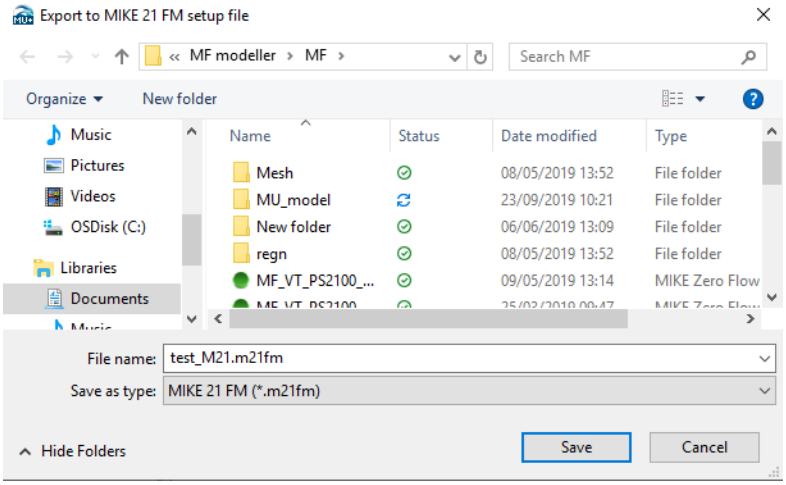


Figure 6.44 The file is saved at a location specified by the user. Per default, the file name is given as the name of the MIKE+ source database but can be modified by the user.

- Press the 'Save' button

6.4.9 Export to EPANET Model File

To export the model to an EPANET *.INP file:

- Select the model type 'Water distribution' in the 'Model type' editor
- Choose File|Export|Export EPANET model

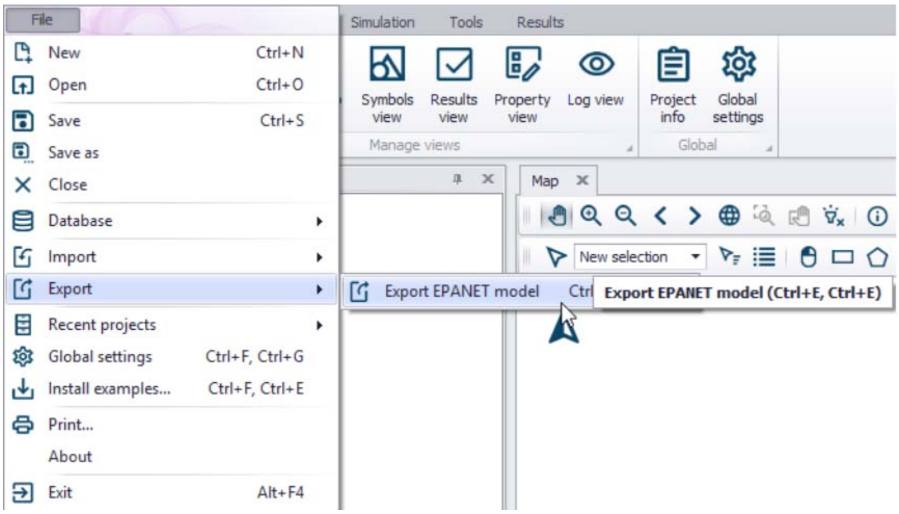


Figure 6.45 Accessing and activating the pre-defined export of MIKE+ WD project to EPANET model file (*.INP).

- Browse the location to export the new EPANET model file
- Optionally change the name of the new export file which has by default the name of the current project

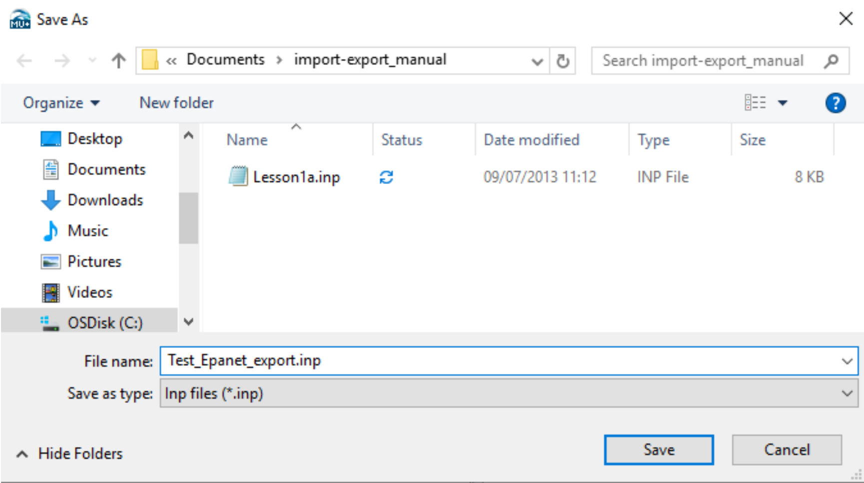


Figure 6.46 The EPANET file is saved at a location specified by the user. Per default, the file name is given as the name of the MIKE+ source database but can be modified by the user.

- Press the 'Save' button

6.4.10 Export to SWMM Model File

To export the model to a SWMM *.INP file:

- Select the model type 'SWMM5 collection system and overland flows' in the 'Model type' editor
- Choose File|Export|Export SWMM model

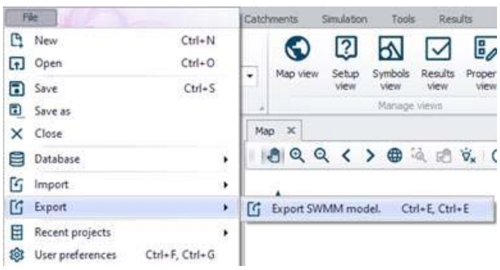


Figure 6.47 Accessing and activating the pre-defined export of MIKE+ SWMM data to SWMM model file (*.INP).

- Browse the location to export the new SWMM model file

- Optionally change the name of the new export file which has by default the name of the current project

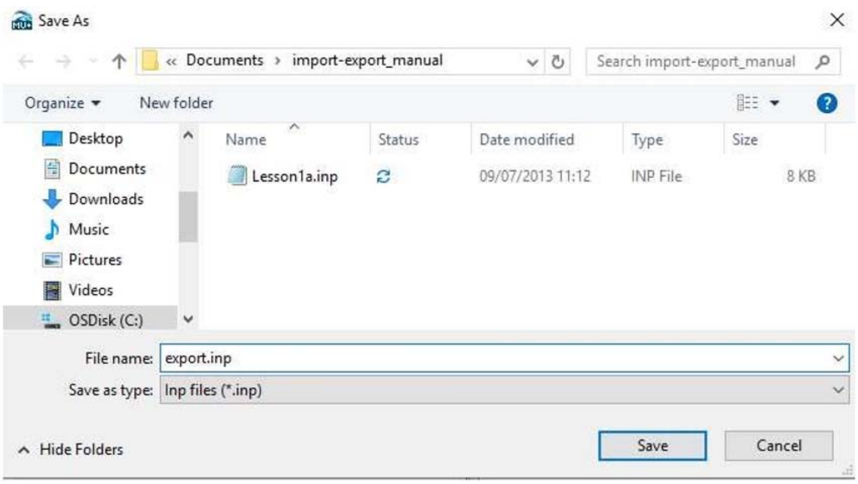


Figure 6.48 The SWMM file is saved at a location specified by the user. Per default, the file name is given as the name of the MIKE+ source database but can be modified by the user.

- Press the 'Save' button

6.4.11 Predefined export from command lines

When working with numerical modelling you often utilize the MIKE+ editor to setup the model and hereafter export the simulation file or execute the simulation using the Run command. However, there are times where it is required to export files and start up a simulation without going through the related editor.

The MIKE+ executable enables you to export simulation files and initiate a model simulation without opening the editor, through command lines. It requires that you have prepared the simulation setups beforehand in MIKE+.

Start by locating the MIKE+ executable named DHI.MIKEPlus.EngineShell.exe in the installation folder. From a command prompt, type the command below to access the description of available options, replacing the … characters by the actual path to the file:

"C:\...\DHI.MIKEPlus.EngineShell.exe" -h

The format of the commands for exporting simulation files or running simulations is:

"C:\...\DHI.MIKEPlus.EngineShell.exe" [Options] [File] where [File] is the path to the *.mupp or *.sqlite file.

The main options are described below:

- -e: Export simulation files (e.g. *.couple and *.m21fm files for a simulation including 2D overland)
- -r: Run the simulation
- -rb: Run the batch simulation
- -id: The simulation ID to be exported or executed (optional). If not specified, the active simulation is used.

Note: Simulations are executed with the parallelisation settings specified in the project, from the 'Parallelisation configuration' dialog.

6.5 Cloning the MIKE+ Database

This functionality creates a new database of the specified type and populates it with the data contained in the selected source database.

For the difference with "Project|Save as", this operation does not create a new MIKE+ project file (*.MUPP), but only a database. Cloning also skips damaged tables, which can be useful to repair a damaged database.

Cloning is also useful as data backup and/or when migrating between different database formats (SQLite -> PostGIS or PostGIS-> SQLite).

To clone a MIKE+ database:

- Choose File|Database|Clone database...

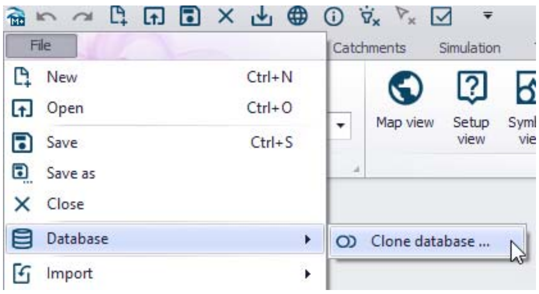


Figure 6.49 Accessing and activating the database cloning functionality

- Choose the source database type
- Select the source database, if not using the currently opened one
- Choose the target database type

- Browse the location to store the database clone and specify the new database name

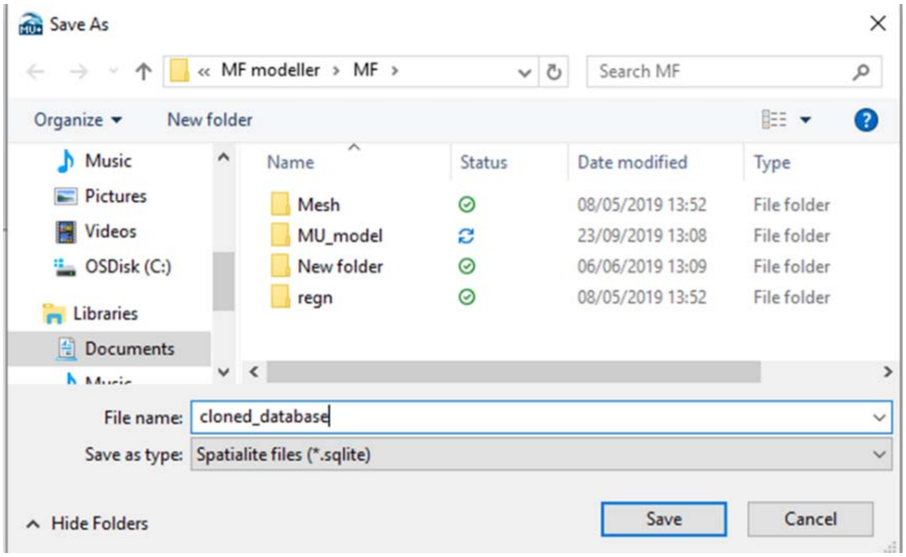


Figure 6.50 The database clone is saved at a location and name specified by the user.

- Press the 'OK' button to clone the database

Note that creating a new PostGIS database requires adequate user privileges.

7 Flagging

7.1 Introduction to MIKE+ Data Flags

MIKE+ provides the ability to track your data in a very flexible way. The tracking is done by assigning user defined 'flags' (status codes) to the data. A user has complete control of how many flags are used, the purpose and associated values.

7.1.1 What are flags?

Flags (Status Codes) are additional data attributes in the database that can be set to an integer value with a user defined meaning, useful for quality assurance, filtering and interrogation of the model. "Status Codes" is an internal table that maintains a customized list of internal values and corresponding names. Typically, when viewed within the user interface, names rather than integer values will be visible to the user.

7.1.2 What can be flagged?

Status codes can be assigned in all tables representing the physical model (i.e. pipes, manholes, nodes, pumps, catchments etc.). Status codes can be defined at two levels:

- **Record level:** This type of flag can be assigned to a record in a table. E.g. a node. A typical use could be to keep track of the data source or a status. The database field ID for this type of flag is 'Element_S'.
- **Attribute level:** The physical attributes of the model elements such as diameter, sizes or levels can have flags defined for individual attributes within a table record. These flags are typically useful for tracking data manipulation and editing operation. The database fields containing these flags contain a suffix "_S". For example, the "Diameter_S" field contains the corresponding flag to Diameter.

7.2 Defining Status Codes

MIKE+ comes with a set of predefined values which can be accessed in a model component table via the "Description" tab, "Status" drop-down list.

This default list of status codes can then be extended to include codes more relevant to your workflow. For example, "Surveyed 01/01/2019". Code numbers do not have be consecutive but must be unique.

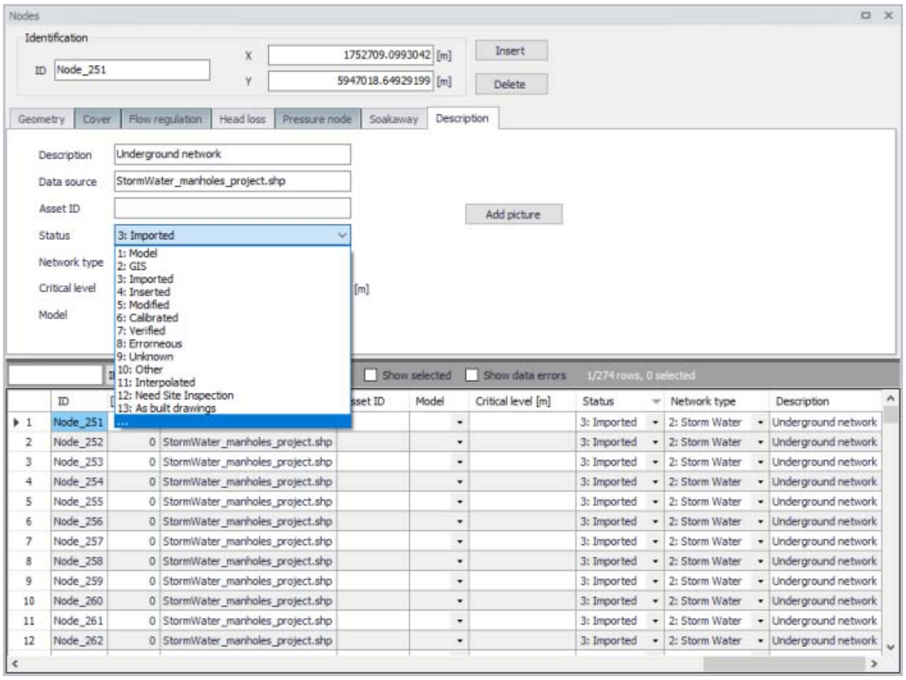


Figure 7.1 Customising status codes

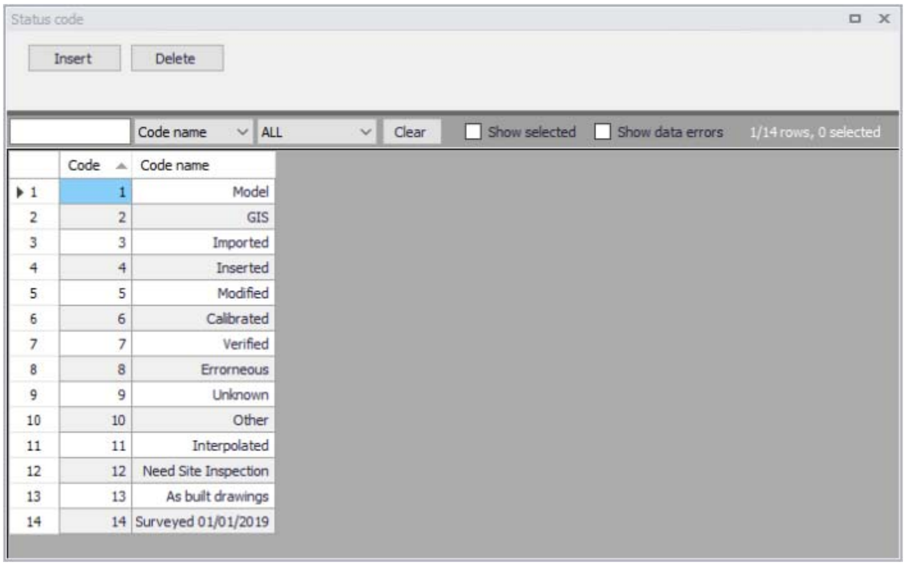


Figure 7.2 Defining status codes

While the underlying database uses the code values (integer), the code names (text) are visible in the MIKE+ graphical user interface. When using the field calculator the integer code value is expected or when accessing the database outside of the MIKE+ interface (ArcGIS, SQLite/PostGIS) only the code value is available.

7.3 Setting a Flag

The default value for all flags is 'null'. The value for a flag may be set by one of several methods:

During Import

Setting a flag value during an import is the easiest way to track the data source in cases where there are multiple data sources. This is done in the assignment specification as presented in Figure 7.3. where the record level flag is set to have a code value of 1 (Element_S = 1). The code value must correspond to the integer value of the status code, rather than the code name.

If your source database has a tracking/flagging field defined, this can be imported using the source data field ID and utilised to keep the two databases synchronised. Refer to the example in Figure 7.4 where "Diameter_S" is set to equal "Data_source".

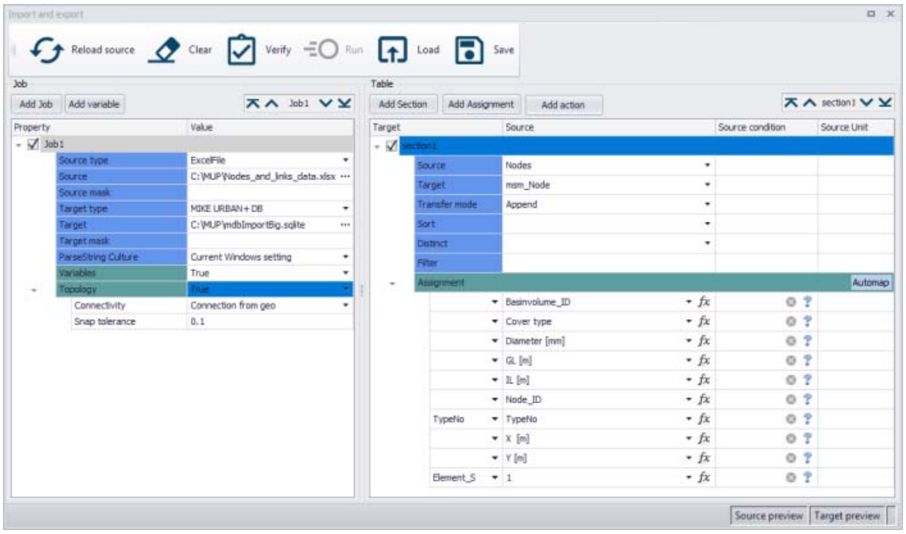


Figure 7.3 Setting the flags when importing data

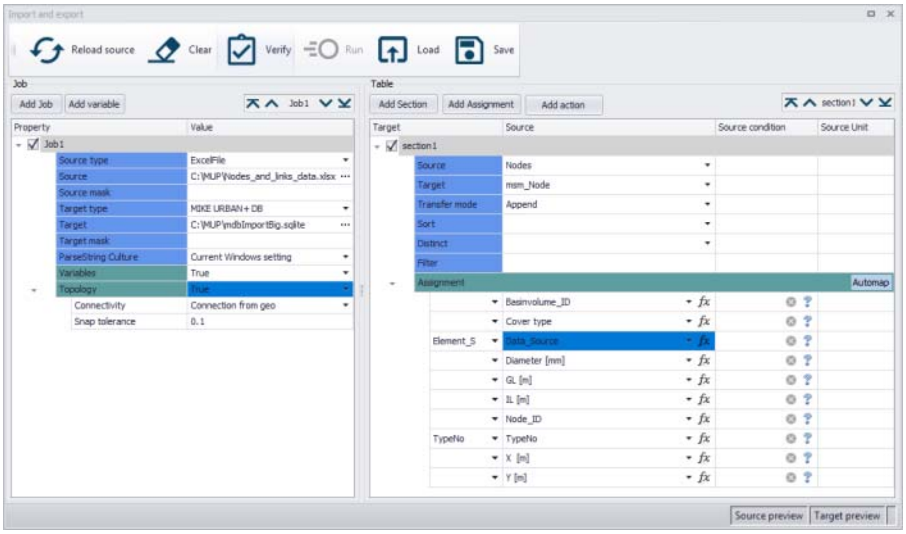


Figure 7.4 Importing flags from external source

Assigning Flags with Bulk Editing Tools

Several of the tools in MIKE+ that modify data en masse will offer to flag the affected records and attributes. By selecting the desired status code when prompted by the tool, the data updated in that session will be flagged, enabling further filtering, processing, checks or tracking.

As an example, the assignment tool is often used to fill data gaps. Figure 7.5 shows how to flag all affected node records as "Model" and all Diameters as "GIS". To force the assignment of a status code, the lower two boxes in the "Overall assignment" screen need to be ticked on, and the desired status code value selected from the drop-down list.

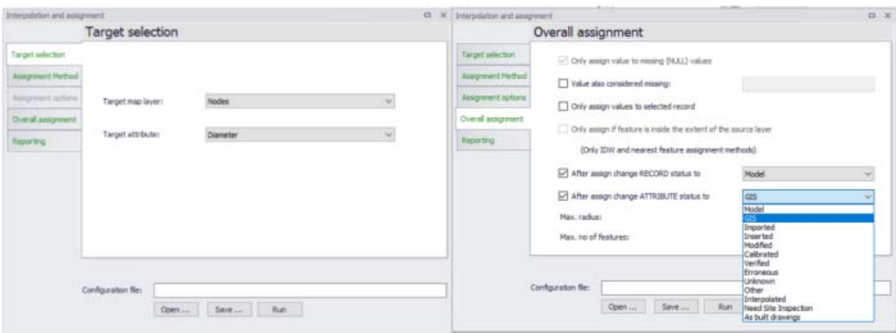


Figure 7.5 Using the assignment tool for setting the flags

Other Means of Setting the Flags

Flags can also be assigned by direct manual editing in the data tables. The values can be either typed directly into the Status field for each record or selected from a drop-down list. The Field Calculator can be used for flagging multiple records at a time. The entered values are automatically validated to ensure that they correspond to those available in the list of status codes.

7.4 Using the Flags

Flags are useful for finding, selecting, filtering, categorizing, reporting, processing and general tracking of model features and data quality.

Using status codes for feature selection is demonstrated in Figure 7.6. Note that only code values are visible in this dialog. Click on "Get Unique Values" to generate a list of all available status codes to use in the selection expression.

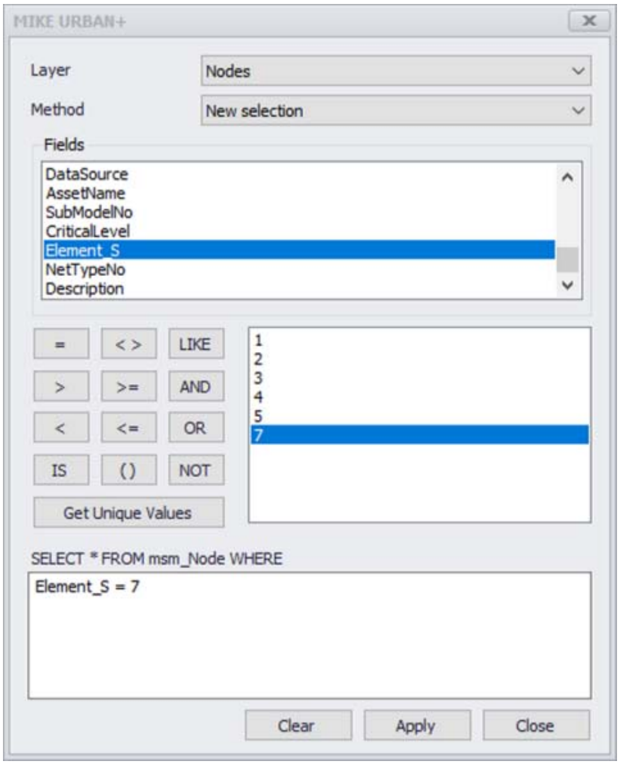


Figure 7.6 Selection by attribute

The flags are also very useful for visually displaying data quality information. For example, highlight areas with a low confidence in data quality. The symbology can be customised to assign different symbols and colours as shown in Figure 7.7. In this way it is easy to provide both an overview and at the same time highlight important features.

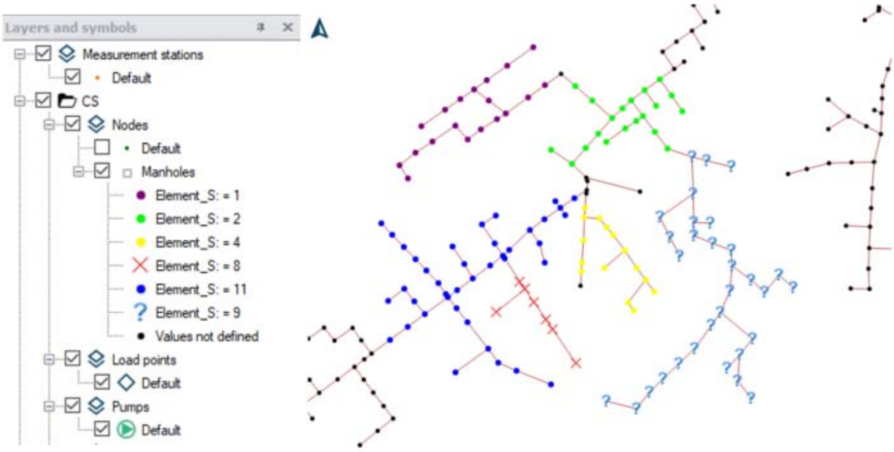


Figure 7.7 Utilizing symbology to display flags

8 Editing Tools

MIKE+ is very flexible in how a water distribution or a collection system model can be developed. Raw data can be brought into the model using a variety of input methods such as importing the data from a range of supported formats (see Chapter 6 Import and Export *(p. 147)*), utilising a range of tools available within the interface (e.g. Chapter 12 Interpolation and Assignment Tool *(p. 267)*), direct data entry into the MIKE+ tables or by visually digitising the pipe network through the MIKE+ Map interface.

8.1 Overview

Graphical editing tools are available for all network components such as:

- Model network elements (e.g. junction nodes, pipes, sewer manholes, storage tanks, pumps, valves),
- Demand points (consumption points),
- Load allocation points, or
- Catchments

Within each MIKE+ database table, functionality exists to efficiently edit the attributes of each model component.

Any alterations or changes made are immediately visible on the map and are automatically applied to the database tables. As each individual edit or update is recorded within a session, unlimited Undo and Redo is available, as long as the application is not closed.

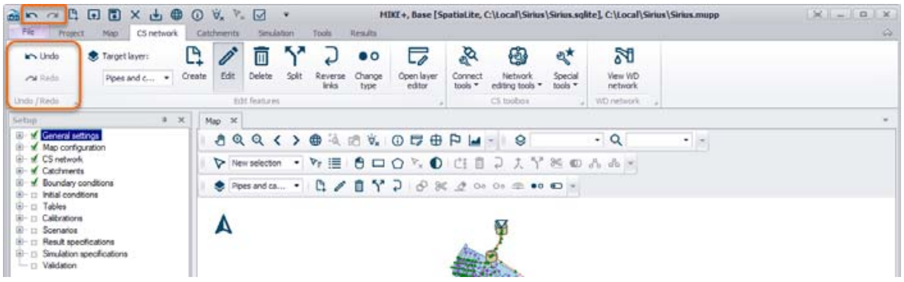


Figure 8.1 MIKE+ Graphical user interface with unlimited Undo and Redo functions

8.2 Graphical Editing

The easiest way of spatially defining a collection system or water distribution network, especially for smaller network additions to a model, is to graphically digitise the elements through the Map view.

To display the Map view, select Project | Map view. The Map view opens a drawing surface using the default coordinate system as defined during the creation of the MIKE+ project.

By utilising background maps (aerial photos, terrain, cadastral, asset layers, etc.) loaded into the Map view, components of a model can easily be graphically constructed.

8.2.1 Toolbars

Graphical editing tools, available both in the ribbon (e.g. in the CS/WD Network tab) or in the map view toolbar are used for interactively defining components of the model setup.

First a "Target Layer" must be selected. The list of target layers available in the map toolbar depends on the active modules in the project. Figure 8.2 and Figure 8.3 show the graphical editing tool bars for collection systems and water distribution systems respectively.

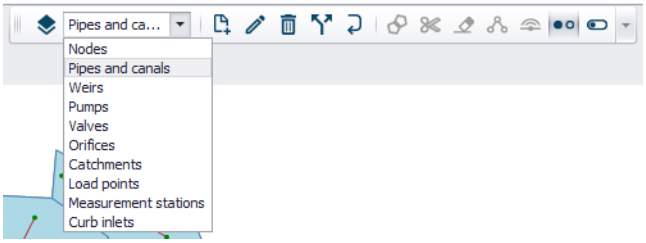


Figure 8.2 Collection System Network editing tools - Map view

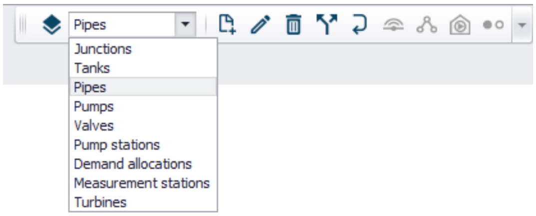


Figure 8.3 Water Distribution Network editing tools - Map view

After selecting a Target layer, the editing tools are available, and the required tool can be activated by clicking on the icon. The tool remains active until the icon is clicked again. Depending on the individual tool, a number of mouse-click actions are available within the Map view. Generally, single left mouse clicks will define extents, double clicks will complete the action, "enter" will finalise an action and "escape" will abort the tool.

A list of editing tools with a short description is summarised below.

Create new feature

This tool activates the ability to add a new network component, depending on the component type (target layer) selection.

Points (e.g. Nodes/Junctions) are added using a single mouse click on the map view. If the mouse click occurs over an existing pipe, the symbol for a new node will appear on the pipe and you will be asked if you would like to split the pipe. "Yes" will create a new node and split the existing pipe in two, automatically connecting the node and pipes. "No" will create a node but not split the existing pipe, therefore the node will not be automatically connected to the network.

Other network components that are represented by a line (pipes, weirs, valves, etc.) require a single mouse click to start the digitisation. If the first click occurs over an existing point feature, a line feature will automatically be connected to this point. The line feature can continue being digitised using single clicks but a double mouse click is required to complete the line. If the double click occurs over a point feature, the network component will automatically be connected to this node. If the double click does not occur over a point feature, a new node/junction will automatically be created.

New polygon features (catchments) are defined with single mouse clicks on the map view to define the polygon shape and a double click (or click "enter") to complete the polygon.

If the digitisation of a new feature needs to be aborted partway through, click the "Esc" button on your keyboard or click on the "create new feature" button in the MIKE+ interface again to deactivate the tool.

Edit

This tool is used to alter the geometry of an existing feature within the Map view. Click on the tool to activate it and then click on the network element to be changed. For point elements, it's location can be shifted. Shapes or lines (e.g. catchments or pipes) can be moved after clicking on the grab icon in the middle of the element. It is also possible to move the points defining a catchment or a pipe and therefore resize or rotate them.

Delete features

This tool is used to delete components within the Map view. Firstly, activate the tool by clicking on the icon. As you move around the visible network in the Map view, the cursor will change to + over an element that can be deleted, based on the selected target layer. Click on the element to delete it. Remember, it is possible to undo a deletion if something is deleted by mistake.

Split

This tool is used to graphically split a feature. To split a pipe, select the tool and click on the location of where the pipe is to be split. The tool automatically inserts a node at the split location. To split a catchment, draw a line across the catchment shape by a single click to start and a double click to complete the line (start and end must be outside the catchment boundary). The tool automatically deletes the existing catchment record and inserts two new catchment records. To split a river or 1D-2D coupling line, click on the location where the line should be split and the part of the line between the two closest vertices in both directions will be removed.

Reverse orientation of a line

This tool is used to swap the orientation of a line feature (from and to nodes/junctions) by clicking on the pipe. It is not possible to visually view the changes unless the pipe symbology has directional arrows included. The "From" and "To" node/junction change will be visible in the database tables.

Change element type

This tool is used to replace the type of point or line element, dependent on the selected target layer. A pipe can be replaced by a weir, pump, valve or orifice. A manhole can be replaced with a basin, outlet and junction.

Open/Close element

This tool is used to open or close a pipe within the network. Closing and opening actions alternate with every click. There is no visual change in the pipe appearance unless a dedicated symbology is used, but the underlying property "Enabled" will change and can be checked in the network tables. This tool is only available in Collection system mode.

Append catchment

This tool is used to insert a new catchment graphically by appending it to the external boundaries of existing catchments. Digitising the new catchment must start and end within an existing catchment. The face of the new catchment will automatically align with the existing catchments

Clip catchment

This tool is used to clip existing catchments defined by a polygon shape, excluding the remainder of the catchment. Activate the tool to draw a polygon (clipping extent). This can span over one or more catchments. Single mouse clicks will define the polygon and a double click will complete the polygon. To finalise the clip, click on "Enter" on your keyboard and then the underlying catchment/s will be clipped, maintaining the same attributes as the original catchments except for the geometrical area.

Erase catchment

This tool is used to define an area of a catchment to be removed from an existing catchment (opposite of the clip tool). Single clicks will digitise the extent of the polygon to be removed, a double click will complete the polygon and once "enter" is selected on the keyboard, a polygon will be deleted from the existing catchments.

Connect catchment

Once a catchment is selected, this tool is used to connect the selected catchment to a node or a link by simply activating the tool and clicking on a node or link. If a node is selected, a new catchment connection will be created, appearing visually on the map and as a new row in the catchment connections table. It is important to note that a catchment can be connected to multiple nodes. If a link is chosen to connect the catchment, the chainages of the link to distribute the catchment load will be requested. For catchments distributed to multiple nodes and links, the proportion of rainfall runoff and population equivalent from the catchment going to the node/link must be defined.

Connect Pump Station (only in WD network)

This tool is used to connect a pump station to the water distribution network.

Connect Demand Allocation (WD)/Connect load (CS)

This tool is used to connect either a demand allocation or a load point to the network.

Connect Station (both in CS and WD)

This tool is used to connect a measurement station to the network.

In addition to using the editing tools described above that allow you to work on an individual network element or define your network layout, there are several bulk editing tools associated with the selection toolbar. These tools, such as "Delete", union of pipes, etc will be executed on all selected network elements. (see Selection *(p. 63)*)

8.3 Graphical Editing Step-by-Step Example (CS)

Select the ‘Rivers, collection system and overland flows’ mode (Project | Model, from the main ribbon at the top of the MIKE+ interface). Make sure that all the modules you will need are turned on (from the ‘Setup’ tab on the left tree view, go to General Settings | Model type, and select the required modules for the ‘Rivers, collection system and overland flows’ mode). E.g. if you will have catchments, make sure that the "Rainfall-runoff (RR)" module is ticked on.

The main components of a sewer network can be defined as points (manholes), lines (pipes, weirs, etc) and polygons (catchments). Either in the Map view (Project, Map view) or in the CS Network and Catchments ribbons, graphical editing tools are available. First, select the "Target layer" from the drop-down list and the available tools for the selected target layer will be visible. Figure 8.4.

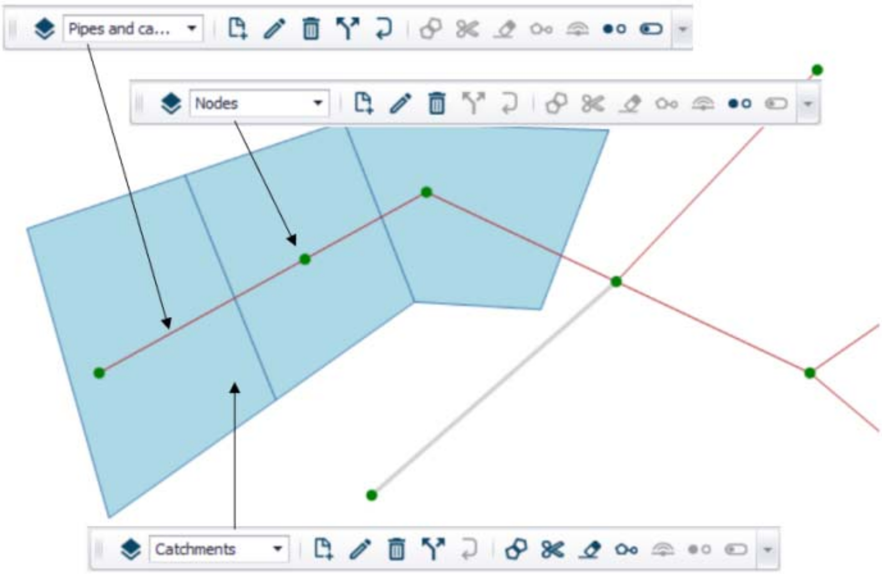


Figure 8.4 Tools activated for different "Target layers" (pipes, nodes, catchments)

To create new network components and add them to the model, first select the appropriate target layer, such as Nodes. The target layer selection will enable an appropriate set of tools.

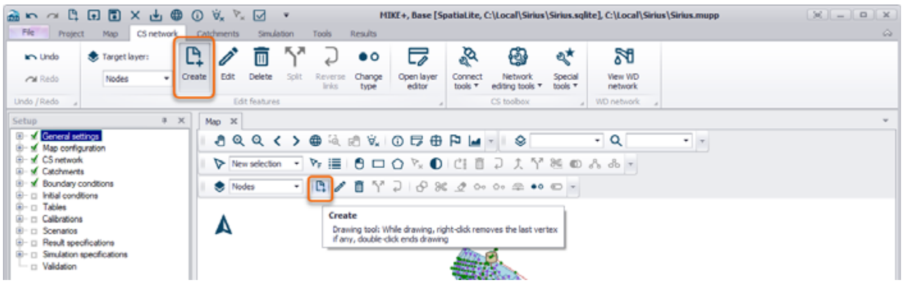


Figure 8.5 Use the Create tool to digitise nodes (manholes) for the collection systems network

Click on the Create tool from the network editing ribbon or floating toolbar, as shown in Figure 8.5. The icon will become active (it will appear pressed down) and you can point and click within the Map window to graphically add manhole locations. When finished, click the Create tool again (it will pop back up) to deactivate the tool.

When adding nodes, you can change the type of node you want to create. For sewer networks, nodes can be manholes, basins, outlets or soakaways (Figure 8.5).

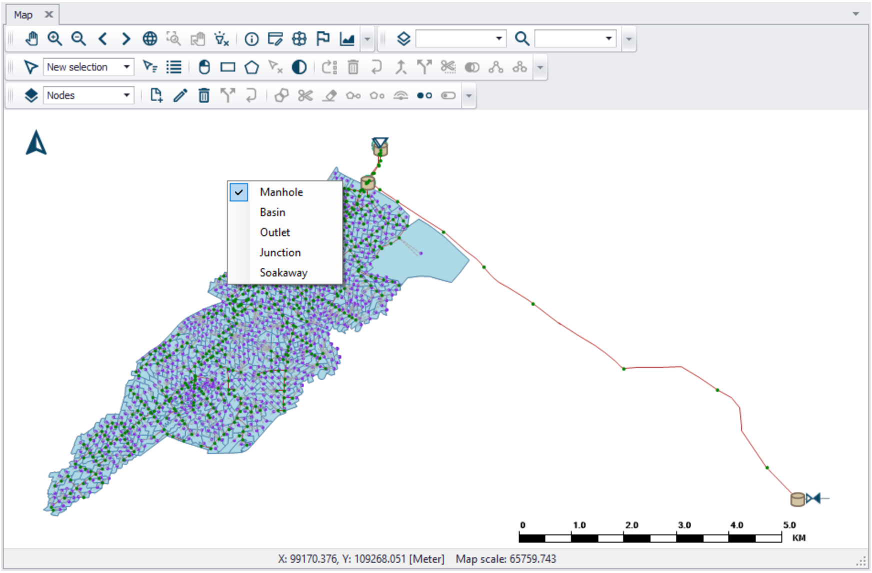


Figure 8.6 Right-click while in the node create mode will prompt you to select what type of node you want to create

To digitise links, select the Pipes and Canals layer from the Target Layer drop down list and click within the Map view to graphically add links using single mouse clicks to define the link and a double mouse click to complete a link. Continue digitising links while the tool is active. Tip: use the "Esc" button on the keyboard to keep the tool active but start digitising in a different area of the model. Note that the cursor changes to a circle when snapping onto existing nodes. If no node exists at the completion of a pipe digitisation, a new node will be created. To finish, click on the Create tool again to deactivate it.

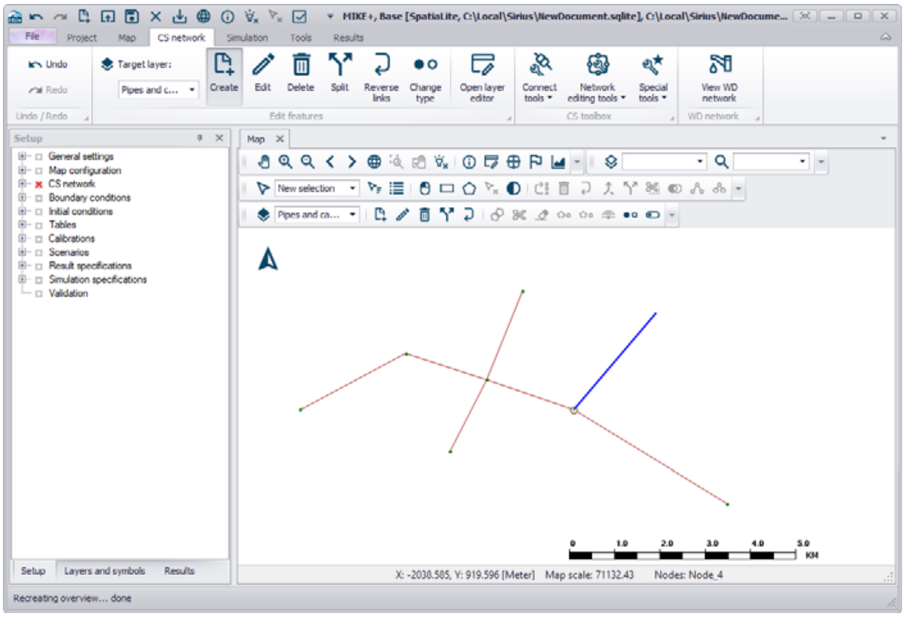


Figure 8.7 Use the Create tool for interactively laying out the network pipes in the CS mode

Catchments are defined as polygons. Digitise the polygon by clicking around the catchment extent to close a shape. Double-click to complete the catchment. To add an adjacent catchment without gaps or overlaps, activate the append polygon tool and then start the new catchment within the existing catchment, digitise the outer boundary (there is no need to digitise along the shared boundary) and then double click back in the existing catchment to complete. A new catchment will be created with the shared face between catchments exactly in line with each other, as shown in the Figure 8.8.

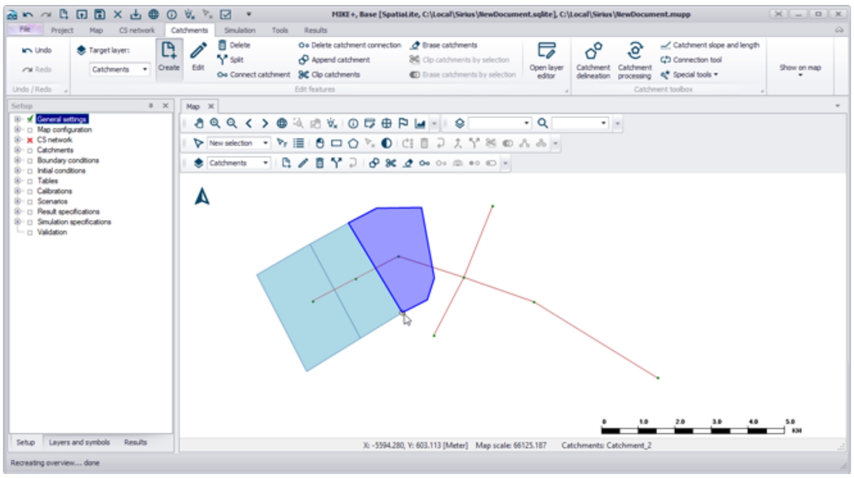


Figure 8.8 Graphically appended catchments

To graphically digitise water distribution network components, activate the water distribution mode by selecting Project | Model | Water Distribution from the main ribbon. The process of graphically adding pipes, junctions or any other network elements is the same as described above for the collection system described above.

8.4 Using the Editors

Once components of a model have been graphically digitised or imported into the model, there is often a need to edit the attributes of an element. This can be done by manually typing attribute data into the Editors or by utilising MIKE+ selection tools (Refer to Chapter 2.9.3 Map Menu *(p. 61)*) and table editing tools to edit the data en masse.

8.4.1 Identify the Location to Edit

A number of different methods exist to locate the attributes of the model to be edited.

MIKE+ has automatic data validation where missing or incorrect attributes are highlighted in orange. Model components where the data validation has identified issues can be summarised by ticking on "Show data errors" as shown in the figure below.

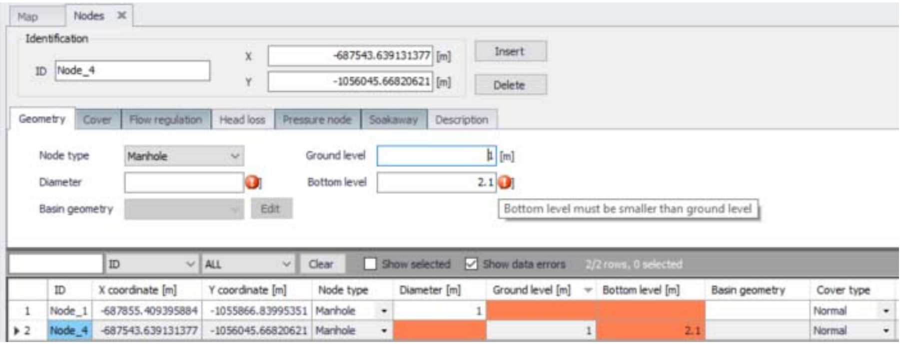


Figure 8.9 Show data errors

As the Map view is synchronised with the tables, locations can be selected in the Map view and the row corresponding to the highlighted element will be highlighted in blue in the table. Conversely, rows selected in the table will be highlighted in the Map view to visualise the locations.

Selected rows can be visualised together by ticking on "Show selected" as shown in the figure below.

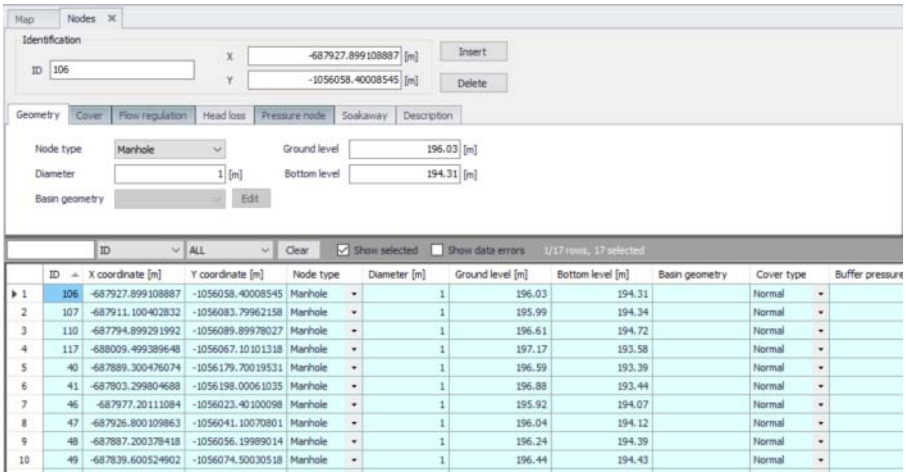
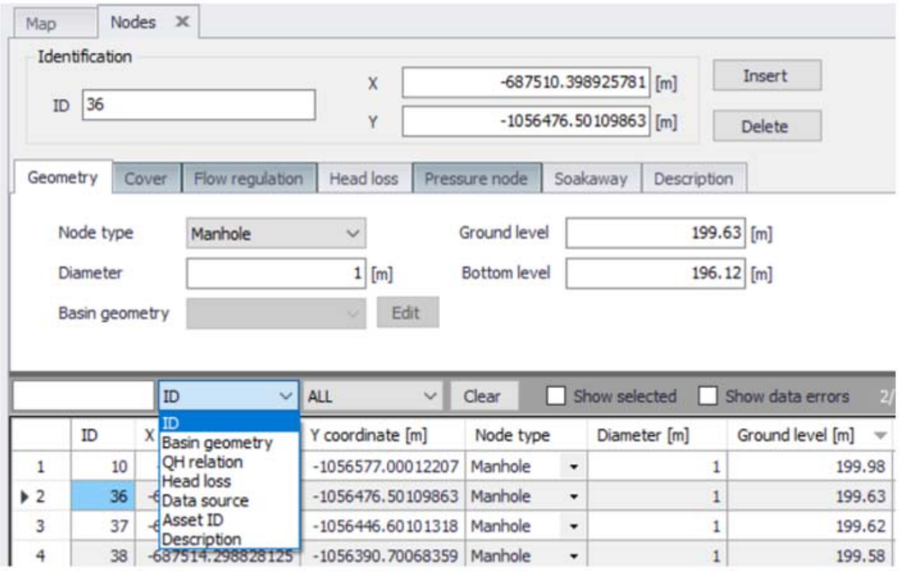


Figure 8.10 Summarise the selected rows by ticking on "Show selected"

A left mouse click on a table heading sorts the data (ascending to descending and vice versa). So outlying values can easily be identified, or a particular value can be found more efficiently.

Filters exist to sort by a table heading, a type of model component (e.g. manhole, basin, outlet or soakaway) or filter by typing part or all of the ID.



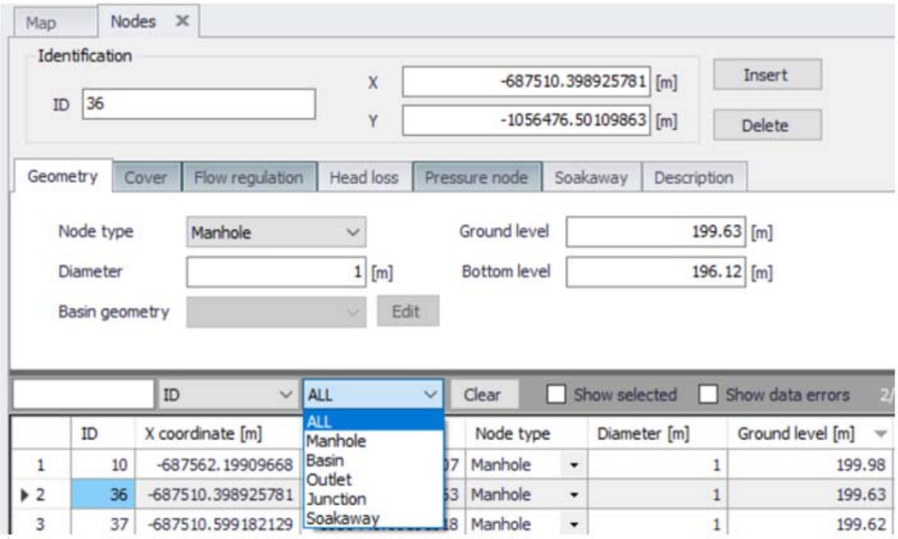
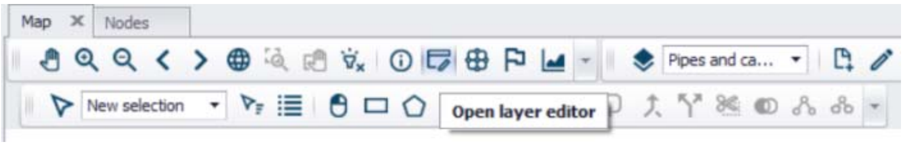


Figure 8.11 Filtering by header or model component type

A right click on a table heading opens the possibility to select by column, or select by attribute. (Refer to Chapter 2.9.3 Map Menu *(p. 61)* for more on selecting by attribute).

One way of automatically opening a table to the correct location (ID) is done through the Map view. Click on the tool, click on a location on the map (e.g. the pipe of interest) and the appropriate table will open in a new tab at the correct location showing the corresponding attributes.



8.4.2 Editing the Data in the Editor Table

Once the location to be edited is identified, there are a few ways to edit attributes.

Information can be manually typed into the form fields, where the input fields displayed correspond to where the small triangle appears on the row ID. Alternatively, values can be manually input into the table at the bottom of the screen.

The most useful method to edit data en masse is to use the Field calculator which is available by right clicking on a table header. An expression editor is then available where simple or complex expressions can be written to edit selected rows. If no rows are selected, the expression will be applied to the entire column.

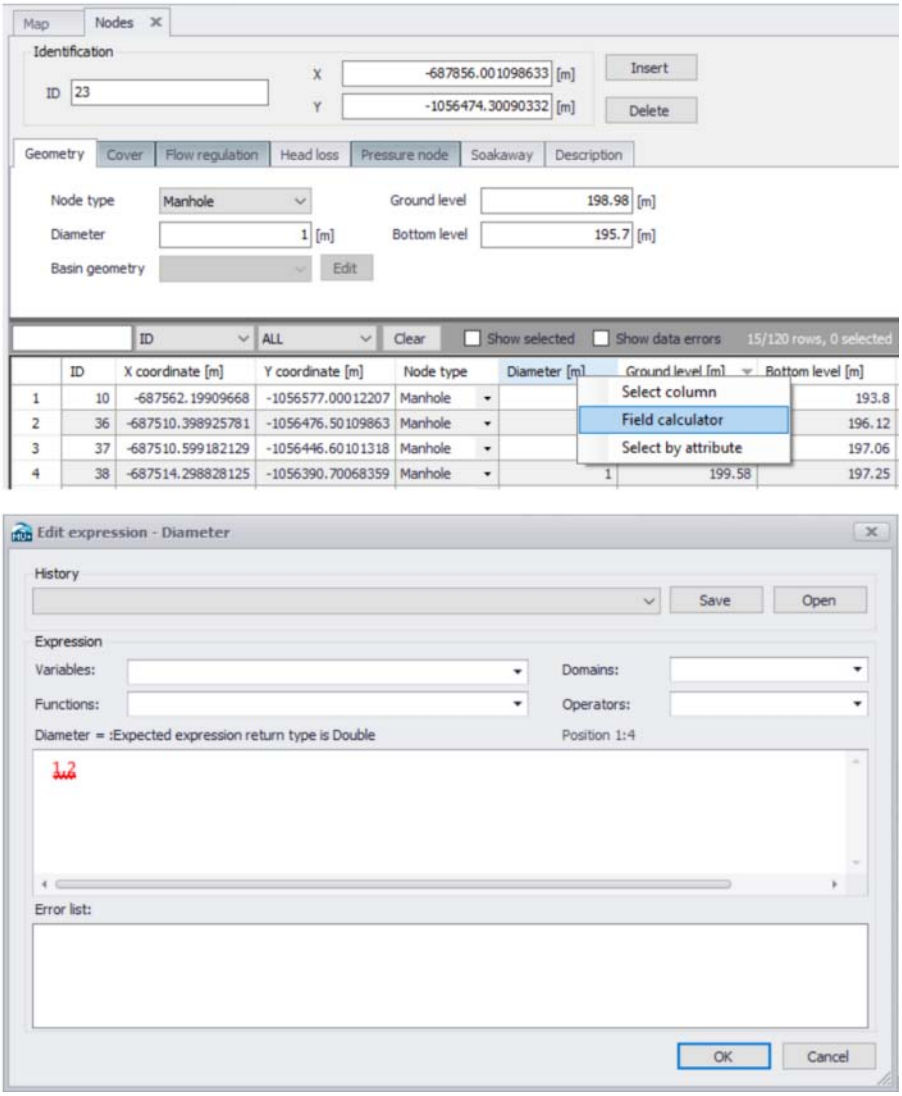


Figure 8.12 Editing data using the Field calculator

Another method of editing database values is to edit values within the Property and Result Explorer. Activate the Identify tool in the Map view, and click on the item of interest (e.g. node). In the table that appears as part of the identify tool, values can be directly edited and will be automatically synchronised with the map and database tables.

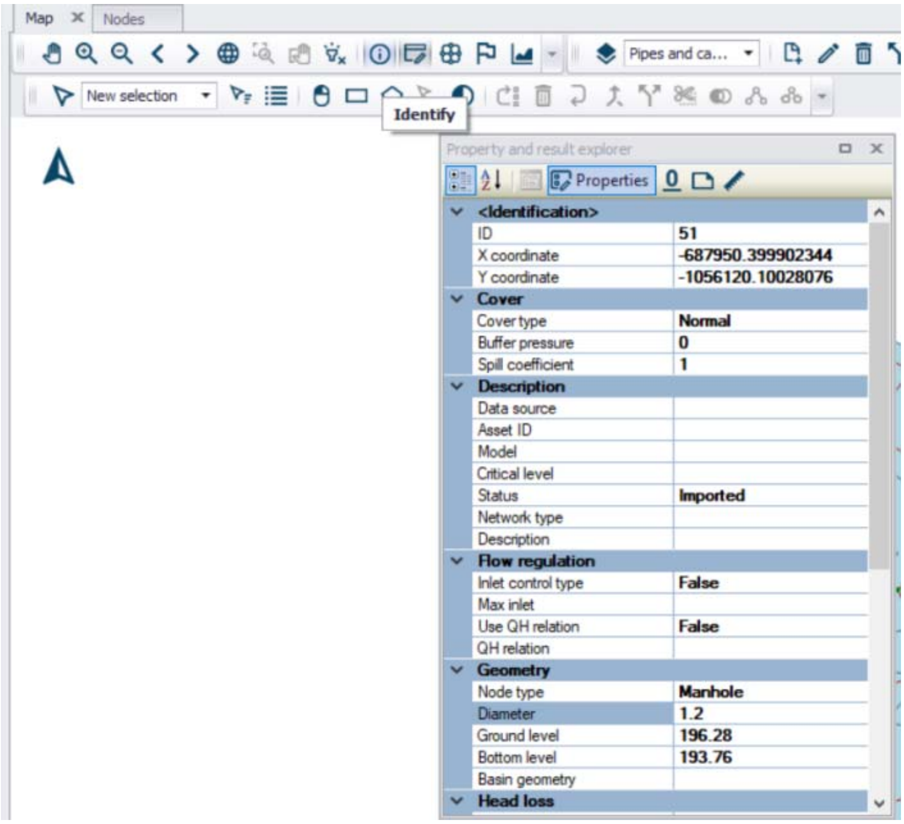


Figure 8.13 The database can be edited within the Identify tool

Note: Applying the Field calculator with a PostgreSQL database from a distant server may be significantly slower than when working with a local database. This is an expected behaviour, and happens when the updated field triggers additional updates in other editors (e.g. when editing IDs, then all connected items must also get updated connection IDs), hence requiring additional communication time with the distant server.

9 Catchments and Catchment Tools

9.1 MIKE+ Catchments

MIKE+ catchments are geographical polygon features representing hydrological urban catchments or wastewater drainage areas. As such, MIKE+ catchments may be used for hydrological modelling or as wastewater sources for MIKE+ Collection System models. Furthermore, MIKE+ catchments may be used in various analyses, independently of any computational model.

MIKE+ catchments are initially independent of any network. It is only after definition of catchment connections to a model network that the catchments become a source of loads for a network model.

9.2 Management of MIKE+ Catchments

MIKE+ catchments can be managed both graphically on the Map and through the Catchments Editor. The two modes complement each other. Joint application of both modes allows for efficient and complete management of catchment data.

The graphical mode allows digitization of catchment extent by tools like 'Create, 'Edit', 'Delete', 'Split', 'Append catchment', 'Clip catchment', and 'Erase catchment'. These tools are accessed via the Catchments ribbon as well as the Map. The graphical mode comes short of specifying catchment attributes.

Figure 9.1 The Catchments ribbon

The Catchment Editor is used for:

- Editing catchment attributes. It is also possible to insert catchments through the Editor; these are given a Default quadratic shape.
- Editing connections to model networks and hydrological data for hydrological models.

Consult MIKE+ Collection System User Guide Chapter 4 "Rainfall-Runoff Modelling" for more details on the Catchments Editor.

The Catchments Editor can be accessed via Catchments| Catchments.

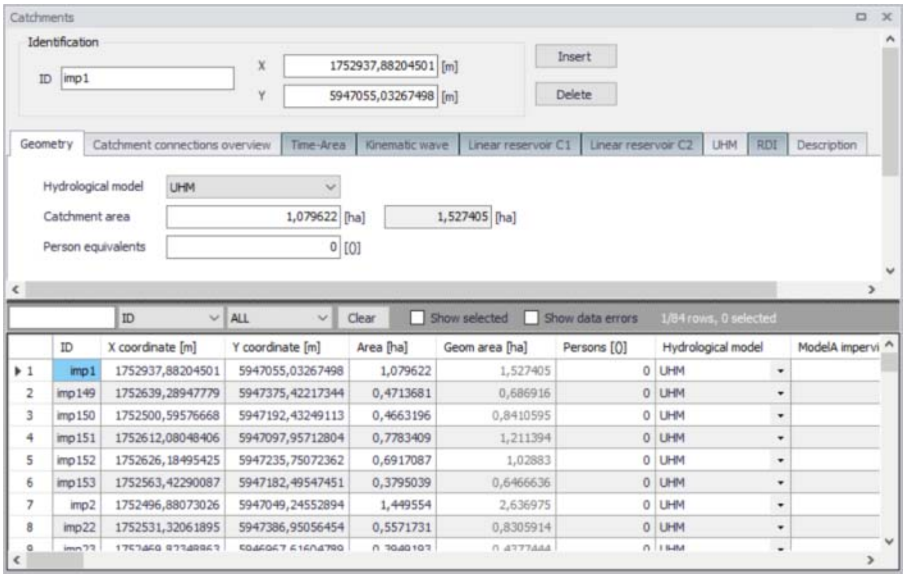


Figure 9.2 The Catchments Editor

Finally, there are also tools for automated delineation, connection and hydrological parameter estimation for stormwater catchments. These tools include fast ways to generate reasonably good input to hydrological models, which can then be modified in the Editors as needed.

9.2.1 Calculated vs. User Specified Values

The system automatically provides values for a number of geographical catchment properties (e.g. centerpoint coordinates, Catchment area). Optionally, 'Catchment area' values may be specified by the user in the Catchments Editor. If present, user-specified values replace program-computed values in model calculations.

9.2.2 Tools for Graphical Catchment Editing

The various tools for graphical catchment editing can be accessed through the Catchments ribbon (Figure 9.1 and Figure 9.3) or through the 'Layer editing tools' toolbar on the Map (Figure 9.4).

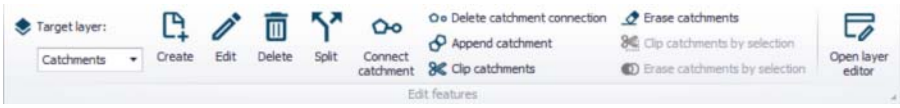


Figure 9.3 The 'Edit features' options on the Catchments ribbon

Activate ‘Catchments’ on the ‘Layer editing tools’ toolbar to graphically edit the catchment layer.

Figure 9.4 The ‘Layer editing tools’ toolbar ready for editing catchments

All tools for graphical editing are fully supported by Undo/Redo functions.

9.2.3 Create Catchment Feature

The ‘Create’ function allows for digitization of perimeters of detached or overlapping catchment polygons. Adjacent catchments can be more precisely digitized using the ’Append catchment’ tool, or by splitting an existing catchment into two parts using the ‘Split’ option.

During catchment creation, the cursor appears as crosshairs. The polygon is digitized by clicking along the wanted catchment perimeter. Right-clicking removes the last vertex added. A double-click completes the current digitization process and the system is immediately ready for the next catchment.

Snapping is available during catchment creation. This allows automatic detection of vertices or edges of nearby shapes if the cursor is within snapping tolerance of existing elements, allowing for inserted points to be created at snapping point locations.

Deactivate the tool by deselecting the tool from the ribbon or by selecting another tool from the ribbon.

Each new catchment polygon is added as a new record in the catchments attribute table. By Default, a name (i.e. identifier) is given as ’Catchment_n’, where ’n’ stands for internal catchment index. The default identifier should normally be changed into some meaningful name.

9.2.4 Edit Catchment Feature

This tool allows for editing existing catchment polygons. When activated, the tool brings a catchment into the editing mode on a mouse click inside the catchment. The catchment turns dark blue and the polygon vertices are highlighted.

Individual vertices can be clicked-on and dragged to a wanted position. The existing vertices can be deleted or new vertices can be inserted, as needed.

Left click on an edge to add a vertex. Double left-click on a vertex to delete it.

Deleting a vertex connects the two closest adjacent vertices along the catchment perimeter by a straight line. Inserting a point inserts a vertex at the closest point on the catchment perimeter. The new vertex can subsequently be dragged to the wanted position.

By clicking the mouse outside the catchment, the editing session is completed (the catchment turns back to original color), and the system is ready for editing another catchment.

Figure 9.5 The original catchment polygon (A); Edited polygon (blue) with highlighted vertices; Catchment after completed editing (C).

9.2.5 Move Catchment

An individual catchment polygon can be moved (translated) to a new position. Crosshairs appear at the polygon centroid when the catchment feature is selected for editing. Click on the crosshairs and move the feature to a desired location.

Figure 9.6 Use the crosshairs at the polygon centroid to move the feature during editing.

9.2.6 Delete Catchment

Activate the ‘Delete’ tool from the Catchments ribbon, and then select a feature on the map to delete it.

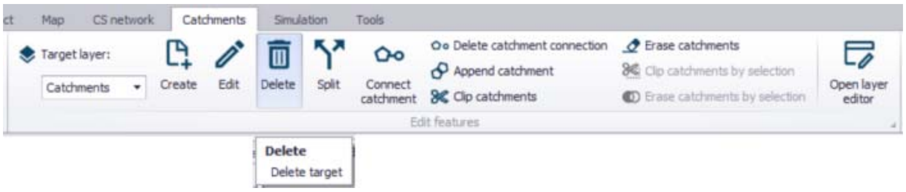


Figure 9.7 The ‘Delete’ tool on the Catchments ribbon

9.2.7 Split Catchment

An existing catchment polygon can be split into two adjacent catchment polygons. The digitization of the split line is started by a mouse click. The first click MUST be outside the polygon to be split. The line is drawn with subsequent clicks until the catchment perimeter is crossed. A double-click outside the polygon (typically on the opposite side) ends the splitting process.

The catchment connection (if any) for the original catchment is kept for the new catchments after a split.

After a split, the system deletes the original catchment record and inserts two new catchment records. The automatically provided identifiers (i.e. _copy*n*) of the new catchments would normally be changed into some meaningful catchment names.

The imperviousness for the new catchments is copied from the original catchment, while Catchment area and Person equivalents from the original catchment are divided proportionally between the split catchments.

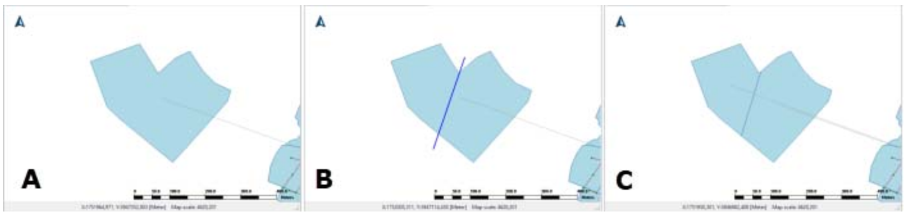


Figure 9.8 The splitting process starts with an existing catchment (A); The split line is drawn along the wanted path (B). The split action is initiated by a double click. The original catchment is split into two adjacent catchments.

9.2.8 Append Catchment

A new catchment can be appended to existing catchment(s). The result of this action is identical as with the 'Create' tool, except that part of the catchment perimeter coincides with the perimeter of the adjacent catchment(s).

The digitization of the new catchment is started by a mouse click. The first click MUST be inside an existing polygon. The catchment perimeter is drawn with subsequent clicks. A double-click inside any of the existing polygons ends the append process.

Note that the new catchment is created based on the digitized perimeter and between the first and last mouse clicks. Hence, if the existing edge (where the catchment is appended) is jagged, care should be taken to digitize the new catchment so it covers the whole area to avoid any gaps between the new catchment and the existing ones.

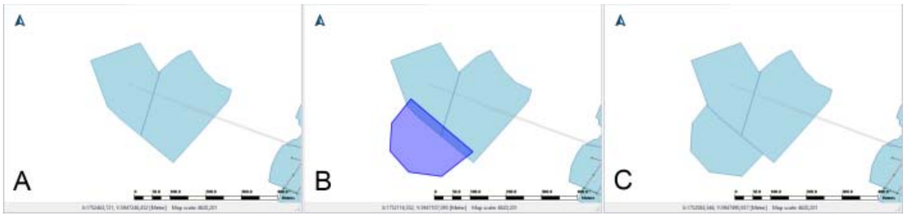


Figure 9.9 Existing catchment polygons (A); Digitization starts and ends INSIDE an existing polygon (B); A new catchment is appended (C).

The automatically provided identifier for the appended feature should be changed into some meaningful catchment name.

9.2.9 Clip Catchments

Existing catchment features may be reshaped using the 'Clip catchments' tool.

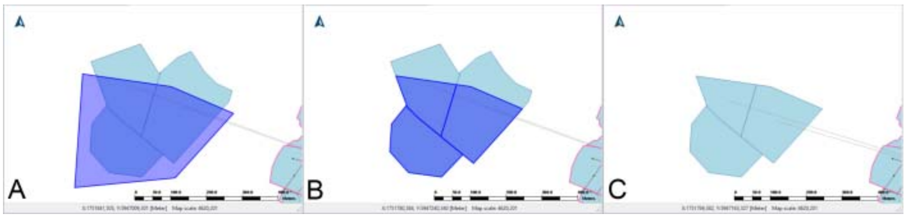


Figure 9.10 Define a clipping polygon on the map (Blue area in A); Areas under the clipping polygon intersecting existing catchment features are highlighted (B); Areas outside the clipping polygon are removed (C).

Activate the tool from the Catchments ribbon. Define a clipping polygon on the map, (left) clicking along the desired perimeter on the map and ending the polygon definition with a double-click. The area under the clipping polygon that intersects existing catchment features are retained.

9.2.10 Erase Catchments

Use the 'Erase catchments' tool to reshape catchment features. Similar to the Clip tool, define a polygon on the map--(left) clicking on the map and ending the polygon definition with a double-click. The area under the defined polygon that intersects with existing catchment features are removed.

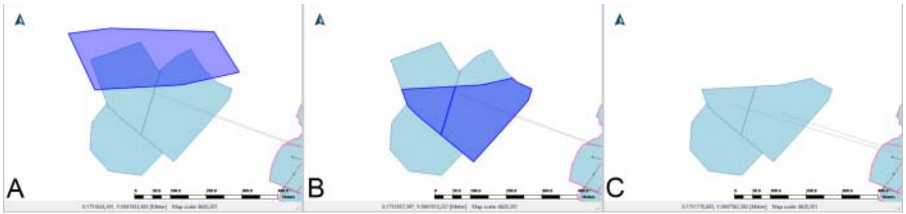


Figure 9.11 Define a polygon on the map (Blue area in A); Areas outside the polygon intersecting existing catchment features are highlighted (B); Areas inside the polygon are removed (C).

9.3 Connecting Catchments to the Drainage/Wastewater Collection Network

In order to utilize the MIKE+ catchments in network modelling, they have to be connected to the network.

The definition and management of catchment connections is supported both through the editors, and by a set of graphical catchment connection tools.

9.3.1 Catchment Connections Editor

For 'Rivers, collection system and overland flows' model type, connections between the catchments and the network are defined and stored in the 'Catchment connections' editor.

The Catchment Connections Editor contains information on all catchment connections in the model.

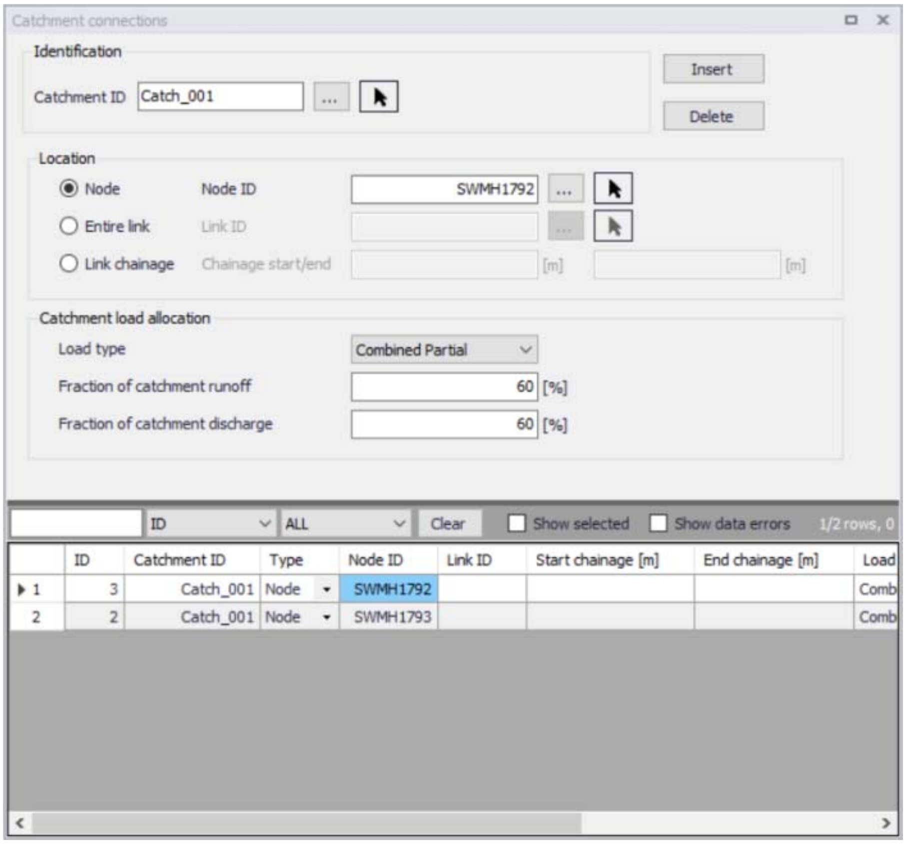


Figure 9.12 The Catchment Connections Editor

For this model type, it is possible to connect multiple catchments to the same network locations, and also to connect the same catchment to multiple network locations in order to distribute its runoff along the network.

Create catchment connections through the ‘Insert’ button. A catchment can be the source of multiple load types (i.e. stormwater and wastewater), and can be connected to multiple network elements and network types.

Table 9.1 Overview of the Catchment Connections Editor fields (Table msm_-CatchCon)

Edit field	Description	Usage	Attribute Table Field
Catchment ID	Unique catchment identifier	Yes	CatchID
Location Type radio buttons	Specifies the type of network element to which the catch-ment is connected. Options are: Node Entire link, or Link chainage	Yes	TypeNo
Node ID	Identifier of a connection node	Yes, if ‘Single Node’ connec-tion type	NodeID
Link ID	Unique identifier for the con-nected link	Yes, If Connection Type = Entire link or Link chainage	LinkID
Chainage start/end	Start and end chainages of the connected link	Yes, If Connection Type = Link chainage	StartChain-age/EndCHain-age
Load Type dropdown menu	Parameter that defines how the loads from the catchment are allocated to the pipe net-work for a connection. Options are: Standard, Wastewater Total, Stormwater Total, Com-bined Partial, Wastewater Par-tial, and Stormwater Partial. These different Load Types are further explained in the text below.	Yes	LoadTypeNo

Table 9.1 Overview of the Catchment Connections Editor fields (Table msm_-CatchCon)

Edit field	Description	Usage	Attribute Table Field
Fraction of Catchment Runoff	Fraction of the catchment stormwater runoff to allocate for the connection	Optional, If Load Type = Combined Partial or Stormwater Partial	RRFraction
Fraction of Catchment Discharge	Fraction of the catchment discharge to allocate for the connection	Optional, If Load Type = Combined Partial and Wastewater Partial	PEFraction

Qualifying a connection according to pipe network type and connection options is important. These Load Types are:

- Standard: This type of load connection applies to combined systems where all the catchment output is connected to a single location. This is the Default type, which corresponds to the MIKE URBAN Classic Single Node connection type.
- Wastewater Total: This type of load connection applies to fully separated systems, where the catchment is connected to a single location in the wastewater network.
- Stormwater Total: This type of load connection applies to fully separated systems where the catchment is connected to a single location in the stormwater network.
- Combined Partial: This type of load connection applies to combined systems where the catchment is connected to multiple locations in a combined network. This is the fully versatile connection type.
- Wastewater Partial: This type of connection applies to fully separated systems, where the catchment is connected to multiple locations in a wastewater network.
- Stormwater Partial: This type of connection applies to fully separated systems where the catchment is connected to multiple locations in a stormwater network.

The User's choice of Load Type affects the Catchment load allocation Editor fields and the internal data validation.

A facility for data validation checks that for each catchment in the Catchment Connections Editor, the sum of the fractions for Catchment Discharge (i.e. PEFraction) and Runoff Discharge (i.e.RRFraction) is close to 100 (99.9<sum<100.1).

For catchments where this sum is not found to be close to 100%, all specified connections will be reported as faulty and marked in red.

Details on defining catchment connections via the Editors are also found in the MIKE+ Collection System User Guide Chapter 4 "Rainfall-Runoff Modelling".

9.3.2 Catchment Connections Overview

The ‘Catchment Connections Overview’ tab in the ‘Catchments’ editor (Figure 9.13) shows a table summarizing the connections of the (active) catchment to the network. The data dynamically link to records in the Catchment Connections Editor.

Add a catchment connection via the ‘Add connection’ button. This will create and open a new connection in the 'Catchment connections' editor.

The summary table shows information on:

- Location. To which type of network element the catchment is connected, and the ID of the element.
- Catchment Runoff. Percentage of the Catchment Runoff from the catchment entering a location.
- Catchment Discharge. Percentage of the Catchment Discharge from the catchment entering a location.
- Action. Offers options for editing or adding connections for the active catchment.
  - Edit. Opens the Catchment Connections Editor, wherein attributes for the existing catchment connection entry can be modified.
  - Add connection. Adds a connection for the active catchment. The new connection is reflected in the overview table and the Catchment Connections Editor.

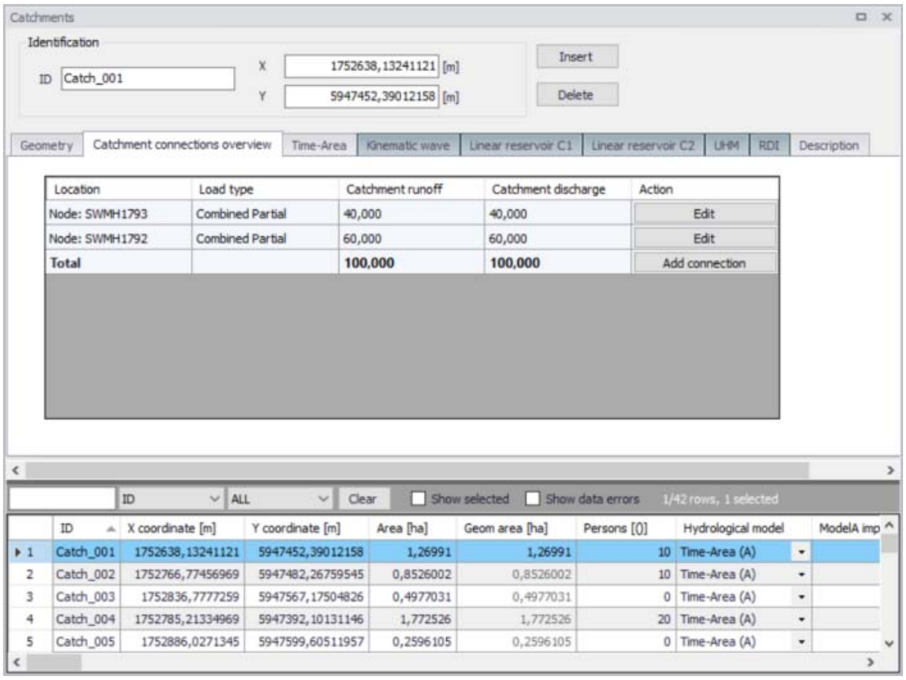


Figure 9.13 The Catchment Connections Overview Tab

9.3.3 SWMM Catchment Connections

For 'SWMM5 collection system and overland flows' model type, connections between the catchments and the network are defined in the 'Catchments' editor, in its 'SWMM catchment connections' tab.

Refer to the MIKE+ SWMM Modelling User Guide Chapter 4.1.3 "Catchment Connections", for more information.

9.4 Graphical Tools for Connecting Catchments to Networks

Figure 9.14 Catchment connection tools in the ‘Catchments’ ribbon

A set of graphical tools supports the process of connecting catchments to networks. These tools can be accessed through the Catchments ribbon. Furthermore, some of the tools are available on the map's toolbars as long as the application is in ’Catchments’ edit mode.

The tools support the option of connecting a catchment to a network element, i.e. nodes and links.

9.4.1 Catchment Dialog

This tool opens the Catchments editor.

9.4.2 Find Catchment Overlaps and Gaps

The ‘Catchment overlaps’ tool highlights catchment overlaps - that is all areas covered by 2 or more catchments.

The highlighted graphics can be removed by pressing the ’Catchment overlaps’ tool again or the ‘Clear highlighted’ tool.

This tool highlights catchment gaps - that is all areas not covered by any catchment, but completely surrounded by catchment polygons.

The highlighted graphics can be removed by pressing the ’Catchment gaps’ tool again or the ‘Clear highlighted’ tool.

The ‘Clear highlighted’ tool removes highlights on areas identified with various ‘Show on map’ tools.

9.4.3 Show Connected Catchments

This tool selects all the catchments connected to the currently active network.

Remove graphical highlights using the ‘Clear highlighted’ tool.

9.4.4 Show Disconnected Catchments

This tool selects all the catchments without connections to the currently active network.

Remove graphical highlights using the ‘Clear highlighted’ tool.

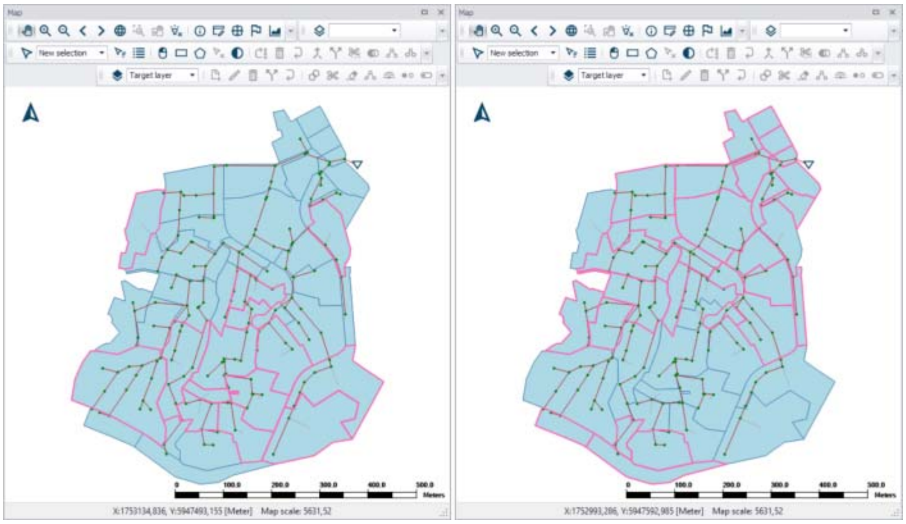


Figure 9.15 Highlighted connected catchments (left) and disconnected catchments (right)

9.4.5 Connect Catchment

This tool allows for connecting one catchment to a network element from the map.

For 'Rivers, collection system and overland flows' model type, click the main button in the ribbon, and select the desired mode:

- Replace connections: this mode will ensure that the selected catchment will have only one connection to the network. When the new connection is made, all previous connections to the selected catchment are removed.
- Add connection: this mode adds a new connection to the selected catchment, keeping all previous connections to this catchment unchanged. After adding a new connection to existing ones, ensure that catchment load types and fractions remain consistent (i.e. total 100%).

For 'SWMM5 collection system and overland flows' model type, execution of this tool will define a connection to the selected node in the 'SWMM catchment connections' tab of the 'Catchments' editor. It will always replace the previous connection, if any.

Click on the catchment to connect on the map, and finally click on the network element (i.e. node or link) to which the catchment shall be connected. The program draws the connection line upon completion.

When connecting to a link, a window to control the start and end chainage of the connection will show up. If the start and end chainages are kept equal, the catchment will connect to this single point location. If the start and end chainages differ, the catchment will connect to the specified span along the pipe, and the runoff will be distributed to all calculation points in this range of chainages.

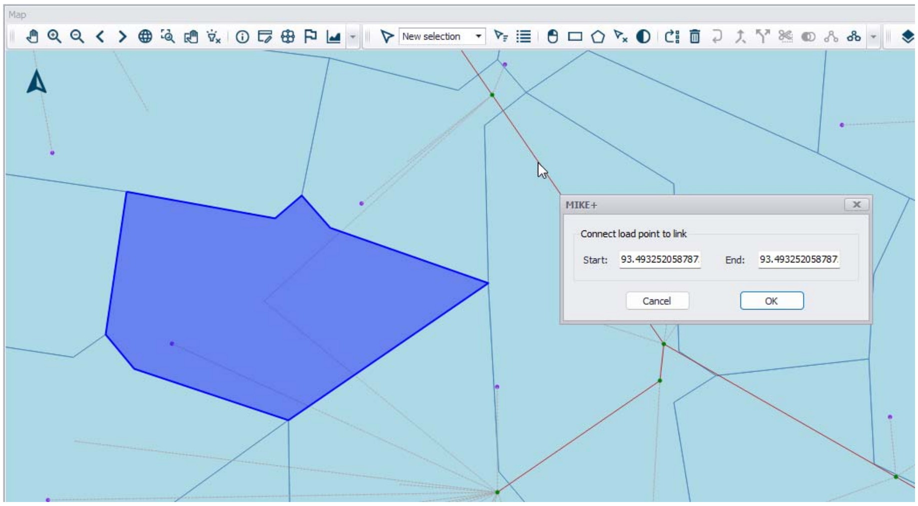


Figure 9.16 Graphical connection of a catchment to a network link

9.5 Automated Catchment Tools

The Catchment Toolbox is a collection of tools that makes delineation of catchments for stormwater networks extremely easy and fast.

Figure 9.17 The Catchment Toolbox

The toolbox includes the following automated tools:

Catchment delineation helps you delineate catchment polygons in an automated and reproducible way. The catchments can be automatically created as Thiessen polygons derived from a layer of points or lines or as polygons derived from a digital elevation model (DEM).

Catchment processing is an automated and reproducible way to calculate imperviousness, time of concentration and other hydrological parameters for your hydrological models - traditionally a very time consuming task with big risk of making errors and inconsistencies. The hydrological parameters can be calculated for MIKE 1D rainfall-runoff Time-Area models.

The catchment slope and length tool is an automated way to calculate the slope and length of a catchment based on a DEM. These parameters are used for MIKE 1D rainfall-runoff Kinematic Wave models.

The (catchment) connection tool automatically connects all selected catchments to network elements based on a number of user specified principles, e.g. to the nearest manhole. For those places where you want the catchments to be connected differently, the connections can be moved using graphical editing tools.

Three additional tools are available under the 'Special tools' list. The '**Create elevation zones from DEM tool**' is used to divide RDI catchments into a number of elevation zones to model snowmelt, based on an input DEM. The ‘**Spatial processing**‘ tool can perform GIS operations such as Merge and Join and export the result to a shapefile. The '**Snap Neighboring Catchments**' tool is used to update the geometry of catchments on the map, to ensure that neighboring catchments are correctly snapped.

9.5.1 Catchment Delineation Wizard

The catchment delineation tool helps delineate catchment polygons in an automated and reproducible way. The catchments can be automatically created as Thiessen polygons derived from a layer of points or lines or as polygons derived from a digital elevation model (DEM).

The tool guides you through the steps of the delineation process (Figure 9.18).

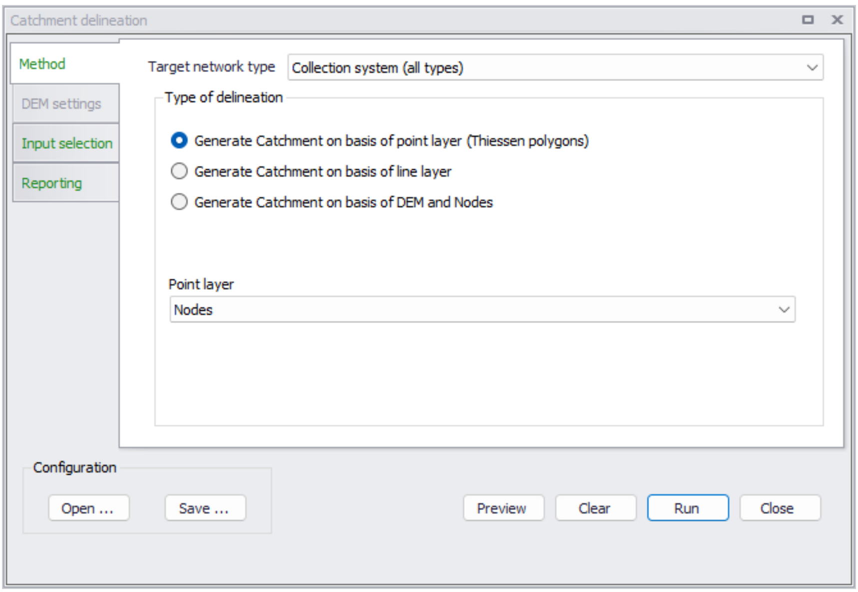


Figure 9.18 The catchment delineation tool

Method

The first step in the delineation process is to select the type of delineation.

For model setups with model type 'Rivers, collection system and overland flows', a target network type can also be selected. For collection system the 'Target network type' can act as a filter, to select which items on the network may be used to delineate catchments. Therefore, the CS network type must be appropriately set for the network items, before applying a specific target type.

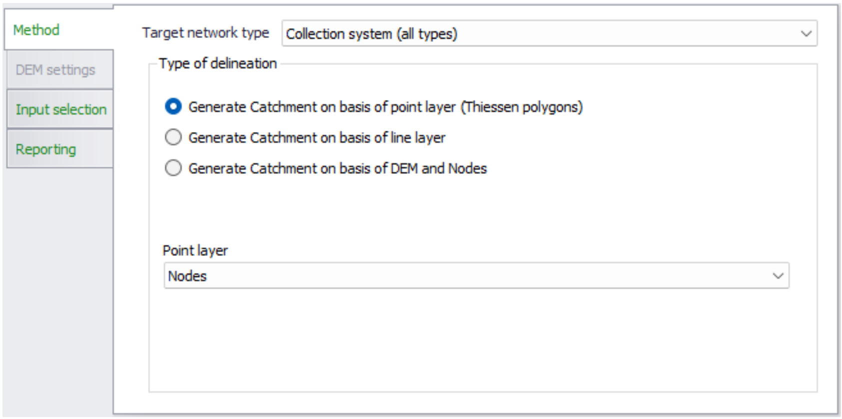


Figure 9.19 Selection of the type of delineation to use with a collection system

When delineating catchments for a collection system network, the three options available are:

- Generate catchment on basis of point layer (Thiessen polygons). Based on Voronoi partitioning, which is a mathematical way of diving spaces into a number of regions.
- Generate catchment on basis of line layer. Also based on Voronoi partitioning principles, but around line segments instead of points.
- Generate catchment on basis of DEM and nodes. The catchments will describe the actual hydrological catchments around inlet nodes, defined based on the slopes on the DEM. This option requires pre-loading a valid DEM layer in the project. A valid DEM must be in *.asc or *.dfs2 file format.

Depending on the delineation type, select the actual layer upon which the delineation shall be based. Available layers relevant to a delineation type are offered in the drop-down menu.

When the catchments are created as Thiessen polygons (or Voronoi cells) you start out by specifying a selection of points or lines to use as an input layer. Typically either all manholes or all links, or only a selection.

The tool will proportionally divide and distribute a point coverage into the polygons known as Thiessen polygons. When a line layer is used as input, the points used are the midpoints of the lines. Each polygon contains only one input feature point. Each polygon has the unique property that any location within the polygon is closer to the polygon's point than to the point of any other polygon.

The Thiessen polygons (or Voronoi cells) are constructed as follows:

- All points are triangulated into a triangulated irregular network (TIN) that meets the Delaunay criterion.
- The perpendicular bisectors for each triangle edge are generated, forming the edges of the Thiessen polygons. The location at which the bisectors intersect determine the locations of the Thiessen polygon vertices.

The outside boundary of the Thiessen polygons needs to be specified. Either as a user specified polygon (created on the fly or loaded in as a layer) or as extent of the points used plus an additional area.

Please note that the underlying Delaunay triangulation method used works best with data in a projected coordinate system.

Note: When working with point layers, outlet nodes are always excluded from the analysis (no catchment delineated around outlets).

When delineating catchments for a river network (only available for model type 'Rivers, collection system and overland flows'), the delineation is always based on the slopes from a DEM. This requires pre-loading a valid DEM layer, in *.asc or *.dfs2 file format, in the project. Three options are available to define the location of delineated catchments:

- Automatically delineate catchments at confluences. This method attempts to split the overall catchment according to the modelled river network, by creating one catchment at each upstream end of the river network and at each confluence between two rivers. At these confluences, the tool will create one catchment for each of the rivers: if the tributary river ends exactly on the main river, then the starting point of the two catchments will have the same location on the DEM, and the tool would delineate the same catchment for the two rivers. To address this issue, the starting point of the catchments is moved upstream by a minimum distance from the confluence. A small value for this minimum distance is recommended, so that the delineated catchment remains close to the confluence. After previewing the delineated catchments, if the tool fails to create different catchments for the two rivers, then this value should be increased.
- Manually locate on map the catchment's connection to the river. This method lets the user identify on the river network (clicking on the map) where catchments should be created.

- Automatically generate catchments on basis of point layer. This method uses a point layer to define the location of the start points of the catchments. When this option is selected, a point layer must be chosen, and a search radius must also be specified. The tool will delineate a catchment for each point in the source layer, and will create a catchment connection to the closest river location (or storage) from each point. If no river or storage is found in the search radius, no catchment connection will be created.

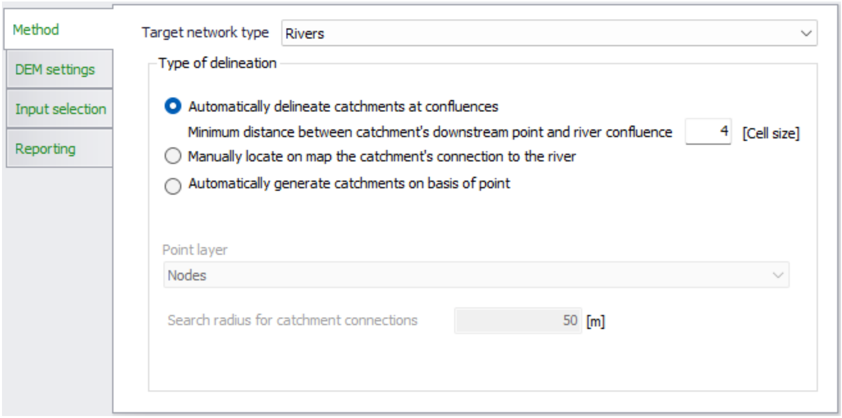


Figure 9.20 Selection of the type of delineation to use with a river network

DEM settings

When the delineation is based on a DEM, the DEM source layer must be selected from the 'DEM settings' tab. The DEM must be added to the map prior to the delineation, and the drop-down list will show all valid DEM files.

The additional controls below are available.

Spatial extent

Two options are available:

- Use whole DEM: the entire extent of the DEM will be included in the analysis.
- Use DEM only inside digitized area: this option allows defining a reduced extent covered by the DEM, from the map. Click the 'Digitize' button to draw on the map a rectangle defining the reduced extent. Once the area is finally defined, right-click on the map to stop the digitization. To edit the reduced area afterwards, use the 'Edit' button and then click and drag the symbols on the map to resize the rectangular area or move it on the map. Right-click on the map to stop editing.

Resampling

Resampling requires specification of a resampling factor. If a resampling factor of 2 is used, then the minimum elevation of a 2x2 set of pixels is assigned to one new pixel with the same area as the 2x2 set. A resampling factor of 3 assigns the minimum value of a 3x3 pixel set to one new pixel with the same area as the 3x3 set, and so on.

Number of pixels

This group shows for information the number of pixels in the source DEM, as well as the final number of pixels actually used in the analysis. They are both provided as number along the X axis and along the Y axis respectively. The final number of pixels is reduced by the spatial extent of the analysis when using the option 'Use DEM only inside digitized area', and is divided by the value of the resampling factor.

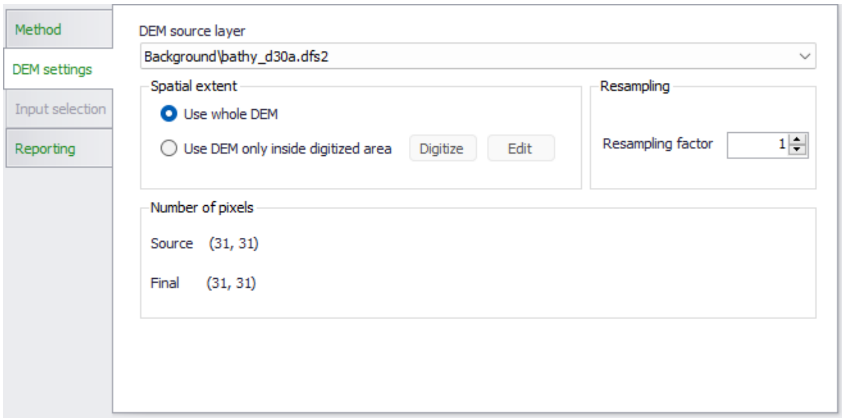


Figure 9.21 Controlling the DEM settings for the catchment delineation

Input selection

Next step is selection of the extent for the delineation, see Figure 9.22.

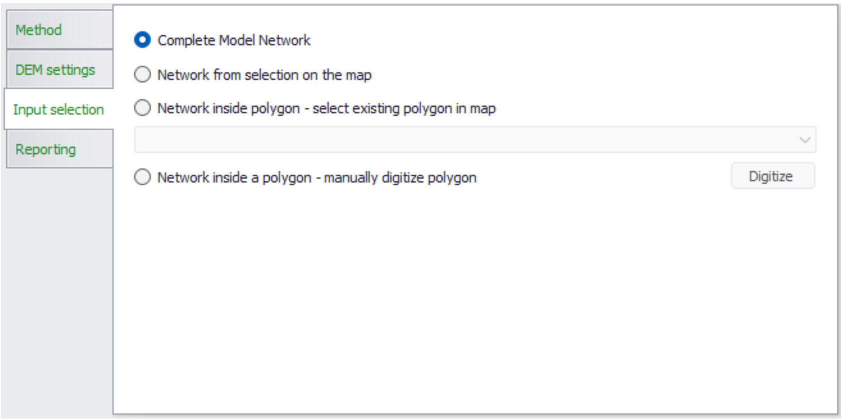


Figure 9.22 Selection of the part of the network to be processed

There are four options:

- Complete model network. Uses a default boundary defined by a rectangle covering the complete network (including a 30-m buffer zone). There is no additional setting for this option. A catchment will be delineated around each network element (node, pipe or river).
- Network from selection on the map. Creates a catchment around each network element currently selected on the map. When delineating catchments for river networks with the method 'Automatically delineate catchments at confluences', catchments are only created at confluences between two selected rivers.
- Network inside polygon - select existing polygon on map. Select an existing polygon from a polygon layer added to the project. If this method is selected, the specific layer to be used is chosen from the dropdown list and the specific feature selected on the map. A catchment will be delineated around each network element within this polygon. After selecting the polygon layer from the list, the message "Please select a feature" will appear: select polygon(s) from the layer to be included by clicking on the map.
- Network inside polygon - manually digitize polygon. Manually digitize the polygon on the map. If this method is selected, use the ‘Digitize’ button to draw the boundary directly on the map, ending the digitization with a double-click. A catchment will be delineated around each network element within this polygon.

Note that for DEM-based delineation, the delineation is performed according to the defined input selection, but catchments covering the total extent of the input DEM will still be generated.

Click on the ‘Run’ button to delineate the catchments according to the specified configuration.

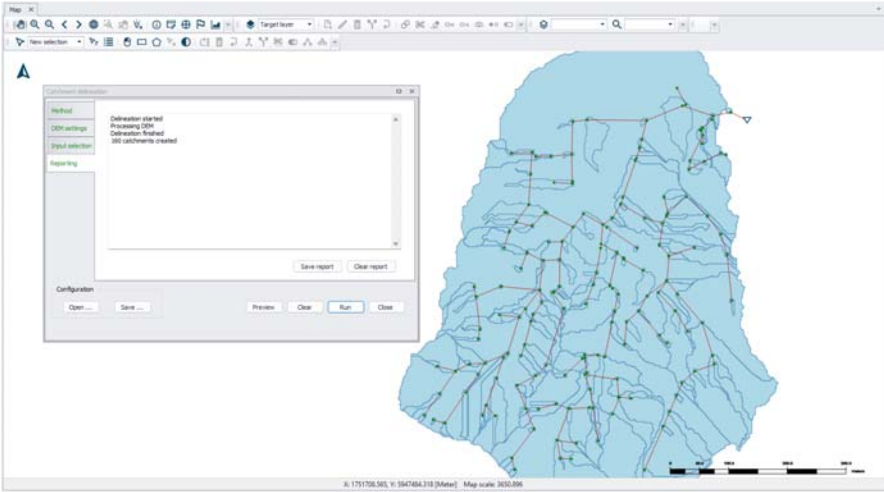


Figure 9.23 Click on the ‘Run’ button to perform catchment delineation

Reporting

This section displays a summary of results from running the delineation tool.

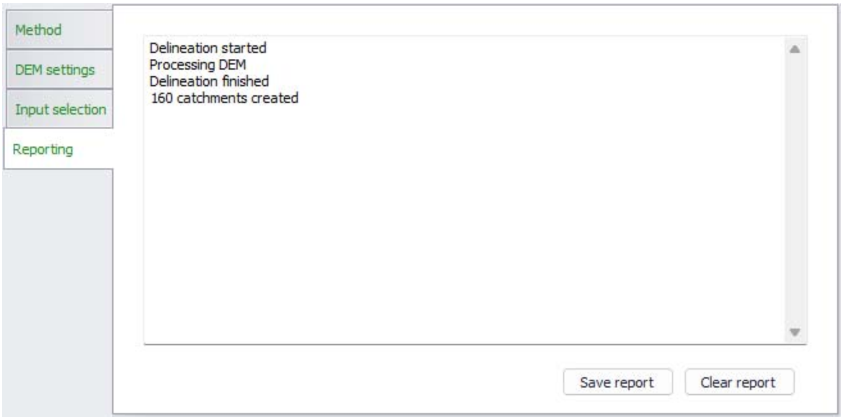


Figure 9.24 Report from the catchment delineation tool

If the reported information is relevant for future use, it can be saved to a text file using the 'Save report' button.

Buttons

The following buttons are available at the bottom of the tool.

‘Open...’ button

Loads a previously-saved catchment delineation *.XML configuration file.

‘Save...’ button

Saves the current catchment delineation configuration into an *.XML file.

‘Preview’ button

Option for viewing preliminary results of a catchment delineation configuration. . If not satisfying, the settings can be modified and the delineation can be previewed again. The previewed results are not saved to the model database until actually executing the tool.

‘Run’ button

Executes the catchment delineation tool following the defined configuration.

‘Clear’ button

Resets the map view by removing highlights or preliminary delineation lines related to result previewing or extent digitization.

9.5.2 Catchment Processing Wizard

The catchment processing tool is an automated and reproducible way to calculate:

- imperviousness, time of concentration and other hydrological parameters for Time-Area runoff models
- distribution of land uses, for Time-Area, Kinematic Wave, Linear Reservoir or New UK / Wallingford runoff models
- imperviousness and catchment width for SWMM hydrological models.

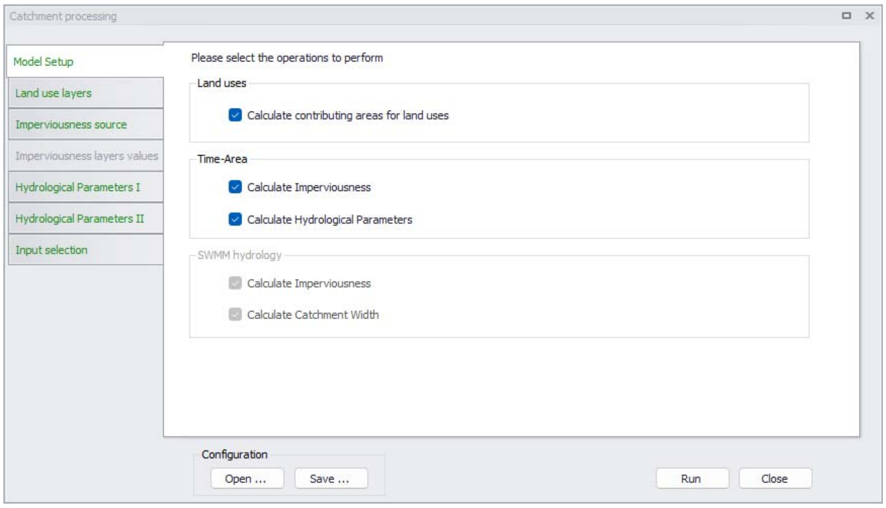


Figure 9.25 The start-up dialog of the catchment processing wizard

Model setup

The first step in the catchment processing is selection of which parameters to calculate. For 'Rivers, collection system and overland flows' projects, the tool can calculate the contributing areas of land uses on each catchment, when catchments are set to work with land uses definition. For 'Time-Area' models specifically, the tool can additionally be used for calculation of hydrological parameters.

For SWMM model setups, the tool can be used for performing the following operations for SWMM hydrological models:

- Calculate Imperviousness. Use the tool to derive or set imperviousness values for catchment .
- Calculate Catchment Width. Catchment width for SWMM catchments is computed as Area / MaxLength, where MaxLength is:
  - If the catchment is connected to a node, this is the distance from the connected node to the farthest point in the catchment.
  - If the catchment is connected to another catchment, the length is the maximum length across the catchment between two opposite points along the border.

Calculating only catchment width using the tool requires no further steps after selection of the option from the 'Model Setup' tab.

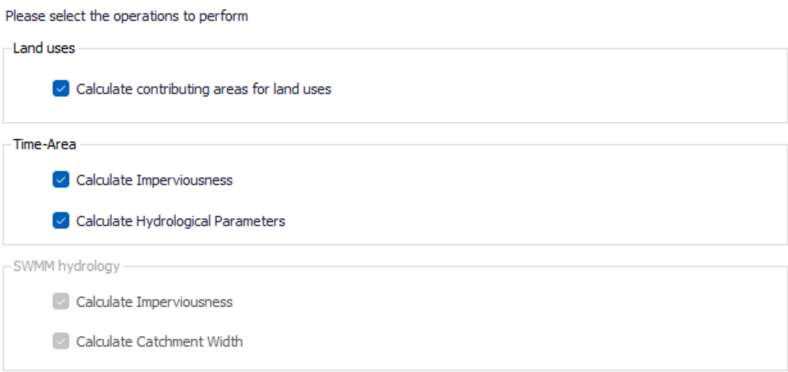


Figure 9.26 Selection of parameters to calculate

Land use layers

The 'Land use layers ' tab is active when the tool is used for calculating the contributing areas for land uses. This function can be applied to catchments with hydrological model Time-Area, Kinematic Wave or Linear Reservoir when their option 'Use land use distribution' is ticked, or New UK / Walling-ford. This option is meant to compare the extents of land use polygons and catchment polygons, and:

- Define the list of land uses covering each catchment
- Estimate the percentage of coverage of each land use.

Land use polygons must be defined in a polygon feature layer loaded on the map prior to using the tool. Two formats are supported for the definition of the various land use layers:

- Either each land use is defined with its own feature layer (e.g. shape file), i.e. all polygons in a given layer will be assumed to describe the same land use
- Or multiple land use types can be defined in the same feature layer, with an attribute of the layer being used to identify the land use type for each polygon.

In this tab, all valid existing land uses defined in the project are listed, and the table is used to select which land uses to process and associate them with polygon layers added to the map. Land uses defined for the 'New UK / Wallingford' model and using the 'New UK' runoff volume type are excluded from the table, because this type of land use is to be defined for the remainder of the catchment not covered by any land use with the 'Fixed coefficient' runoff volume type.

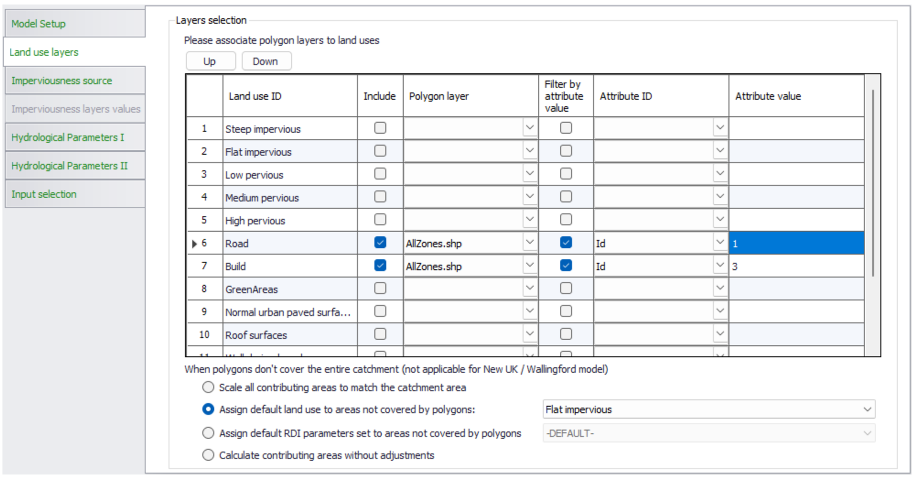


Figure 9.27 Polygon layer selection for calculation of land use distribution

The following settings are controlled in the table:

- Land use ID: this column lists all valid land uses defined in the project, in the 'Land uses' editor, and all RDI parameter sets, defined in the 'Parameters RDI' editor.
- Include: this check box controls which land uses (including RDI parameters) are being processed by the tool. Land uses which are not included will be ignored by the tool and won't be assigned to catchments.

- Polygon layer: this column is used to select the polygon layer to associate with land uses included in the operation. The drop-down list shows all available polygon layers (shape files or feature classes from geodatabases).
- Filter by attribute value: when this is unticked, all polygons within the selected polygon layer will be associated with the corresponding land use. When it is ticked, an attribute from the polygon layer will be used as a filter, and only the polygon for which the attribute matches a user-defined value will be associated with the corresponding land use.
- Attribute ID: When 'Filter by attribute value' is ticked, this list shows all attributes available in the selected polygon layer, and is used to select the attribute which will be used for the filter.
- Attribute value: When 'Filter by attribute value' is ticked, this is the value that the selected attribute must have for polygons to be associated with the corresponding land use.

Please note that the order of the layers is important. If some of the polygons overlap, only the uppermost layer (i.e. higher on the list) is considered in the overlapping area.

Unless the catchments are fully covered by the polygon layers, there may be parts of the catchment which are not covered by any of the land uses. The radio buttons at the bottom are used to control how these remaining areas are processed. The available options are:

- Scale all contributing areas to match the catchment area: for each catchment individually, the contributing percentage of each land use is multiplied by a correction factor such that the sum of all land use contributing percentages matches 100% for the catchment. This method is usually to be used when the fraction of the catchment not covered by any polygon layer is small (e.g. when the catchment is supposed to be fully covered by polygons but some uncovered areas exist due to inaccuracies in the polygons digitization).
- Assign default land use to areas not covered by polygons: for this option, select the default land use to apply, from the drop-down list. This option allows to provide polygons only for some land uses (e.g. impervious areas like buildings and roads), and associate the rest to a default land use (e.g. natural areas).
- Assign default RDI parameters set to areas not covered by polygons: this option is to be used for catchments combining either 'Time-Area + RDI', 'Kinematic wave + RDI', 'Linear reservoir C1 + RDI' or 'Linear reservoir C2 + RDI'. With this method, all parts of the catchment not covered by any polygon will be associated with the RDI hydrological model and with the selected parameters set.

- Calculate contributing areas without adjustments: this method keeps the computed land use contributing areas unchanged. If some parts of the catchment are not covered by any polygon, it will usually result in having these areas not contributing to runoff on the catchment.

These radio buttons are only applicable for rainfall-runoff models Time-Area, Kinematic Wave and Linear Reservoir. For New UK / Wallingford models, the part of the catchment not covered by any polygon will be applied the single land use with the 'New UK' runoff volume type, which is selected in the catchment.

Imperviousness source

The 'Imperviousness source' tab is active when the tool is used for deriving or setting catchment imperviousness values.

The second step in the calculation of imperviousness is specification of the source of imperviousness values during processing.

Imperviousness for MIKE+ catchments can be calculated as a constant value or as a weighted average of imperviousness of multiple polygon layers. The layers should be pre-loaded in the project to be selectable in the wizard.

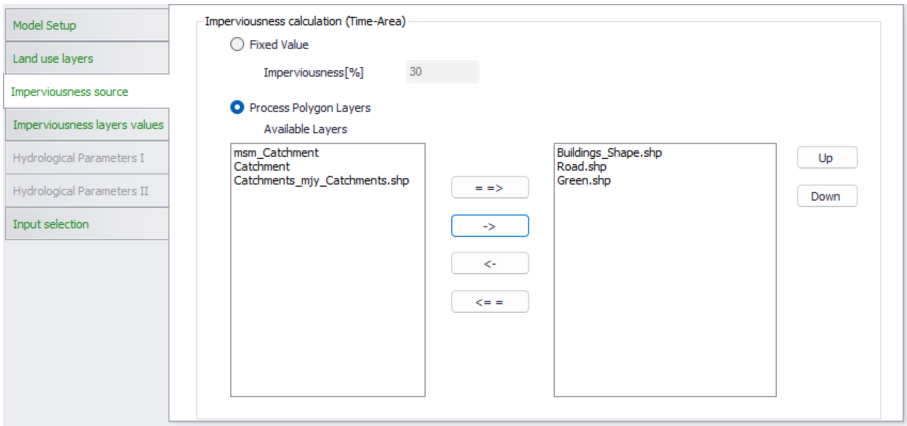


Figure 9.28 Polygon layer selection for calculation of imperviousness

Imperviousness layers values

The next step in the calculation of imperviousness is specification of the imperviousness for each selected layer. Please note that the order of the layers is important. If some of the polygons are overlapping, the value from the uppermost overlapping layer (i.e. higher on the list) is prioritized.

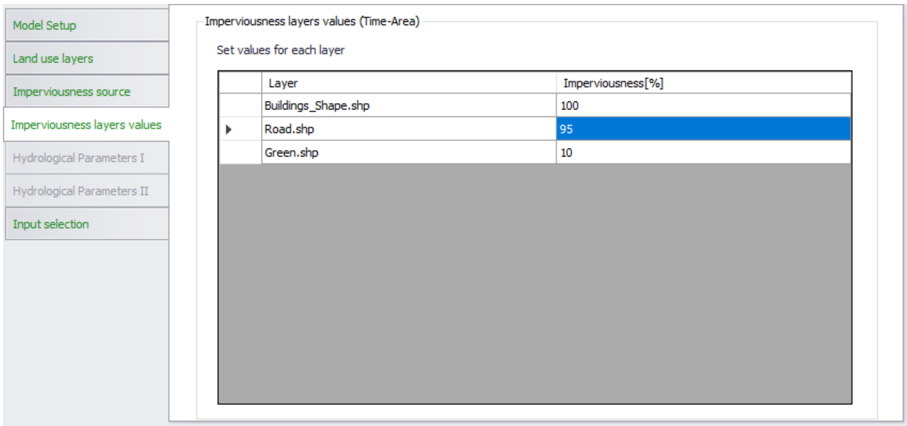


Figure 9.29 Specification of parameters for calculation of imperviousness

During the computation, areas of the catchments not covered by any layer will be assumed to be highly pervious (imperviousness = 0%).

Hydrological parameters

Several hydrological parameters for MIKE+ Time-Area runoff models can be calculated. The configuration is split in two tabs, with the principles explained in the dialogs.

Note that catchments must be connected to the pipe network in order to compute these parameters, as the connection is required for the computation of time of concentration.

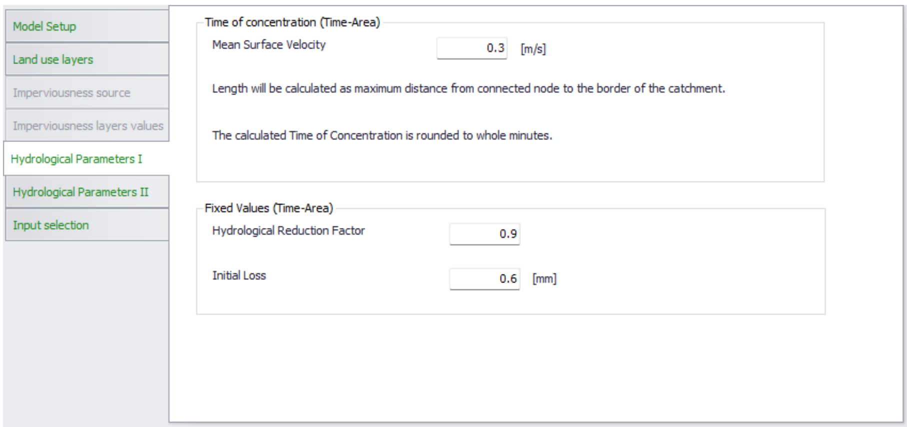


Figure 9.30 Specification of the first set of hydrological parameters for Time-Area runoff models

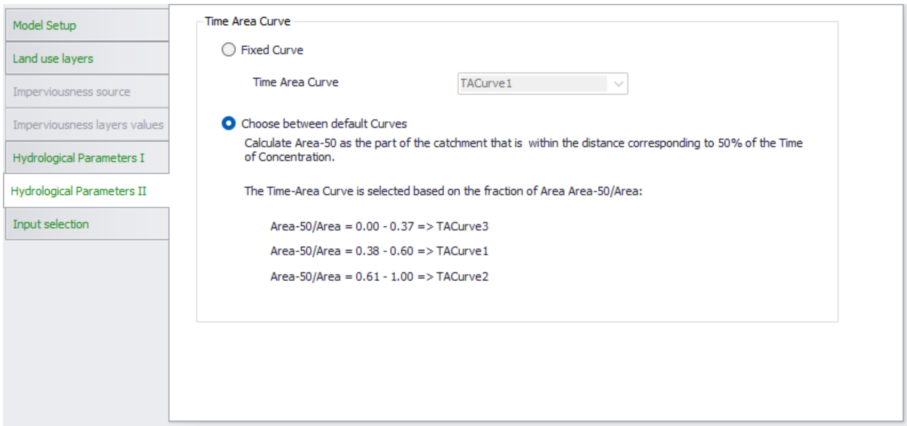


Figure 9.31 Specification of the second set of hydrological parameters for Time-Area runoff models

Input selection

From this tab, two options are available to control which catchments will be processed by the tool:

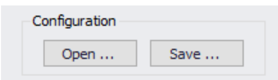
- All catchments: land use distribution will be computed for all catchments in the project
- Catchments from selection on the map: only the catchments selected when the tool is executed will be processed.

Running the tool

The final step is to execute the tool using the ‘Run’ button at the bottom of the window.

Configuration

A section for saving or loading a Catchment Processing configuration *.xml file. Use the **Save** button to save the current processing configuration into an *.xml file. The **Open** button loads a previously-saved *.xml configuration file.



9.5.3 Catchment Slope and Length Tool

As part of the hydrological modelling, the catchment slope and/or length may need to be estimated for some rainfall-runoff models. The tool performs automatic estimation of hydrological parameters for each catchment in a consistent, documented and reproducible way.

Based on delineated catchments, a DEM, and lines for the flow path inside a catchment, the slope and length can be automatically estimated for each catchment using the Catchment Slope and Length tool. Note that the length is not computed for the catchments with 'SWMM hydrology' model.

The tool is initiated from the Catchment Toolbox.

To calculate the slope and length, the typical flow path within the catchment must be digitized (i.e. the slope lines). These can be drawn from the load point or towards the load point but a consistent methodology should be used in a project. A multiple number of slope lines can be defined for each catchment. The slope lines must be a line feature in MIKE+ either from a background layer or an existing (unused) layer in the database.

The slope and length are calculated as an average slope and length of the lines that are completely contained within the catchment.

An example of slope lines are shown in Figure 9.32.

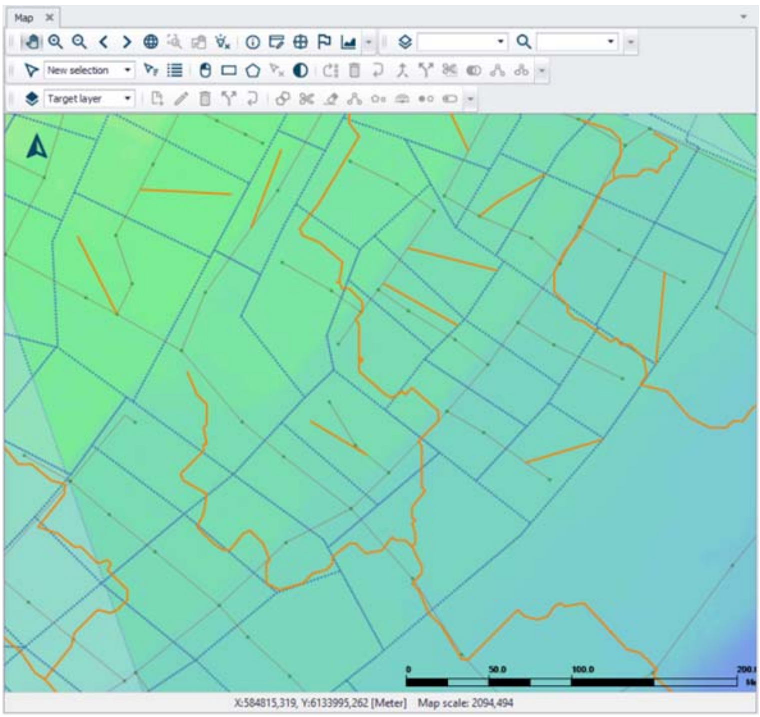


Figure 9.32 Example of slope lines (solid yellow lines) following surface flow paths overlaying catchments (broken blue lines) and the DEM (coloured surface), which are used in catchment length and slope derivation.

When the tool is opened, the slope line layer must be specified together with the direction the lines were digitized to obtain the correct sign for the slope. A minimum slope is also specified that will be assigned to all catchments with smaller slopes. The DEM and slope line layer must be added as a background layers in the MIKE+ project to be available in the tool, see Figure 9.33.

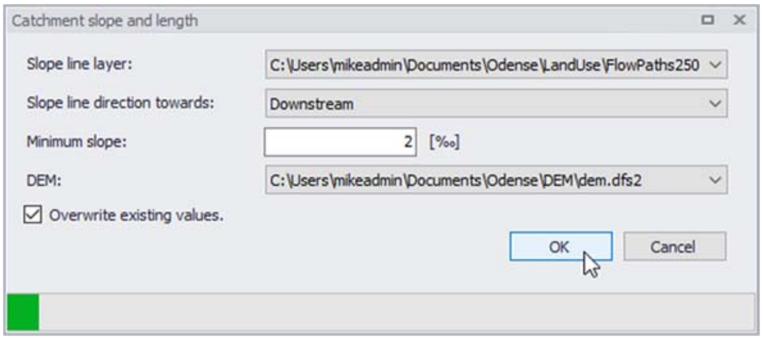


Figure 9.33 Catchment Slope and Length Tool

Click the OK button to run the tool.

The tool will calculate the length and/or slope for all selected catchments. If no catchments are selected, the length and slope will be calculated for all catchments containing slope lines. The results are saved under the 'Kinematic Wave' tab in the Catchments Editor.

Activate the 'Overwrite Existing Values' option if the computed catchment slopes and lengths shall replace existing values, if any.

9.5.4 Create elevation zones from DEM tool

The purpose of the 'Create elevation zones from DEM' tool is to automatically divide catchments into a number of elevation zones to model snowmelt in RDI hydrological models, and also to estimate the area of each of the elevation zone based on an input DEM.

The tool analyses the DEM elevations within the extent of the catchment, as defined by its polygon on the map. Based on the minimum and maximum elevations found in the catchment, and a user-specified maximum zone's height, the tool defines a number of elevation zones for each processed catchment. Then the tool also estimates the area associated with each elevation zone by searching the areas on the DEM with elevations in the range associated with the zone.

The list of elevation zones and their areas are then filled in the set of RDI parameters associated with the catchment. When modelling snowmelt with elevation zones, and especially when using this tool, it is therefore recommended that each catchment is associated with its own set of RDI parameters in order for each catchment to use a local definition of elevation zones.

Note: This tool is primarily intended for catchments using the RDI (solo) hydrological model. The tool will work the same way for other hydrological model types combined with RDI (e.g. 'Time-Area + RDI'), however for such catchments the tool cannot estimate where on the catchment the RDI model applies and where the elevation zones should be computed. Therefore, manual adjustments / corrections may be needed after running the tool for such catchments.

DEM layer

This is the source DEM used to create the elevation zones within the catchment polygons. If valid DEMs are already loaded on the map, they can be selected from the drop-down list. The '…' button can be used to load a new DEM file.

Maximum zones' height

Each zone is associated with a range of elevations (height). This maximum height controls the number of elevation zones which will be obtained.

Each zone is given the same height, which is lower or equal to the specified 'Maximum zone's height'. The number of zones is therefore estimated so that the minimum number of zones, fulfilling this condition, is created,.

For example for a catchment with elevations varying within 100m and a maximum zone's height of 30m, four zones will be created, each with a height of 25m.

In the list of elevation zones in the 'Parameters RDI' editor, the reported elevation of the zone is the average elevation of the zone.

Create a new individual set of RDI parameters for each processed catchment

When this option is unselected, the tool creates the elevation zones in the sets of RDI parameters which are currently used by the RDI catchment (as selected in the 'RDI' tab of the 'Catchments' editor). This solution can however create conflicts if multiple catchments share the same set of RDI parameters.

When this option is selected, the tool instead creates a new set of RDI parameters for each processed catchment. The new set of RDI parameters is initially created as a copy of the original parameters set used by the catchment, in which the elevation zones are created.

Input selection

The tool can process either:

- All catchments: elevation zones will be computed for each catchment in the model setup.

- Catchments from selection on the map: elevation zones will be computed only for the currently selected catchments.
- Single catchment: elevation zones will be computed only for the selected catchment.

Regardless of the selected option, only the catchments using the RDI hydrological models will be processed.

Configuration

- Open: loads the settings from a file created during a previous use of the tool.
- Save: saves the current settings in the tool to a file for later re-use.

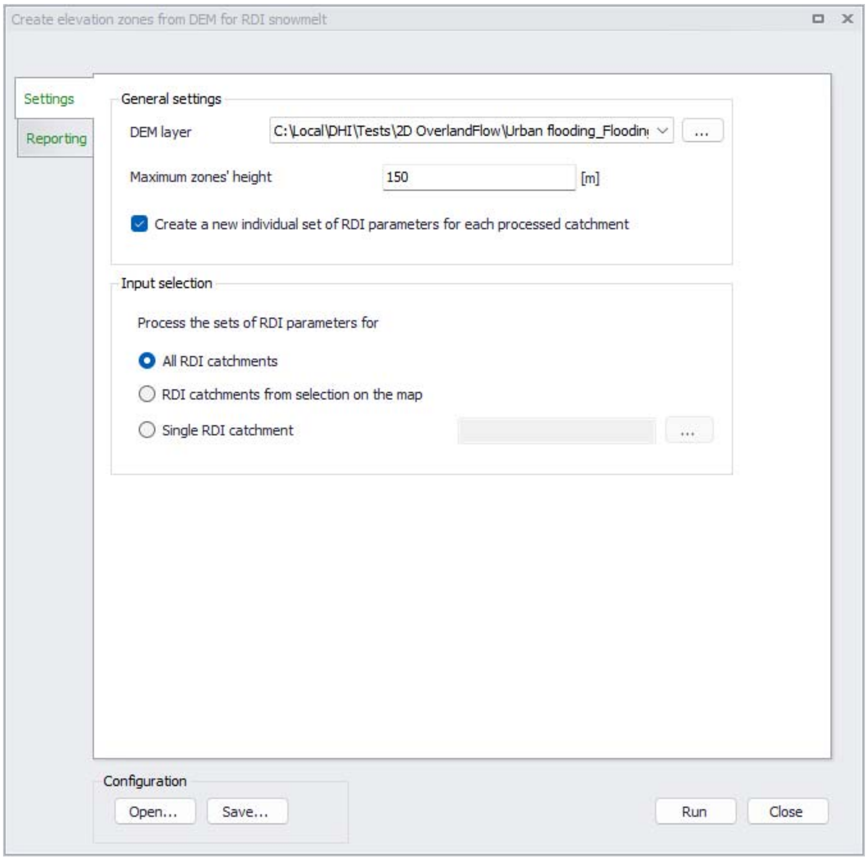


Figure 9.34 The Create elevation zones from DEM tool's dialog

After running the tool, the 'Reporting' tab will report the list of processed catchments, as well as errors and warnings if any.

Note: The use of snowmelt and elevation zones in the processed RDI parameters is not automatically activated when running the tool. If these options are not enabled in the processed RDI parameters before running the tool, they must be activated manually afterwards for the elevation zones to take effect.

9.5.5 Spatial Processing Tools

The Catchment Toolbox also offers specialised tools i.e. for Spatial Processing. These are accessed via the 'Special tools' menu on the Catchments ribbon.

Spatial analysis tools allow the user to perform several GIS-processing operations on various polygon and line layers available in the project. These layers are either model element layers, or shapefile layers loaded into the project.

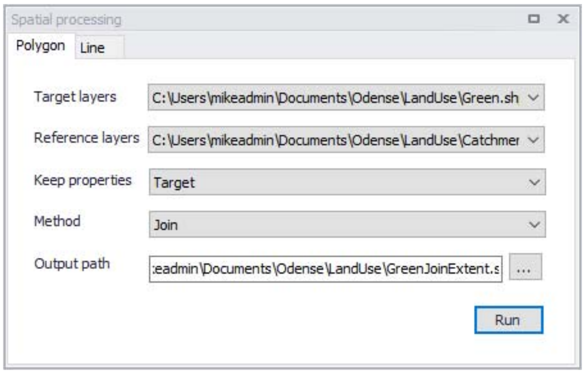


Figure 9.35 The Spatial Processing dialog

Polygon

Operations such as Merge and Clip may be performed between various polygon layers. The analysis results are saved in a new shapefile and automatically loaded into the project as a new layer. It may also be imported as a model element feature.

Table 9.2 Parameters for Polygon Spatial Processing

Parameter	Description
Target layers	Polygon feature to be modified (i.e. to which the operation will be done)
Reference layers	The second polygon layer used in modify the target layer
Keep properties	Information on the feature attributes used for resulting layer

Table 9.2 Parameters for Polygon Spatial Processing

Parameter	Description
Method	Dropdown menu for selecting spatial operation to use: **Clip** = Extracts from target the areas intersecting the reference. Features in target not intersecting the reference are also kept. **Erase** = Removes from target the areas intersecting the reference. **Merge** = Features in both layers combined, where intersecting features are fused in new features. **Join** = Extracts from target the areas intersecting the reference.
Output path	Use the ellipsis button "..." to specify the path and file name for the resulting feature layer from the operation
Run button	Button for executing the spatial processing

Line

Operations such as Merge and Clip may be performed between various polygon layers. The analysis results are saved in a new shapefile and automatically loaded into the project as a new layer. It may also be imported as a model element feature.

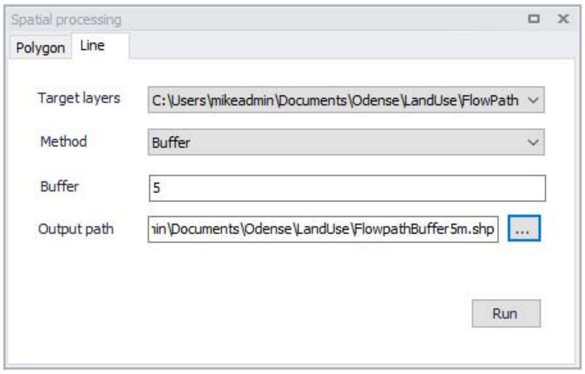


Figure 9.36 Spatial processing for lines

Table 9.3 Parameters for Line Spatial Processing

Parameter	Description	Usage
Target layers	Line layer to be modified (i.e. to which the operation will be done)	Yes
Method	Dropdown menu for selecting spatial operation to use: **Buffer** = Creates buffer polygons around target layer features according to a buffer distance. **To Polygon**= Converts the line features to polygons. Note that polyline features **must be closed** with overlapping start- and end-vertices to be converted to pol-ygons.	Yes
Buffer	Distance around the line features that will be buffered.	If Method = Buffer
Output path	Use the ellipsis button "..." to specify the path and file name for the resulting fea-ture layer from the operation	Yes
Run button	Button for executing the spatial process-ing	-

9.5.6 Snap Neighboring Catchments Tool

The 'Snap Neighboring Catchments' tool is accessed via the 'Special tools' menu on the Catchments rib-bon.

This tool can be used to update the geometry of catchments on the map, to ensure that neighboring catchments are correctly snapped. It is especially useful to prepare catchments before attempting to merge them with the 'Net-work simplification' tool, because distant catchments cannot be merged even if the distance between them is negligeable.

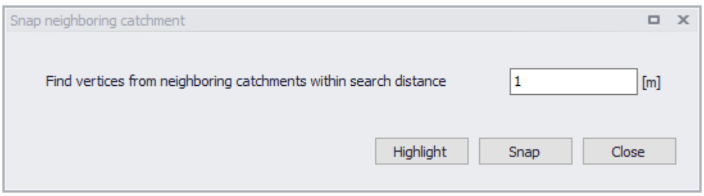


Figure 9.37 The Snap Neighboring Catchments Tool

The tool will edit catchments when their boundaries are within the specified distance from another catchment.

Clicking 'Highlight' will show on the map the catchments which will be updated. Clicking 'Snap' will execute the tool and update the catchments.

10 Connection Tool

The Connection Tool is a generic wizard which can be used to connect catchments, load points, demand allocations or measurement stations to the network.

The tool automatically connects all selected catchments to nodes, pipes or rivers, based on a number of principles, e.g. to the nearest node. For places where it is desired to connect catchments differently, the connections can be moved easily using graphical editing tools.

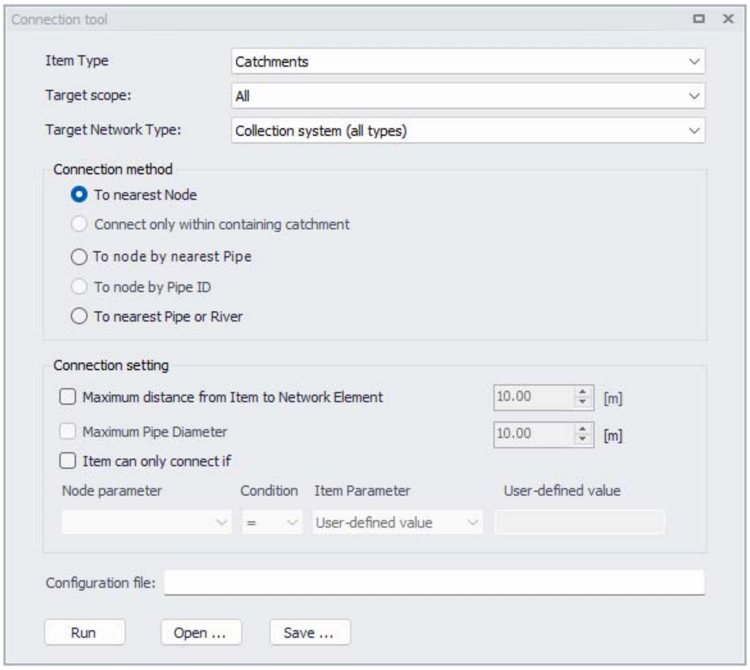


Figure 10.1 The Connection Tool dialog

Use of the Connection Tool requires first defining the 'Item Type' to be connected, which depends on the type of project. It can either be catchments, load points, demand allocations or measurement stations.

For SWMM collection system projects, the tool can only connect catchments to nodes, although SWMM catchments may be connected to other catchments. If catchments shall be connected to other catchments, this should be done manually through the 'Catchments' editor.

Secondly, define the 'Target scope' i.e. the group of items included in the connection process:

- **All**: all the appropriate model items.

- **Current Selection**: only currently selected model items.

Then, define the 'Target Network Type' (only used with some item types) to only consider target network element of certain Network Types in the processing. The target network type acts as a filter to select which items on the network may be connected. Note that catchments will not connect to nodes which have undefined Network Type when applying to a specific target network type (other than ALL). Therefore, the CS network type must be appropriately set for the network items before applying a specific target network type.

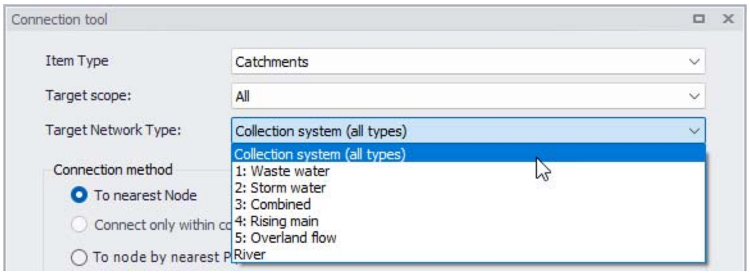


Figure 10.2 Selection of Network Type for target to which the items should be connected

10.1 Connection Method

Define the Connection Method to use:

- **To Nearest Node**: connect to node nearest the item location (or polygon's centroid, for a catchment).
- **Connect only within containing catchment**: connect load points only to nodes in the same catchment as the load points.
- **To Node by Nearest Pipe**: connect to the nearest end node of the nearest pipe to an item location/centroid.
- **To node by Pipe ID**: connect the demand allocation point to the closest node from a pipe with the same ID as the demand allocation (only available for Water Distribution networks). This option is relevant when demand allocation points have previously been named according to their connected pipe
- **To nearest Pipe or River**: connect to nearest pipe or river from the item location or centroid.

The available methods are different depending on the selected 'Item type'.

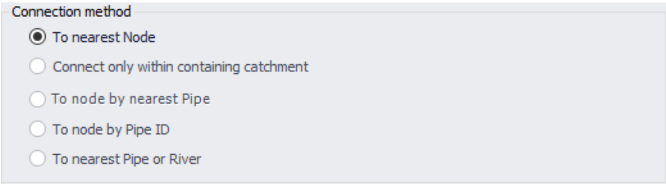


Figure 10.3 Selection of Connection Method

10.2 Connection Settings

Connection Settings are optional and may be used to include extra criteria for connecting to the network.

- **Maximum Distance from Item to Network Element**: maximum search distance to find nearest node element from the item location/centroid.
- **Maximum Pipe Diameter**: maximum pipe diameter to involve in the search for nearest pipe element from the item location/centroid. E.g. for Method = To Node by Nearest Pipe. If the nearest pipe's diameter is too large, the program will skip this pipe and will keep searching for another pipe. This is e.g. suitable in order not to allocate demand points to main or transmission pipes.
- **Item Can Only Connect If**: option for setting a user-defined condition in search for nearest elements.
  - **Node / Pipe Parameter**: node or pipe parameter to use for additional filter criterion.
  - **Condition**: mathematical condition for filter criterion.
  - **Item Parameter**: item parameter to use for building the conditional statement for the filter criterion.
  - **User-defined Value**: used when the criterion is defined with a user-defined value instead of an item parameter.

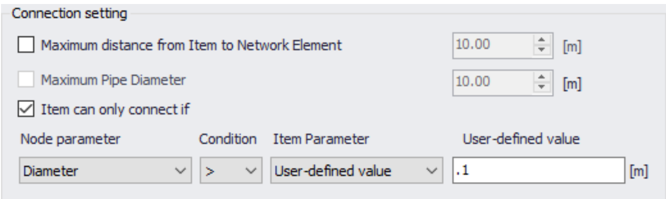


Figure 10.4 The Connection Settings section in the Connection Tool

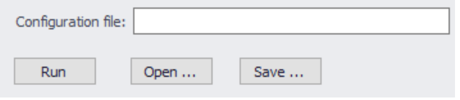
10.3 Running the Tool

Finally, click on the **Run** button to run the Connection Tool.

10.4 Configuration

The Configuration File input box shows the file name for a saved or loaded/opened connection configuration *.XML file. The path and file name for a new configuration may also be specified in the input box. Note that only specifying a file name will save the file in the user's Documents folder by Default.

Use the Save button to save the current processing configuration into an *.XML file. The Open button loads a previously-saved processing *.XML configuration file.



11 Load Allocation Through Geocoding

MIKE+ supports the allocation of geographically determined load points to the nodes of a collection system model. The allocated loads may then constitute a component of the overall network load definition for a collection system hydraulic model.

The load points are geographical point features, typically representing water, pollution and/or sediment sources (households, factories, etc.). Each point can be assigned a source type. E.g. Domestic Wastewater, Industrial Wastewater, etc.

To be used in MIKE+, each point must be attributed by the load size (volume or mass per unit time). The present MIKE+ release supports the water point loads only, i.e. the water quality properties for the point loads (if available) cannot be utilized. A typical origin for useful sets of point loads would be water consumption records, normally available in GIS applications managed by urban water utilities. A specific source of point loads is the demands allocation table found in each MIKE+ Water Distribution project. I.e., the water demand allocations can be directly imported into the Collection System project, to be used as the Collection system network load points.

11.1 Management of Point Loads

The management of Point Loads for collection system hydraulic models consists of the following distinct steps:

- Create/Edit/Import of load points
- Connect loads to the nearest node
- Aggregation of load allocations.

Generally, the load points are managed either through a customized import from a MIKE+ Water Distribution model or through a user-defined import from various external sources using the import and export tool in MIKE+ (Tools | Import and export). Load points can be managed both graphically and through the "Load Points" editor. The two modes complement each other.

The graphical editing mode ("Feature Edit") allows load points to be edited with the functionality "Insert", "Move" and Delete". A special graphical tool is available for the load allocations (connections) to the network nodes.

The "Load Points" editor is primarily used for reviewing and editing the load points attributes, deleting obsolete load points and for access to the vital related tools for Geometry and aggregation.

11.2 The Load Points Editor

The "Load Points" editor can be accessed through the menu (Project | Setup view | Boundary Conditions | Load Points), as shown in Figure 11.1 and Figure 11.2. Also, the "Load Points" table includes a tool for direct access to the "load point connection”.

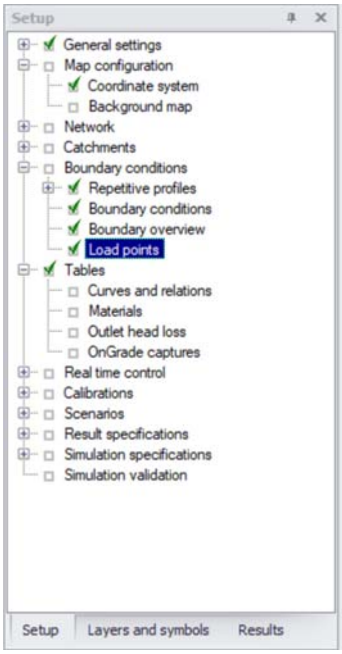


Figure 11.1 "Load points" database

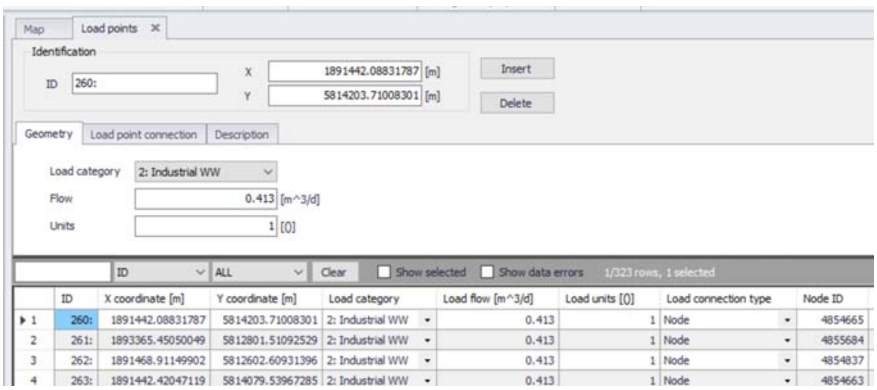


Figure 11.2 "Load Points" editor

Table 11.1 Overview of the editable Load Points attributes

Edit field	Description	Usage	Attribute Table Field
ID	Reference to the load point identifier in the original source.	Generated automatically but can also be adjusted manually	ReferenceName
X co-ordinate	X co-ordinate of the load point. Provided automatically by the system	Calculated	Identification
Y co-ordinate	Y co-ordinate of the load point. Provided automatically by the system	Calculated	Identification
Load Category	Classifies the load point into one of the available load categories. Relevant when several categories of point loads are to be distinguished in the project.	Optional	Geometry
Flow	Defines the load amount as volume/ times (flowrate) i.e. m3/day	Mandatory	Geometry
Load Point Connection	Used to either view, edit and connect a specific load point to the correct node/ load/	Mandatory	Load point connection
Description	Describes the site and information relating to the load point etc.	Optional	Description
Data Source	Used for identification of the date source (file and path name)	Optional	Description

Edit field	Description	Usage	Attribute Table Field
Asset ID	Reference to the load point identifier in the original source.	Optional	Description
Owner	Identifies the "owner" (e.g. water consumer) of the load point.	Optional	Description
Location	Identifies the site (e.g. mailing address) associated with the load point.	Optional	Description
Date	Specifies the date of entry or load validity period.	Optional	Description
Picture	The user can add a picture of the load location.	Optional	Description

11.3 Importing Load Points

Load point data may be imported from a variety of sources. Use the Import/Export tool functionality in MIKE+. See Chapter 6 Import and Export *(p. 147)* for more details on importing data into MIKE+.

11.3.1 Importing Load Points from MIKE+ Water Distribution

In MIKE+, projects which include both water distribution and collection systems, the point loads for the collection system would typically be imported from the water distribution part of the project. Namely, the water demands are specified for the water distribution network as demand points, equivalent to the collection system's load points. I.e. water demands are turned into the collection system loads.

MIKE+ supports this transfer through the Import and Export tool (Tools | Import and export), by choosing the Source data as “Demand allocation” and target as the “Load points”. If the demands point data exist in Water Distribution, the tool copies the point features and relevant attributes from the Water Distribution layer "Water Demand - Demand Allocation"

11.3.2 Importing Load Points from External Sources

In this case, load points typically originate as a layer in a GIS application or as tabulated data in database tables, spreadsheets or ASCII files. They usually represent water consumption records or wastewater and/or pollution emissions according to discharge permissions.

If the load points data is part of GIS, then the geographical information is intrinsically present. When stored in any other tabular format, the table must include the columns with X and Y co-ordinates. In both cases, for correct overlay of the network data and the load points, it is essential that the coordinate systems of the current MIKE+ project and the external GIS are identical.

11.4 Graphical Editing of Load Points

The tools for graphical editing of load points can be accessed through the ribbon menu in the CS network tab. Select the target layer to be “Load points” from the drop-down menu and the relevant tools will become active (create, edit, delete, connect load point). Alternatively, the same editing tools are available in the map view after selecting the target layer to be “Load points”.

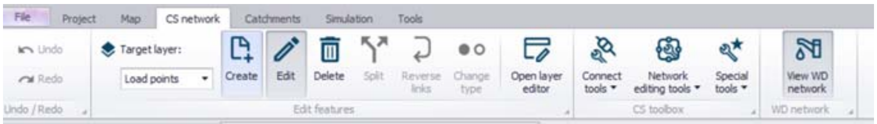


Figure 11.3 Layer editing tools activated after selecting the Load points target layer

All tools for graphical editing are fully supported by the "Undo" function.

11.4.1 Create a Load Point

Select the create feature tool to digitise load points. When this tool is active, the cursor appears as a + sign. Load points are digitized by a left mouse click at the desired location/s on the map. The tool is deactivated by clicking on the create feature tool again or selecting some other tool.

Each new load point is added as a new record in the ‘Load Points’ table. Per default, a name (i.e. identifier) is given as "Load_Point_n", where "n" stands for internal load point index. If required, the default identifier can be changed into a more meaningful name.

11.4.2 Edit/Move Load Point

An individual load point or a group of load points can be moved (translated) to a new position using the edit feature tool. Once the tool is selected, the mouse cursor will change symbol when it is directly over a load point. Single mouse click over a load point to select the load point to be moved (the symbol on the load point will change), then drag the load point to the new location. Right click on the load point to finalise the new location. The Edit tool remains active until it is deselected or some other tool is activated.

11.4.3 Delete Selected Load Point

Select the delete feature tool, the cursor will change to a + symbol and then click on a load point on the map to be deleted.

11.5 Allocating the Load Points to the Model Network

Three load allocation methods are available:

1. Manual load allocation
2. Graphical Load Allocation
3. Automatic Point Allocation by GIS Geocoding

11.5.1 Manual Load Point Allocation

Individual load points may be allocated to the collection system nodes through the “Load points” editor, by selecting a specific node to connect load point to, see Figure 11.4. This method is appropriate for individual corrections and/or for smaller sets of load points.

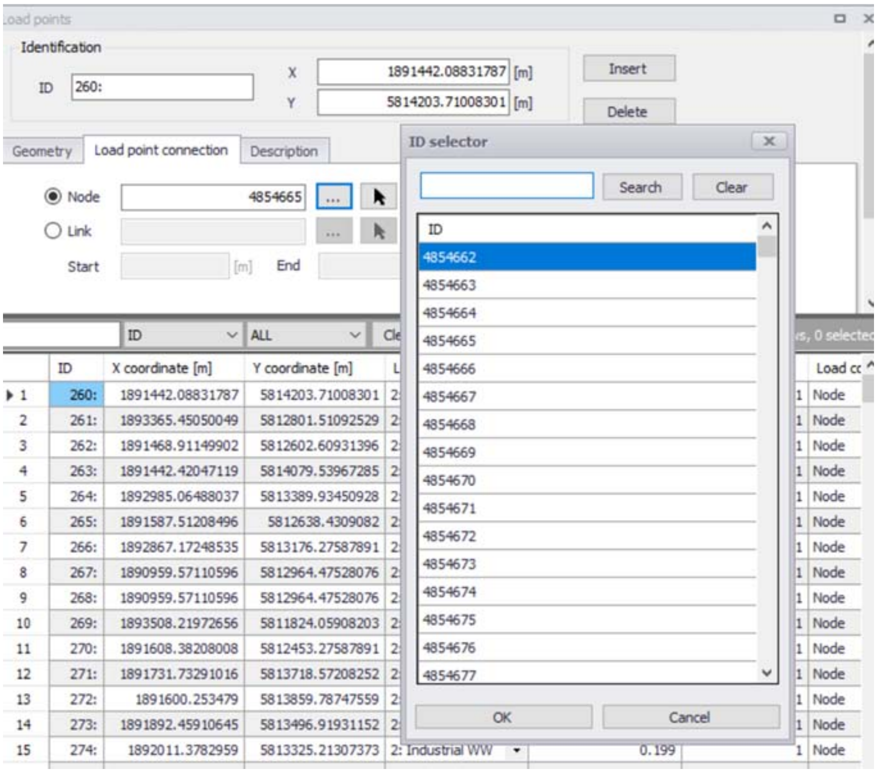


Figure 11.4 Manual allocation of the load point to a node - Example

11.5.2 Graphical Load Point Allocation

Individual load points may be allocated to the collection system nodes through the "Connect load" graphical tool. This can be activated by clicking on CS network | Connect tools | Connect load, click on the desired load point in the map and then select a node to connect it to.

Also, when adding a new load point into the map using the ‘create’ feature, you can use the connect tool to “connect load” tool to connect the unconnected load point to the desired node.

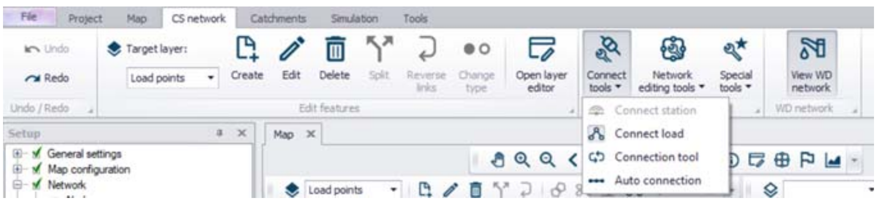


Figure 11.5 The Load Allocation Toolbar

The work process for the geographical load allocation is as follows:

1. Select the "Load points" layer from the target layer list, in the CS network toolbar.
2. Click on Connect tools | Connect load
3. Click on the desired load point in the map
4. Click on the desired node, and the connection will be automatically generated

MIKE+ plots the connection line between the load point and the selected node. If the current load point has already been allocated to some other node, the confirmation of the allocation action would re-connect the load to the current node.

This method is appropriate for individual corrections and/or for the smaller sets of load points.

11.5.3 Automatic Load Points Allocations by GIS Geocoding

MIKE+ supports automatic allocation of load points to collection system nodes through a GIS geocoding process. The geocoding process is initiated and controlled through the connection tool (Figure 11.6). This dialog is opened by clicking on CS Network | Connect tools | Connection tool

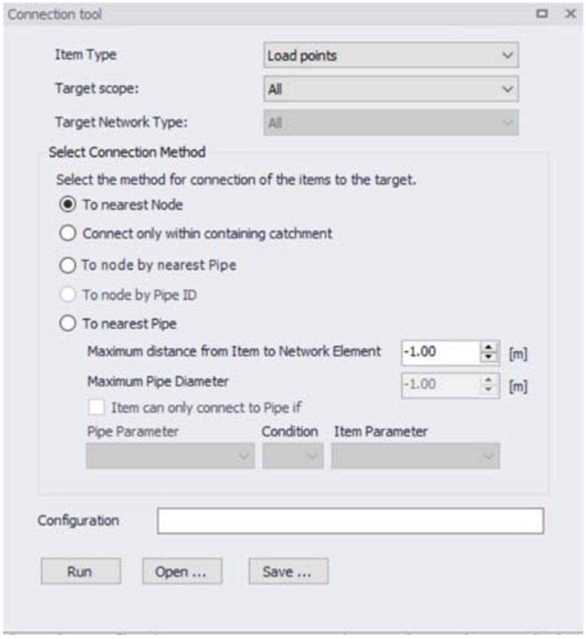


Figure 11.6 The connection tool

The following parameters affect the geocoding process:

Network Type: The load points are allocated only to the network elements (nodes and pipes) of a specified network type (optional field available in the description section of a network element). Thus, it is avoided that wastewater loads are allocated to the storm drainage network present in the same project.

Load category: A load point may be classified according to the available load types. Sometimes (e.g. in cases when pollution associated with each category is defined separately), it might be necessary to maintain the various load categories separately, so that the model boundary conditions can be defined properly. Selecting some records (e.g. based on the category) the connection tool then allows to perform the geocoding only on the selected load points by choosing target scope as “Current selection”.

Geocode method: There are two geocoding methods available for connecting load points to the MIKE+ model:

- To the nearest node: The load point is allocated to the MOUSE node (manhole or basin) which is geographically closest to the load point
- connect only within a contained catchment:
- to Nodes by nearest pipe: The load point is allocated to the downstream node of a link which is geographically closest to the load point.
- to nearest pipe: The load point is allocated to the nearest pipe, using two additional parameters which affect the geocoding process in this method:
  - Maximum distance from Item to Network Element (snap tolerance radius): The specified value (in map units) determines the largest distance for which is the geocoding performed. All load points which are not within the specified snapping distance to any pipe will remain non-allocated.
  - Maximum Pipe diameter: The specified value (in units for pipe diameter) limits the largest circular pipe which is eligible for geocoding. i.e. all larger the pipes (presumably trunk sewers) are assumed not to receive any direct loads.

Note: The geocoding process works on the selected set of load points or on the entire set.

The user should be aware that geocoding of a large set of load points is a computationally intensive process and may take some time. If the geocoding is attempted for already allocated points, the existing allocations will be cleared and replaced.

12 Interpolation and Assignment Tool

12.1 Introduction

The field assignment and interpolation tool is a tool that will assign values to any field in the MIKE+ database either by taking the attribute value directly from another feature/attribute or by interpolating between a number of other features.

Examples of the tasks that may be performed with this tool are:

- Assign ground elevation values from a raster layer representing the DEM to nodes.
- Assign the diameter of manholes to be equal to the largest pipe entering the manhole.
- Calculate missing values for manhole invert levels from a point theme using Inverse Distance weighted spatial interpolation
- Calculate pipe levels by interpolating values following the network (pipes).
- Assign a value to a construction year and or contractor based upon a polygon theme giving city areas.

The source of the data (i.e. the features where data is taken from) may be any layer in the MIKE+ map view, including layers that have been added as background layers. Any compatible data value can be assigned to almost any field in the database. This also means that it should be used with some care as it obviously also can make completely non-sense assignment if the wrong fields or names are specified.

The tool is accessed through the MIKE+ ribbon, WD network or CS network tab (depending on the project mode), Network Editing Tools, Interpolation and assignment.

The tool is set up as a workflow with the following steps:

- Target selection
- Assignment Method
- Assignment options (depending on the method chosen)
- Overall assignment
- Reporting

Each of the above steps are described in detail in the following sections.

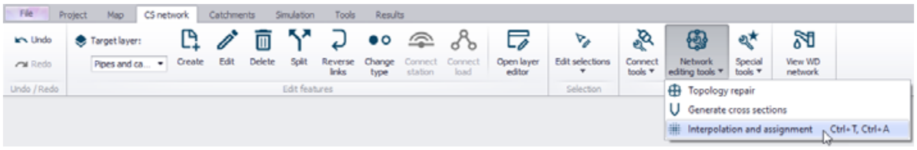


Figure 12.1 Accessing the interpolation and assignment tool

12.2 Target Selection

In the first step of the workflow, select the target attribute for the assignment. A target map layer (network component) must first be selected followed by a target attribute from the selected network component. For example, nodes layer, ground level attribute.

Once the empty fields are populated, MIKE+'s data validation functionality changes the "Target Selection" section of the workflow heading colour from red to green.

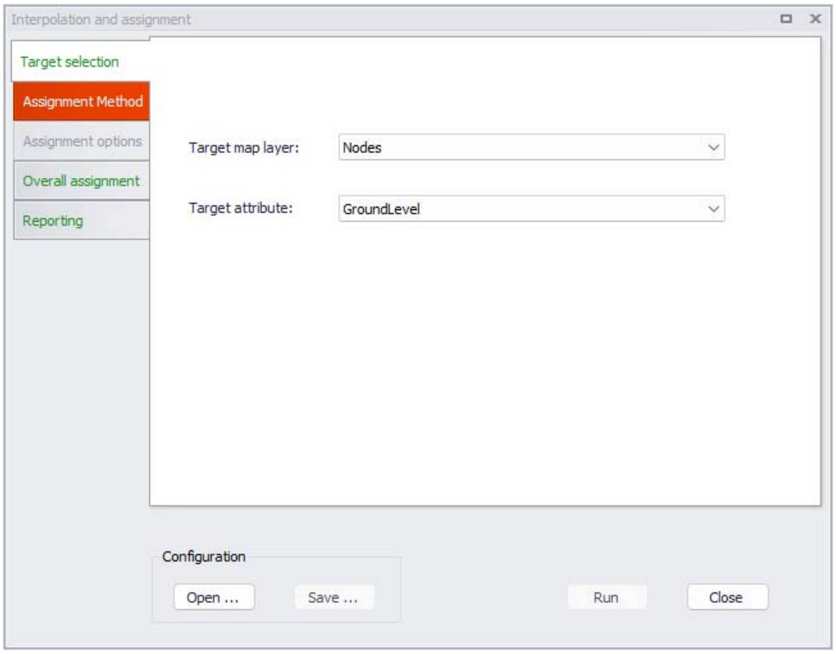


Figure 12.2 The Target selection dialog

12.3 Assignment Method

The next stage of the workflow defines the method to assign values to the target and the data source.

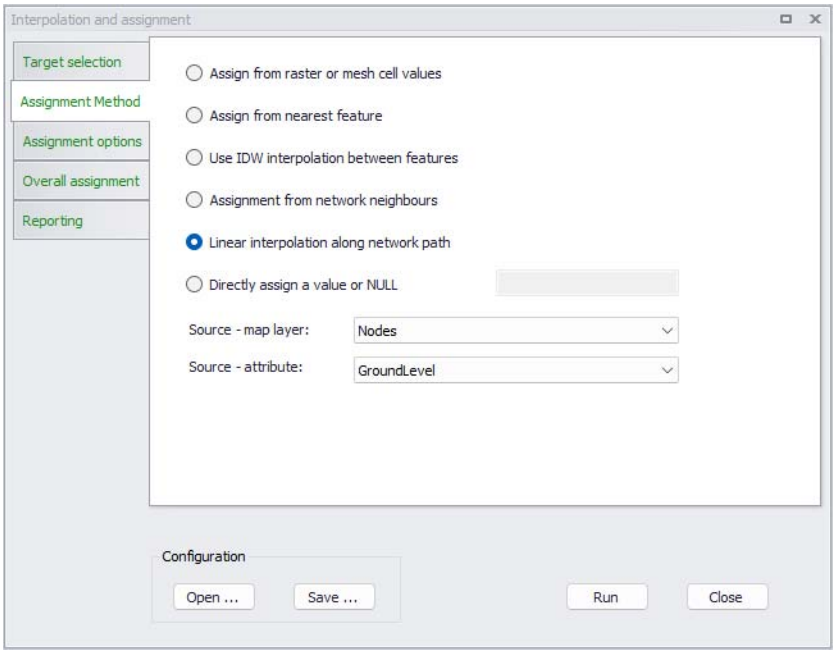


Figure 12.3 The Assignment Method dialog

First you must select the method as this will influence the valid choices for the data source. A number of methods exist:

- Assign from raster or mesh cell values - this will assign a value from the raster or mesh (DEM) cell in which the target data is located. For example, assign node ground levels based on levels in a raster. If the target is a polyline or polygon the tool will use the centroid position to determine the correct cell. No interpolation is done. The supported raster formats are .dfs2 files, ESRI text files (.txt, .asc), Arc/Info binary grids, GeoTIFF files (.tif, .tiff). The supported mesh formats are .mesh and .dfsu files. With a .mesh file, the tool assigns the average value from the nodes defining the element in which the target item is located. When assigning from rasters, points laying outside the raster's extent will be assigned the "No data" value. When assigning from meshes, points laying outside the mesh will not be updated).
- Assign from Nearest Feature - In this case the tool will locate the feature from the source layer that is closest to the feature in the target layer. If lines or polygons are used the centroid position is used for calculating distances.
- Use IDW interpolation between features - this option will make an Inverse Distance Weighted (IDW) interpolation between features in the source layer to determine the value for each target feature. The IDW parameters are fixed to the following: max number of points is 12 and the max distance away from the target feature is 300 (map units).

- Assignment from Network Neighbours - This option will take the source value from a network neighbour to the feature being updated. This obviously requires both the target and the source to be included in the same network. For example, assign manhole diameters from other manhole diameters nearby. Assignment will only be done if the immediate neighbour has the requested value i.e. the network will not be traced.
- Linear interpolation along network path - This option will do a distance weighted interpolation along the path of the network. If the direct neighbours do not contain values (null) the network is traced until a value is reached or the number of 'hops' (number of network nodes traced through) exceed a given maximum.
- Directly assign a value or NULL - This option allows to assign a specific value or to delete the content of an attribute (by assigning the NULL value).

Depending upon the choice of assignment method, the two selection boxes for the source data will be filled with layers/attributes compatible with the choice of method (i.e. only raster layers will be shown for raster assignment) or greyed out in the case of the last option.

12.4 Assignment Options

When the assignment method is “Assignment from network neighbours” or “Linear interpolation along network path”, extra parameters need to be specified in the next stage of the workflow in the section “Assignment Options”.

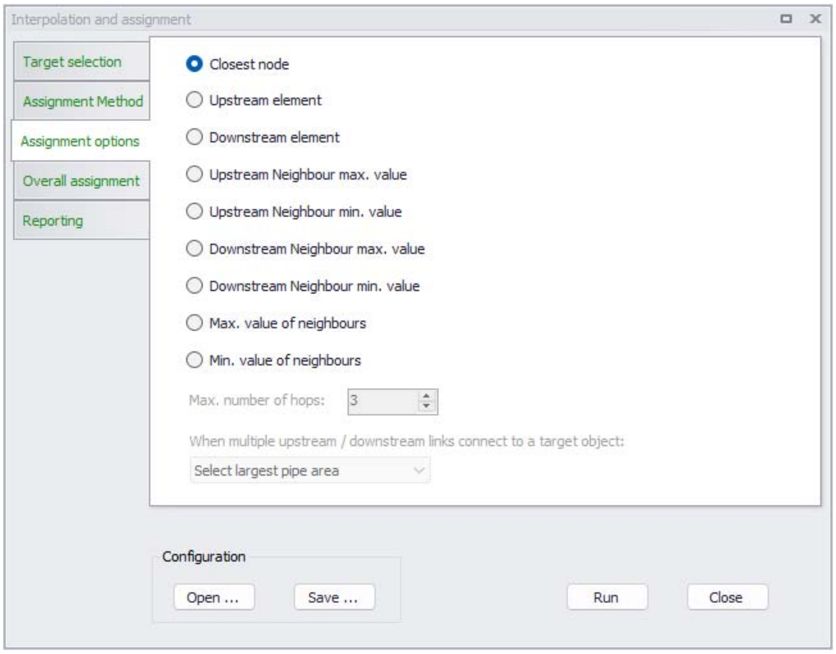


Figure 12.4 The assignment options dialog

For the 'Assign from network neighbours' assignment method, the following assignment options are activated to define how the assignment is to take place:

- Closest Node - This will use the node that is closest to the one being assigned to. This option is only relevant if both target and source are nodes.
- Upstream Element - This option will assign from the upstream element (upstream/downstream is as defined by the GIS geometric network and may differ from the actual flow direction (which may not be constant).
- Downstream Element - This option will assign from the downstream element (upstream/downstream is as defined by the GIS geometric network and may differ from the actual flow direction (which may not be constant).
- Upstream/Downstream Neighbour Max. Value - These two options will scan the connected network neighbours upstream/downstream and use the maximum source value found as data source. Example: for assigning ground level and diameters.
- Upstream/Downstream Neighbour Min. Value - These two options will scan the connected network upstream/downstream neighbours and use the minimum source value found as data source. Example: for assigning invert levels.
- Max. Value of Neighbours- This option will scan the connected network neighbours and use the maximum source value found as data source. Example: for assigning groundlevel and diameters.

- Min. Value of Neighbours - This option will scan the connected network neighbours and use the minimum source value found as data source. Example: for assigning invert levels.

For the ‘Linear interpolation along network path’ option, the following assignment options are activated:

- Maximum number of hops. This allows you to control how many network 'hops' the interpolation will search for a value. The search continues until the max number is reached or a non-null value is found. When the value is set to 5 or higher it may cause instability (particularly in looped networks). A value of 0 means that only immediate neighbours are taken into consideration. Large values may be time consuming if a large number of features are selected for update.
- Path selection method - The linear interpolation method interpolates between two features along the network. At junctions between multiples links, several paths can be selected (affecting which source features to interpolate from), and this option is used to control this selection. The method 'Select shortest link' will select the shortest link upstream or downstream the node, to define the path along which to interpolate. The method 'Select largest pipe area' will select the pipe with the largest flow area upstream or downstream the node. The following limitations apply to this 'Select largest pipe area' method:
  - If two pipes with the same flow area connect to the same node, the tool will select the one with the smallest length
  - Other link types than pipes are ignored, i.e. the path only select pipes
  - Pipes with missing flow area values (typically diameter) are ignored
  - In collection system networks, natural channels are ignored
  - In SWMM networks, only circular and closed rectangular conduits are supported by this method.

In the example below, values in Node_2 will be interpolated between Node_1 and Node_3 with the method 'Select largest pipe area', and between Node_4 and Node_3 with the method 'Select shortest link'.

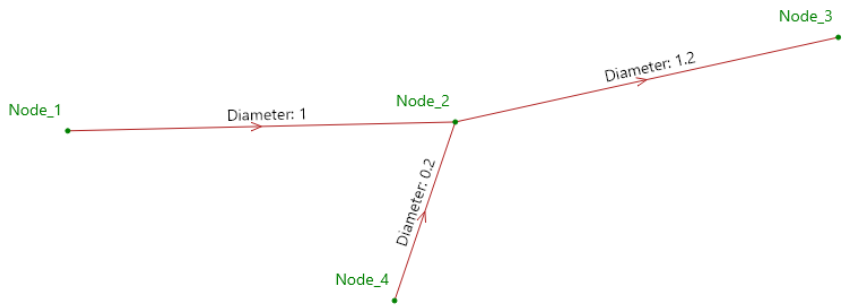


Figure 12.5 Interpolation path selection options

12.5 Overall Assignment

In this step of the workflow, as shown in Figure 12.6, you can control which features are taken into account for the assignment operation.

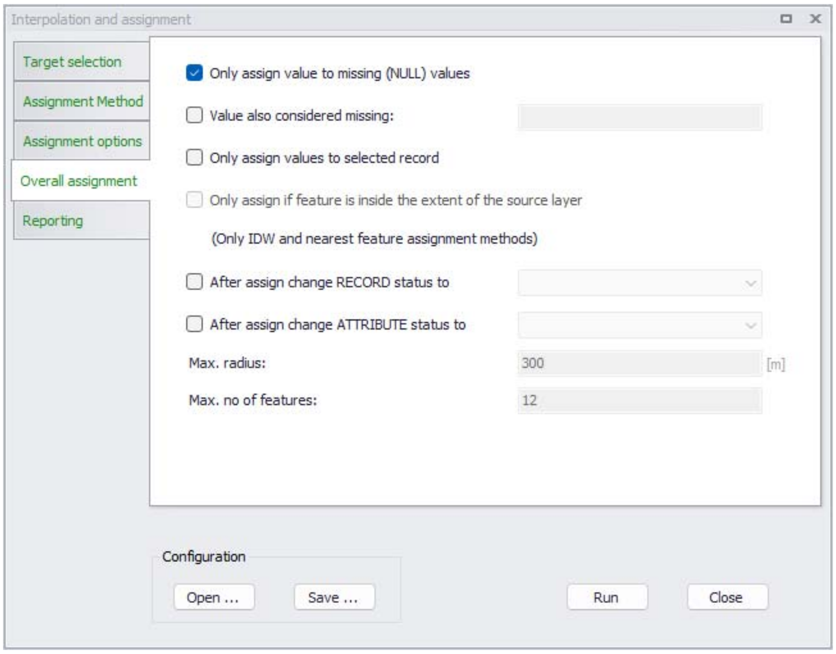


Figure 12.6 The Overall assignment dialog

The following options are available:

- Only assign value to missing (NULL) values - means that features that already have a value in the target field will not be updated. Removing this tick mark will overwrite any existing attribute values.

- Only assign values to selected records - this means that only records that are selected before the wizard was started are taken into consideration for updates.
- Only assign to features inside the extent of the source layer - this option prevents the tool from extrapolating outside the boundaries of the source layer when looking for the closest feature or when doing IDW interpolation.
- After assign change RECORD status to - this option changes the status of the modified records (e.g. nodes), by applying the predefined status selected from the list. This is the main status for the record (e.g. the nodes), which is typically found in the 'Description' tab.
- After assign change ATTRIBUTE status to - this option changes the status of the modified attribute (e.g. ground level), by applying the predefined status selected from the list. Every record is defined with multiple attributes, and this option will change the status for the updated attribute only. This attribute's status (e.g. the node's ground level) is found in the Property view, under the 'Status' menu.

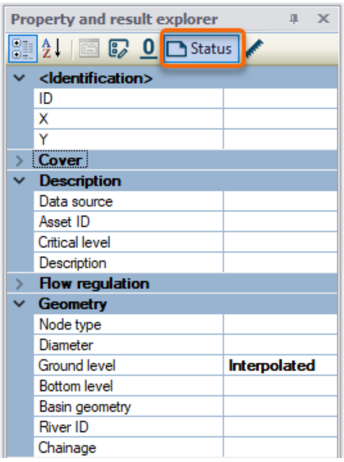


Figure 12.7 Accessing the attributes' status

12.6 Finishing the Wizard

To update the model with the interpolation/assignment, click on “Run”. The last section ‘Reporting’ gives a summary of the features that have been updated.

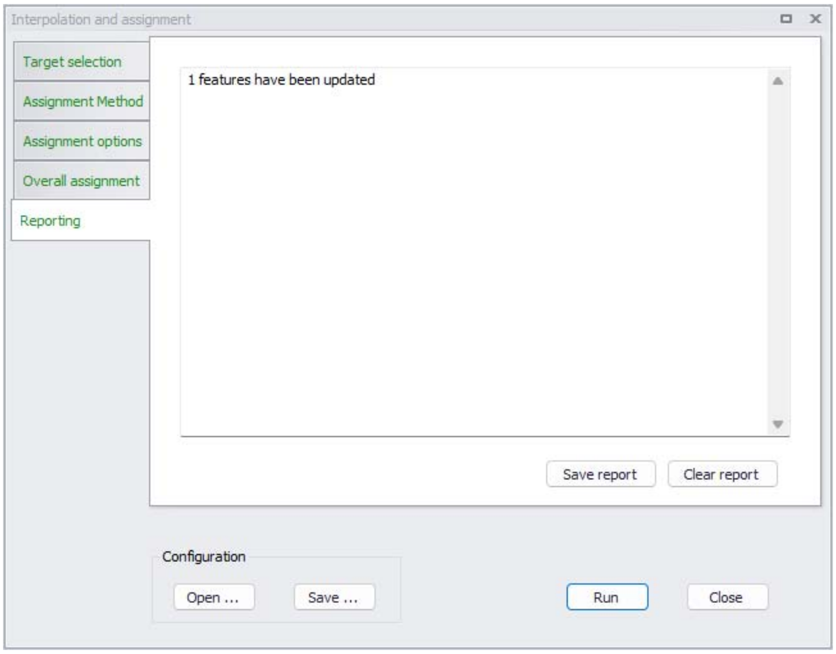


Figure 12.8 The report dialog

12.7 Configuration File

As with other MIKE+ tools, it is possible to save the tool setup configuration (Save button located near the bottom of the tool). A configuration file is created in a *.XML format and can be reused later (using the Open button).

13 Create Valves from Points Tool

13.1 Introduction

This tool is used to create multiple valves at once, by finding their locations and optionally their properties in a point shape file.

It creates the new valves at the closest location on the pipe network from the original point, within a maximum search distance. When the valve is inserted in the middle of a pipe, this pipe is automatically split in two new shorter pipes. When the valve is inserted at the end of the pipe, this pipe is simply shortened and no new pipe is created on the other side of the valve.

The tool is accessed through the MIKE+ ribbon, WD network tab, Network Editing Tools, Create valves from points.

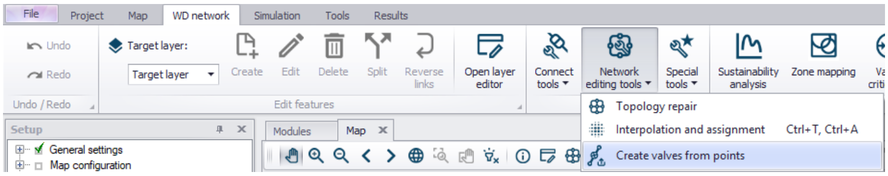


Figure 13.1 Accessing the Create valves from points tool

13.2 Configuration

The input shape file, with the points locating the valves to be created, must be selected in the 'Configuration' tab.

In the 'Input attributes' group, it is possible to import the main valves' properties from an attribute from the selected shape file. To achieve this, select the relevant attribute from the list, for the corresponding valve's property. Note that not all attributes are valid for each property: for example, only the attributes with numerical values will be listed for numerical properties. The details for each property are:

- Valve ID: any attribute can be selected to specify the valve ID. Note that this ID is expected to be unique for each valve: if the attribute contains an ID already in use for another valve, then the new valve will be assigned a default ID, thus differing from the name in the attribute.
- Valve type: the valve type can be imported either from attributes containing text data (in which case the valve type should be the same text as in the valves editor) or from attributes containing integers (in which case the valve type's value should correspond to the value of mw_-Valve.TypeNo). For example, both an input value '2' and an input string 'PSV' would be imported as valve type 'PSV'.

- Fixed status: the Fixed status can be imported from attributes containing integers corresponding to the value of mw_Valve.StatusNo.
- Diameter: any attribute with numerical values can be imported.
- Setting: any attribute with numerical values can be imported.
- Description: any attribute can be imported.

Selection of any of these attributes is optional. When no attribute is selected, the corresponding valve's property will be given a default value. Similarly, when an attribute is selected but when it contains an invalid value for a point, then the created valve is also given a default value and a warning will be provided.

The 'Maximum search radius' is the distance around a point in the shape file within which the tool will look for a pipe. If no pipe is found within this distance, the valve is not created.

The 'Created valve length' is the distance between the new nodes to which the valves are connected. This length therefore controls the length by which the pipes are reduced after inserting the valves.

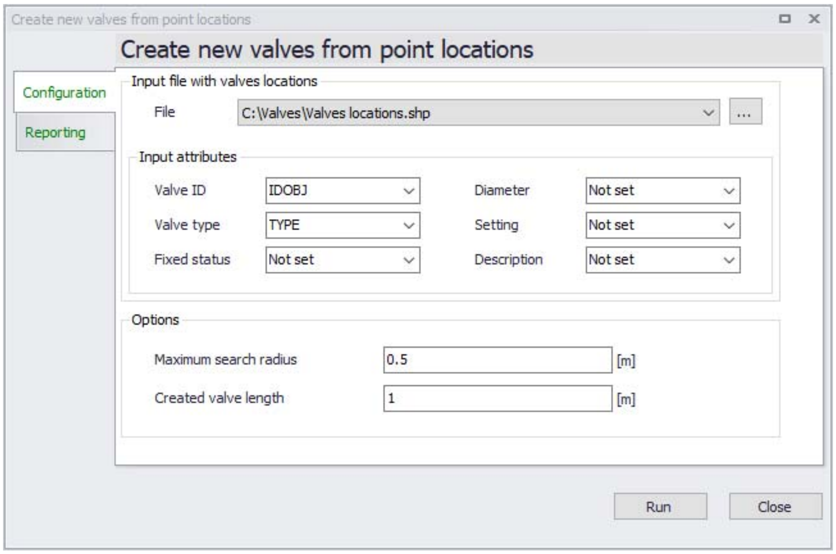


Figure 13.2 Configuring the Create valves from points tool

13.3 Running the tool

To update the model with the new valves, click on "Run". The 'Reporting' tab shows the warnings, if any, which are provided when valves could not be created or when properties could not be imported from the shape file.

14 Simplification Tool

14.1 Introduction

'Model simplification' is the term associated with the process of removing disconnected and unnecessary model elements, removal of model parts outside an area of interest and eliminating internal nodes which appear as redundant and insignificant for the hydraulic computations or for any other use of the model data.

Simplification reduces the complexity of a model which improves the efficiency of the computations. Correct simplification shall not compromise the integrity of the model and shall not affect the model's accuracy significantly.

Simplified models are used in different contexts - for the computations where time-efficient computation is of crucial importance, such as online model applications, long term simulations, strategic scenario analyses, etc., or for the presentation purposes.

14.2 Launching the Tool

The MIKE+ model simplification tool works with WD and CS models. The tool is found in the ribbons under 'WD Network' or 'CS Network' , under | Special tools | Network simplification (Figure 14.1).

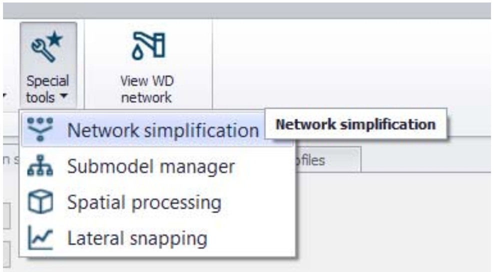


Figure 14.1 Launching the Network simplification tool

When activated, a wizard-like tool opens (Figure 14.2). The wizard includes several pages, each dedicated to a specific stage in the simplification process. Access to various pages goes through page selector in the left side. The pages accessible through the page selector depend on the actual context.

The sequence of pages in the menu suggests the preferred workflow.

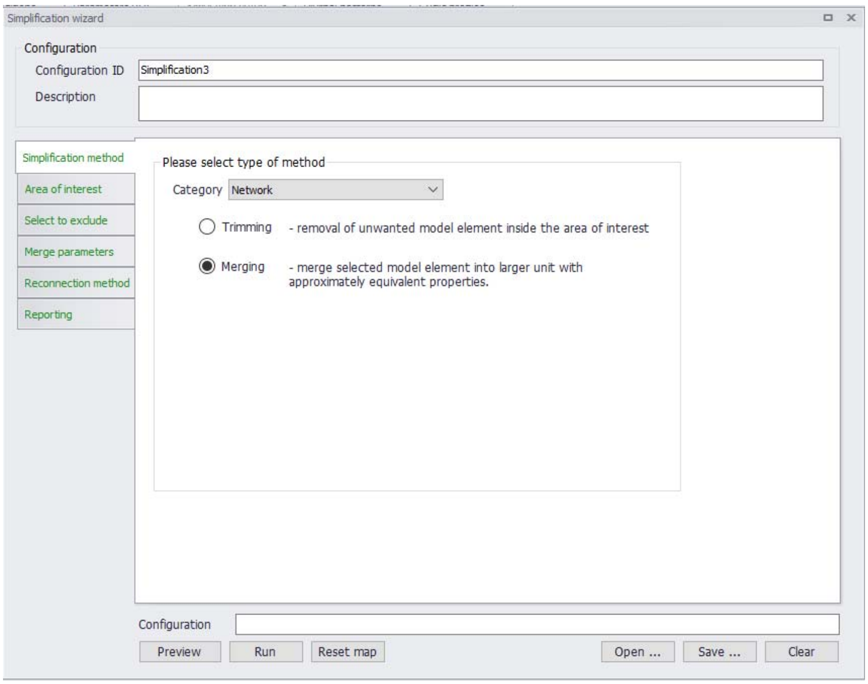


Figure 14.2 Simplification wizard

14.3 Simplification Categories and Methods

The simplification tool supports the following categories:

- Network: this category deals with reducing the network's complexity by removing nodes and pipes. The applicable methods are:
  - Trimming: This is about removing disconnected and regular parts of the network, selected by spatial or attribute-based filters, and/or manually. The catchments of CS models and boundary conditions attached to the removed network elements are automatically reconnected to the remaining network.
  - Merging: This is about removing nodes along a stretch of conduits and merging several conduits into one, equivalent conduit. The catchments of CS models and boundary conditions attached to the removed nodes are automatically reconnected to the remaining nodes or directly to the conduit.
- Catchments: this category deals with catchments. The only simplification method available for catchments is merging of multiple sub-catchments to larger units. The original parameters for the hydrological model are recalculated for the merged catchment. The merged catchment is re-connected automatically, according to user's specification.

- Surrogate: this category deals with converting a simplified "skeleton" network model into an equivalent network composed of orifices connecting basins.

The network simplification methods ("Trimming" and "Merging" are available for both CS and WD networks.

"Catchments" and "Surrogate" simplification categories are only available for the CS network.

14.4 Simplification Procedure

The simplification procedure includes several steps, each associated with the wizard pages:

1. **Simplification Method:** Here the simplification category and method are selected. Depending on the choices on this page, other pages are made accessible or hidden
2. **Area of interest:** In this page several various filters (both inclusive and exclusive) are available to select parts of the model that will participate in the simplification operation. This is relevant for all simplification categories and methods.
3. **Select to exclude:** Selection filters in this page operate on previously selected "Area of interest", to EXCLUDE the selected model elements from the simplification process. Typically, excluded elements are those that are essential for the model integrity. This is relevant for all simplification categories and methods.
4. **Trimming Parameters:** Includes a collection of parameters that control the network trimming process. This is only relevant when the selected simplification category is "Network", and the method is "Trimming"
5. **Merge parameters:** Includes a collection of parameters for the merge operation. Separate pages are available for network merge and for catchment merge operations. This is relevant when the selected simplification category is "Network" or "Catchments", and the method is "Merging"
6. **Surrogate parameters:** Includes a collection of parameters for the creation of "surrogate" hydraulic model. This is only relevant when the selected simplification category is "Surrogate"
7. **Reconnection method:** Contains specification for reconnecting catchments and boundary conditions, orphaned after removal of their original connection locations
8. Reporting

Each of the above steps are explained in detail in the following sections.

It is possible to save the simplification settings from the current session into an *.XML configuration file. Also, previously stored configurations can be open in the wizard, so that the simplification operation can be repeated on a different/updated model.

This functionality is available at the bottom of the network simplification wizard. When the *.XML file is opened, the settings in all dialogs will be filled to reflect the saved configuration.

14.4.1 Simplification method

The first thing to do is to select the wanted simplification category, and in case of "Network", the wanted simplification method.

The choices made in this page control the accessibility the remaining relevant pages with parameters for the configuration of the simplification operation. Content of some pages is adjusted automatically, depending on the actual context.

14.4.2 Area of interest

The parts of the model area that will be subject to the selected simplification method are defined on the tool's page "Area of Interest". User has several mutually exclusive and inclusive filters at disposal. The types of filters that are available depend on the actual simplification category.

"Area of interest" for the "Network" and "Surrogate" simplification categories

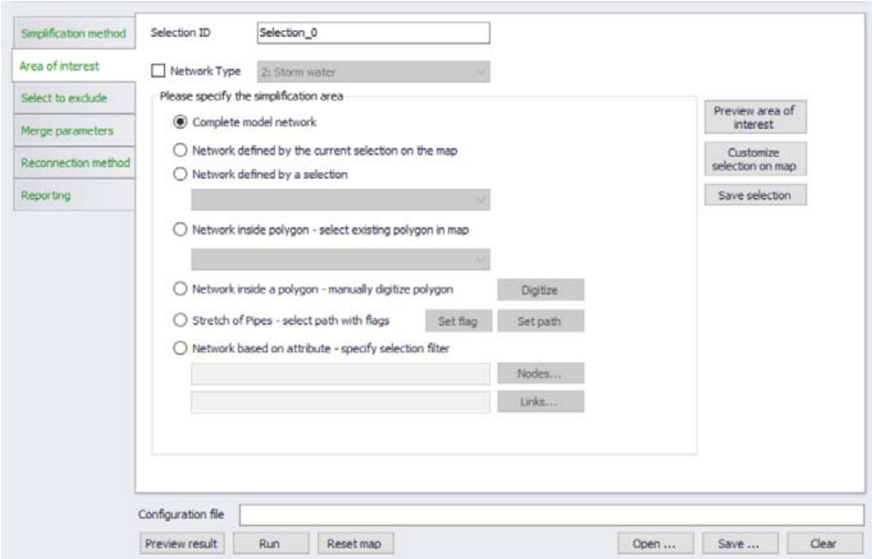


Figure 14.3 Area of interest page for "Network" and "Surrogate" simplification category

For the "Network" (both CS and WD networks) and for CS "Surrogate" categories, the area of interest is based on the selections of network elements - nodes and links. The following filters are available:

- **Network type**: Per default, this is not activated. This means that the chosen simplification method will act on all otherwise included model elements, independently of their "network type" attribute. This is recommendable for the models comprising only one type of drainage network. When activated by a checkmark, this filter limits the simplification operation to the selected type of network. Obviously, this requires that the "network type" attributes are consistently and correctly applied to all model elements.
  The "network type" filter works jointly with the other available filters.
- **Geographical and attribute-based filters**: These filters are mutually exclusive, i.e. the specified "area of interest" is based on one of the filters only.
  - **Complete model network**: This is the tool's default setting. This is an unfiltered selection
  - **Network defined by the current selection on the map**: When selected, area of interest is limited to the model elements that are marked as selected on the map, independently on how this selection has been created

- **Network defined by a selection**: This filter includes a reference to an existing selection in the "Selection manager"
- **Network inside existing polygon**: This filter limits the simplification to model elements inside e.g. selected sub-catchment polygons.
- **Network inside manually digitized polygon**: This filter limits the simplification to model elements inside a polygon digitized "on-the-fly".
- **Stretch of pipes** - select with flags: This filter is applicable for the network merging method and for "Surrogate" simplification. It limits the merging or surrogate operation to a specified stretch of pipes.
- **Network based on attributes**: Predefined filters (for nodes and/or links) in SQL format can be loaded and employed.

Pre-viewing and customizing the "Area of Interest" selection

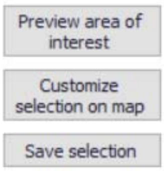


Figure 14.4 Action buttons on the "Area of interest" page

Action buttons "Preview area of interest", "Customize selection on map / Commit selection on map" and "Save selection" provide (optionally) useful functionalities, that support the selection process.

"**Preview area of interest**" highlights on the map all the model elements selected by the activated filters.

If this is not satisfactory, pressing "**Customize selection on map**" turns the highlighted element to the actual selection, allowing that user "manually" (i.e. using the map selection tools) customizes the selection generated by the active filters, by adding or removing some elements from the selection.

While in "customize" mode, the button changes to "**Commit selection on map**". When pressed upon completed customization, this button turns the current selection to highlights.

If wanted, the currently highlighted or selected elements are saved into the "Selection manager" as a new selection by "**Save selection**" button. This new selection is given a generic ID, that needs to be replaced by some meaningful ID for easy identification.

"Area of Interest" for the "Catchments" category

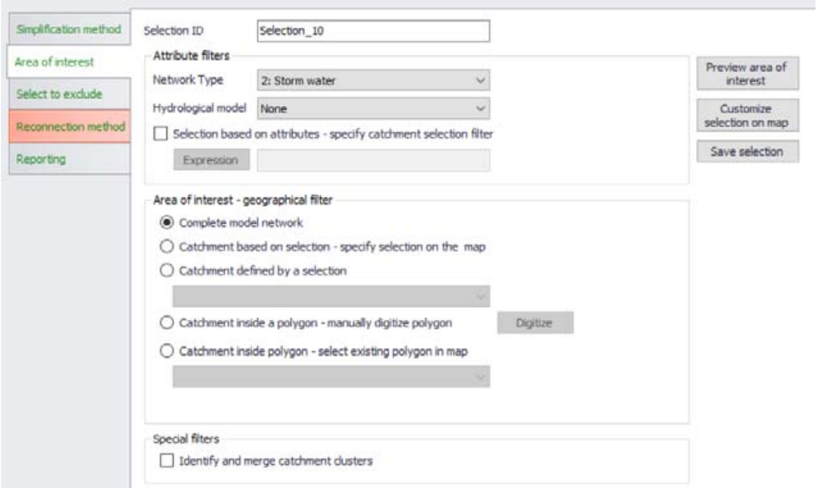


Figure 14.5 Area of interest page for "Catchments" simplification category

For the "Catchments" simplification category, the area of interest is based on the selections of catchments. The following filters are available:

- **Network type**: This filter limits the catchment merging operation to the catchments attributed to belong to one network type at a time (as per the 'Network type' defined in the 'Catchments' editor, in the 'Description' tab). This means that the catchment merging in the models containing catchments belonging to various network types would require several subsequent runs of the tool. Obviously, this tool requires that the catchments are consistently attributed by network type or, alternatively, remain undefined.

  This filter works jointly with the hydrological model filter, the generic attribute filter, and the activated geographical filter.
- **Hydrological model**: The tool only merges catchments set up for the same type of hydrological (i.e. Rainfall-runoff) model. I.e. if two or more catchments are to be merged, they all must be set up for the computation with the same hydrological model.

  This filter works jointly with the network type filter, the generic attribute filter, and the activated geographical filter.
- **Selection based on attributes**: When activated by checkmark, this filter allows for an additional filtering based on one or more catchment attributes.

  This filter works jointly with the network type filter, the hydrological model filter, and the activated geographical filter.

- **Geographical filters**: These filters are mutually exclusive, i.e. the specified "area of interest" is based on one of the filters only.
  - **Complete model network**: This is the tool's default setting. This is an unfiltered geographical selection that includes all catchments in the model
  - **Catchments defined by the current selection on the map**: When this is chosen, the area of interest is limited to the catchments that are marked as selected on the map, independently on how this selection has been created
  - **Catchments defined by a selection**: This filter includes a reference to an existing selection containing catchments in the "Selection manager"
  - **Catchments inside existing polygon**: This filter limits the simplification to model elements inside a selected existing polygon feature.
  - **Catchments inside manually digitized polygon**: This filter limits the simplification to the catchments inside a polygon digitized "on-the-fly".

- **Special filter (catchment clusters)**: A "**catchment cluster**" is defined as a set of catchments, all set to use the same hydrological model, defined with the same network type, and all connected to the same location in the network. The catchment cluster would typically emerge after a trimming operation, where insignificant, peripheral parts of the network get removed and the orphaned catchments get reconnected to the remaining part of the network.

  This filter identifies "clusters" and merges all catchments belonging to one cluster into one catchment.

14.4.3 Select to exclude

"Select to exclude" filters out the model elements that are to remain intact by the actual simplification. This is achieved by several mutually inclusive filters that are applied to the parts of the model previously defined by "Area of interest".

The types of filters and their default settings depend on the actual simplification category and method.

“Select to exclude” for CS network

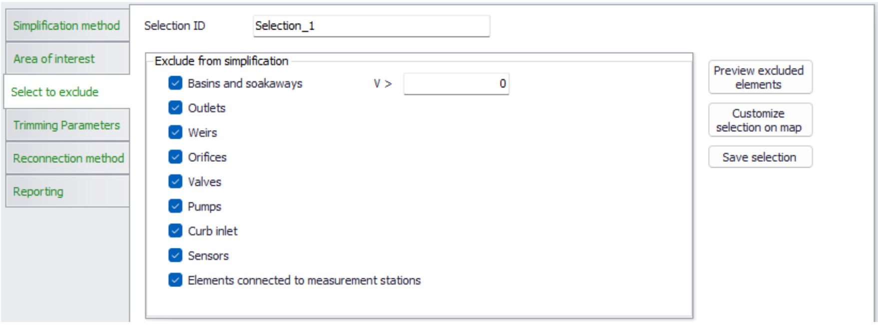


Figure 14.6 "Select to exclude" filters for CS network trimming

Per default, in CS network trimming all structures and the associated FROM/TO nodes, all basins and all outlets nodes, as well as all associated links are excluded from the trimming operation. Also, all elements containing sensors or connected to a measurement station are excluded per default. This is to avoid a loss of important functionality of the model in the simplified version.

Optionally, user can relax the default selection by unchecking some of the element types or to limit the max. size of basins to be excluded.

For CS network merging and surrogate, only the filters for basins & soakaways and for sensors can be deactivated. This means that merging operation will only remove plain nodes (manholes and junctions) without any structure attached and, optionally, basins under the specified volume threshold.

For "Surrogate" category all basins are unconditionally excluded from the simplification.

"Select to exclude" for catchment merging

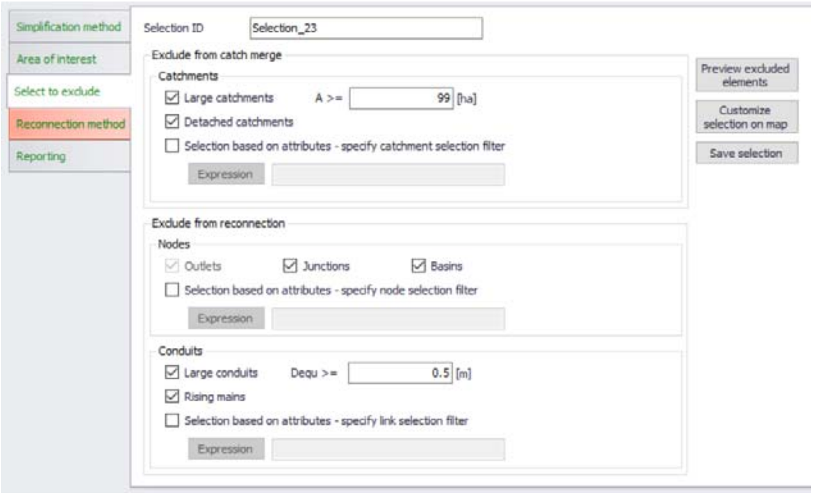


Figure 14.7 "Select to exclude" filters for catchment merging

This page contains two sets of exclusion filters:

- **Catchment exclusion filters**: this serves to exclude certain catchments from the merge process, similarly as it is the case with network elements in network merging or network trimming operations.
  - **Large catchments (default A >= 99 ha)**. When activated, this filter will leave out all catchments of the size larger than the specified threshold. I.e. the operation will be limited to smaller catchments only.
  - **Detached catchments**. Catchments that are not properly "snapped", will not be merged.
  - **Selection based on attributes**. This excludes from the merge operation all catchments filtered out by one or more catchment attributes specified here
- **Reconnection exclusion filters**. After the catchment merge operation, the resulting catchment shall be connected to the network according to the specification (see "Reconnection method" further below). Target for the catchment connection are both nodes and links. Some of these are not eligible for catchment connections by default and some may be excluded optionally by filtering.
  - **Node (Outlets)**: Outlet nodes are excluded per default, as connection of catchments to outlet nodes is not allowed.
  - **Node (Junctions)**: Junctions typically represent pipe joints without actual connection to the surface. Therefore, junctions are per default excluded as catchment connection points. If wanted, this filter can be de-activated

– **Node (Basins)**: Basins represent water storage facilities, frequently built as underground structures. In such cases they do not have actual connection to the surface, i.e. runoff. Therefore, basins are per default excluded as catchment connection points. If wanted, this filter can be de-activated.
– **Node (selection based on nodes attributes)**: This excludes from the catchment connections any nodes filtered out by one or more node attributes specified here
– **Link (Large conduits)**: When activated, this filter will leave out all pipes of the size larger than the specified threshold (i.e. of the equivalent diameter). So, the merged catchments will be connected to smaller pipes only. Reasoning behind this filter is that the large sewer pipes normally serve as transport conduits, without actual connections to the catchments. If wanted, this filter can be de-activated, or any other threshold size applied.
– **Link (Rising mains)**: per default, rising mains are excluded as targets for catchments connections. If wanted, this filter can be de-activated.
– **Link (selection based on links attributes)**: This excludes from the catchment connections any links filtered out by one or more links attributes specified here

"Select to exclude" for WD network

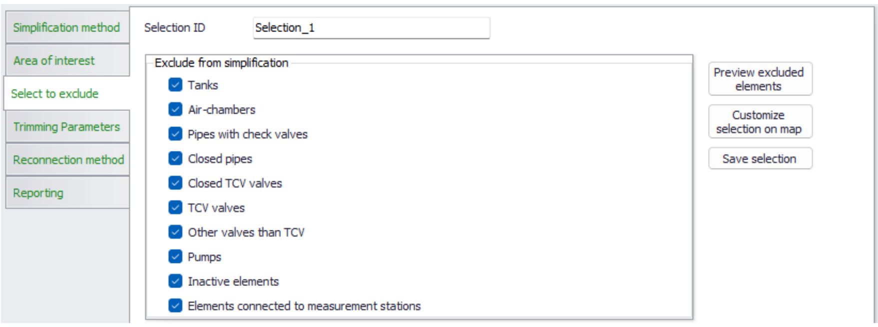


Figure 14.8 "Select to exclude" filters for WD network trimming

Per default for WD network trimming, several feature types like tanks, air-chambers, closed pipes, TCV valves, pumps, etc. are excluded from the simplification. Also, all elements connected to a measurement station are excluded per default. This is to avoid a loss of important functionality of the model in the simplified version.

Optionally, user can relax the default selection by unchecking some of the element types.

For WD network merging, only the filters for pipes with check valves, closed pipes, inactive elements and elements connected to measurement stations can be deactivated.

Pre-viewing and customizing the selection

Similar to the "Area of Interest" page, the "Select to Exclude" page provides functionalities for pre-viewing and customizing the selections obtained by the activated filters.

This is achieved by action buttons "Preview excluded elements", "Customize selection on map / Commit selection on map" and "Save selection". The functionality of these buttons is described under "Area of Interest".

14.4.4 Trimming parameters (CS Network and WD network)

After defining the area of interest and after excluding some model elements from the trimming operation, the actual trimming operation must be configured by setting the trimming parameters.

This is achieved on the "Trimming parameters" page.

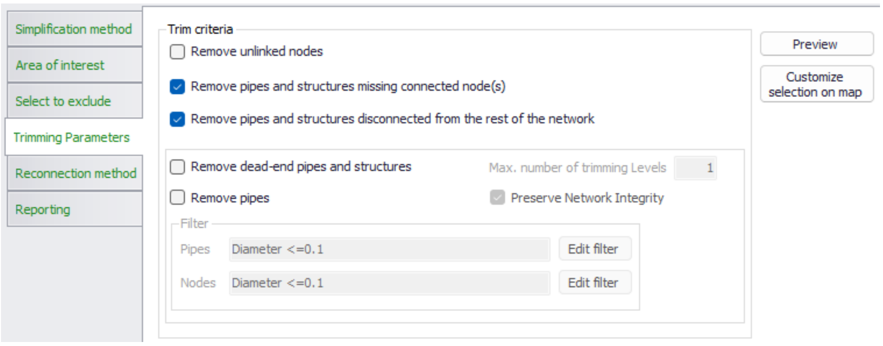


Figure 14.9 The "Trimming Parameters" page

"**Trimming**" means removal of the unwanted model elements from the periphery of the model. Typically, this would include disconnected nodes and pipes, dead-end pipes, small diameter pipes and nodes.

Any selection for trimming is made from the currently defined area of interest, further reduced by the elements specified as "selected to exclude".

The trimming operation is controlled by the following parameters:

- **Remove unlinked nodes**: "Unlinked nodes" are nodes disconnected from the rest of the network, i.e. with no connection by at least one link (conduit or structure)
- **Remove pipes and structures missing connected node(s)**: these are links missing a 'From' node, 'To' node, or both.
- **Remove pipes and structures disconnected from the rest of the network**: these are single links, which are not connected to any other link.

- **Remove dead-end pipes and structures: "Dead-end pipes"** are ending pipes with upstream node type "manhole" or "junction. **"Dead-end structures"** (CS network) are only relevant if not excluded by the user in the "Select to exclude" page.
- **"Remove pipes"**: This affects any pipes fulfilling the set pipe and node filters, independently on its position in the network. If both pipe and node filters are specified, then both criteria shall be fulfilled to include the pipe in the "Remove pipe". If node filter is specified, it must be fulfilled for both nodes for the pipe to be included in "Remove pipe".
- **"Preserve network integrity"** is a default option that controls the working of "**Remove pipes**". When activated, it prevents that the trimming removes pipes that secure the connectivity of the network! E.g. if a pipe fulfils criteria to be removed (D =<= 0.2 m) but is located somewhere inside the network, it should never be removed.

Note that if any dead-end pipe is removed, then its upstream ("dead-end", i.e. "FROM") node must also be deleted. Alternatively, if pipe orientation is not correct, a remaining orphan node shall be removed, independently if it is defined as "FROM" or "TO".

Removing of any pipe that is not a dead-end pipe, does not affect any of the connecting nodes.

"**Max no. of trimming levels**" is of relevance for both dead-end pipes and for general "Remove pipes". If number of trimming levels is set to more than 1, then initially internal pipe may become dead-end pipe in second round. Also, a pipe which initially was internal (and as such if removed by "Remove pipes" would destroy network integrity), may become ending pipe in the second trimming level and as such eligible to be removed.

"**Remove pipes**" is typically used to eliminate small peripheral pipes (e.g. house connections or some local sewers with small diameter).

Any catchment and boundary condition which becomes "orphan" after a pipe and its orphan node have been removed must be reconnected to a remaining node, according to the specification in "Orphan connections".

Pre-viewing and customizing the "Trimming" selection

Similarly, as the "Area of interest" page and the "Select to exclude" page, the "Trimming parameters" page provides for pre-viewing and customizing the selections obtained by specified trimming parameters.

This is achieved by action buttons "Preview" and "Customize selection on map / Commit selection on map". The functionality of these buttons is described under "Area of interest".

14.4.5 Network merging parameters (CS Network)

After defining the area of interest and after excluding some model elements from the network merging operation, the actual merge operation must be configured by setting the pipe merging parameters

This is achieved on the "Merge parameters" page.

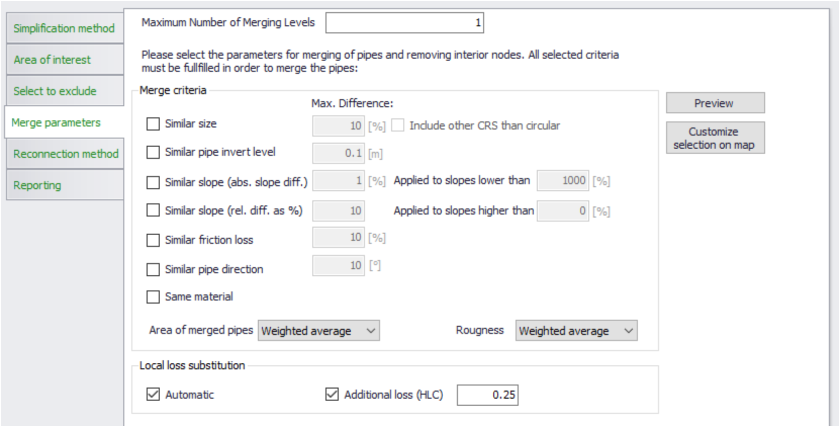


Figure 14.10 The CS Network "Merge parameters" page

"Merging" means joining two or more conduits along a network path into one conduit with equivalent hydraulic properties.

The orphan nodes remaining after the "merge" operation shall be removed. Also, any catchment or boundary condition attached to these nodes must be reconnected either to the remaining nodes or to the merged link.

General properties of merged conduits are:

- Slope is uniform and is calculated as a ratio between the connection level difference between the upstream and downstream node connection of the merged conduit and the total length of the merged conduits
- The merged conduit has the same shape of the cross section and the same size as calculated by the set criteria, along its entire length.
- If the sizes and types of standard conduits shapes (circular, rectangular, egg, O) or if the generic shape ID are identical for all included conduits in one stretch to be merged, then the merged conduit retains this size and type or the same generic shape ID.
- If the stretch of conduits to be merged contains conduits of different type and size in any combination, the merged conduit will be a circular pipe, with calculated equivalent diameter and Manning number.
- Conduits defined as "Natural channels" cannot be merged.

- The friction loss is defined with local data and is always calculated so that the conveyance of the merged conduit is identical to the weighted average conveyance.
- Horizontal layout of the conduit is unchanged, just the removed nodes are replaced by the link vertices.
- MUID of the merged conduit is constructed as follows:
  MUID = concatenate(<FromNode MUID> + "-->" + <ToNode MUID>)

All activated merging criteria must be fulfilled simultaneously if the two conduits are to be merged.

The merging operation is performed in one or more levels. This is necessary as the set criteria may be fulfilled after the first level merging operation has been completed.

Before any user-specified merging criteria are evaluated, some general conditions must be fulfilled if two or more conduits are to be merged. These are:

1. Included conduits must constitute a continuous flow path

AND

2. Any of the internal nodes in the included stretch of pipes must be connected to only two conduits. I.e. any junction or splitter node with more than two links connected cannot be eliminated by merge operation.

In the following, workings of the specified criteria are described in detail:

Similar size

Two or more conduits will be merged because of "similar size" in the following case:

- For circular pipes if the difference in size (diameter) is smaller than the specified limits
- For circular, rectangular, egg, O shape, or any closed generic shape in any combination, if the difference in size (i.e. full-flowing conduit area) is smaller than specified limits
- Pipes with open generic shape with the same generic shape ID.

Similar pipe invert level

Two or more conduits will be merged because of "similar pipe invert level" in the following case:

- The pipe invert level difference for the two adjoining pipes is smaller than the specified limit

Same material

Two or more conduits will be merged because of "Same material" in the following case:

- The friction loss for the included pipes is based on pipe material

AND

- The pipe material is the same for all pipes included in the stretch

Similar friction loss

Two or more conduits will be merged because of "Similar friction loss" in the following case:

- The friction loss computation for the included pipes is based on the same formulation

AND

- The friction loss for the included pipes is based on pipe material or locally specified value for all pipes included in the stretch

AND

- The friction loss (Manning number or C-W coefficient) difference is within the specified limit

Similar slope (relative slope difference)

Two or more conduits will be merged because of "Similar slope (relative slope difference" in the following case:

- The slope of the included conduit is inside limit for the specified minimum and maximum slope

AND

- The relative difference of slope of the involved conduits is within the specified limit

Similar slope (absolute slope difference)

Two or more conduits will be merged because of "Similar slope (absolute slope difference" in the following case:

- The slope of the included conduit is inside limit for the specified minimum and maximum slope

AND

- The absolute difference of slope of the involved conduits is within the specified limit

Similar pipe direction

Two or more conduits will be merged because of "Similar pipe direction" in the following case:

- The angle (direction change) in degrees, between the last segment (i.e. the last vertex and the "TO" node) of the upstream link, and the first segment (i.e. "FROM" node to the first vertex) of the downstream link of the two conduits is inside the specified angle limit

Pipe merging methods

When a stretch of two or more conduits fulfils all the activated criteria for merging, they will be merged into one "equivalent" circular pipe, with the following properties:

- Upstream (invert) connection level is set to the upstream connection level of the upstream-most conduit in the stretch
- Downstream (invert) connection level is set to the downstream connection level of the downstream-most conduit in the stretch.
- Length is set to the sum of all conduits in the stretch. The upstream and downstream connection levels and the total pipe length determine the uniform slope of the merged "equivalent" pipe.
- Cross section type, size, and generic shape ID:
  - If all conduits in the stretch to be merged have the same type and size, or the same generic shape ID, these properties are retained by the merged conduit.
  - When conduits with different sizes and closed cross section types are included, the merged conduit type is set to be a circular pipe. Equivalent diameter for the merged conduit is calculated with one of the following methods, selected by the user:
    - Weighted average cross section area. This is applicable for circular pipes, rectangular, egg, O-shape, generic shapes (closed) and any combination of the above. This method gives the equivalent pipe's volume equal to the volume of conduits included in the stretch. This is the default method.
    - Max. cross section area (This is applicable for circular pipes, rectangular, egg, O-shape, generic shapes (closed) and any combination of the above): equivalent diameter for the merged pipe is calculated from the largest value of the cross section area included in the merged conduit.
    - "Min. cross section area (This is applicable for circular pipes, rectangular, egg, O-shape, generic shapes (closed) and any combination of the above). Eq. diameter for merged conduit is calculated from the smallest value of the cross section areas included in the merged conduit.

    The calculated equivalent diameter is rounded off to 1 cm.
- Friction loss (roughness): Equivalent roughness for the merged conduit is calculated by one of the following methods:

- The weighted average hydraulic grade. This method sets the equivalent roughness (Manning's number) of the merged conduit so that in full-flow conditions it generates the same friction loss as the original conduits before merging. This method is appropriate for the pipe stretches that are dominantly full flowing.
- The weighted average roughness. This method calculates the equivalent roughness (Manning's number) of the merged conduit as a weighted average for the included conduits. The weighting is based on the conduit lengths. This method is appropriate for the pipe stretches that at dominantly free-surface-flowing.

Local (minor) head loss substitution

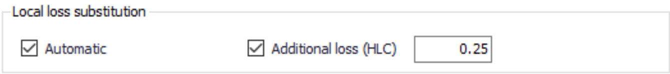


Figure 14.11 Activating local loss substitution

Some of nodes which get removed by the merging operation, may have included definition of local (minor) losses due to direction change, invert drop or flow contraction at the outlet. The loss of flow resistance due these head loss definitions in the removed nodes is (optionally) compensated by two methods:

- An automatic head loss substitution
- User-specified additional loss

Both methods are based on adjusting the merged pipe's equivalent Manning number, i.e. increasing the pipe's roughness., by the value that causes the additional friction loss along the merged pipe equivalent to the local head loss in the removed nodes, in conditions of full-flowing pipe.

The head loss substitution is activated by setting the checkmarks in the "Local loss substitution" box.

The head loss substitution is subject to the following:

- Nodes with method "No head losses" are excluded, i.e. no correction of Manning number in the merged pipe is needed
- For all nodes with local loss defined by the methods "Classic" or "Mean energy approach", a correction of Manning number in the merged pipe can be (optionally) applied
  - If the loss coeff. type is "Total HLC", the specified value is used directly in the calculation

- If the loss coeff. type is "Contraction HLC", the specified value is added to the calculated drop loss and direction change loss. The summed-up value represents the total HLC which is used in the calculation
- If the loss coeff. type is "Km", the specified value of Km is used to calculate contraction HLC. This is then added to the calculated drop loss and direction change loss. The summed-up value represents the total HLC which is used in the calculation.

A user-specified head loss is optionally specified as an alternative to the automatic substitution, or as a supplement to account for any local losses along the stretch of conduits to be merged, not included by the automatic method.

The user specified loss coefficient is used exactly in the same way as the losses substituted by increased friction loss by the automatic method.

14.4.6 Network merging parameters (WD Network)

Merging of pipes in WD networks is based on similar principles as described for CS network conduits.

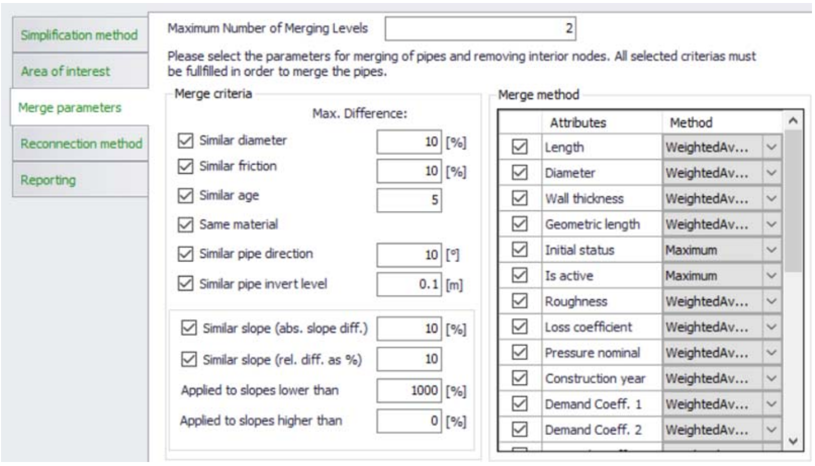


Figure 14.12 The WD Network "Merge Parameters" page

But, while all attributes for the CS network conduits are set or calculated by a predefined method, some attributes of WD pipes can be set or calculated by one of the following methods:

- Weighted average: the attribute is calculated as a weighted average
- Minimum
- Maximum
- Sum

14.4.7 General catchment merging parameters

General catchment parameters are subject to automatic merge operation that cannot be changed by the user. I.e. the user just needs to understand how does the merge operation act on the involved catchments.

Total geometric area is calculated as a geometric area of the merged catchment.

Total catchment area (user specified) for the merged catchment is calculated as follows:

- If none of the included catchments have user specified "Catchment area" defined, this attribute remains empty
- If all selected catchments have user specified "Catchment area" defined, the "Catchment area" of the merged catchment is calculated as a sum of all "Catchment area" values
- If only some of the selected catchments have user specified "Catchment area" defined, the "Catchment area" for the merged catchment is set as a sum of user-specified "Catchment area" values (where available) and the geometric areas (for catchments with undefined "Catchment area").

Person equivalents for the merged catchment is calculated as a sum of "Person equivalents" values for the selected catchments. If none of the selected catchments has "person equivalents" defined, this remains undefined also for the merged catchment.

14.4.8 Catchments merging parameters for hydrological models

For different hydrological models, the actual catchment merge operation must be configured by the user. This includes selecting the preferable automatic method for setting up the values of catchment merge parameters or setting up the values directly. This is achieved on the catchments "Merge parameters" pages, separately for each type of hydrological model.

The merge operation works for one runoff model type at a time, as specified on the "Area of interest" page. Accordingly, the page(s) containing the respective model parameters are made accessible.

Time-Area merge parameters

"Imperviousness" is calculated as a weighted average imperviousness, weighting based on the total catchment area. This is the default and only method, that cannot be changed.

Other time-area model parameters can be calculated or set according to the chosen method. The page containing time-area model parameters is accesible when merging catchments with runoff models "Time-Area (A)" and "Time-Area (A) + RDI".

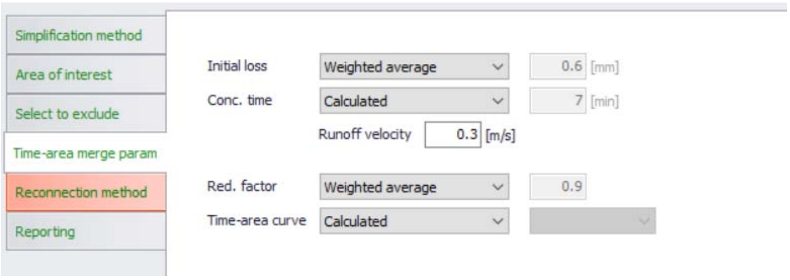


Figure 14.13 Time-Area merge parameters

The automatic calculation of model parameters is based on the local parameter values (where activated) or on the values from the applied parameter set for the actual catchments.

Time-area model parameters to be automatically set by the catchment merge operation are:

- **Initial loss**
  Methods available for setting this parameter are:
  - Weighted average (default): Initial loss for the merged catchment is calculated as a weighted average initial loss, weighting based on the impervious catchments' area. If all included catchments have imperviousness zero, weighting is based on the catchments' total area.
  - Minimum: Initial loss for the merged catchment is set to the lowest initial loss of the included catchments.
  - Maximum: Initial loss for the merged catchment is set to the highest initial loss of the included catchments.
  - User-specified: Initial loss for the merged catchment is set to the user-specified value.
- **Concentration time**
  Methods available for setting this parameter are:
  - Calculated (default): Concentration time for the merged catchment is calculated as a product of the longest distance inside the merged catchment's polygon and the user-specified runoff velocity.
  - Weighted average (default): Concentration time for the merged catchment is calculated as a weighted average concentration time, weighting based on the catchments' geometric area.
  - Minimum: Concentration time for the merged catchment is set to the lowest concentration time of the included catchments.
  - Maximum: Concentration time for the merged catchment is set to the highest concentration time of the included catchments.
  - User-specified: Concentration time for the merged catchment is set to the user-specified value.
- **Reduction factor**
  Methods available for setting this parameter are:

– Weighted average (default): Reduction factor for the merged catchment is calculated as a weighted average reduction factor, weighting based on the impervious catchments' area. If all included catchments have imperviousness zero, weighting is based on the catchments' total area.
– Minimum: Reduction factor for the merged catchment is set to the Reduction factor of the included catchments.
– Maximum: Reduction factor for the merged catchment is set to the highest Reduction factor of the included catchments.
– User-specified: Reduction factor for the merged catchment is set to the user-specified value.

- **Time-Area curve**
  Methods available for setting this parameter are:
  – Calculated (default): This method creates a new, 11-points time-area curve. The new curve's ordinates are calculated as weighted averages for the T-A curves of the included catchments. The weighting is based on the geometric catchments' area.
  – User-specified: Time-area curve for the merged catchment is selected from the list of the available T-A curves.

Kinematic wave model merge parameters

"**Contributing area**" for all five surface types is calculated as a weighted average, weighting based on the total catchment area. This is the default and only method, that cannot be changed.

Other kinematic wave model parameters can be calculated or set according to the chosen method. The page containing kinematic wave model parameters is accesible when merging catchments with runoff models "Kinematic Wave (B)" and "Kinematic Wave (B) + RDI".

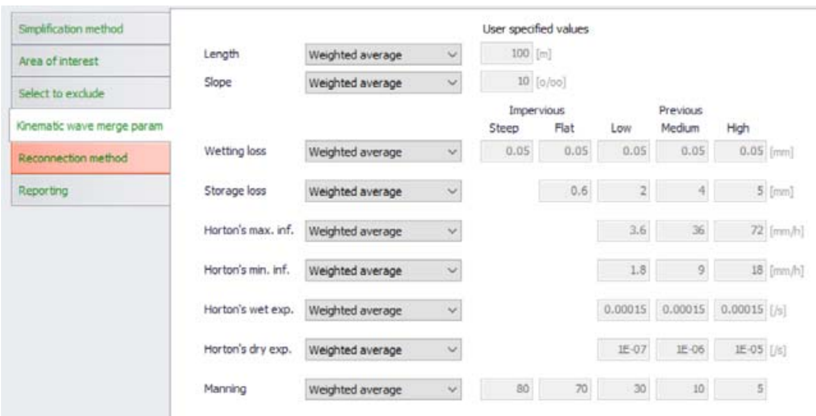


Figure 14.14 Kinematic wave model merge parameters

The automatic calculation of model parameters is based on the local parameter values (where activated) or on the values from the applied parameter set for the actual catchments.

Kinematic wave model parameters to be automatically set by the catchment merge operation are:

Catchment-wide parameters:

- **Length (m)**
- **Slope ($^{o}/_{oo}$)**

Parameters for various contributing surface types:

- **Wetting loss (mm)**
- **Storage loss (mm)**
- **Horton's max. inf. Capacity (mm/h)**
- **Horton's min inf. Capacity (mm/h)**
- **Horton's wet weather exponent (/s)**
- **Horton's dry weather exponent (/s)**
- **Manning number**

For all these parameters, methods available for setting the model parameter are:

- Weighted average (default): The catchment-wide parameters for the merged catchment are calculated as weighted averages, weighting based on catchments' geometric area. The parameters for contributing surfaces are calculated as weighted averages, weighting based on catchments' surfaces contributing areas.
- Minimum: Parameter for the merged catchment is set to the lowest value of the included catchments.
- Maximum: Parameter for the merged catchment is set to the highest value of the included catchments.
- User-specified: Parameter for the merged catchment is set to the user-specified value.

Linear reservoir model merge parameters

Effective area (%) (model C1) and **Imperviousness (%) (Model C2)** are calculated as a weighted average, weighting based on the total catchment area. This is the default and only method, that cannot be changed.

Other linear reservoir model parameters can be calculated or set according to the chosen method. The page containing linear reservoir model parameters is accessible when merging catchments with runoff models "Linear reservoir

(C1)", "Linear reservoir (C2)", "Linear reservoir (C1)+RDI" and "Linear reservoir (C2) +RDI".

Depending on the chosen type of linear reservoir model (C1 or C2), the relevant attributes get activated or de-activated.

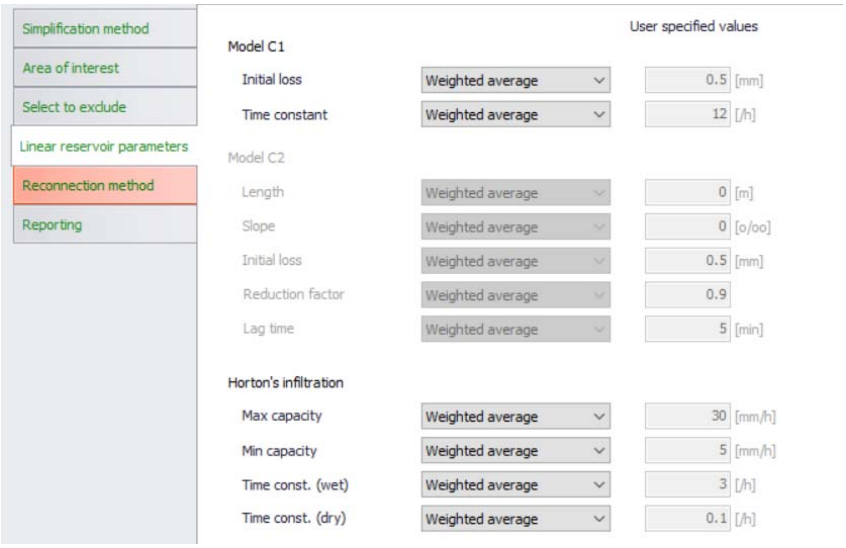


Figure 14.15 Linear reservoir model merge parameters

The automatic calculation of model parameters is based on the local parameter values (where activated) or on the values from the applied parameter set for the actual catchments.

The model parameters can be calculated or set by one of the following methods:

- Weighted average (default)
- Minimum: Parameter for the merged catchment is set to the lowest value of the included catchments.
- Maximum: Parameter for the merged catchment is set to the highest value of the included catchments.
- User-specified: Parameter for the merged catchment is set to the user-specified value.

The default method (weighted average) is different for various catchment-wide parameters, i.e. the weighting is based either on the contributing area or on the geometric area.

For the following parameters, weighting is based on "Effective area" (Model C1) and on "Imperviousness" (Model C2):

- **Initial loss (C1)**
- **Initial loss (C2)**
- **Reduction factor (C2)**
- **Horton's Min capacity (C1 and C2)**
- **Horton's Max. capacity (C1 and C2)**
- **Horton's time constant (wet) (C1 and C2)**
- **Horton's time constant (dry) (C1 and C2)**

For the following parameters, weighting is based on geometric areas:

- **Time constant (C1)**
- **Length (C2)**
- **Slope (C2)**
- **Lag time (C2)**

UHM model merge parameters

All parameters for the UHM model can be calculated or set according to the chosen method. The page containing UHM model parameters is accesible when merging catchments with runoff models "UHM" and "UHM+RDI".

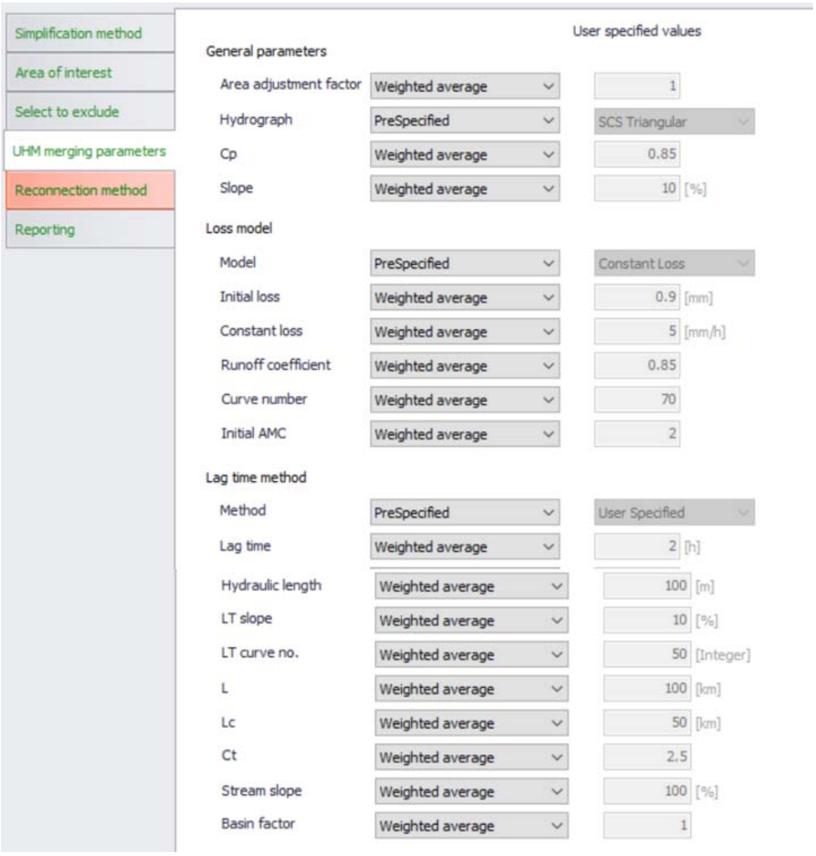


Figure 14.16 UHM model merge parameters

The UHM model's numeric parameters are calculated or set by one of the following methods:

- Weighted average, weighting based on the catchments' geometric area (default)
- Minimum: Parameter for the merged catchment is set to the lowest value of the included catchments.
- Maximum: Parameter for the merged catchment is set to the highest value of the included catchments.
- User-specified: Parameter for the merged catchment is set to the user-specified value.

Non-numeric parameters, which represent the choice among available options are:

- **Hydrograph type**
- **Loss model type**
- **Lag time method**

These are set as to one of the following options:

- **Pre-specified (default)**: this option is valid and available when each of the selected catchments has the same option set in the original setup. I.e. the merged catchment gets the same setting of the parameter as the original catchments included in the merge operation.
- **User-specified**: this option can be actively chosen in any situation, but when the selected catchments have different parameter settings, it is the only option available. User must set the wanted value of the parameter.

RDI model merge parameters

Parameter "**RDI area**" (%) is calculated automatically as a weighted average, based on the geometrical catchment areas of the included catchments. This is default operation and cannot be changed by the user.

Parameter "**Additional flow**" (m^3/s) for the merged catchment is calculated as the sum of additional flows for the involved catchments (only with "additional flow" activated). This is default operation and cannot be changed by the user.

All other parameters for the RDI model can be calculated or set according to the chosen method.

The RDI parameters are presented in two pages (see below). The pages containing RDI model parameters are accesible when merging catchments with runoff models "RDI (solo)" and RDI model in any available combination with surface runoff models.

Figure 14.17 RDI model merge parameters (page 1)

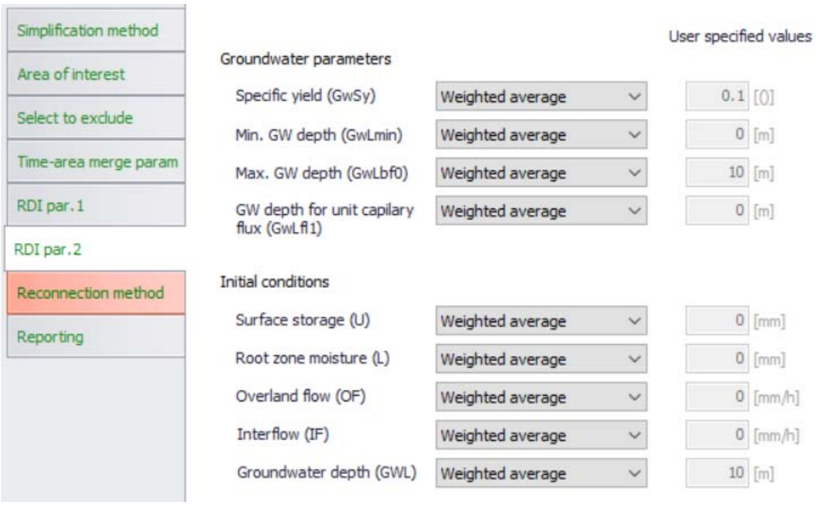


Figure 14.18 RDI model merge parameters (page 2)

All other parameters are calculated or set by one of the following methods:

- Weighted average (default)
- Minimum: Parameter for the merged catchment is set to the lowest value of the included catchments.
- Maximum: Parameter for the merged catchment is set to the highest value of the included catchments.
- User-specified: Parameter for the merged catchment is set to the user-specified value.

All parameters are weighted by the actual RDI area.

Weighting of the time constants **CK**, **CKif** and **BF** also includes the overland flow coefficient **CQof** , that accounts for the distribution of the RDI components.

14.4.9 Parameters for the surrogate model simplification

Applying the surrogate model simplification means replacing a prismatic conduit (e.g. a pipeline) by an orifice and a basin.

I.e., instead of a conduit between two nodes, an orifice is inserted. The orifice has a shape and size identical to the conduit's cross section.

The upstream node of the conduit is converted to a basin, and its volume is represented by the conduit's area-elevation curve, assuming a zero slope (i.e. conduit slope is not accounted for).

Normally, an outset for a surrogate model will be a model which is simplified by "Network trim" and "Conduit merge" methods, with the following remaining elements:

- Any important structures and nodes (basins, pumps, weirs, orifices, valves…)

- Merged conduits connecting model nodes, so that the main network layout and water transport ways are preserved. Distances between the remaining nodes (i.e. lengths of the remaining conduits) shall not be too long.

Ultimately, the final resulting surrogate model will be the model containing only important nodes and structures, including orifices and basin volumes representing major conduits. I.e. an ultimate surrogate model is a hydraulic model without any conduits.

While other types of structures remain in their original form, pumps will usually require modifications to "constant flow" pumps or similar, to avoid the pumping dynamics and therefore shortening simulation time step.

Accordingly, the surrogate simplification operation includes the following steps:

- Selected links are removed and replaced by additional volumes in upstream nodes and by orifice functions, according to specified parameters
- Any boundary conditions and catchments connected to the removed links are reconnected to the remaining nodes, according to the specified method.

Parameters that control the surrogate model simplification are found on the page "Surrogate parameters". This page is accessible when surrogate simplification category is chosen.

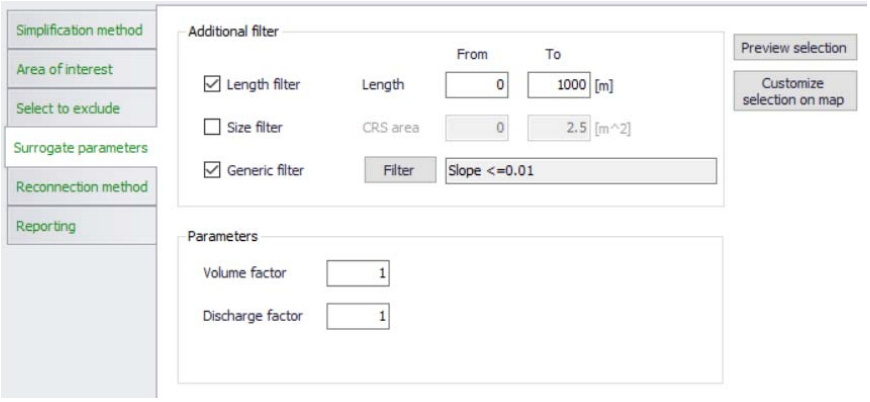


Figure 14.19 Surrogate model parameters

"**Additional filter**" focuses on easy selection of links to be converted to orifices. The filters activated here act on top of the selection defined by the "Area of interest" and "Select to exclude".

Length filter: allows to select the pipes within a specified range of length

Size filter: allows to select the pipes within a specified range of cross section area

Generic filter: allows for a specification of any generic filter

"**Parameters**" are the calibration factors for the two important parameters of the surrogate model:

- **Volume factor**: scales the volumes automatically included in the volume definition
- **Discharge factor**: Scales the automatically calculated discharge coefficient for the orifice.

14.4.10 Reconnection methods for network and surrogate simplification categories.

In order to preserve the integrity and completeness of the loads to the simplified CS model, any catchments (i.e. runoff and catchment loads associated with these catchments) and network loads connected to the nodes and links removed from the model by any of the network simplification methods must be re-connected to the nodes or links remaining in the CS model.

Similarly, in order to preserve the integrity and completeness of the demands to the simplified WD model, any demands connected to the nodes and links removed from the model by any of the network simplification methods must be re-connected to the nodes or links remaining in the WD model.

This reconnection is done automatically, according to the specified reconnection method. The reconnection process is controlled from the page "Reconnection Method"

Reconnecting catchments (CS), loads (CS) and demands (WD) after network trimming

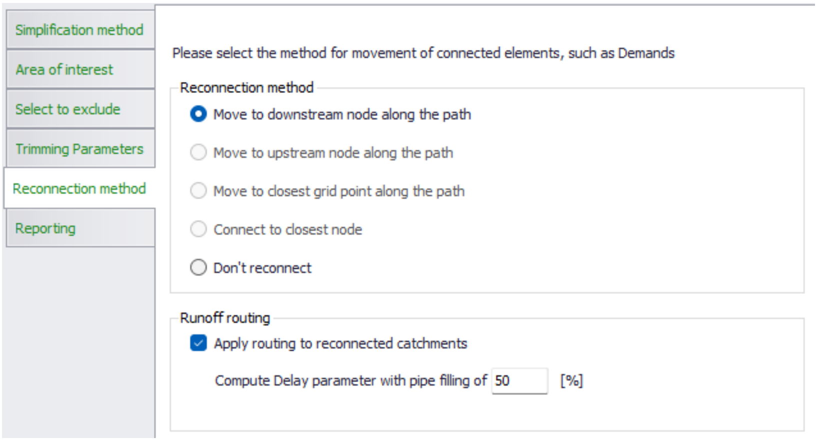


Figure 14.20 Reconnection methods for network trimming

The only reconnection method available for CS catchments, CS network loads and WD network demands left disconnected ("orphaned") by trimming is the connection to the downstream node along the path. E.g., if a dead-end pipe and its upstream node are removed, any CS catchment, CS network load or WD network demand originally attached to the upstream node or the pipe itself, in the simplified model will be attached to the pipe's downstream node - the one that remains in the model.

Alternatively, "Do not reconnect" would leave such CS catchments, CS network loads and WD network demands "orphaned", i.e. they would not contribute to the loads/demands of the simplified network.

Reconnecting the network loads in CS networks implies a loss of flow travel time, network volume and wave attenuation in the removed network. To account for this, a runoff routing can be activated during the CS network trimming operation. When the option 'Apply routing to reconnected catchments' is selected, the runoff routing (defined in the Catchment connections editor) will be activated. An attenuation of the runoff hydrograph will then be computed before it enters the trimmed network. This routing (attenuation) is computed using the Muskingum method, using the parameters defined in the Catchment connections editor. During the simplification process, the Delay parameter for each of these catchment connections will be updated, adding a flow time along all links trimmed downstream the catchment connection (i.e. between the original connection location and the new connection location after trimming). This flow time is computed using a flow velocity estimated with the Manning formula and with an assumption regarding the pipe filling, which is defined in the reconnection method settings.

Reconnecting CS catchments, CS loads and WD demands after network merging

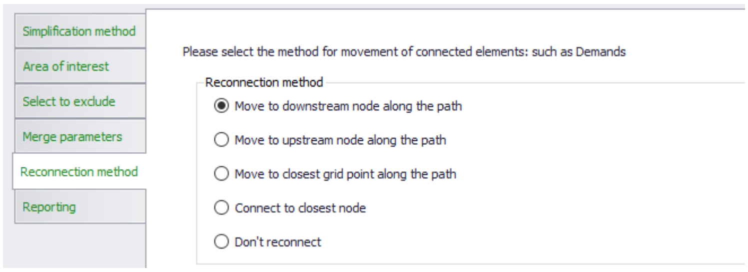


Figure 14.21 Reconnection methods for network merging

The reconnection methods available for "orphaned" CS catchments CS network loads and WD network demands by pipe merging are the following:

- **Move to downstream node along path**: This method reconnects any CS catchment, CS network load and WD network demand originally connected to the removed nodes and/or to the substituted conduits to the downstream node of the merged pipe.
- **Move to upstream node along path**: This method reconnects any CS catchment, CS network load and WD network demand originally connected to the removed nodes and/or to the substituted conduits to the upstream node of the merged pipe
- **Move to closest grid point along the path**: This method reconnects each CS catchment, CS network load and WD network demand originally connected to the any of the removed nodes and/or to any of the substituted conduits to the grid points of the merged pipe that are the closest to the original location of the connection.
- **Connect to closest node**: This method reconnects any CS catchment, CS network load and WD network demand originally connected to the any of the removed nodes and/or to any of the substituted conduits to the closest node in the network after merge operation.

Alternatively, "Do not reconnect" would leave such CS catchments, CS network loads and WD network demands "orphaned", i.e. they would not contribute to the loads/demands of the simplified network.

Reconnecting the CS network loads and CS catchments to the locations relatively far from the original connection location may imply change of flow time. In the current version, the simplification tool does not compensate for the lost flow time.

Reconnecting catchments and loads after Surrogate simplification

Reconnection options for this simplification type are similar as those available for pipe merge method.

The only difference is that (for obvious reasons) reconnection to grid points is not possible.

14.4.11 Reconnection methods for CS catchment merge simplification

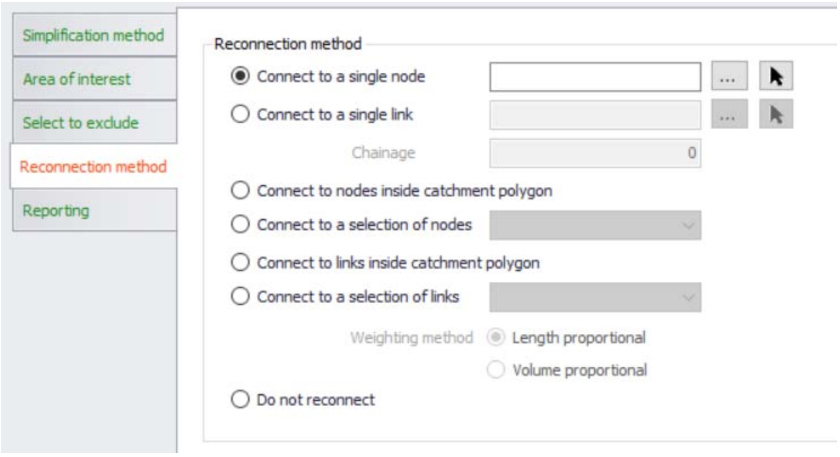


Figure 14.22 Reconnection methods for catchments merging

The reconnection methods available for merged CS catchments are the following:

- **Connect to a single node**: This method connects the merged CS catchment to the user-specified node.
  The specified node must be of the same "Network type" as the current setting in "Area of interest". 100% of runoff and/or catchment discharges generated on the catchments included in the merge operation will flow into the specified node. All original catchment connections associated with the involved catchments are deleted. The new catchment connection is generated as type "Standard", "WW Total" or "SW Total, depending on the currently set "Network type".

- **Connect to a single pipe**: This method connects the merged CS catchment to the user-specified link, to a grid point that is closest to the specified chainage (i.e. distance from the FROM node).

  The link must be of the same "Network type" as the current setting in "Area of interest". 100% of runoff and/or catchment discharges generated on the catchments included in the merge operation will flow into the specified node. All original catchment connections associated with the involved catchments are deleted. The new catchment connection is generated as type "Standard", "WW Total" or "SW Total, depending on the currently set "Network type".
- Connect to nodes inside the catchment polygon: This method connects the merged CS catchment to all eligible nodes inside the catchment polygon.

  All original catchment connections related to the included catchments are deleted.

  The runoff and catchment discharge are distributed to the nodes inside the merged catchment polygon with fractions corresponding to the area fractions obtained by Thiesen polygons method performed around the connection nodes. The sum of the fractions is 1 (i.e. 100%).

  The new catchment connections are of the type "Combined partial", "WW partial" or "SW partial", depending on the actual "network type" currently set.
- **Connect to a selection of nodes**: the merged CS catchment is connected to a set of nodes defined by a selection from the "Selection manager".

  All original catchment connections related to the included catchments are deleted.

  The runoff and catchment discharge are distributed to the selected nodes with fractions corresponding to the area fractions obtained by Thiesen polygons method performed around the selected nodes. The sum of the fraction is 1 (i.e. 100%).

  The new catchment connections are of the type "Combined partial", "WW partial" or "SW partial", depending on the actual "network type" currently set.

- **Connect to links inside the catchment polygon**: This method connects the merged CS catchment to all eligible links inside the catchment polygon.

  "Eligible links" are those of the current "Network Type" and those that are completely inside the catchment polygon. I.e. links which cross the catchment boundary are not included.

  The runoff and catchment discharge are distributed to the links inside the merged catchment polygon with fractions corresponding to one of the following options:

  – Length-proportional: the load is distributed proportionally to the links' length
  – Volume-proportional: the load is distributed proportionally to the links' volumes (calculated for full flow)

  All original catchment connections related to the included catchments are deleted.

  The new catchment connections are of the type "Combined partial", "WW partial" or "SW partial", depending on the actual "network type" currently set.

- **Connect to a selection of links**: This method connects the merged CS catchment to the eligible links defined by a selection in the "Selection manager".

  "Eligible links" are those of the current "Network Type".

  The runoff and catchment discharge are distributed to the connections links with fractions corresponding to one of the following options:

  – Length-proportional: the load is distributed proportionally to the links' length
  – Volume-proportional: the load is distributed proportionally to the links' volumes (calculated for full flow)

  All original catchment connections related to the included catchments are deleted.

  The new catchment connections are of the type "Combined partial", "WW partial" or "SW partial", depending on the actual "network type" currently set.

Alternatively, "Do not reconnect" would leave the merged CS catchment disconnected.

Reconnecting the merged CS catchment using different reconnection options may imply the change of flow patterns. In the current version, the simplification tool does not compensate for these changes.

14.5 Saving the Configuration

Once the simplification method has been defined, it is possible to save the configuration into an *.XML configuration file.

This file can then be re-opened to be reused by clicking on the 'Open' button and browsing to the saved *.XML file.

14.6 Previewing the simplification results and generating the simplification report

By clicking on the button "Preview result" at the bottom of the simplification tool, the model features that are to be included in the specified simplification operation will be highlighted on the map and a summary with overview of the effects of the simplification will be generated in the "Reporting" page.

Example for a CS network Pipe merging simplification setup is shown in Figure 14.23 and Figure 14.24.

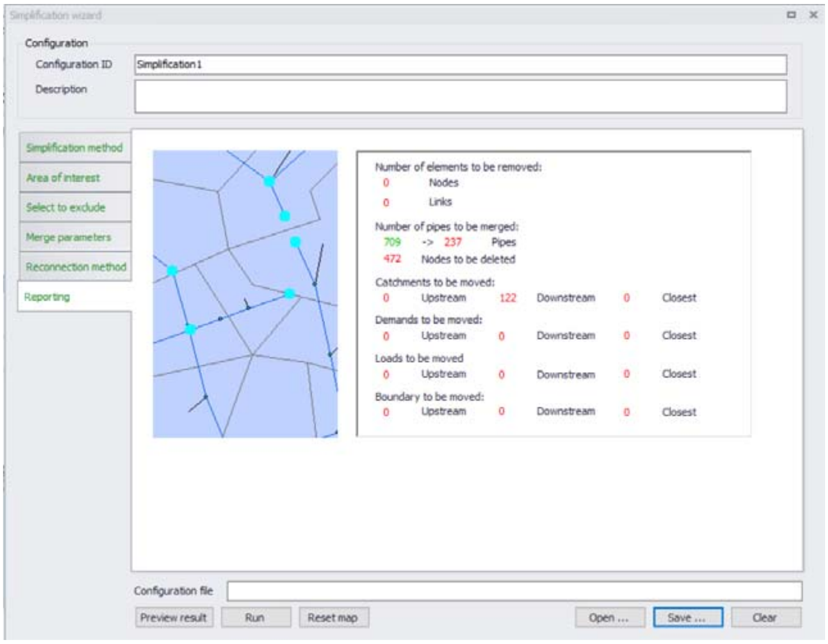


Figure 14.23 Summary report of simplification results

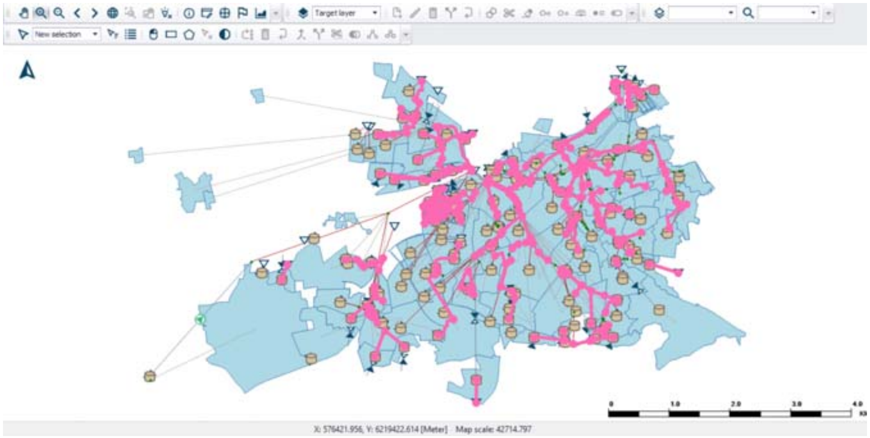


Figure 14.24 Highlighted model elements affected by the set pipe merge simplification

14.7 Executing the simplification

The actual simplification is executed by pressing the "Run" button. An example of a CS network before and after a simplification (pipe merging) operation is shown in Figure 14.25.

An original model configuration can be recovered after any simplification operation by "Undo" function.

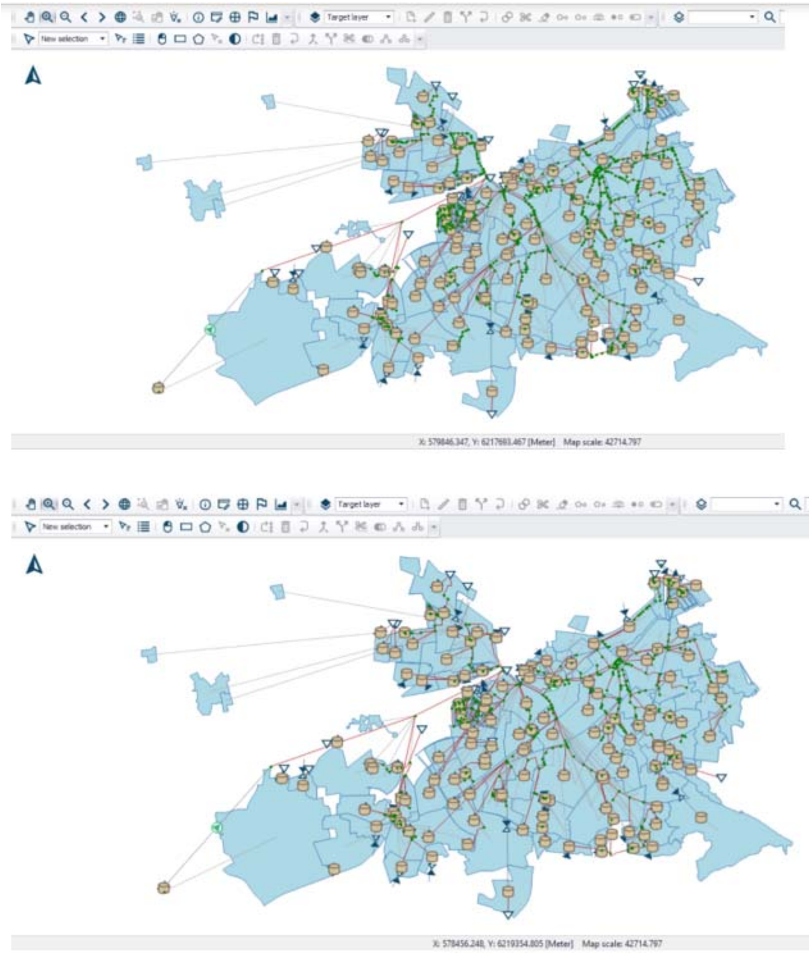


Figure 14.25 A CS network before (above) and after the pipe merge simplification. The original network has 1280 pipes, and the simplified network has 808 pipes.

14.8 Executing from command lines

The simplification tool can be configured and executed from the user interface as described in the previous chapters. However, there may be situations where it is required to automate the simplification without going through the related editors.

The MIKE+ executables enable you to execute some tools without opening the editor, through command lines. It is possible to run the simplification tool in this manner, assuming you have prepared the simplification configuration file beforehand in MIKE+.

Start by locating the MIKE+ executable named DHI.MIKEPlus.ToolShell.exe in the installation folder. From a command prompt, type the command below to access the list of supported tools, replacing the … characters by the actual path to the file:

"C:\...\DHI.MIKEPlus.ToolShell.exe" -h

The format of the command for executing a simplification configuration is:

"C:\...\DHI.MIKEPlus.ToolShell.exe" SimplificationTool -db [MIKE+ file] -f [Configuration file]

Where [MIKE+ file] is the path to the .mupp or to the .sqlite file and [Configuration file] is the path to the .xml configuration file.

15 Scenario Management

Water distribution and free-surface flow models are commonly used for system performance analysis and planning studies. The complexity of the involved systems, the various uncertainties about future conditions and usually huge costs associated with maintenance, rehabilitation and development necessitate a thorough investigation of alternative system configurations in a search for the technically feasible, environmentally sound and economically efficient solution.

These alternative configurations - scenarios - may differ by the system's physical layout, loading conditions, operational strategies, etc. Various projects, such as development of a Sewerage Master Plan, Wastewater Transportation Strategy, an Overflow Abatement Strategy, a River Restoration Stategy, a Flood Protection Scheme, etc. would typically result in a large number of scenarios, either representing alternative system configurations at a given time and/or representing the system at various development stages. Testing of each scenario against the prescribed legislation or the standards of service that the authorities provide requires a numerical model on its own.

These scenarios are always related to each other through the common origin ('Existing Case' or 'Base') and the differences typically encompass only a smaller part of the total data. Moreover, scenarios representing a development of the system through time are subject to the dependencies propagating along with the timeline.

Analysis of the scenarios as separate projects creates major inconveniences, such as:

- Large number of models, even when differences between them are minor;
- Missing an efficient overview over the entire set of solutions;
- Inability to maintain the existing dependencies between the individual scenarios automatically. Thus, the updating of the models with additional information requires editing of multiple files to change the same element, e.g. if a pipe diameter is found to have been incorrectly registered in the source data, it will have to be updated multiple times in each of the scenario project file;
- Unable to easily visualise differences between scenarios.

In other words, working with the scenarios as separate projects is inefficient and cumbersome.

Instead, the MIKE+ Scenario Manager provides an easy way of managing multiple scenarios, within a single MIKE+ project (i.e. a single database).

15.1 What is a Scenario Manager?

The MIKE+ Scenario Manger is accessed via the Setup tree ‘Scenarios'.

The Scenario Manager enables the definition, organisation, management and reporting of alternative model scenarios, such as:

- Augmentation of existing trunk sewer mains;
- Increased wastewater loading from increased population;
- Increased water demands from increased population;
- Alternative design loads, e.g. rainfall-runoff of different return period;
- Alternative/new alignment of water, sewer and storm mains;
- Building of a new sewer trunk and water supply mains in order to cater for a new development area;
- A range of riverbed roughness values for sensitivity analysis purposes;
- Modified 2D overland roughness for land use changes impact management e.g. as a result of flooding;
- Etc.

All within the same MIKE+ project.

With the MIKE+ Scenario Manager, a user can work with an unlimited number of scenarios in a single MIKE+ project.

15.2 Design of the MIKE+ Scenario Manager

15.2.1 Data Groups, Alternatives and Scenarios

The MIKE+ Scenario Manager is based on the concept of Data Groups, Alternatives and Scenarios.

In this context, a Data Group is a set of database tables which form a meaningful set. E.g. all database tables containing collection system network data belong to the data group " CS Network Data". Every database table relevant for the scenario manager is included in one of the Data Groups.

Each Data Group can appear in the MIKE+ project in any number of Alternatives. The initial alternative is named with a default name 'Base Alternative'. Any further alternatives are created upon user request and can have a user-specified name. The Alternatives for a certain data group are organised in a tree-like structure, where dependencies propagate along the branches - from the "parent" to all the "heirs" i.e. "child" alternatives.

A scenario contains a collection of one alternative from each Data Group. Individual alternatives are used as building blocks for constructing scenarios.

For example, modelling a new development area could have new alternatives for "CS Network", “Loads and boundaries" and "Catchments and hydrology" data groups, while the remaining data groups remain as the base case. The moderate number of data groups allows for a manageable structure of scenarios, while ensuring a high level of flexibility.

The initial scenario is named with a default name 'Base', and consists of the 'Base' alternative of each data group. Any further scenarios are created upon user request and can have a user-specified name. The scenarios can be organised in a tree-like structure of "parents" and "children". A new scenario is created in the "Setup" menu, by a right click on a scenario (e.g. Base) and selecting "Create child scenario". Select the new scenario and tick on the relevant alternatives for the selected scenario.

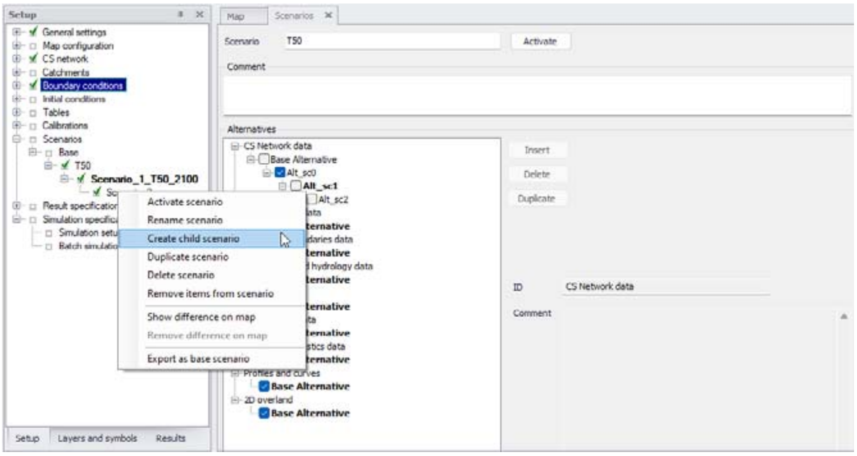


Figure 15.1 Create a new scenario by right clicking on an existing scenario (e.g. Base) and selecting "Create child scenario"

Right-click on a scenario and select "Activate scenario", or click the 'Activate' button at the top of the 'Scenarios' editor, to modify the database for this collection of alternatives.

15.2.2 Alternatives

As described in the previous section, alternatives represent components of scenarios. The various alternatives contain the actual data belonging to a certain data group. Each subsequent alternative only contains information on the differences relative to its immediate "parent", while the rest of the data is inherited from the "parent" through the principles of inheritance.

Grouping of various alternatives belonging to different data groups into scenarios is sometimes subject to limitations, because the data groups have not been formed on the basis of data independency, but rather following the logical data grouping, recording differences according to hierarchy. E.g. An alternative of the "Catchment connections" CS data group, which specifies a catchment connection to node 'A', cannot be used with the alternative of the "Network data" data group where node 'A' has been renamed or deleted. Obviously, the catchment would remain disconnected. It is therefore prudent to plan the scenarios and alternatives before commencing a project, where possible.

15.2.3 Base Data vs. Child Data

When the scenario manager is activated for the first time, the system provides 'Base' alternatives for each data group automatically. The 'Base' data contains the original model database and is the "trunk" for all the alternative branches.

A 'Base' alternative for any data group can be empty, if no data are specified in any of the tables belonging to this data group. E.g. no control rule may be specified for structures, thus leaving the 'Base' alternative of the ‘Control rules data’ empty. So, although the control rules are a part of the 'Base' scenario, it does not necessarily mean that any control rule data are specified.

After making a scenario active (click the "Activate" button in the scenario manager) all the alternatives that are a part of the scenario are automatically made active and can thus be edited. Changes made to the database will be recorded within the alternative for each data group as differences to the parent alternative. If a base alternative is active for a data group, the changes made to this group will apply to this base alternative and will therefore propagate to all child scenarios.

15.2.4 Inheritance Principles

With the inheritance from 'parent' alternatives to 'child' alternatives, some considerations must be kept in mind.

- Making a change to an alternative will affect all descendent ('child') alternatives of that alternative. This means that it will impact all the scenarios where either the alternative or the children of that alternative are applied. The benefit of this feature is that it ensures that if one value needs updating it will be updated in all the scenarios where the alternative is applied (e.g. if a pipe diameter is found to have been incorrectly registered in the GIS data during the course of a project then the pipe diameter can be changed one place only, regardless of the number of scenarios and alternatives that reference to this alternative).
- The chain of inheritance for a certain data record stops where any change (or deletion) of that element has occurred in earlier work. E.g. if a bottom level of a node 'A' has been edited in a child alternative, a later update of the bottom level in the 'Base' will only propagate through the alternative tree until it reaches the alternative containing the first change.

- Adding an element (e.g. a node) in the 'parent' with an ID that already exists in one or more of its descendants ('children') will overwrite the content of the 'child' element
- If adding an element (e.g. pump/link) in the parent that cannot be added to all the children (because some parts may have been deleted/changed there), the element is added where possible and omitted elsewhere.

15.2.5 Data Not Specific to any Alternative/Scenario

There are some data tables which are not included in the Scenario Manager.

These are typically tables containing data of general usability, i.e. data without a reference to the current network - e.g. in MIKE+ CS these include cross sections, parameter sets, etc. These data should be understood as belonging to a general project database.

There are some single record tables containing various parameters (e.g. water quality parameters) that are not part of the Scenario Manager, in order to allow the ability to apply various parameters within the same project.

The data not included in the Scenario Manager can be accessed from any scenario, regardless of the alternatives that make up that specific scenario.

Please note that the computed values are not part of Scenario Manager (all fields ending with _C) and are not automatically re-computed after switching scenarios.

15.3 Managing Scenarios and Alternatives

The Scenario Manager contains two main windows:

- The 'Setup' view, showing the list of scenarios and their relationships
- The ‘Scenarios’ editor, showing the list of alternatives and their relationships.

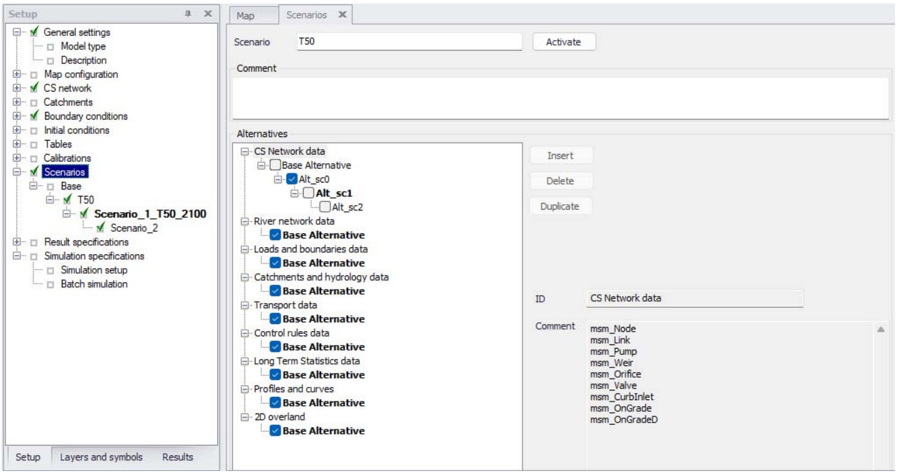


Figure 15.2 Example of the scenario window for the scenario manager - 'Scenarios' in the Setup on the left and 'Alternatives' window on the right.

15.3.1 Scenarios

The scenario section, in the Setup tree view, is used for creating, editing, and managing scenarios. Per default there will be one built-in scenario, i.e. the Base scenario. The Base scenario cannot be edited or deleted. An unlimited number of additional scenarios can then be added to cover the various 'What if' scenarios.

A mouse right click on a scenario enables multiple actions on the scenario.

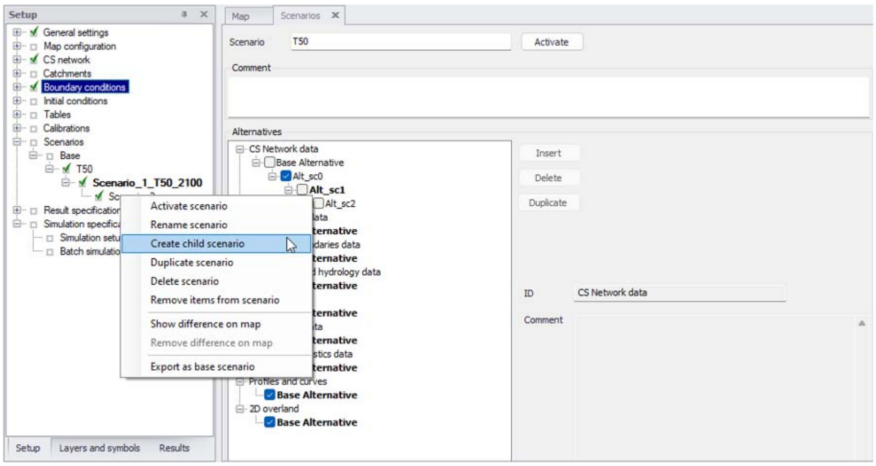


Figure 15.3 Scenario options are visible by a mouse right click on a scenario.

Activate scenario

The activate scenario option will load the scenario, i.e. the project data is manipulated so that all editors contain the appropriate data corresponding to the collated alternatives for the scenario. Depending on the size of the project this may take some time.

Rename scenario

The rename scenario button will make the scenario name editable so it can be easily renamed. Alternatively, the scenario can be renamed in the Scenarios editor.

Create child scenario

The create child scenario option adds a scenario that is a child of the clicked scenario (not to be confused with the active/current scenario), i.e. to begin with, the alternatives of a new scenario will be that of the clicked scenario. A name for the new scenario is suggested by default. The name can be changed by using the rename scenario option.

Duplicate scenario

The duplicate scenario option will make a copy of the selected scenario. This means that all the alternatives that make up the original scenario will be transferred to also be applied to the new scenario. Once the new scenario has been made, the original and the duplicate scenario are edited independently of one another.

Delete scenario

The delete scenario option will remove the selected scenario. The Base scenario cannot be deleted. Note that deleting a scenario will not delete any data as the alternatives hold the data (the scenarios just refer to alternatives). The comments for the scenario being deleted, however, will also be deleted.

Remove items from scenarios

This option is used to remove the alternatives' data for some items (e.g. nodes or pipes), so that they inherit again their settings from the parent alternatives.

For example, if a pipe's diameter has been changed in an alternative, this value is no longer inherited from the parent alternative. Even if the diameter is changed again to be the same as in the parent alternative, MIKE+ still considers the two alternatives to hold different values, therefore a change of diameter in the parent alternative no longer propagates to the edited alternative. So, this option can remove this pipe diameter from the list of changes of the alternative, so that it is inherited again from the parent alternative.

Note that this function only applies to items being updated in the alternative i.e. with different attributes values than in the parent alternative. Items inserted or deleted in the alternative are not removed / restored using this function.

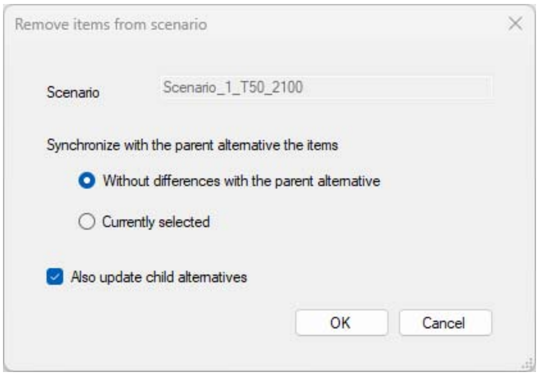


Figure 15.4 The 'Remove items from scenarios' window

This option opens the 'Remove items from scenarios' window, showing the following settings:

- Scenario: this field shows the selected scenario being updated. All alternatives (other than 'Base') used by this scenario will be analysed and possibly updated.
- Synchronization method: two options are available to control which records / items should be synchronized with the parent alternative.
  - Items without differences with the parent alternative: with this method, all items which are updated in the selected scenario are compared with the parent alternative. If an item is found to have exactly the same attribute values as in its parent alternative, then this item is removed from the alternative (i.e. it becomes inherited from the parent).
  - Items currently selected: with this method, all items which are currently selected are removed from the alternative (even if the selected items have different attribute values than in the parent alternative). No action is performed for items which are selected but not modified as part of the selected scenario.
- Also update child alternatives: if this option is selected, items removed from the selected scenario are also removed from the child alternatives. With the method 'Items without differences with the parent alternative', items will be removed from the child alternatives only these alternatives also have the same attributes values as their own parent. With the method 'Items currently selected', all selected items will be removed from all child alternatives.

Show difference on map

The show difference on map option is very useful to graphically display differences between scenarios. Differences are shown on the map view with a color code and will show differences between the activated scenario and the selected (right-clicked) scenario.

The color coding is as follows:

- Green: items added to the active scenario, compared to the clicked one
- Yellow: items edited in the active scenario (at least one of the item's properties has been changed)
- Red: items deleted in the active scenario, but present in the clicked one
- Others: unchanged items in the active and the clicked scenario.

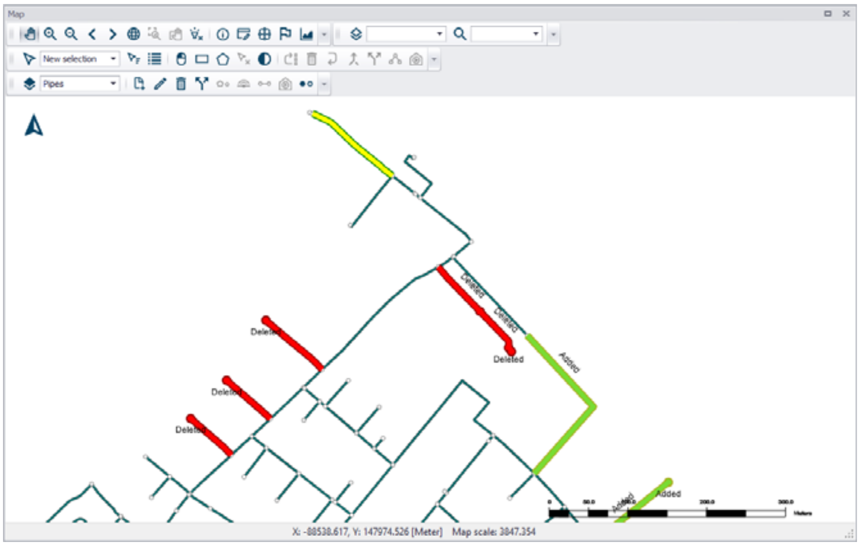


Figure 15.5 Graphical display presenting the differences between scenarios

Note that these differences can also be listed in a report. See following chapter for more information.

Remove difference on map

This clears the differences shown on the map.

Export as base scenario

This option creates a new database, where the clicked scenario becomes the Base scenario. This is useful e.g. in case a past scenario has become the reference situation, and it's no longer required to keep the parent scenarios.

During this operation, all children scenarios of the clicked scenario (the new Base scenario) are kept in the new database. Children alternatives are also kept even if they are not used by any of the children scenarios.

15.3.2 Alternatives

Alternatives can be edited only once the corresponding scenario is made active.

Alternatives can, however, be added to the tree view in the manager regardless of the active scenario. When a scenario is activated, the project data is manipulated so that all editors contain the appropriate data corresponding to the alternatives for the scenario.

The list of tables / editors included in a given group of alternative data can be visualized by clicking on the Base alternative: the list is then shown in the 'Comment' field.

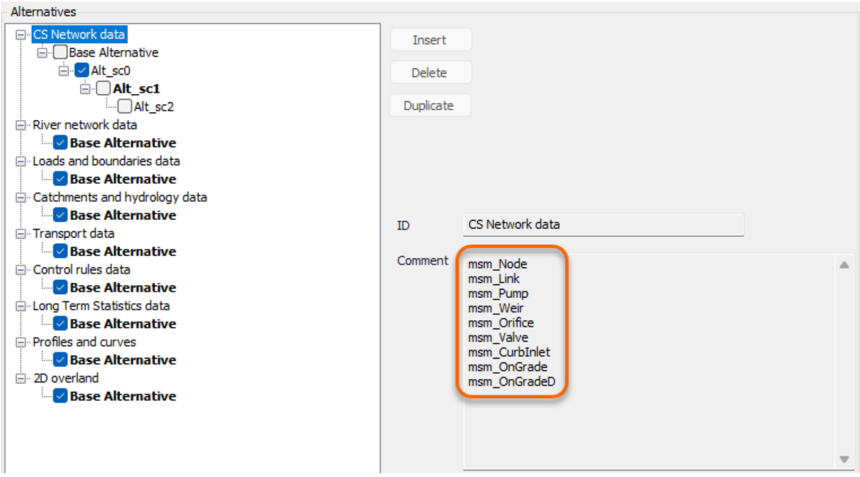


Figure 15.6 Viewing the list of tables in a selected group of alternative data

The alternatives that correspond to the scenario selected in the Setup tree (which is not necessarily the active scenario) are ticked in the tree of alternatives.

The alternatives being currently edited (which are part of the active scenario) are displayed in bold. The name of the active alternative is also shown in the title of the editors, when it is different than the Base Alternative.

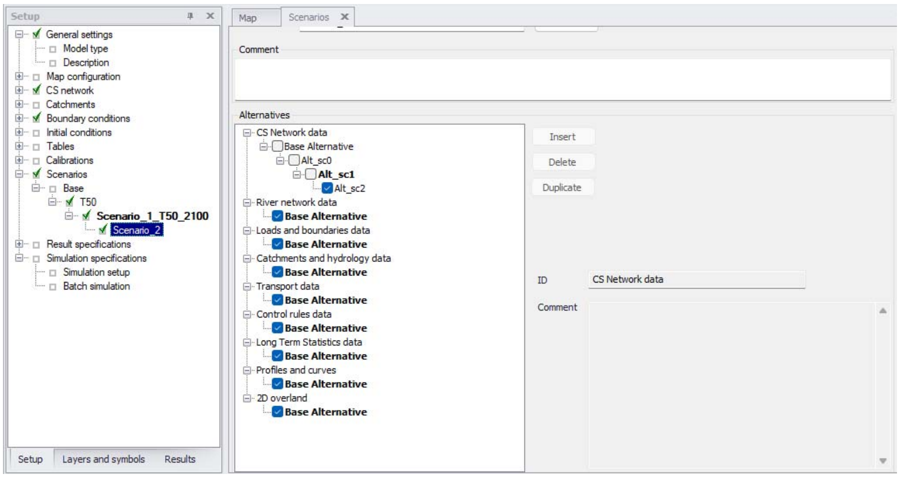


Figure 15.7 Alternatives included in the selected scenario are ticked. Alternatives currently active are shown in bold.

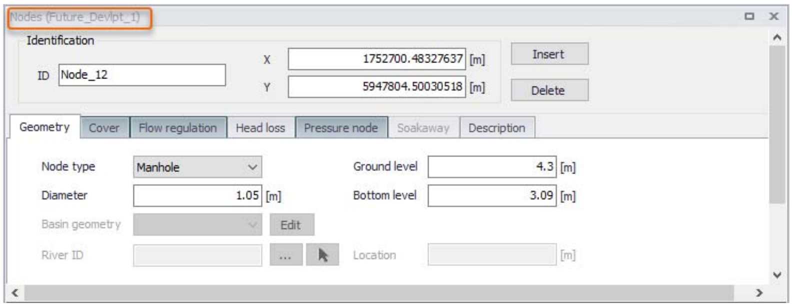


Figure 15.8 Editor showing the name of the alternative being edited

The alternative part of the dialog consists of three buttons: "Insert", "Delete" and “Duplicate” along the right side.

Insert

The insert button adds an alternative that is a child of the selected alternative (not to be confused with the active/current alternative). A name for the new alternative is suggested per default.. By a single left mouse click on the alternative, the alternative can be renamed.

Delete

The delete button will remove the highlighted alternative. The alternatives must be deleted by starting at the end of the trees until the root is reached (the alternatives can only be deleted one by one starting by the latest child). The Base alternative cannot be deleted. Remember: Deleting an alternative will delete the changes made to that alternative.

Duplicate

The duplicate button creates a copy of the selected alternative. The new copy is created as a sibling of the selected alternative, i.e. they share the same parent alternative.

Child alternatives are not duplicated.

ID

This field shows the name of the alternative being selected in the Alternatives tree. The ID can be customised for other alternatives than the Base ones.

Comment

A comment describing the selected alternative can be inserted.

15.3.3 Scenario Simulation

Each simulation, as defined in the 'Simulation setup' editor, is associated with a specific scenario. To be able to run a simulation for a particular scenario, it is therefore necessary to ensure that this scenario is correctly selected in the simulation to be executed.

If a new simulation has to be defined for a scenario, the typical steps to follow are:

- Activate the relevant scenario. This is done by selecting the scenario in the "setup" view, scenarios section, and then clicking the "activate" button available in the scenario manager window.
- Insert a new simulation. From the 'Simulation setup' editor, click on the "Insert" button to insert a new simulation or 'Copy' to copy an existing simulation (e.g. associated with another scenario). The active scenario will appear automatically in the "Scenario" field of the window.
- Adjust the simulation settings as necessary.
- Run the simulation. Once a simulation is created, it can be run for the Scenario ID, even when the active scenario is changed. i.e. when the simulation is run, MIKE+ will automatically activate the relevant scenario and run the model. In this way, multiple scenario simulations can be set up and run.

15.3.4 Example

To investigate how upsizing certain pipes and adding some real time control to the system can affect the performance of the system, start by making two child alternatives: one for the CS network data (as the pipes are a part of this group) and one for the control rules data (as the real time control is a part of that group). Then, create a scenario that applies the new CS network alternative and the new control rules data alternative and then activate this scenario.

Start editing the data in the MIKE+ tables (e.g. upsizing the pipes and adding real time control).

Once the data are edited, insert a new simulation to correspond to the active scenario. Run the model and compare the results to the original setup to see the effect of the changes.

You can also choose to make a new scenario that contains e.g. the network alternative (but not the control rules alternative), to see what change in performance the pipe upgrades alone will have.

15.3.5 Reporting Changes

When setting up multiple alternatives and scenarios one of the most important aspects is keeping track of the changes that have been done. The Model and Result Report tool (In the MIKE+ ribbon, select Tool |Model and Result Report) can be utilised to track and document changes made between scenarios. (Refer to Chapter 20.14 Reports *(p. 473)* for further details). The reports are all in XML format but can also be imported into a word document.

In the Model and Result Report tool, create a new template and select the Content to be compared Scenario section. Click "Run" to present the comparison. The table can be exported to a variety of formats. E.g. Word, Excel, *.PDF, *.XML ,etc. The report style can be utilise the default 'MUReport' format, or an imported style.

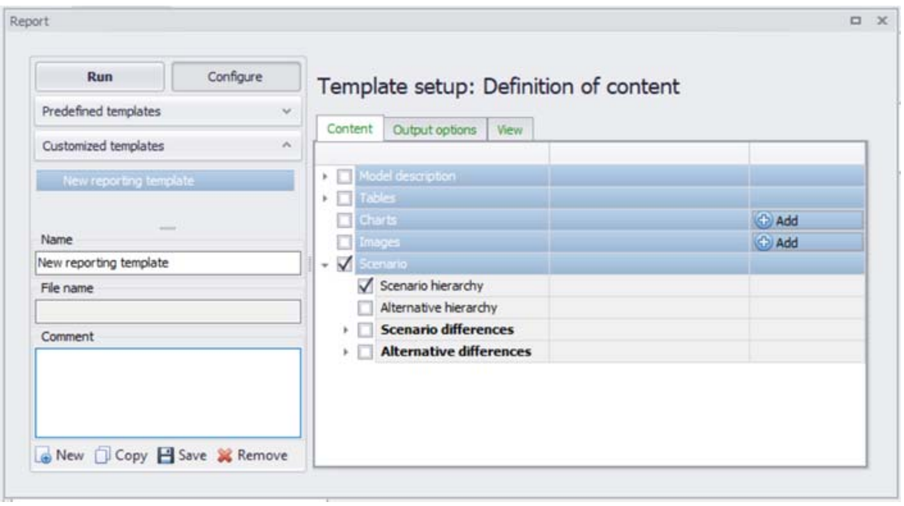


Figure 15.9 The Model and Result Reporting tool can be utilised to keep track of scenarios and alternatives.

Within a report, color coding is used to signify the origin of the record:

- White - original record, no changes

- Green - record added
- Yellow - record has been changed (updated)
- Red - record has been deleted

Scenario hierarchy

Will create a table with scenario IDs, active scenario, parent of the scenario and comments.

Alternative hierarchy

Will create tables for each data group with alternative IDs, active alternative, the parent alternative, a comment, and the scenario the alternative is associated with.

Scenario differences

Scenario #1 and #2 are compared to each other, selected from a drop-down list of all the scenarios in the model. Comments in the scenario specification can be included in the comparison as an option. To narrow the comparison, specific data groups can be selected, and a choice can be made whether or not to present a comparison of everything in a report or "only include changed values that differ" between scenarios.

Alternative differences

When comparing two different alternatives, the data group to be compared must be chosen from a drop-down list of all data groups. Then two alternatives from within the specified data group can be selected to be compared to each other, selected from a drop-down list of all the alternatives within the data group. Comments in the alternative specification can be included in the comparison as an option. and a choice can be made whether or not to present a comparison of everything in a report or "only include changed values that differ" between alternatives.

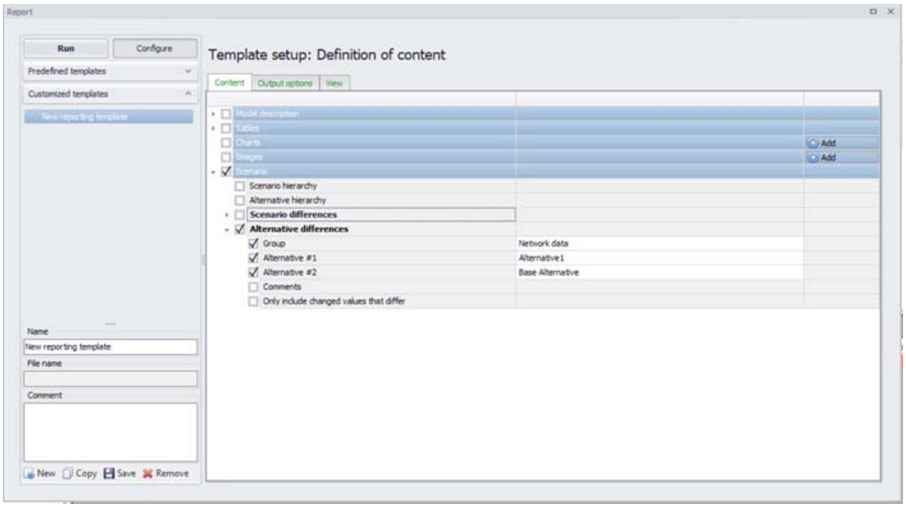


Figure 15.10 Reporting differences between Alternatives using the Model and Result reporting tool

15.4 Step-by-Step Guide to Creating a Scenario

1. In the setup view, go to the "Scenarios" editor and click on an existing scenario (e.g. 'Base') to display the editor
2. Create a child scenario by right clicking on the existing scenario (e.g. ‘Base’) and selecting 'Create child scenario'.
3. In the ‘Scenarios’ editor, select the alternative group that you wish to add an alternative to and press the 'Insert' button in the Scenario editor window
4. You can now rename it and/or continue to make alternatives
5. Once you have created the alternatives that you need, highlight the scenario you created and tick on the alternatives that you wish to include in the scenario, one for each data group;
6. Activate the scenario that you wish to work with (right click on the scenario ID in the setup view and select 'Activate scenario' or click on the 'Activate' button next to the ID of the selected scenario in the Scenario editor window). The activated scenario is displayed in bold font. Equally, all the alternatives that relate to the active scenario, are displayed in bold in the list of alternatives.
7. Edit the model, making sure to only edit the data associated with the new activated alternatives.
8. Create a new simulation for the active scenario (Simulation, Simulation setup in the ribbon view, or via the "Setup" view, Simulation, Specifications, Hydrodynamic simulation).

9. Run the new scenario and compare results to other scenarios, e.g. using the 'Compare' option in the 'Results' tree.

16 Submodel Manager

16.1 Introduction

The 'Submodel Manager' tool is used to split and combine models in an easy manner. The tool was designed for cases where a detailed model exists within a large area (city-wide model), while the issue to be analysed by the model is only in one small area. Therefore, a simplified model can be created (e.g. using the simplification tool) and then parts of the detailed model and simplified model can be combined. i.e. detailed area of interest with the remainder of the model being simplified to account for upstream and downstream effects.

An additional use for the submodel manager tool is when distributing modelling resources. Modellers can work on different sections of a larger model and then use the submodel manager to combine the final model together.

Based on user specified polygons, different models are split into submodels of the network. The desired submodels can be merged into a new model containing one version of an area from the original models e.g. a detailed or simplified submodel. The tool has two main functionalities:

- Extract submodels
- Merge submodels

An example of the submodel manager concept is shown below Figure 16.1, where two original models are split into submodels based on polygons and then a different combination of submodels are merged together to create a new model.

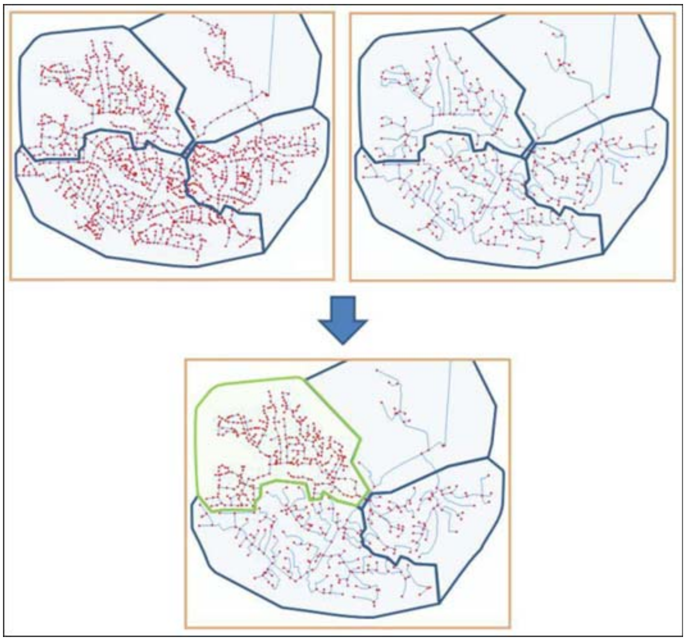


Figure 16.1 Submodel manager concept

A submodel will contain all nodes inside the submodel polygon and all elements connected to these nodes even if they are outside the polygon (e.g. catchments). Areas that cross the borders defined by the polygons must be identical in the models (detailed and simplified version of the model area) to be able to be merged later. It is possible to create a selection file of elements that exist in more than one submodel. Before merging, it is important to make sure IDs between the models are unique.

The two options are detailed in the following sections. They can be executed separately or in sequence.

16.2 Extract Submodels

A model can be extracted from a base model based on a polygon background layer in MIKE+. Every polygon must contain a unique ID which will be used as a reference to the individual submodel. The background layer must cover the entire model area and the polygons should only intersect with pipes, structures and connections (catchments, demands etc.).

A submodel will contain all junctions, pipes etc. inside the polygon and all elements connected to these junctions. e.g. a zone located outside the submodel polygon will still be included in the submodel if it is connected to a junction located inside the submodel area.

In order to extract the submodel, a polygon layer and submodel ID must be provided to specify the extent and name of the submodel. The directories for the detailed and simplified submodels in the dialog specify where the created submodels will be located. If the directory is specified without a full path the directory is a relative path to the database currently loaded in MIKE+.

In the 'Extract' tab the detailed and simplified models must be specified, see Extract Submodels. The selected polygon shape under the selection tab defines the area for submodel #1, which is the submodel to be extracted. Submodel #2 is given by region outside the selected polygon. To create the submodel, add a name for the selection list and click on ‘Create and clip submodels’.

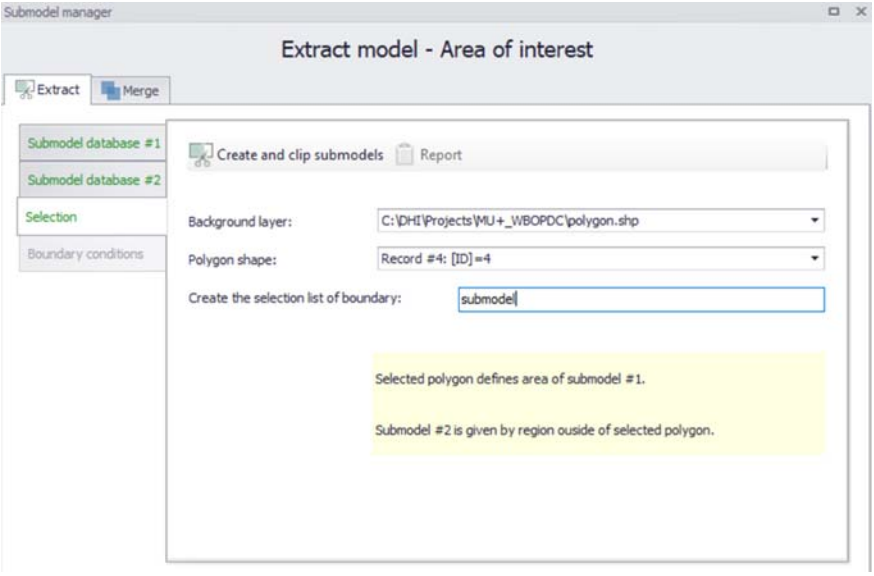


Figure 16.2 Extract Submodels

It should be noted that the created submodels do not function as individual models but parts for a complete model. Hence a submodel may not be valid for a simulation without further manual editing.

16.3 Merge Submodels

When merging submodels the same polygon layer that was used to extract sub models should be loaded into MIKE+. For each polygon, the user needs to specify which model to use within the area. E.g. from the detailed or simplified model.

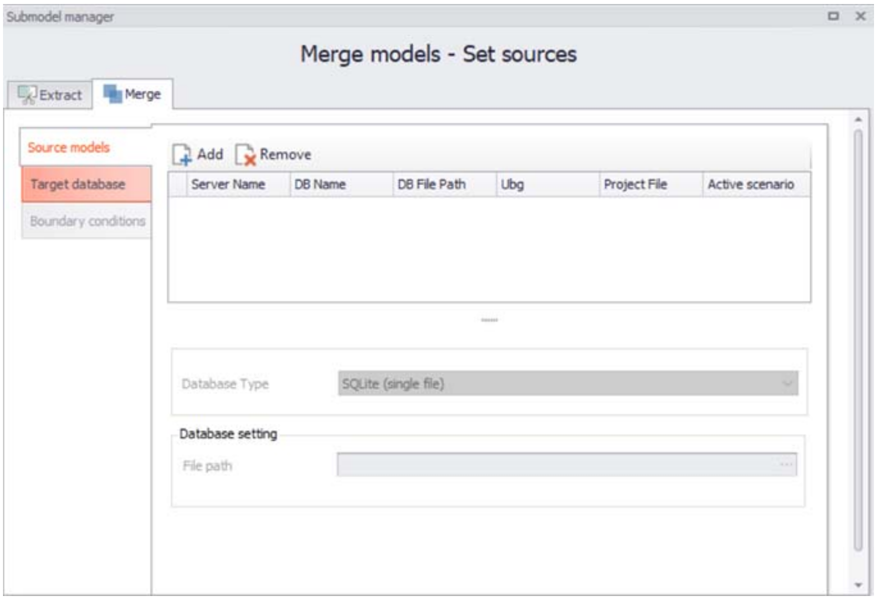


Figure 16.3 Merge Submodels

In order to merge submodels, Click on Merge within the sub model manager dialog as shown in Figure 16.3. The source models must be added (Add button), along with a new file path / name. The target database must be specified. Then click on 'Create and Merge' and a report file can be produced and exported through the Report tab. The report will include information of when the model was created and which version of each submodel was used.

If the location of the submodels is specified without a full path the directory use a relative path to the database currently loaded in MIKE+.

When a new model is created, it is based on a combination of submodels originating from the detailed model and from the simplified model. In order to ensure that the individual submodels fit together in both versions there are some limitations to which changes are allowed between the detailed and simplified model.

17 Versions Management

17.1 Principles and Definitions

The Versions Management tool is designed to support a cost-effective model maintenance. The tool can identify, report and visualize differences between any two versions of a model setup, as well as it can automatically update any model with the identified differences. Additionally, the Versions Management tool facilitates the organization of various model versions into a tree-like dependency structure that reflects the actual models' mutual relations and evolution.

As such, this labor-saving tool can be used for a variety of tasks and operations in the model's life cycle:

- **Managing model versions and instances:** The versions Management tool allows for the creation and maintenance of the models "Version tree", as a structured format for keeping track of various model versions dependencies and evolution. Modelers typically develop and maintain several versions of models representing the modelled system. The model versions may differ by the level of modelling detail, by the functionality, by geographical extent, etc. All these versions have in common the fact that they are built on the data from the same source and that they describe the same system.
  Each of the model versions may contain several instances. Instances are "versions of versions", typically reflecting certain model version at different times, e.g. non-updated and updated instance of the same model version.
  Dependencies and evolution of model versions and instances is represented by the "version tree" with the "master version" in the root and other versions emanating from the root as "children" or "grandchildren". Position of any model version in the version tree defines its origin and relation to its predecessor ("parent") or successors ("children").
  The version tree may be established ad-hoc by a modeler, as a helping structure to keep the models organized. Or, it can be created, maintained and strictly controlled by the corporate model administrator, authorized to release the models for editing, to append new versions and to perform updates.

- **Identifying, reporting and visualizing changes in the asset data system that is used as a source of model data:** This is relevant when the model data are maintained outside the MIKE+ model e.g. in an asset database. The data in the asset database are kept up to date with the changes in the actual system by adding new data records, by editing already existing data and by deleting obsolete data. Documenting these changes by importing the relevant data from the asset database into MIKE+ with the 'Import and export' tool at regular intervals and analyzing the differences between the current and any previous imported data set, helps to keep track of the database updates and provides QA feedback for the updating process.
- **Applying the identified data changes in the asset database for update of any model version developed from the asset data and documenting the updates:** Working MIKE+ models are frequently created by importing the relevant data from asset database into MIKE+ and subsequently performing "manual" data modifications as appropriate. Such modifications can include addition of data not available in the asset database, supplementing the missing attribute values, reconfiguration of complex structures, removal of small pipes or an area which is outside the area of interest, etc. This is a time-consuming process that requires a solid expert know-how. If these models are to maintain their value, they must be kept up to date with the changes in the actual system they represent, i.e. with the changes in the asset database. A manual updating process is costly and complicated, which frequently results in models that are outdated and as such not representative for the current system. The version management tool automates the updating process by analyzing the differences between the previous and the current version of the asset database (actually, between the previous and the current imported data set) and applying these differences to the working model. The tool identifies and resolves possible conflicts on individual data record level by letting the modeler to accept or reject the data update.
- **Applying the identified data changes in any MIKE+ model setup for update of any other model version and documenting the updates:** Various MIKE+ model versions are typically created by making a copy (a "child") of an existing model version and editing the copied MIKE+ database. A new version may have a different purpose and functionality, but essentially it represents the same system as its "parent" model version with the bulk of same data. When a "parent" model is edited or updated (e.g. by applying the updates from the asset database), these updates must also be applied to any "child" models as well, to preserve consistency. This can be achieved in the same way as the previous task (updating of a MIKE+ model from the asset database), with only difference that the differences are analyzed for the current and previous instance of the "parent" model and then applied to a new instance of the "Child" model.

- **Identifying, reporting and visualizing differences between any two model setup versions or instances:** Different MIKE+ model versions and instances may arise as a consequence of a continued modelling process. It is often relevant to identify and document the differences between two MIKE+ databases. For example, comparing today's and yesterday's instance of a MIKE+ database, provides documentation for the model editing performed during the time interval that separates their date & time. Similarly, comparing a basic version of a model with its extended version that includes control rules provides a documentation on the control data that have been added to the controlled version.

Note: Operation of the Versions Management tool relies on unchanged data identifiers. Renaming the data records (i.e. changing their IDs) either in the asset data system or in any MIKE+ model version breaks definitely the links between the corresponding data and prevents correct functionality of the versions management tool.

In the following chapters, the definitions below are used:

- **Asset data system:** this is the original source of model data, before they are imported into MIKE+. It usually includes a main asset database, plus some supplementary data sources in various formats. The asset data are maintained continuously to represent the current physical system in the best possible way. It is this continuous evolution of the asset data that needs to be included in the existing models.
- **Master database:** this is the main MIKE+ database, obtained after importing the data from the asset data system. It is usually not a working model used to run simulations. When an updated version of the asset data system is available, a new master database can be created by importing the latest asset data in a new MIKE+ database, using the 'Import and export' tool.
- **Model version:** in this context, a model version is a modified version of the master database or any other MIKE+ model version. Multiple model versions may exist, each tailored to a specific purpose.
- **Model instance:** an instance represents a master database or any model version database at a given point in time. The master database and model versions usually have multiple instances, each instance reflecting the asset or model data at a specific date. When a model database is to be updated, its former instance is kept unchanged, and a new instance is created with the updates. New instances of the master database are created by repeating the import from the asset data system. New instances of model versions can be created automatically using the Versions Management tool.

The core purpose of the Versions Management tool is to create a new (updated) instance of a model version, by finding and applying the latest updates from the master database.

Note: The Versions Management tool is designed to work with the Base scenario only. Differences that may arise in other scenarios are not identified and reported. Updates are performed on the data in Base scenario only, and their propagation to other scenarios is controlled by the functionality of Scenario Manager.

17.2 Model versions and instances management

The 'Versions management' tool is accessible from the ribbon, in the 'Tools' tab. It can be used even if no model database is opened.

Its main window allows organizing the different model versions as well as their instances.

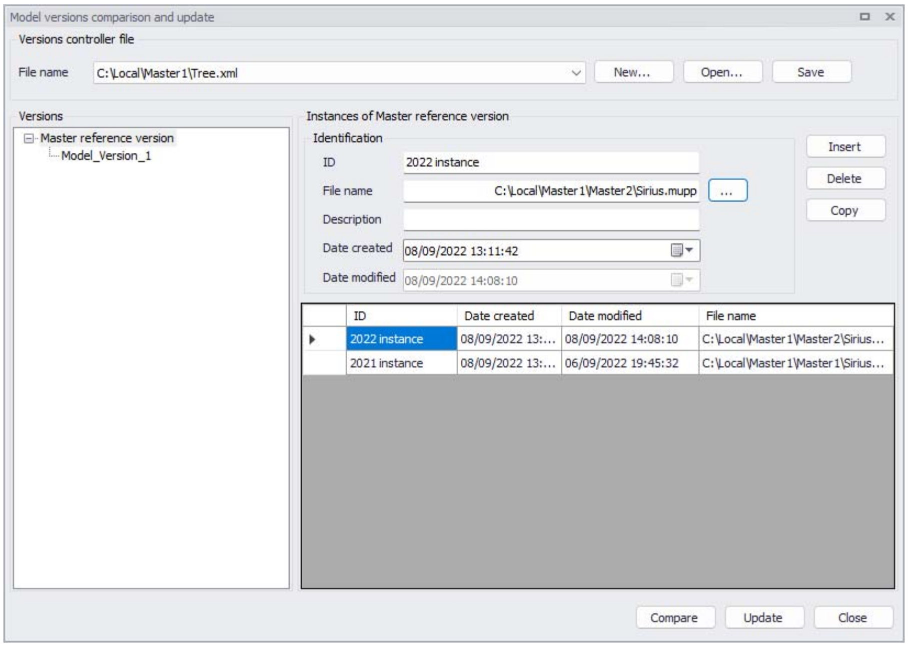


Figure 17.1 The main window of the 'Versions management' tool

17.2.1 Versions controller file

At the top of the window, a .xml file must be selected. This file stores all data defined in this window, related to model versions and instances. It is possible to work with several files, in case multiple sets of model versions should be used one after the other.

Recently-used files can be selected from the drop-down list. The buttons to the right are also used to manage this file:

- New: creates a new file. When a new file is created, the user is asked to select an existing master model database. The versions tree will then initially contain only this master database with a single instance, and other model databases will have to be inserted manually afterwards.
- Open: opens an existing file to show its list of model versions and instances.
- Save: saves the changes to the list of model versions and instances to the file.

17.2.2 Versions

The left-hand side of the window contains a tree view representing the hierarchy between the various model versions and their master database.

The first version at the root of the tree is always the master model database. Any number of versions can be added to the tree, and each version can have its own "sub-versions".

Several options are available in the context menu (right-click on a selected item in the tree):

- Create new child: inserts a new child below the selected item. Two options are available:
  - Create the new child as a copy of the last instance of its parent. After selecting this option, the user is asked to provide a location and file name for the new child's model database.
  - Associate the new child to an existing model database. After selecting this option, the user is asked to select the existing model database.
- Rename: renames the model version in the tree view.
- Delete: removes the selected item from the tree and all its children versions. It does not delete the associated MIKE+ model files.
- Compare: opens the tool to report differences with another model version.
- Update: opens the tool to update the model version using previous instances and updates to the master database.

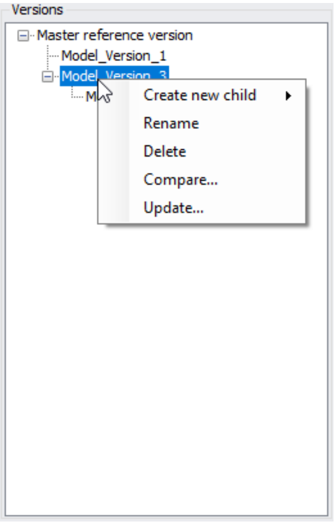


Figure 17.2 The tree view to manage model versions and their hierarchy

17.2.3 Instances

The right-hand side of the window contains a list of instances for the selected version in the tree, shown in the lower table. Each model version must contain at least one instance, and each instance is associated to a MIKE+ model database.

The following details are shown above the table for the selected instance:

- ID: the identification text of the instance.
- File name: the path to the corresponding MIKE+ model setup. It can be selected using the '…' button to the right.
- Description: an optional descriptive text.
- Date created: the date of creation of the instance. It can e.g. be used to sort the instances chronologically in the table.
- Date modified: the date at which the selected .mupp file was last saved.

The following buttons can be used to manage instances:

- Insert: creates a new and empty instance in the list. The corresponding model database must be selected manually afterwards. This option should be used when the model database for the instance already exists.
- Copy: creates a new instance in the list, being a copy of the selected instance in the table. The new created instance can then be modified manually. When using this button, the user is asked to provide a location and file name for the new instance's model database.

- Delete: removes the selected instance from the list. Note that the corresponding MIKE+ model database is not deleted.

Note: After running the 'Update' tool, a new instance is automatically inserted and associated to the created model database.

17.3 Compare tool

The 'Compare' tool is either opened from the 'Compare' button in the main window, or from the context menu of the tree view.

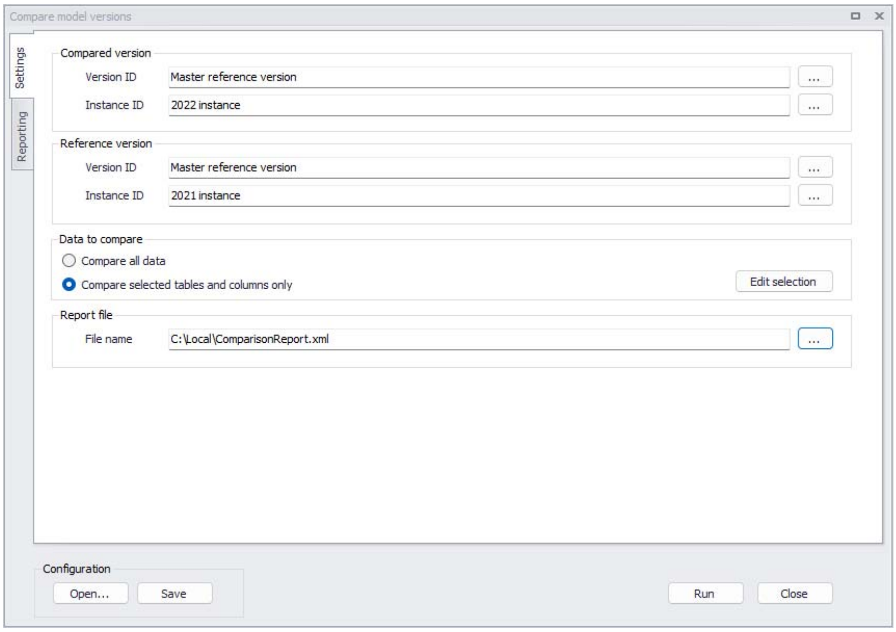


Figure 17.3 Comparing two model versions

The purpose of versions comparison is to identify differences between two data sets. Differences are listed in a report and can be shown on the map. The comparison tool does not perform any change to the model databases.

The differences are categorized as:

- Added: data records that are present in the compared version, but do not exist in the reference version.
- Updated: data records that are common to both versions but have at least one attribute value or geometry that is different.
- Deleted: data records that do not exist in the compared version, but are present in the reference version.

Settings

Before running the comparison, the two model versions must be selected: the compared version and the reference version; and for each of them the instance to be used must also be selected.

Under 'Data to compare', two options are available to control the scope of the comparison:

- When selecting 'Compare all data', the comparison tool compares all database tables, and all columns in these tables.
- When selecting 'Compare selected tables and columns only', the comparison tool compares only the tables and columns selected from the 'Edit selection' button. This option can avoid reporting differences of no interest, when they are related to data types not (or no longer) used in the project, or when users consider that changes in some columns are not relevant for the comparison.

When the latter option is used, a selection window shows up, with a default list of tables to compare. Tables can be added or removed using the 'Add' and 'Remove' buttons on the right. For each table in the list, the table's name can be edited from a drop-down menu or by typing the table name. Each table can be expanded to access:

- The list of columns in the specific table: from this list, it is possible to tick only the columns which should be compared, while the others should remain unselected. All columns from the selected table are listed, except the MUID column which is always compared and used as comparison key between the two model databases.
- A filter expression: when this filter is active, an expression must be defined in the second column. Clicking the expression field (in the second column) will automatically open the Expression Editor *(p. 533)*, where the expression used as filter can be entered using the available variables or functions. Only the records fulfilling the specified expression, in both the reference and compared versions, will be compared. This filter can be used, for instance, to compare a limited geographical area (filtering nodes by coordinates) or filtering records with specific IDs, etc. Note that the filter only applies to items existing in both the reference and the compared versions, therefore all items added or deleted in the compared version will be listed in the report even if they don't fulfill the expression.

The 'Save' button on the right allows saving the configuration of this selection (saving the list of included tables, columns and filter expressions), for later re-use in future comparisons. The 'Load' button is then used to load this configuration file.

During the comparison process, if the option 'Compare selected tables and columns only' is active, the tool will only compare and report differences for tables which are selected in this window, and for columns also selected and fulfilling the filter, if any.

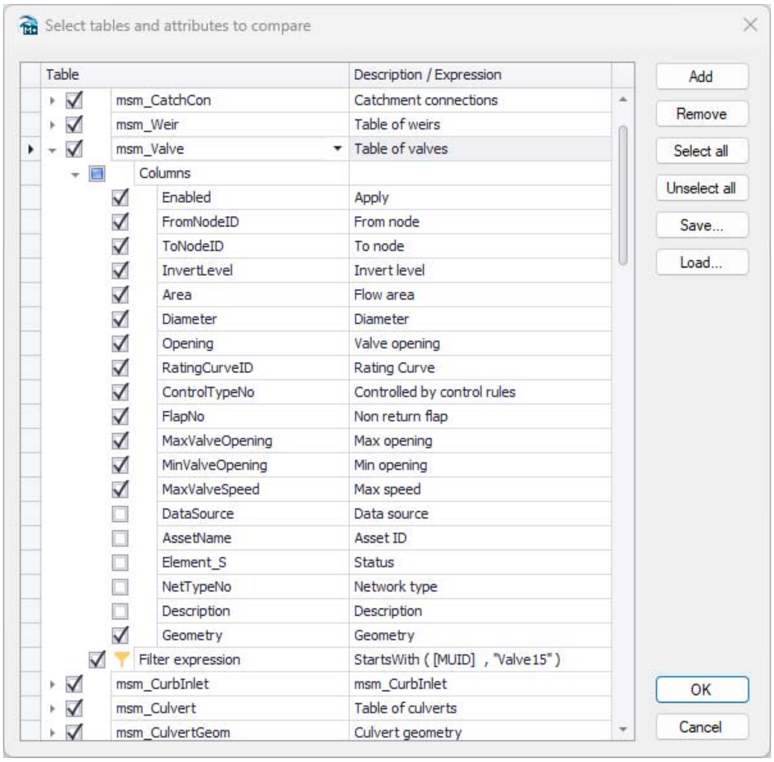


Figure 17.4 Specifying tables and columns to compare and filter expressions

The tool compares records based on their unique ID, therefore the item IDs should ideally not be changed between the two compared databases in order to produce a relevant report. If an item has been renamed in the compared version, the original item (with original ID) from the reference database will be reported as deleted, whereas the new item (renamed) from the compared database will be reported as added, even if they are identical otherwise.

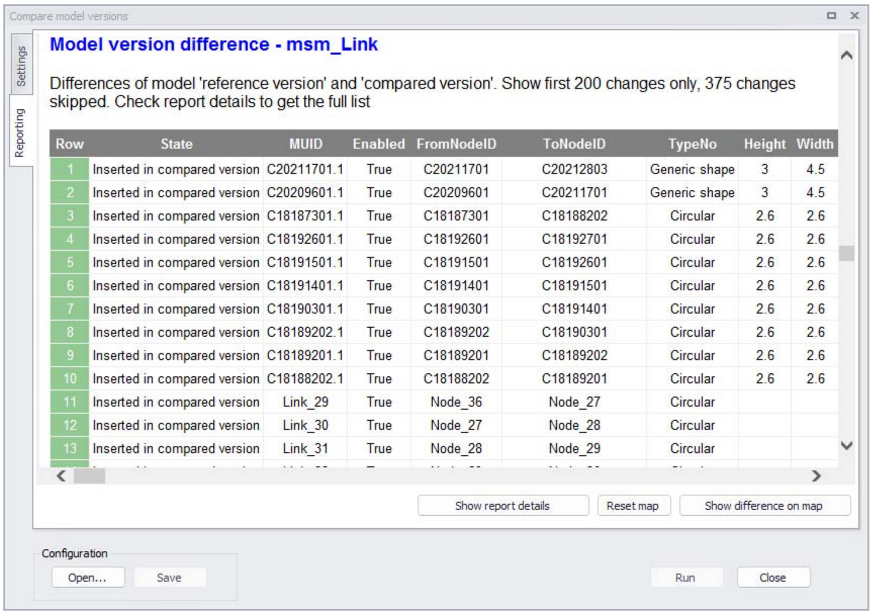
The path to the report file must also be specified. This report will automatically show up in the 'Reporting' tab once the comparison is completed.

Reporting

When the comparison is executed, the report appears in the 'Reporting' tab. For all compared tables, it will list the differences:

- Added records are highlighted in green.
- Updated records are highlighted in yellow. Updated columns are also highlighted in yellow (if the option 'Compare selected tables and columns only' is active, only the value of the compared columns is reported).

- Deleted records are highlighted in red.



Row	State	MUID	Enabled	FromNodeID	ToNodeID	TypeNo	Height	Width
1	Inserted in compared version	C20211701.1	True	C20211701	C20212803	Generic shape	3	4.5
2	Inserted in compared version	C20209601.1	True	C20209601	C20211701	Generic shape	3	4.5
3	Inserted in compared version	C18187301.1	True	C18187301	C18188202	Circular	2.6	2.6
4	Inserted in compared version	C18192601.1	True	C18192601	C18192701	Circular	2.6	2.6
5	Inserted in compared version	C18191501.1	True	C18191501	C18192601	Circular	2.6	2.6
6	Inserted in compared version	C18191401.1	True	C18191401	C18191501	Circular	2.6	2.6
7	Inserted in compared version	C18190301.1	True	C18190301	C18191401	Circular	2.6	2.6
8	Inserted in compared version	C18189202.1	True	C18189202	C18190301	Circular	2.6	2.6
9	Inserted in compared version	C18189201.1	True	C18189201	C18189202	Circular	2.6	2.6
10	Inserted in compared version	C18188202.1	True	C18188202	C18189201	Circular	2.6	2.6
11	Inserted in compared version	Link_29	True	Node_36	Node_27	Circular		
12	Inserted in compared version	Link_30	True	Node_27	Node_28	Circular		
13	Inserted in compared version	Link_31	True	Node_28	Node_29	Circular		

Figure 17.5 Reporting differences between two model versions

This report will show a maximum of 200 differences per table. Pressing 'Show report details' will open another window with the full list of reported differences and with additional options to export the report to other formats.

Note: after closing the tool, it is possible to re-open the .xml report file from the 'Model and result report' tool available in the ribbon (select the file from the 'View' tab).

Pressing the button 'Show difference on map' will highlight on the map the modified items, with the same color code. Note that this requires that the compared model version is opened in MIKE+ beforehand.

The button 'Reset map' will clear these highlights on the map.

Buttons

Use the 'Save' button to save configuration settings in *.xml format for later reuse 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.

Note: If one of the two compared databases contains a network simplified using the 'Network simplification' tool, the comparison can take into account the simplification information, in order to e.g. report pipes as "Merged" instead of "Deleted". This requires that the model database with the simplified network, which stores simplification information (relationships between IDs of original and merged pipes), is opened when the 'Compare' tool is executed. It also requires, if the option 'Compare selected tables and columns only' is active, that both nodes and links tables are included in the comparison with all their columns.

17.4 Update tool

The 'Update' tool is either opened from the 'Update' button in the main window, or from the context menu of the tree view.

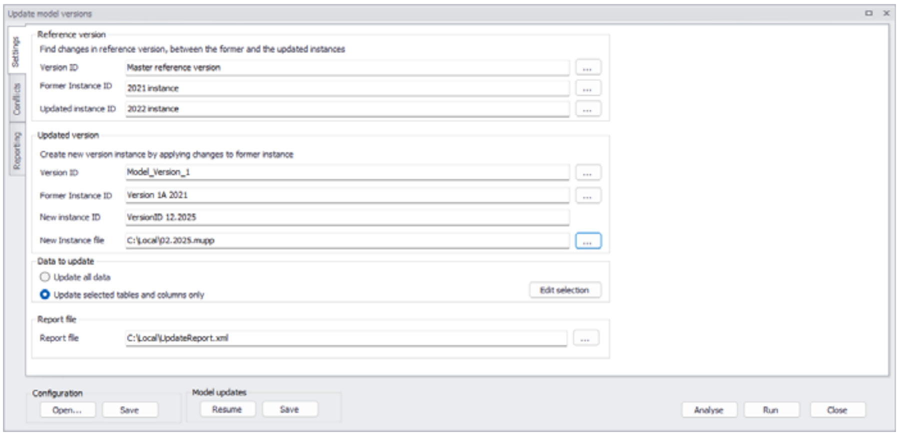


Figure 17.6 Creating an updated instance of a model version

The purpose of versions updating is to identify updates (differences) between two data sets (typically two instances of the master database) and then apply these updates to a model version. This creates a new instance of the model version.

The update is performed in successive steps:

1. Identify updates, by finding differences between two different instances of a reference version (e.g. two instances of a master database).
2. Identify changes to the model version to be updated, compared to the former reference version it was created from.
3. Compare the updates with the changes made to the model version. If the update does not conflict with changes made to the model version, the update will be performed when creating the new instance. If the update does conflict with changes made to the model version, this update will be listed as a conflict and different actions may be applied.

4. List all conflicts. For each of them, the user can decide which action to perform.
5. Update the model version in a new instance, using the selected actions for the conflicts.

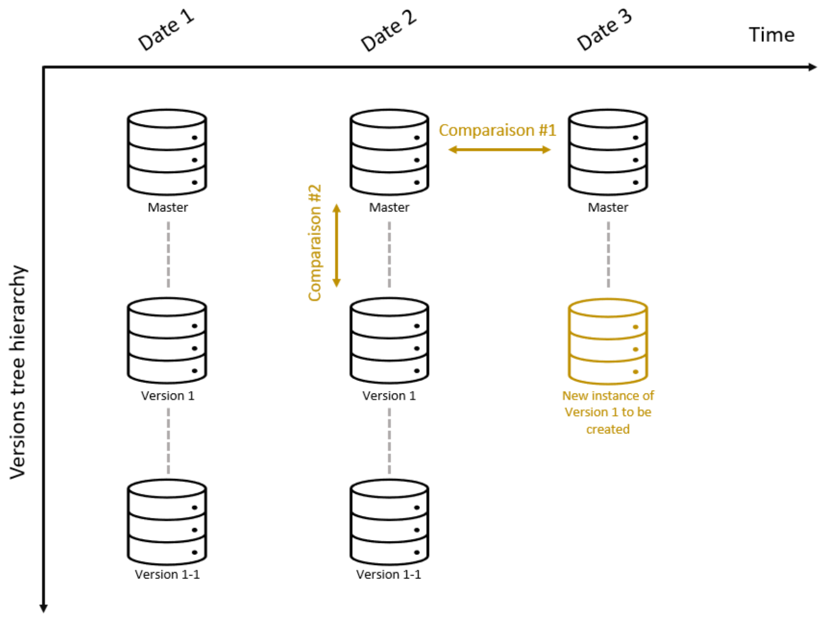


Figure 17.7 Creating a new instance of a model version using previous instances and updates to the master database

Settings

In the Settings tab, three input model databases must be selected:

- The two instances of the reference version (typically the latest and the former instances of the master database). They are selected in the 'Reference version' group.
- The former instance of the model version, which is to be updated. It is selected in the 'Updated version' group.

Besides, a path to an output model version must be specified in the 'New instance file'. This is the updated instance which will be created by the tool for the updated model version. A 'New instance ID' is also required: this is the ID which will be shown in the list of instances, in the main window.

Under 'Data to update', two options are available to control the scope of the update:

- When selecting 'Update all data', the update tool compares and updates all database tables and all columns in these tables.

- When selecting 'Update selected tables and columns only', the update tool compares and updates only the tables and columns selected from the 'Edit selection' button. This option can avoid identifying differences of no interest, when they are related to data types not (or no longer) used in the project, or when users consider that changes in some columns are not relevant for the comparison.

When the latter option is used, a selection window shows up, with a default list of tables to compare. Tables can be added or removed using the 'Add' and 'Remove' buttons on the right. For each table in the list, the table's name can be edited from a drop-down menu or by typing the table name. Each table can be expanded to access:

- The list of columns in the specific table: from this list, it is possible to tick only the columns which should be compared and updated, while the others should remain unselected. All columns from the selected table are listed, except the MUID column which is always compared and used as comparison key between the different model databases.
- A filter expression: when this filter is active, an expression must be defined in the second column. Clicking the expression field (in the second column) will automatically open the Expression Editor *(p. 533)*, where the expression used as filter can be entered using the available variables or functions. Only the records fulfilling the specified expression, in both the reference and compared versions, will be compared and updated. This filter can be used, for instance, to compare a limited geographical area (filtering nodes by coordinates) or filtering records with specific IDs, etc. Note that the filter only applies to items existing in all model versions, therefore all items marked as 'added' or 'deleted' will be listed in the report even if they don't fulfill the expression.

The 'Save' button on the right allows saving the configuration of this selection (saving the list of included tables, columns and filter expressions), for later re-use in future comparisons. The 'Load' button is then used to load this configuration file.

During the update process, if the option 'Compare selected tables and columns only' is active, the tool will only compare, update and report differences for tables which are selected in this window, and for columns also selected and fulfilling the filter, if any. Tables and columns which are not selected will be kept unchanged (same as in the former model version being updated).

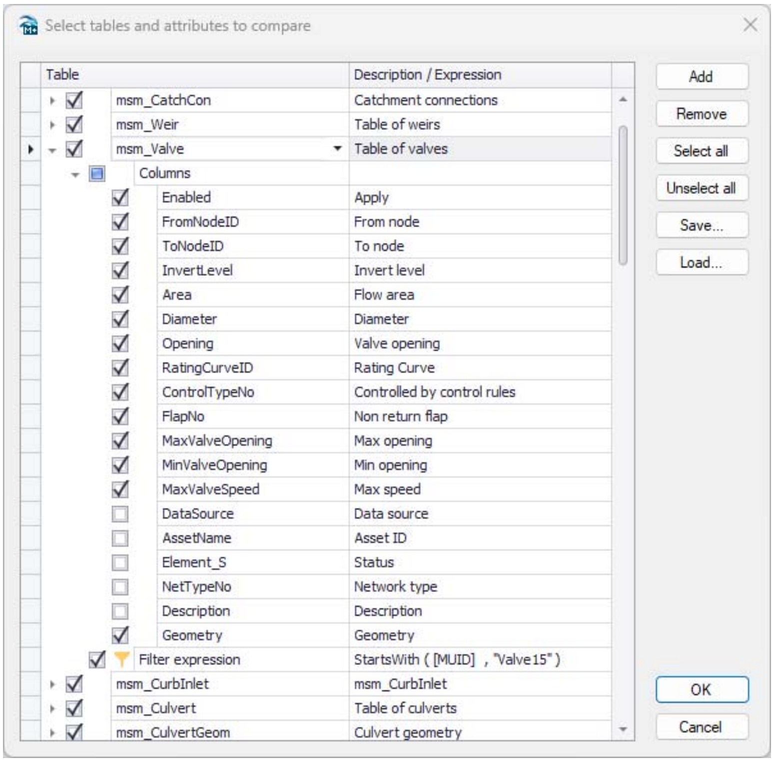


Figure 17.8 Specifying tables and columns to compare and filter expressions

Finally, the path to the report file must also be specified. This report will automatically show up in the 'Reporting' tab once the update is completed.

When settings are correctly configured, pressing the 'Analyse' button will analyse the input databases and list all the conflicts in the 'Conflicts' tab.

Conflicts

The table in the 'Conflicts' tab shows all updates (identified by comparison between the two instances of the reference version) conflicting with the changes made to the model version being updated (compared to the former reference version). For these conflicts, multiple actions are possible. This table should therefore be reviewed carefully and the action for each conflict should be modified as necessary.

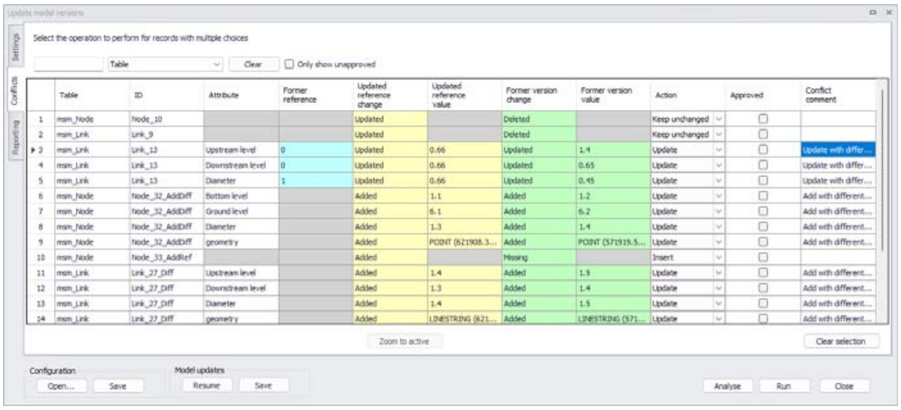


Figure 17.9 The list of identified conflicts

The table of conflicts contains the following columns:

- Table: the name of the database table containing the item
- ID: the MUID of the compared item
- Attribute: when an item is added or updated in both the new reference version and the model version but with different attribute values, this column lists all the attributes with different values
- Former reference: the value of the attribute in the former reference version, shown when it has been updated in the new reference
- Updated reference change: it shows the changes in the updated instance of the reference version, compared to the former reference:
  - Updated: data records that are common to both instances but have at least one attribute value or geometry that is different.
  - Added: data records that are present in the updated instance, but do not exist in the former instance.
  - Deleted: data records that do not exist in the updated instance, but are present in the former instance.
- Updated reference value: the value of the attributes in the updated instance of the reference version. This is shown only when the conflict is handled for individual attributes (not when e.g. the entire item with all its attributes is to be updated or deleted).
- Former version change: it shows the changes in the version to be updated compared to the former reference version:
  - Added
  - Deleted
  - Updated
  - Missing
  - Unchanged

- Former version value: the value of the attributes in the model version to be updated. This is shown only when the conflict is handled for individual attributes (not when e.g. the entire item with all its attributes is to be updated or deleted).
- Action: this shows the selected action to be applied for each record. Depending on the changes identified in the reference version and in the model version, the possible actions are:
  - Update
  - Insert
  - Delete
  - Keep unchanged (remains identical to the previous instance of the model version being updated)
- Approved: an optional check box which can be used to track the progress of the reviewed and approved actions. It has no effect on the actual update process, i.e. all actions from the table will be applied to the created new model version even if all records are not approved.
- Conflict comment: optional description of the conflict.

The different cases which can be encountered are listed in the following table. Note that only the cases where different actions can apply are listed in the table of conflicts. Therefore, this table of conflicts does not show all updates to be performed to the model version.

Table 17.1 The list of cases and their possible actions

Change in reference	Change in model	Comparison	Possible action	Use case	Shown in conflict table
Added	Added	Unchanged	Keep unchanged	User has already inserted a record to the model database during the modelling process. The master database has subsequently been updated by inserting a new record with exactly the same attributes as in the model version.	No
	Added	Updated	Update Keep unchanged	User has already inserted a record to the model database during the modelling process. The master database has subsequently been updated by inserting à new record, but geometry and/or some attributes are different than in the model version. User has a choice of two options: to update the geometry/attributes or to keep them unchanged.	Yes
	Doesn't exist		Insert Keep unchanged	The master database is updated by a new record. Per default, the model version is updated by inserting the new record. Optionally, the model version can be kept unchanged.	Yes
Deleted	Unchanged	Deleted	Delete Keep unchanged	A record is deleted from the master database. Per default, the model is updated by deleting the record. Optionally, the record may be kept in the model version.	Yes
	Deleted	Unchanged	Unchanged	A record is deleted from the master database. The same record has already been deleted from the model database, so there is no need for further update action.	No
	Updated	Deleted	Delete Keep unchanged	User has edited a record in the model version during the modelling process. The master database has been updated by deleting this record. User has a choice between two options: delete the record, or keep it unchanged.	Yes
Updated	Deleted	Deleted	Keep unchanged Insert	User has deleted a record in the model version during the modelling process. This record in the master database has been updated (new attribute values or geometry). Per default, no update action is done. If the record has become relevant for the model, it can be inserted.	Yes
	Updated	Unchanged	Keep unchanged	User has edited a record in the model version during the modelling process. This record in the master database has also been updated (new attribute values or geometry), so that it is exactly the same as in the model version. No action is needed.	No
		Updated	Keep unchanged Update	User has edited a record in the model version during the modelling process. This record in the master database has also been updated but with different attribute values or geometry. The user has two options: either update the model version, or keep the record unchanged.	Yes
	Unchanged	Updated	Update	A record in the master database has been updated with new attribute values or geometry. The update is automatically transferred to the the model version.	No

To help searching for specific conflicts, it is possible to apply a filter in the search field above the table, where the search applies to the column also selected above the table. Alternatively, it is possible to select conflicts by attributes, by right-clicking on a column's header to enable the 'Select by attributes' tool, or to load a selection from the Selection Manager from the opened project (only available if a MIKE+ database is currently opened). Use the 'Clear selection' button to clear the selection.

It is also possible to assign the same action to all the selected conflicts, using the Field Calculator on the 'Action' column.

Once your configuration is complete, run the tool using the 'Run' button to create the new updated instance.

Mergers

The 'Mergers' tab will show up if the tool detects that the updated database contains a network with pipes merged using the 'Network simplification' tool. This tab is similar to the 'Conflicts' tab, and allows controlling possible actions for the parts of the network that have been merged.

This requires that the model version with the merged pipes, which stores simplification information (relationships between IDs of original and merged pipes), is opened when the 'Update' tool is executed. It also requires, if the option 'Update selected tables and columns only' is active, that both nodes and links tables are included in the update selection with all their columns.

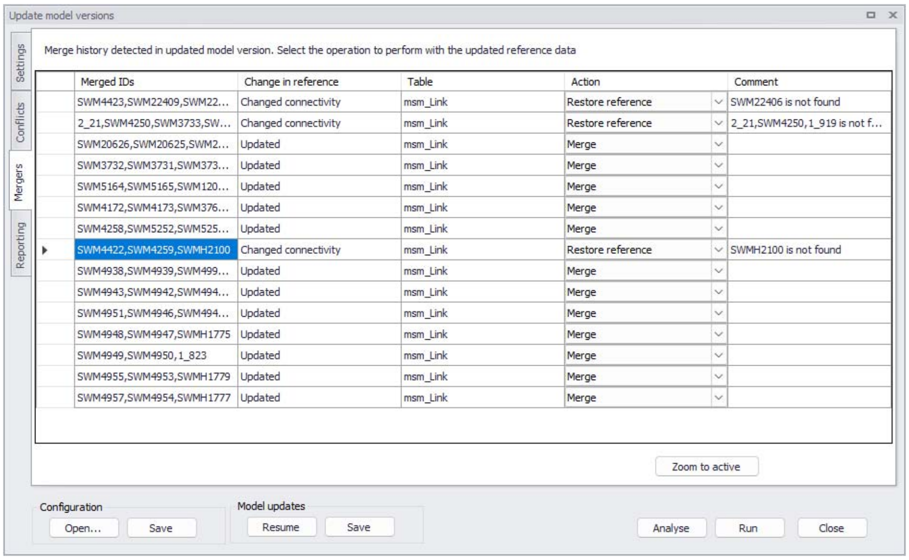


Figure 17.10 The list of identified mergers

Depending on the changes identified in the reference versions and in the model version, the possible actions are:

- Merge: this performs the "merge" operation, using updated attributes and geometry of the pipes as available in the updated reference version.
- Keep unchanged: this keeps the merged pipe, from the former model version being updated, unchanged.
- Restore reference: this removes the merged pipe and replaces it with the original pipes and nodes from the updated reference version. These original pipes won't be merged.

When the changes in reference versions include changes to the connectivity between pipes and nodes, it may require a visual review and possibly require to manually change merge parameters, so restoring the reference data may be the preferred option because automatic re-merge can in some cases lead to unexpected results. In some cases (e.g. if one of the original pipes being merged cannot be found anymore), it is no longer possible to merge the pipes and the 'Merge' action won't be available in the list.

Reporting

When the update is executed, the report appears in the 'Reporting' tab. For all updated tables, it will list the changes applied to the new instance:

- Added records are highlighted in green.

- Updated records are highlighted in yellow. Updated columns are also highlighted in yellow (if the option 'Update selected tables and columns only' is active, only the value of the selected columns is reported).
- Deleted records are highlighted in red.

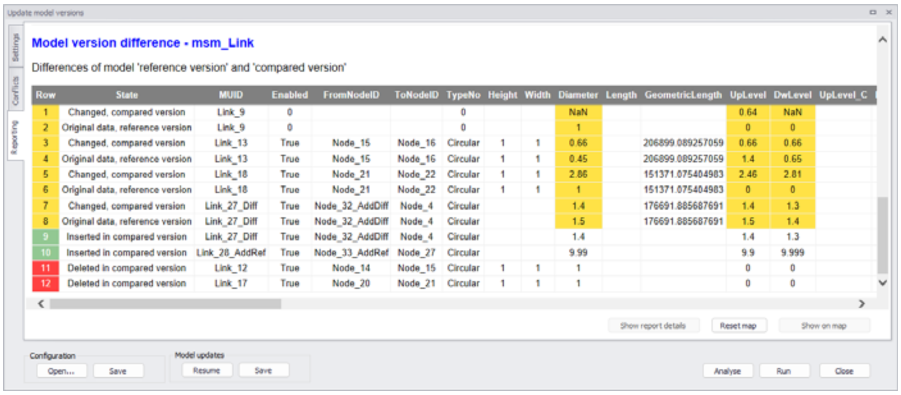


Figure 17.11 Reporting changes in the new instance

This report will show a maximum of 200 differences per table. Pressing 'Show report details' will open another window with the full list of reported differences and with additional options to export the report to other formats.

Note: after closing the tool, it is possible to re-open the .xml report file from the 'Model and result report' tool available in the ribbon (select the file from the 'View' tab).

Pressing the button 'Show on map' will highlight on the map the modified items, with the same color code. Note that this requires that the former instance of the updated version is opened in MIKE+ beforehand.

The button 'Reset map' will clear these highlights on the map.

Configuration

Use the 'Save' button from the 'Configuration' group to save configuration settings in *.xml format to reuse later or in another model. This will save all the configuration of the 'Settings' tab.

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

Model updates

Use the 'Save' button from the 'Model updates' group to save your updates to the list of conflicts in a *.xml file, in order to continue reviewing this list of conflicts at a later stage. This action actually also saves the configuration of the 'Settings' tab to another *.cfg.xml file.

To later resume the review of the conflicts, re-open the 'Update' tool and click 'Resume' to select the *.xml file with previously saved updates to the list of conflicts: this will re-load the databases to compare and run the analysis, and will then load the changes made to the list of conflicts.

18 Results differences Tool

18.1 Introduction

The 'Result differences' tool is designed for comparing results from different variants of hydraulic network simulations, and report any significant difference of result. This may e.g. be used when comparing results from a former version of the model and results from a new version updated with the latest information from an asset management system. The tool allows you to:

1. Quickly identify locations where results differences are observed
2. Visualise and compare results at the identified locations, to verify whether the differences are acceptable or not.

The 'Result differences' tool is accessible from the 'Results' tab in the ribbon. It is not necessary to have a model database opened for using the tool.

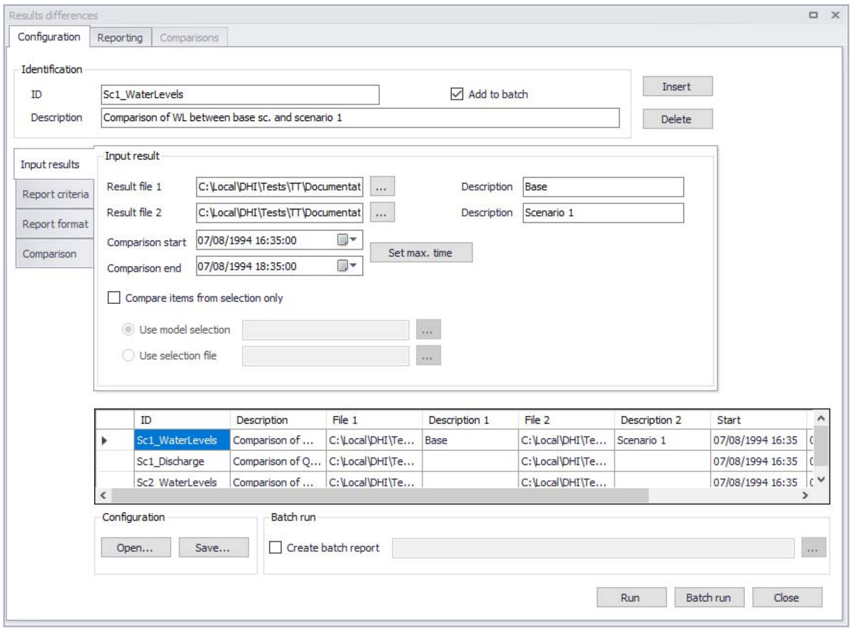


Figure 18.1 The Results differences tool

Note: If one of the two compared result files contains a network with pipes merged using the 'Network simplification' tool, the comparison can take into account the simplification information, in order to compare results in the original pipes and the merged pipe even though they have a different ID. This requires that the model database with the simplified network, which stores simplification information (relationships between IDs of original and merged pipes), is opened when the 'Results differences' tool is executed.

18.2 Running the tool

The tool can handle one or multiple comparison jobs. One comparison job can compare only one result item (water level, discharge, etc.) between two result files. If more result files and/or more result items should be compared, then extra comparison jobs must be used. Comparison jobs are added or removed using the 'Insert' and 'Delete' buttons at the top, and are listed in the table at the bottom of the dialog.

To start using the tool, a first comparison job must be inserted.

Each comparison job is given an ID, and can optionally contain a description, specified at the top of the dialog.

The 'Run' button will run only the active comparison job from the table.

The 'Batch run' button will run all the comparison jobs which have their option 'Add to batch' active.

Each comparison job gets its own report, but for a batch run it is also possible to get an overview report for the entire batch run. This is enabled by ticking the option 'Create batch report' and selecting the path and name of the batch report file. The batch report shows the number of time series for which the criteria were exceeded during the comparison.

Once the tool has been configured, it is possible to save its configuration to a file using the 'Save…' button for later re-use. This configuration can later be loaded again using the 'Open…' button, or can be used to execute the tool from a command line.

18.3 Input results

The 'Input results' tab contains information controlling the selection of time series to be compared.

Result files 1 and 2

These are the paths to the result files selected for the comparison. Press the '…' buttons to select the files. The supported file types are:

- .res1d (MIKE 1D result files from collection systems and/or river networks, excluding catchments results)
- .res (Water Distribution / EPANET result files)
- .out (SWMM result files).

Result file 1 and Result file 2 must be of the same file type for a given comparison job. Other file types (typically .resx, .msxr and .whr) cannot be compared in this tool.

The comparison is done based on the network element ID, i.e. nodes or pipes / rivers should have the same ID in the two result files in order to be compared.

Result file 1 is the reference file. The two result files don't have to store results at the same date and time: when they differ, the results from the result file 2 are linearly interpolated to match the dates and times in result file 1.

Descriptions

Optional descriptions for the result file 1 and result file 2, respectively.

Comparison start and end

The time interval for which the comparison is executed can be set here. It is possible to execute the comparison for a limited period, shorter than the complete overlap between the files.

The 'Set max. time' button can be used to set the common time interval for the two result files as the comparison period.

Compare items from selection only

By default, the tool will compare all result time series for locations found in the two compared result files.

It is also possible to reduce the number of time series being compared by activating the option 'Compare items from selection only' and choosing a selection containing the list of items to be compared. Two options are available to choose the selection:

- Use model selection: this option is used to pick a selection defined in the 'Selection manager', accessible from the Map tab in the ribbon. This option requires that a model database is opened, to access the list of selections. When no model database is opened, this option is therefore disabled.
- Use selection file: this option is used to pick a selection defined in a text file. This text file can be created by saving a selection defined in the 'Selection manager', accessible from the Map tab in the ribbon. This option is always available, even if no model database is opened.

18.4 Report criteria

The 'Report criteria' tab contains information controlling the reported differences in the report.

Result item

Only one result item can be selected for a given comparison job. The list of available compared items depends on the result files type. For .res1d result files, the possible compared items are:

- Discharge: this result item is not available at nodes, and will therefore be compared on links only.
- Water level: this result item is available in both nodes and links.
- Flow velocity: this result item is not available in nodes nor in structure reaches, and will therefore be compared on regular links only.
- Volume: this result item is available in both nodes and links.
  Note that the 'Volume' result item is not included by default in result files and must be added manually before running the simulation.

For .res result files, the possible compared items are:

- Flow: this result item is not available at junctions, and will therefore be compared on links only.
- Velocity: this result item is not available at junctions, and will therefore be compared on links only.
- Pressure: this result item is compared in junctions and tanks.
- Head: this result item is compared in junctions and tanks.
- Water quality: this result item is compared in nodes and links.
- Water demand: this result item is compared in junctions and tanks.

For .out result files, the possible compared items are:

- Discharge: this result item is not available at nodes, and will therefore be compared on links only.
- Water depth: this result item is available in both nodes and links.
- Flow velocity: this result item is not available in nodes, and will therefore be compared on links only.
- Volume: this result item is available in both nodes (result item called 'Node volume stored & ponded') and links ('Link volume').

Each option represents the instantaneous value of the result item in the compared calculation point.

Acceptance criteria

It is possible to select and combine various criteria for the comparison. In general, the result computed for the individual criterion is based on the absolute difference between the two time series being compared. This means that the result indicates if the time series deviate, but the result does not indicate which time series has e.g. the largest maximum value.

In an ideal case, when the two time series are identical, the value computed by each criterion should be zero. The only exception is the 'Confidence band' criterion, which results in a value of 100 when comparing two identical time series. In practice, when comparing two different time series, values close to zero (or close to 100 for 'Confidence band') indicate a good similarity. On the contrary, if the computed values are significantly far from zero (or from 100), the similarity of the two time series is weak.

When a criteria is selected, it is included in the comparison process. If the specified criteria is fulfilled (e.g. Peak error <= 2%), then the difference between the two time series is considered acceptable. When one or more criteria is not fulfilled, the comparison is "rejected" and the time series is listed in the report.

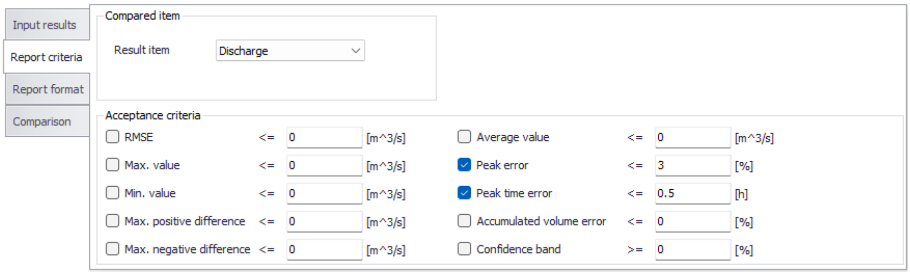


Figure 18.2 The Report criteria tab

If the computed value indicates that the time series deviate, then a more detailed inspection of the time series is recommended to determine the importance of the difference.

The criteria are described below in more details. In these descriptions, y_1 always refers to the instantaneous value of the time series from Result File 1, while y_2 refers to the value from Result File 2.

Root Mean Square Error (RMSE)

The Root Mean Square Error (RMSE) criterion can be applied as a measure for the magnitude of the deviation between the two time series over the period being investigated.

$$\text{"RMSE"} = \sqrt{\frac{\sum_{i=1}^{n}(y_{2,i} - y_{1,i})^2}{n}} \qquad (18.1)$$

The values for the second time series (Result File 2) will be linearly interpolated to get values at the date and times matching the Result File 1.

Maximum value

The criterion provides a value for the difference in maximum values found in the two compared time series.

$$\text{"Max Value"} = |max(y_2) - max(y_1)| \qquad (18.2)$$

It should be noticed that the maximum value found in the two time series does not necessarily occur at the same point in time in the two series.

Minimum value

The criterion provides a value for the difference in minimum values found in the two compared time series.

$$"Min\ Value" = |min(y_2) - min(y_1)| \tag{18.3}$$

It should be noticed that the minimum value found in the two time series does not necessarily occur at the same point in time in the two series.

Maximum positive difference

This criterion computes a value indicating how much the first time series (Result File 1) is above the second time series at the point in time where this difference has its maximum.

$$"Max\ Positive\ Difference" = |max(y_2 - y_1)| \tag{18.4}$$

The values for the second time series (Result File 2) will be linearly interpolated to get values at the date and times matching the Result File 1.

Maximum negative difference

This criterion computes a value indicating how much the first time series (Result File 1) is below the second time series at the point in time where this difference has its maximum.

$$"Max\ Negative\ Difference" = |min(y_2 - y_1)| \tag{18.5}$$

The values for the second time series (Result File 2) will be linearly interpolated to get values at the date and times matching the Result File 1.

Average value

The average value is computed for both time series. Each value is given weight according to the actual time step. Values are assumed valid for the time interval since the previous value. As a consequence, the first value is ignored.

$$"Average\ Value" = \left| \frac{\sum_{i=1}^{n} y_2 \cdot (t_{2,i} - t_{2,i-1})}{t_{2,n} - t_{2,1}} - \frac{\sum_{i=1}^{n} y_1 \cdot (t_{1,i} - t_{1,i-1})}{t_{1,n} - t_{1,1}} \right| \tag{18.6}$$

Peak error

This criterion computes a value for the relative error for the maximum values.

$$"Peak\ Error" = \left|1 - \frac{max(y_2)}{max(y_1)}\right| \cdot 100 \tag{18.7}$$

It should be noticed that the maximum value found in the two time series does not necessarily occur at the same point in time in the two series.

Peak time error

This criterion indicates how far in time the two maximum values a located away from each other. This criterion can be used to clarify if the criteria 'Max Value' and 'Peak Error' actually compare the same event.

$$"Peak\ Time\ Error" = \left|t_{2,max(y_2)} - t_{1,max(y_1)}\right| \tag{18.8}$$

Accumulated volume error

This criterion is only available when comparing a discharge or water demand result item. It computes the deviation (in percentage) of accumulated volume through nodes and grid points from the discharge time series.

$$"Volume\ Error" = \left|1 - \frac{volume(y_2)}{volume(y_1)}\right| \cdot 100 \tag{18.9}$$

Where *Volume(y)* is the accumulated volume in a calculation point computed from the discharge time series:

$$Volume(y) = \int_0^n Q(y)_t dt \tag{18.10}$$

Confidence band

The purpose of this criterion is to verify that two time series are identical at all points. It is accepted that the two time series are shifted in time by maximum one time step and a tolerance (dx) is accepted.

$$Inside\ Band = \begin{cases} 1 & for \quad y_{2,i} \geq min(y_{1,i-1}; y_{1,i}; y_{1,i+1}) - dx \\ & and \quad y_{2,i} \leq max(y_{1,i-1}; y_{1,i}; y_{1,i+1}) + dx \\ 0 & \end{cases} \tag{18.11}$$

$$"Confidence\ Band" = \frac{\sum_{i=1} Inside\ Band}{n} \cdot 100 \tag{18.12}$$

The values for the second time series (Result File 2) will be linearly interpolated to get values at the date and times matching the Result File 1. Tolerance (dx) is set to 0.01.

Note that for this criterion the ideal value is not zero but 100.

Note: When a model database is opened, the criteria's units are controlled by the selected unit system in the model setup. If no model database is opened, the unit is controlled by the 'Preferred unit system': see Chapter 2.9.1 File Menu *(p. 51)* for more information.

18.5 Report format

The 'Report format' tab contains information controlling the format of the reported differences, for the active comparison job.

Report
This is the path to the report file, for the active comparison job. The report is saved to a *.htm file, which can be opened in a web browser.

Comment
An optional description of the active comparison job, which will appear in the report.

Report differences only for gidpoints where criteria are exceeded
By default, this option is active and the report will list only the locations where the acceptance criteria are not fulfilled (i.e. where differences are significant). If this option is unselected, all locations will be reported, therefore also providing the comparison values for the locations where the differences are small.

Create shape file with differences where criteria are exceeded
When this option is selected, the locations where the acceptance criteria are not fulfilled (i.e. where differences are significant) are saved to a shape file. This makes it easy to visualize the locations of significant differences on a map.

Two shape files can be created:

- A point shape file storing nodes locations.
- A line shape file storing links locations.

The specified file name is used by the lines shape file. The nodes shape file has the same file name but with a suffix '_Nodes'.

If the location of the link or node on the map differs between the two result files, the shape file will show the location from the first result file.

Graphics

This option controls if time series plots are included in the report or not. Three options are available:

- Don't include time series: no time series is added to the report.
- Include time series without difference: for the reported locations, a time series plot will show the superimposed time series from the two result files.
- Include time series with difference: for the reported locations, a time series plot will show the superimposed time series from the two result files plus an extra time series showing the differences between the two files, on a secondary Y-axis.

Note: Including time series in the report may significantly increase the results comparison time as well as the report file size, depending on the number of included graphics.

18.6 Comparison

The 'Comparison' tab contains information controlling optional additional comparison plots, which can be provided after running the comparison job.

The following additional plots can be activated.

Scatter plot

The scatter graph is an analysis plot where the horizontal axis is the magnitude of the results from the first file and the vertical axis is the magnitude of the results from the second file. At each time step saved in the first result file, the result from the second file is interpolated so that it can be plotted as an X,Y point on the graph. If the model and data are in perfect match, then the point is plotted on a 45-degree line. If the result file 1 is low by comparison with the result file 2, then the point will be plotted above the 45-degree line.

The red line in the scatter plot is the line of best fit. The value 'a' is the slope of the line and value 'b' is the Y-axis intercept.

The scatter plot visually shows if there is high or low behavior in specific value ranges, and also the width of the scatter gives a qualitative estimate of the amount of variability at a given value range. The analysis therefore allows the modeller to observe where the bias occurs in general areas of the modeled behavior.

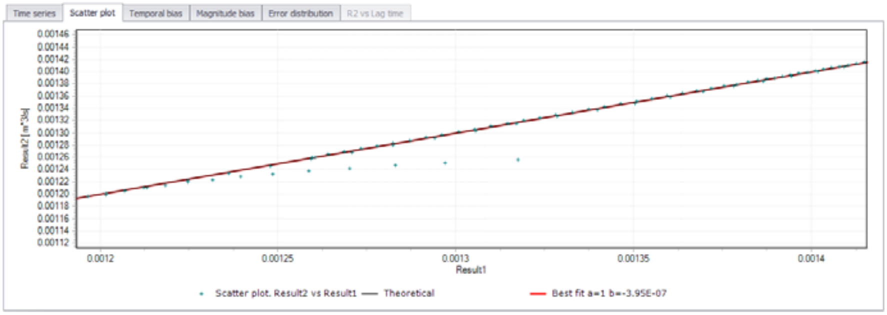


Figure 18.3 The Scatter plot

Temporal bias

The temporal bias is defined as the difference between the two result files at the same point in time, expressed as (Result file 2 - Result file 1). This plot indicates when this temporal bias occurs, and if it is a regular or random occurrence. This plot tends to show if there are certain time periods where errors occur. If there are multiple instances of the same behavior, then it is not due to an isolated event and is something which repeats.

A line of best fit is calculated, and if the slope on the line is zero (i.e. the line of best fit is parallel to the horizontal axis), then there is no trend of bias in the comparison. In the case where the slope is zero but the Y-intercept is non-zero, then there is probably a baseflow error.

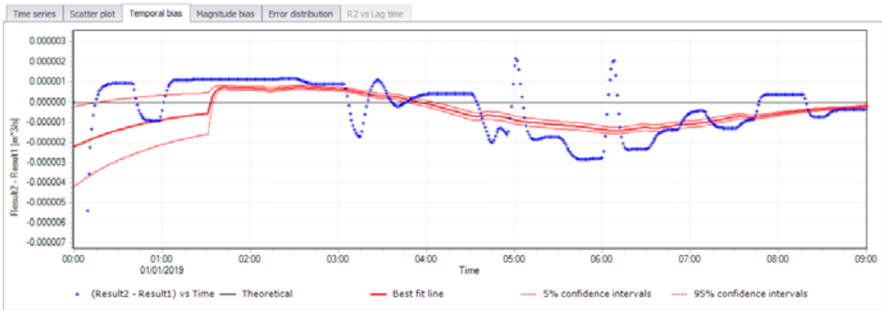


Figure 18.4 The Temporal bias plot

When activating the temporal bias plot, a time interval must be specified. The temporal bias plot is divided into a number of time periods, and the values in the time interval are used as a population to calculate mean and 5% and 95% confidence intervals. The confidence intervals are based on the assumption that the error is normally distributed.

Magnitude bias

The magnitude residual plot is like the scatter graph but normalized to a horizontal axis. By plotting the difference between the two files on the vertical axis and the average of the two files on the horizontal axis, the line of best fit becomes a horizontal line intercepting the Y-axis at zero. Hence this plot shows more clearly the width of the scatter at certain values ranges, and will signal wide errors at certain hydraulic conditions or when certain thresholds are exceeded.

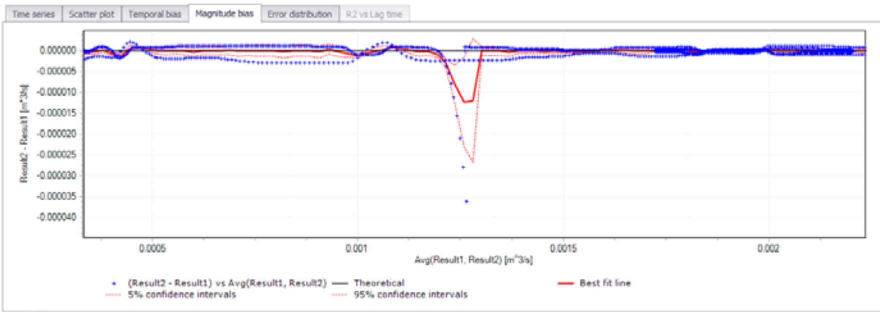


Figure 18.5 The Magnitude bias plot

When activating the magnitude bias plot, a number of intervals must be specified.

Error distribution

This plot does not perform any analysis, but it gives an indication of where the two result files diverge on a percentage basis. The error distribution value yi is expressed like this:

$$yi = \begin{cases} \dfrac{y2i - y1i}{y1i} \times 100 & , |y1i| > 1e^{-8} \\ 0 & , |y1i| \le 1e^{-8} \end{cases} \tag{18.13}$$

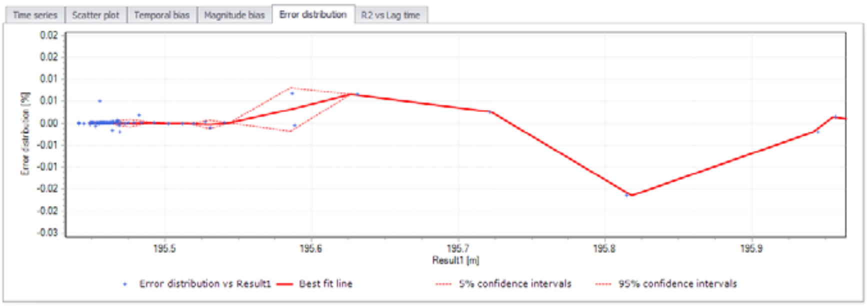


Figure 18.6 The Error distribution plot

When activating the error distribution plot, a number of intervals must be specified.

R2 vs. Lag time

This plot is specifically designed to analyze if there are lags in one of the result files which would otherwise provide a good fit. The analysis is therefore very useful for determining if there are travel time errors.

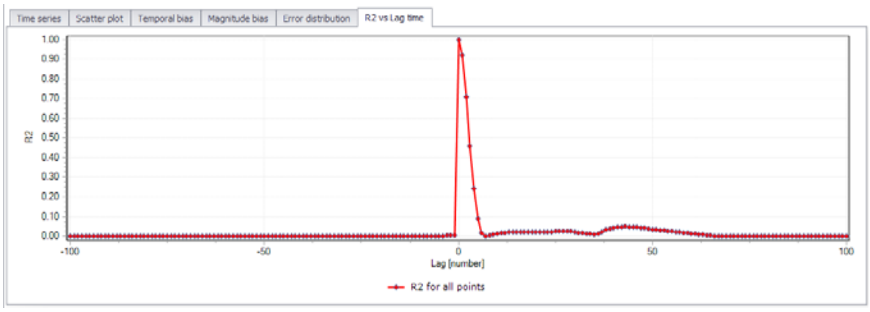


Figure 18.7 The R2 vs Lag time plot

When activating the R2 vs Lag time plot, a number of time lags to be analysed as well as the lag duration must be specified. The tool then shifts one time series both forward and backwards in time compared to the original position, and calculates the coefficient of determination R^2 for each position. The plot produced is a plot of number of lags on the horizontal axis (both negative and positive) and the coefficient of determination R^2 on the vertical axis. The plot can be used to determine if there are fundamental time shifts in the information.

Note: Activating the R2 vs Lag time plot requires much more computational resources than for the other plots. Therefore, when this plot is included in the analysis, the computational time may increase significantly.

18.7 Reporting

After running a single comparison job or a batch run, the 'Reporting' tab is opened

Reports

The upper table shows a list of the report files generated during the run. Each file can be opened using the 'Open' button.

Run status

The different steps of the comparison are listed here, along with possible errors encountered during the run.

18.8 Comparisons

The 'Comparisons' tab shows the result of the comparison job.

Location table

The table in the upper left corner shows the list of locations where time series have been compared. The active record in this table selects at which location the criteria values are reported on the right, and at which location the time series are plotted at the bottom.

The table contains three columns:

- Type: the network element type (link, node, structure type).
- ID: the ID of the network element.
- Chainage: the chainage / distance of the calculation point along its link. Not applicable for nodes.

Two types of filters are available above the table, to help searching an item by reducing the displayed list:

- A Search field: type here the expected text to search in the ID column. Press the 'Clear' button to clear this filter.
- A type selection: use the list on the right above the table, to show only the locations for a given type (Node, pump, orifice, etc.).

ID

In the 'General' group, the ID shows the ID of the comparison job being displayed.

Result item

The 'Result item' shows the item being compared in the displayed comparison job

Show time series of differences

By default, the 'Time series' tab shows the superimposed time series from the two compared result files, only. When this option is active, it also shows an extra time series plotting the difference between the two, on the secondary Y-axis.

Show rejected gridpoints only

By default, all locations are shown in the location table on the left. When this option is active, the table shows only the locations where one or more criteria are not fulfilled.

Result statistics

This group shows the result statistics values for the active location in the location table (upper left table), when they have been selected as acceptance criteria for the comparison.

Comparison criteria

This group shows the calculated comparison values for the active location in the location table (upper left table), when they have been selected as acceptance criteria for the comparison.

It also shows the coefficient of determination R2, also known as Nash-Sutcliffe efficiency, that measures how well the result values in File 1 and File 2 match. This criterion is widely used to evaluate model performance in hydrological modelling. It ranges from minus infinity to 1 with larger values indicating a better fit. An important special case is R2 = 0, which can be obtained if the mean value from result file 1 equals the mean value from result file 2, indicating that the average of the result file 1's values in this case is as good a predictor as the result file 2. Thus, one would most likely require that R2 > 0 for the two files to be fit. The R2 criterion measures the one-to-one relationship between the two files' values, and hence it is sensitive to bias and proportional effects. It should be emphasized that R2 is based on the sum of squared residuals, and hence provides the same information on goodness-of-fit as the RMSE measure.

$$R2 = 1 - \frac{\sum_{i=1}^{N} (y_1 - y_2)^2}{\sum_{i=1}^{N} (y_1 - \overline{y_1})^2} \tag{18.14}$$

Where $\overline{y_1}$ is the mean value of the time series from result file 1.

The values for the result file 2 are linearly interpolated to get values at the same date and times as in result file 1. The calculation of this criterion is valid only when the result file 1 contains a constant time step.

Plots

The lower part of the dialog shows time series plots, for the active location in the location table (upper left table).

The 'Time series' tab is always active and shows the superimposed time series from the two compared result files. When 'Show time series of differences' is active, it also shows an extra time series plotting the difference between the two, on the secondary Y-axis.

The other tabs show optional extra plots, when they have been activated in the comparison configuration before the run.

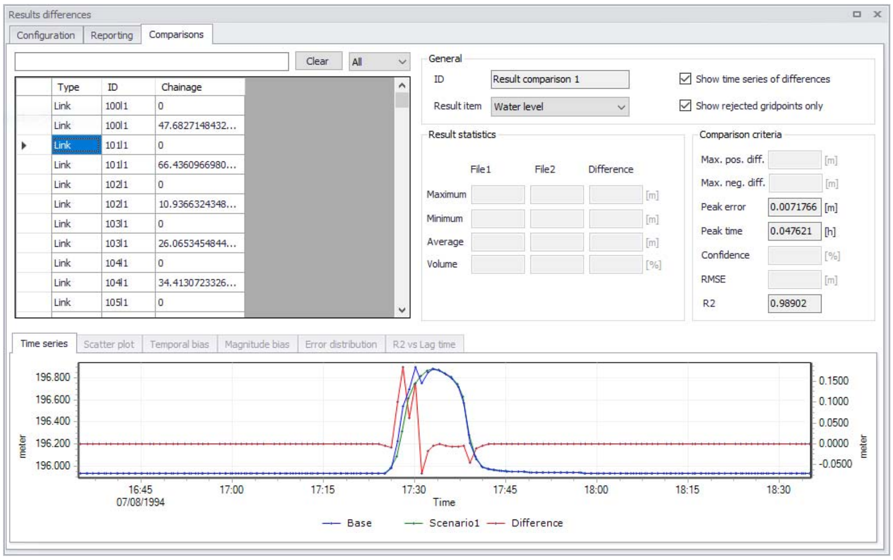


Figure 18.8 Results of the comparison job

18.9 Running the tool from command lines

When working with numerical models and their results, you often utilize the MIKE+ editor to access all the tools, including the 'Results differences' tool. However, there are times when it is required to compare result files in an automated way without opening the tool in the user interface.

The MIKE+ executables enable you to execute some tools without opening the editor, through command lines. It is possible to run the 'Results differences' tool in this manner, assuming you have prepared the comparison configuration file beforehand in MIKE+.

Start by locating the MIKE+ executable named DHI.MIKEPlus.ToolShell.exe in the installation folder. From a command prompt, type the command below to access the list of supported tools, replacing the … characters by the actual path to the file:

"C:\...\DHI.MIKEPlus.ToolShell.exe" -h

The format of the command for running the 'Results differences' tool is:

"C:\...\DHI.MIKEPlus.ToolShell.exe" resdiff -f [Configuration file] [Option]

Where [Configuration file] is the path to the *.xml configuration file.

The only option available is: -c [Comparison ID]. This option may be used to execute only a specific comparison job from the list of jobs in the selected configuration file. When this option is not included, all comparisons added to the batch will be executed.

19 CS Network Specific Tools

19.1 Introduction

Urban stormwater flood modelling can be carried out using a 1D/1D or a 1D/2D approach. The 1D/1D approach has a simplified representation of the overland surface hydraulics compared to the 1D/2D approach.

In recent years coupled 1D/2D modelling applications have been widely used in urban stormwater analysis. The advantage of the 1D/2D approach is that the model is faster to configure, and it provides a more realistic hydraulic description of the surface flow. Model simulation times are however generally long, which makes the design and testing of stormwater improvements and installations difficult and time-consuming.

The advantage of the 1D/1D approach is that the simulation time is considerably faster than the 1D/2D approach and is therefore more suitable for detailed design option runs. However, configuring the 1D/1D model is more time-consuming than the 1D/2D model.

A 1D/1D stormwater model typically comprises three main components: the sewer/stormwater system, the overland flow system, and rainfall-runoff hydrology.

The set of tools is developed to specifically address some normally time-consuming aspects of building a 1D/1D stormwater model.

For a catchment study a staged approach is usually needed, for example:

- The first stage involves catchment-wide modelling in a combined 1D/2D model to assess the performance of the system and identify potential improvement locations;
- For the second stage, a full 1D/1D model is built only for the improvement location including the 1D overland flow path network. The 1D/1D model runtimes are fast and allow for quicker assessment of stormwater improvements.

The stormwater tools focus on building the stormwater model within a 1D framework, which include:

- Cross section extraction from the DEM
- Lateral snapping of nodes according to the DEM
- Auto connection of overland network to stormwater network
- Sequential labelling of nodes

The tools can be activated from the menu under the CS network tab.

19.2 Generate Cross Sections Tool

When overland flow paths are defined as MIKE+ links, a cross section (CRS) needs to be assigned to each of these overland flow links. Often, a standard road profile will fulfil the modelling requirements. However, in other urban flood modelling situations, individual cross sections are required for open spaces, rural areas, park areas, etc.

The Generate cross sections tool uses cross section alignments drawn in a line feature layer to extract cross sections from a DEM for links intersected by the alignments, see Figure 19.1. It generates cross sections for each link and sets the reference between the link and the generated cross section ID.

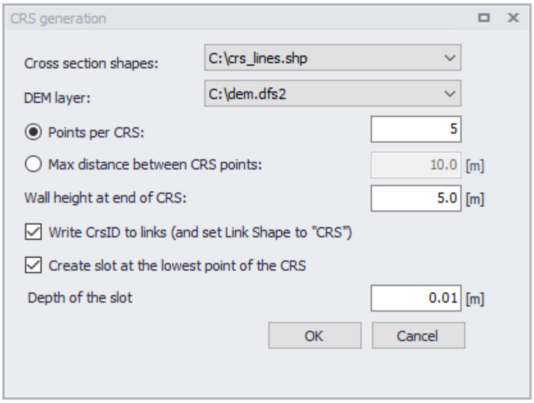


Figure 19.1   Cross Section Generation tool

Note: The DEM and cross section shapes need to be previously added. ‘Add layer…’ under Layer and symbols editors. The DEM layer needs to be a *.DFS2 file type (add layer ‘MZ raster layer’ type).

The manual digitalization of the CRS alignment lines can be guided by MIKE+ FLOOD simulations results (1D/2D results) or an uncoupled 2D model where the precipitation is applied to the surface assuming that the subsurface net-work is completely full. Overland flood results can be used as a background layer in MIKE+. The only requirement for the digitized CRS lines is that they intersect the pertinent MIKE+ links. An example is shown in Figure 19.2.

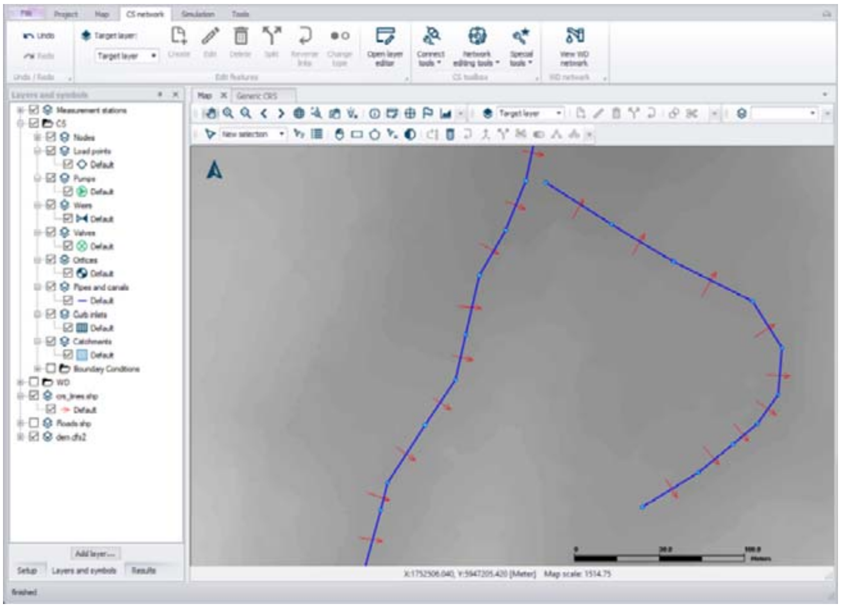


Figure 19.2 Example of defined CRS lines

Either the number of points or the maximum distance between points for a CRS can be specified. To fully capture the information from the DEM the resolution of the CRS should correlate to the resolution of the DEM.

The ID of the new CRS will be identical to the ID of the corresponding link. The CRS will be created with the type X-Z open and with a width equal to the length of the corresponding CRS alignment line. The pertinent link will also be updated to use the new CRS by default, but this option can be disabled in the tool.

If the water level in a CRS rises above the defined CRS height, the simulation will stop. This is very common in stormwater models, so the tool allows for side-walls to be automatically added. Use a side-wall of 1 to 2 meters for an overland flow path such as a road, and 3 to 5 meters for waterways. The default value is set at 5m. Adding side-walls adds 2 points to the number of CRS points specified in the tool.

Another consideration is that if a CRS is very flat, too much numerical water will be generated when the link is running dry. Thus, a slot can automatically be inserted at the lowest point of the cross section.

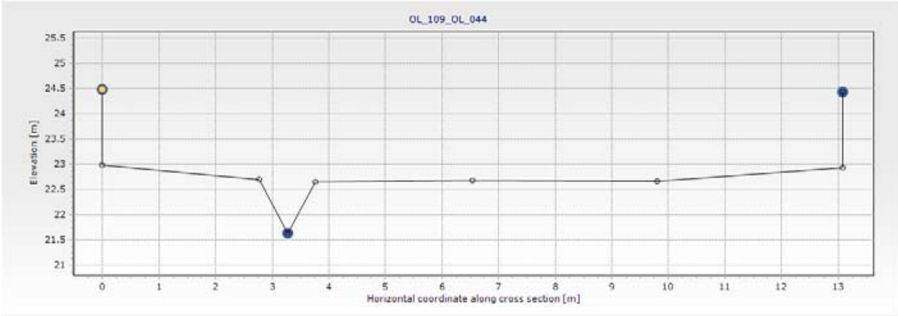


Figure 19.3 Example of CRS with inserted slot

The tool will generate CRSs for selected links or for all links intersecting alignments if no particular link is selected.

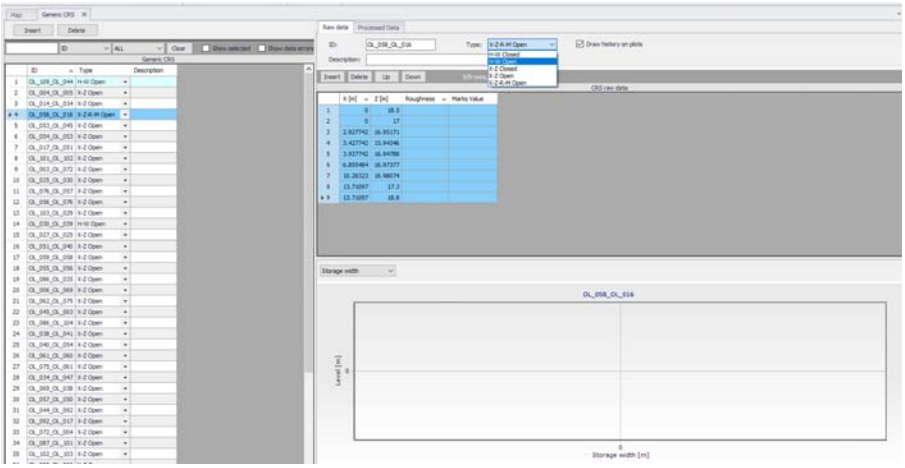


Figure 19.4 Automatic drawing CRS parameters

19.3 Lateral Snapping Tool

It can be difficult to exactly place nodes along the invert when digitizing an overland flow path whether it is along the gutter in a road or the invert of a waterway.

The Lateral Snapping tool, shown in Figure 19.5, is used for automatically moving nodes and snapping them laterally to the lowest DEM value along a lateral snap alignment, which is shown in Figure 19.6. The length of the lateral snap alignment is specified in the tool.

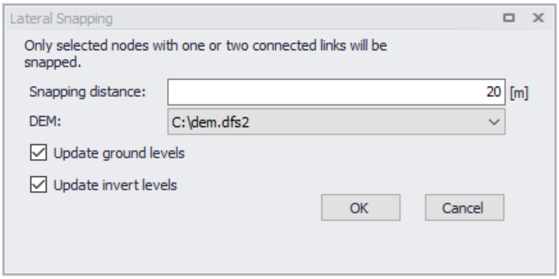


Figure 19.5 Lateral Snapping tool

When the selected node is connected to two links, the direction of the lateral snapping alignment will be created such that the internal angles between the upstream and downstream links are equal θ1θ = θ2θ, see Figure 19.6. If the selected node is only connected to one link, the direction of the lateral snapping alignment will be perpendicular to the link.

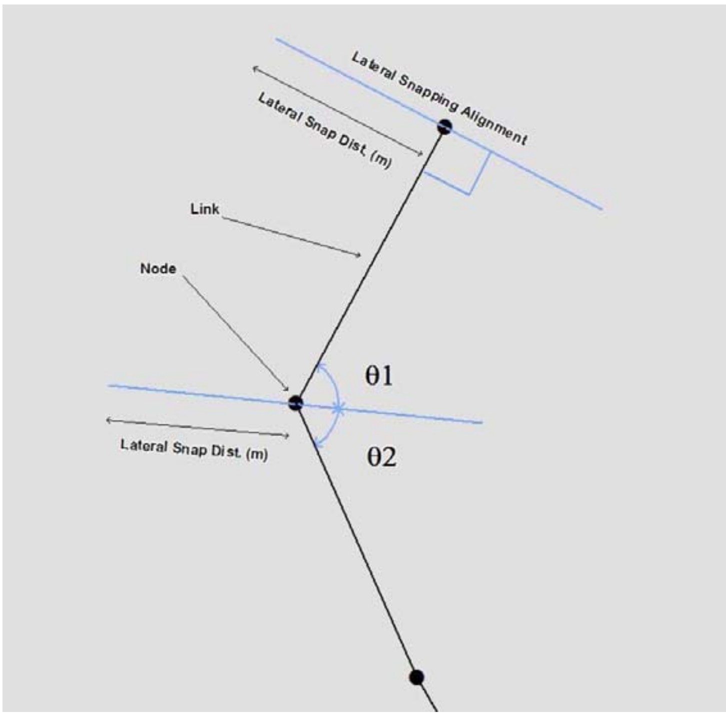


Figure 19.6 Lateral snapping concept

When nodes are moved to the lowest point, the tool can update either the ground level or the invert level according to the DEM. Note that the ground level should be updated for nodes belonging to the subsurface network and the invert level for nodes belonging to the overland flow network.

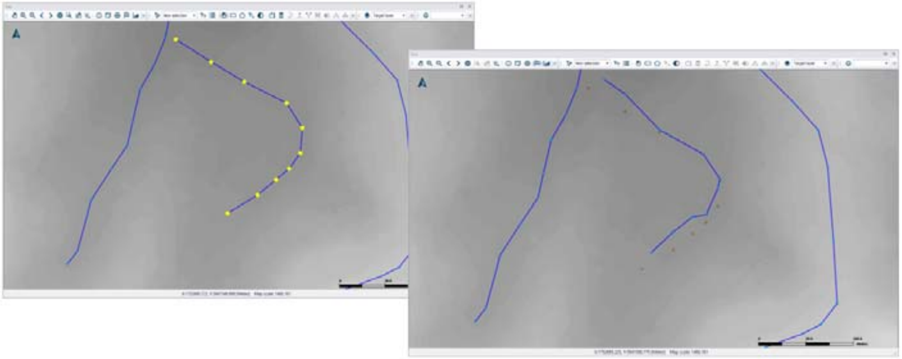


Figure 19.7 Example of nodes moved with the lateral snapping tool

The tool will only laterally snap selected nodes.

19.4 Auto Connection Tool

When overland flow paths have been digitized and snapped to a correct alignment, the overland flow links have to be connected to each other (overland to overland) or connections have to be made between the overland flow network and the subsurface network.

The tool can generate both connections between the same type of network and connections between two different types of network. A network can be defined as i.e. Stormwater, Combined Sewer, Sanitary Sewer or a user-defined network type. A user-defined network could be Overland Flow network (OF) or Rising Main (RM). User-defined network types are set in the ‘Pipes and canals table’ under the “Description” tab and “Network type” (select the “…” to obtain the Status code editor).

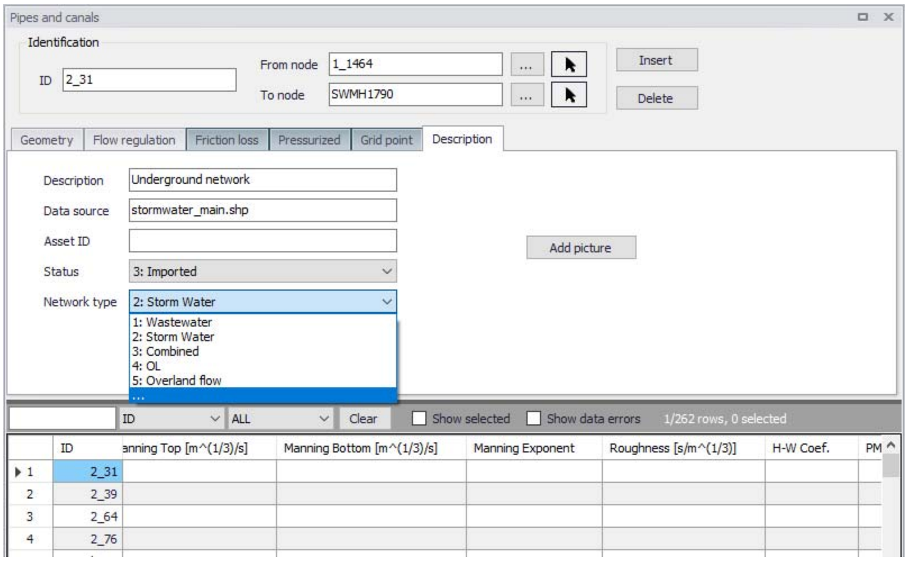


Figure 19.8 Editing network type

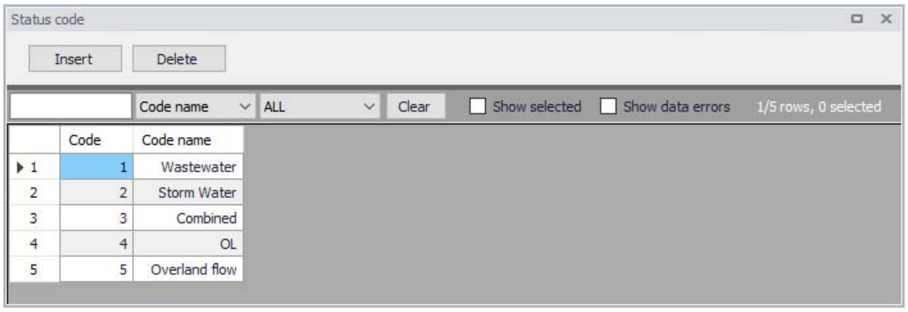


Figure 19.9 Domain Editor

The nodes in the model must be provided with the appropriate network type before proceeding to set up the connections.

A connection can be a weir, orifice or curb-inlet for a MOUSE network, and a weir or orifice for a SWMM network. Connections between an overland flow network and a subsurface network are usually either orifices or curb-inlets while connections between two overland flow networks are usually weirs. The connective structure is created according to the direction specified in the tool. When connecting an overland flow network with a subsurface network, the direction should be specified from the overland network to the subsurface network.

A user-specified search radius is used around the nodes from the network listed under “From nodes” to control where connections can be made. If the two different networks are connected, the connection will be made to the closest node with the type of the “To nodes”. If identical networks are connected, then connections will be made between all nodes within the search radius that are not already connected.

The structures are created with an invert level according to the network, but other parameters for the individual structures must be set manually. If two different networks are connected, the default invert for the structure will come from the invert of the upstream node. If two identical networks are connected, the default invert for the structure will be the maximum of the upstream and downstream node inverts. An offset for the invert can be specified in the tool. It is recommended to insert a small offset of i.e. 0.05 m between overland flow networks and subsurface networks to ensure that flow is not always occurring through the connection.

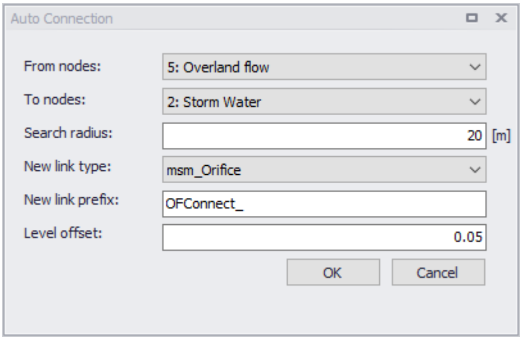


Figure 19.10 Auto Connection Tool

An example from a 1D/1D model is provided in Figure 19.11 where the blue network represents the stormwater network and the red network represents the overland flow network. The overland flow network is connected to the stormwater network by orifices. The overland flow network elements are connected together at junctions by weirs.

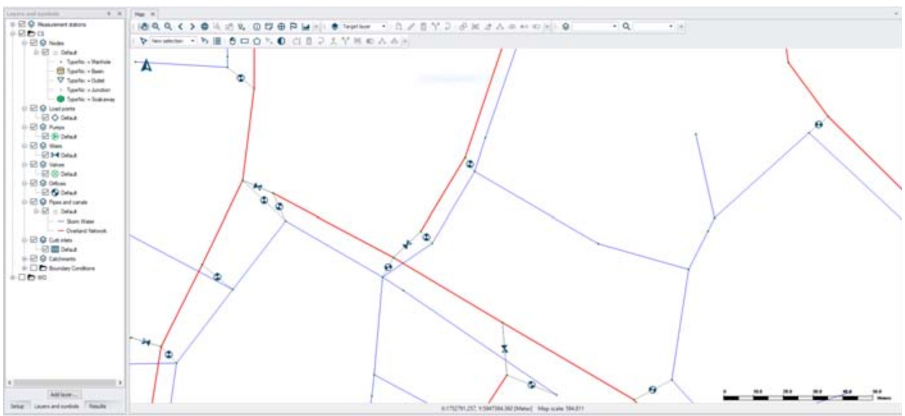


Figure 19.11 Example of connections

The tool will try to make connections for selected nodes, and if no node is selected, connections will be attempted for all nodes according to the specifications.

19.5 Sequential Labelling Tool

When an overland flow path or other network is digitized, it is often useful to provide IDs that are descriptive and logical. The IDs can be made up by street name, type of network, sub-catchment identifier, etc. and the sequential labelling tool can automatically achieve this. The sequential labelling tool is located in the ribbon of the tool tab.

A format for the automatic IDs can be specified for selected nodes or links in the map. A label prefix and suffix may be given together with sequential numbering parameters.

Only selected items of the chosen Map layer (nodes or links) will be given new IDs.

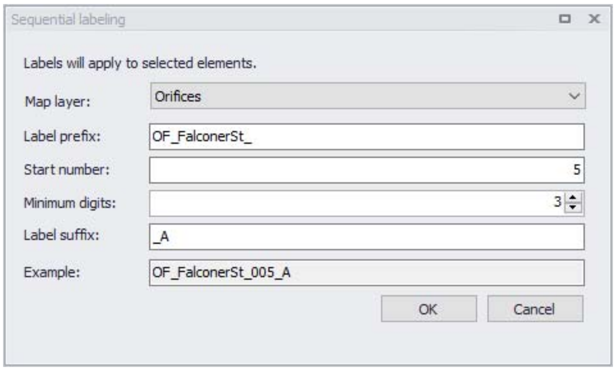


Figure 19.12 Sequential Labelling Tool

Note: Applying the tool with a PostgreSQL database from a distant server may be significantly slower than when working with a local database. This is an expected behaviour, and happens when the updated IDs trigger additional updates in other editors (e.g to update connection information for all connected features), hence requiring additional communication time with the distant server.

19.6 Set Pumps Critical Levels Tool

19.6.1 Introduction

This tool is used to assign a critical level at pumping stations' wet well nodes ('From node' of the pumps). This value is applied to the 'Critical level' field in the 'Description' tab from the Nodes editor.

The tool uses network tracing to trace backwards from each pumping station to find nodes, weirs and orifices where water can potentially leave the network; and stops when it reaches the user-specified trace distance, or earlier if it meets another pump.

For each pump, the tool will assess the critical level for each node found on the backward-traced path. For each of these nodes, the critical level is the lowest level amongst:

1. The nodes' ground level minus the specified freeboard
2. The weir crest level minus the freeboard, if any weir is connected to the node and doesn't have any 'To node' (i.e. if the weir discharges out of the modelled network)
3. The orifice invert level minus the freeboard, if any orifice is connected to the node and doesn't have any 'To node' (i.e. if the orifice discharges out of the modelled network).

Finally, the actual 'Critical level' assigned to the pump's wet well node is the lowest of all critical levels from all these nodes on the backward-traced path.

The tool is accessed through the MIKE+ ribbon, CS network tab, from the 'Special tools' button.

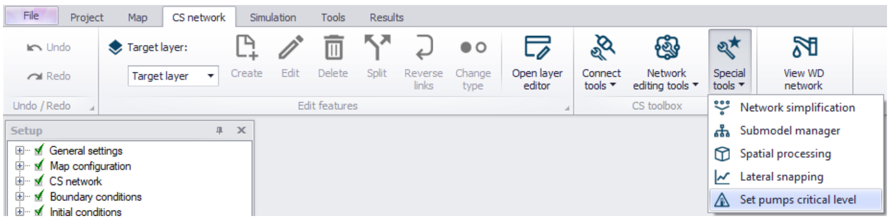


Figure 19.13 Accessing the Set pumps critical level tool

The tool works per default with all pumping stations in the model or, if some wet well nodes are selected, only with the pumping stations connected to the selected wet well nodes.

19.6.2 Settings

The 'Critical level identification' group contains option controlling the selection of upstream nodes and the estimation of the critical level. The following controls are available:

- Backward tracing distance: The distance, starting from the pump's wet well node, along which the nodes will be selected during the backward tracing.
- Freeboard: The depth at which the critical level is defined below the ground level, for a node. For a weir or an orifice discharging out of the modelled network, the critical level is defined at the weir crest level or the orifice invert level, minus the freeboard. The freeboard is defined with a positive value.
- Ignore sealed nodes and junction nodes: When a node is a 'Junction'-type of node or when it's sealed, no overflow can occur and no critical level is assessed using this node's ground level, when this option is selected. It is however possible to force including the ground level for these types of nodes in the analysis, by unselecting this option.
- Update existing critical levels: When this option is not selected, any critical level already defined for the pumps wet well nodes will be kept when running the tool. When it is selected, existing critical levels will be replaced by a new value computed by the tool.

The 'Reporting' group contains options to control how many nodes and their critical levels are reported along the backward-tracing distance from each pump. This does not affect the critical level assigned to the pumps' wet well nodes. Per default, all nodes included in the analysis for a pump are reported. By unticking the check box, it is possible to select a maximum number of nodes to report.

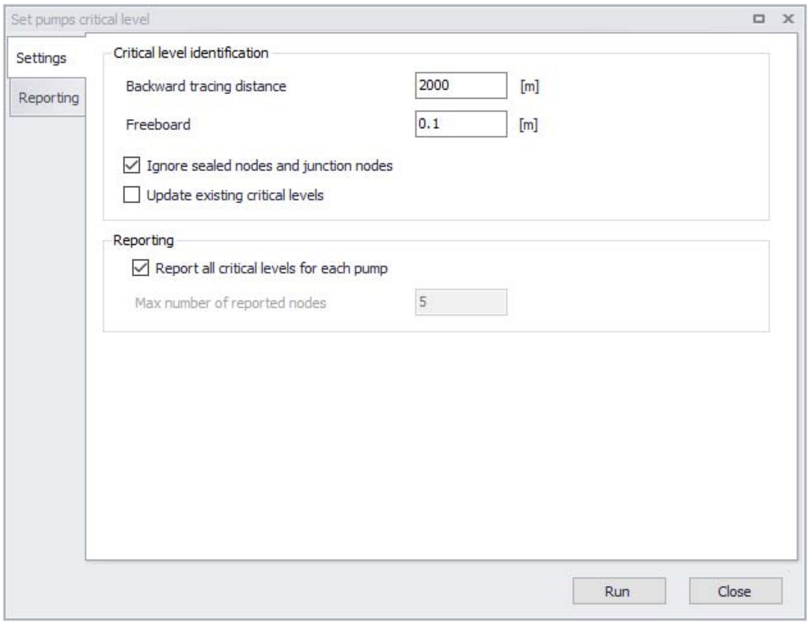


Figure 19.14 Configuring the Set pumps critical level tool

19.6.3 Running the tool

To assign critical levels to the pumps' wet well nodes, click on "Run". The 'Reporting' tab shows a formatted "report" describing the settings applied in the analysis, and detailed information for each pump. For each pump, the report gives the critical level for each of the nodes found on the backward-traced path, sorted by increasing 'Critical level' value. If 'Report all critical levels for each pump' was unticked, then only the user-defined number of nodes is reported for each pump.

The 'Source type' indicates, for each node, where the lowest level is found between the ground level and any connected weir or orifice discharging out of the modelled network. The 'Source level' reports the corresponding minimum level in the node, and the 'Critical level' is the source level minus the freeboard.

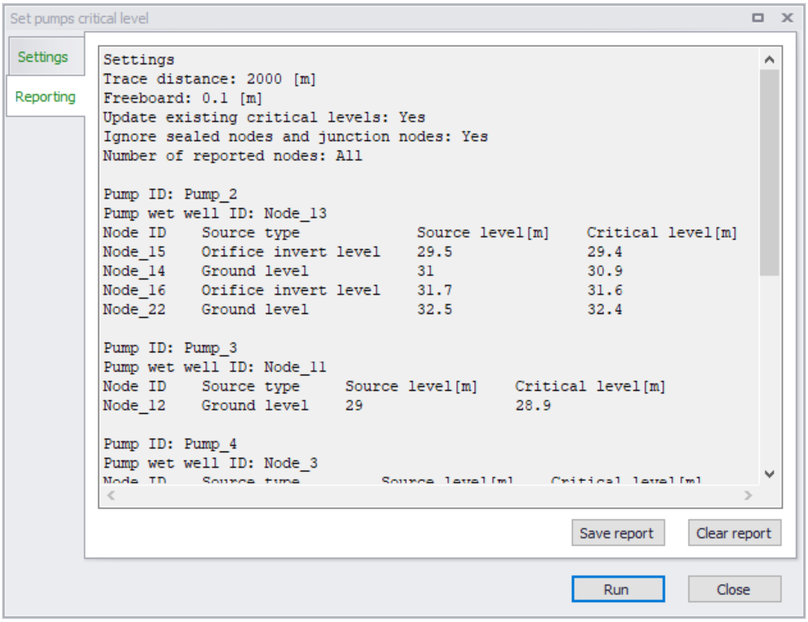


Figure 19.15 Report from the Set pumps critical level tool

The 'Save report' button will save the content to a text file, and the 'Clear report' button with clear the text box.

19.7 Transfer MIKE 1D data to SWMM tool

It is possible to transfer model data from MIKE 1D Collection System tables to SWMM tables. That means that Collection System data used in the working mode 'Rivers, collection system and overland flows' are converted to data for use in the working mode 'SWMM5 collection system and overland flows'. This tool is accessed from the 'CS network' tab in the ribbon, under the 'Special tools' button.

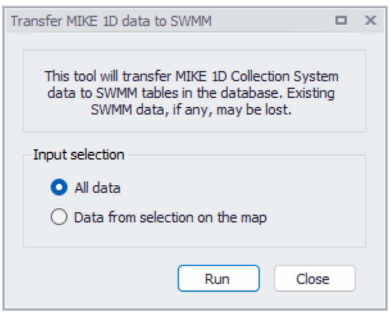


Figure 19.16 Transferring data from MIKE 1D to SWMM

Two options are also available to control which data are being transferred:

- All data
- Data from selection on the map: for this option, only the MIKE 1D data being selected when the tool is executed are transferred. This requires that the active working mode is 'Rivers, collection system and overland flows' in order to select the MIKE 1D data. If the active mode is 'SWMM5 collection system and overland flows', this option is therefore unavailable. Note that this will transfer all selected data, even though some are actually not displayed on the map (e.g. curves and relations data).

The following statements apply during this data conversion:

- Both node types 'Basin' and 'Soakaway' are converted to 'Storage unit' type. Node types 'Junction' and 'River junction' are not supported in SWMM and are not transferred.
- ‘Initial depth' in SWMM nodes is always set to 0, regardless of the actual initial condition defined in the MIKE 1D simulation.
- 'Ponded area' in SWMM nodes is computed assuming that 'Node cover expansion coefficient' = 1000 in the 'MIKE 1D engine configuration' settings.
- Links with type 'Natural channel' are not transferred, because they use a varying cross section along the link, which is not supported by SWMM.
- Orifices with geometry type 'Generic shape' are not supported by SWMM, and are converted to rectangular orifices with dimensions undefined.
- Weirs with formula 'Weir formula' are transferred with all their attributes, however weirs with other formulae ('Energy loss coefficient', 'Q-H' and 'Weir formula, custom shape') are transferred with an undefined weir coefficient: they must be reviewed and defined manually.
- Weir crest levels in MIKE 1D are converted to crest height in SWMM, relative to the invert level of the 'From node' of the weir.
- Only the 'Generic shapes' with type 'X-Z Open' and 'X-Z-R-M Open' are transferred to SWMM, while the others are not supported and skipped.
- From the list of 'Curves and relations', only the following types of curves are transferred:
  - Capacity Curve QH: converted to curve type 'Pump curve 2 (Depth-Flow)'. Note that values are transferred unchanged, and should be converted manually afterwards to convert absolute values of H (levels) used by MIKE 1D to heights relative to the manhole invert, as used by SWMM.
  - Capacity Curve QdH: converted to curve type 'Pump curve 3 (Head-Flow)'
  - Basin Geometry: converted to curve type 'Storage'.

- If multiple connections are defined for a catchment in MIKE 1D, only the first one will be transferred in SWMM, and it will be transferred like a 'Standard' connection accepting all runoff.
- Hydrological parameters are taken from the 'Kinematic wave' hydrological model (impervious steep, impervious flat and pervious medium values).
- Boundary conditions are not transferred.

19.8 Transfer SWMM data to MIKE 1D tool

It is possible to transfer model data from SWMM tables to MIKE 1D tables. That means that SWMM data used in the working mode 'SWMM5 collection system and overland flows' are converted to data for use in the working mode 'Rivers, collection system and overland flows'. This tool is accessed from the 'CS network' tab in the ribbon, under the 'Special tools' button.

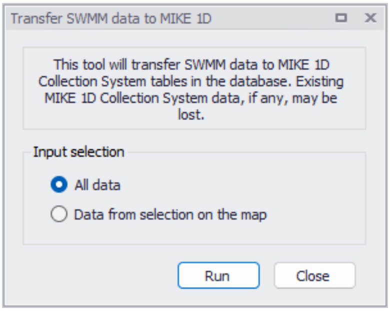


Figure 19.17 Transferring data from SWMM to MIKE 1D

This means that if you e.g. create or import a SWMM5 model into MIKE+, it is then possible to convert the model data into the MIKE 1D format in order to continue the modeling with the MIKE 1D engine instead.

Two options are also available to control which data are being transferred:

- All data
- Data from selection on the map: for this option, only the SWMM data being selected when the tool is executed are transferred. This requires that the active working mode is 'SWMM5 collection system and overland flows' in order to select the SWMM data. If the active mode is 'Rivers, collection system and overland flows', this option is therefore unavailable. Note that this will transfer all selected data, even though some are actually not displayed on the map (especially transects).

The following details apply during this data conversion:

- Rainfall-Runoff parameters from SWMM are transferred to the Kinematic Wave (B) model in MIKE1D.
- The operation creates one parameter set per catchment in the table msm_HParB, even if those parameter sets have the same attributes. It populates some attributes of Impervious Steep, Impervious Flat and Pervious Medium, while leaving the other attributes with value 0.
- The operation creates one catchment connection of load type 'Standard', which accepts all runoff and catchment discharge.
- The transfer creates one boundary condition per rain gauge, with boundary type 'Rainfall'. Each rainfall boundary is applied to a list of catchments, which contains all catchments associated with the corresponding rain gauge. The temporal variation of the boundaries refers to a file "Rainfall.dfs0", which the user has to create manually. Each rain gauge should be placed in a separate column in this time series.
- The following SWMM data are converted to Generic shapes:
  - Transects.
  - Conduits' geometry, for conduit types Trapezoidal, Open rectangular, Triangular, Horizontal ellipsoid, or Modified baskethandle.
  - Curves defined in the 'Curves and relations' editor with the type 'Shape'. This shape is scaled with the height of the first SWMM conduit using this shape: if multiple SWMM conduits use the same shape but with different heights, then additional generic shapes must be created manually for MIKE1D, after running the tool.
  - Weir geometries for weir types V-Notch and Trapezoidal.

20 Results Presentation

20.1 Introduction

With MIKE+ you can present your results in a number of ways:

- Maps (see Displaying Results on a Map, page 394)
- Time series plots (see Time Series Plot, page 403)
- Tables (see Results Table, page 410)
- Profile plots (see Profile Plots, page 417)
- Cross section plots (see Cross section Plots, page 456)
- Bar charts (see Bar Chart, page 444)
- Q-H plots (see Pump Q-H Plot and Hydrant Q-H Plot, page 468)
- Scatter plots (see Scatter Plot, page 463)
- Calibration and statistics (see Calibration Plots and Reports, page 523)

When working with time-varying results, all these results presentation windows are always synchronized to show results from the same date and time. See Animations (page 471) for more information.

You can visualize results with an active project, which allows visualizing both input model data and results at the same time. MIKE+ can also be used for standalone results viewing, i.e. without any opened project / database. To achieve this, simply open MIKE+ and then open the Results panel to load result files.

Before displaying results, result files must first be loaded in MIKE+, in the 'Results' tree. Results are, by default, automatically loaded into the MIKE+ project after a simulation. They can however be added manually using the 'Add file' buttons in the 'Results' tree or in the ribbon.

Note: the automatic loading of simulation results may be disabled via the 'User preferences' (page 54) dialog.

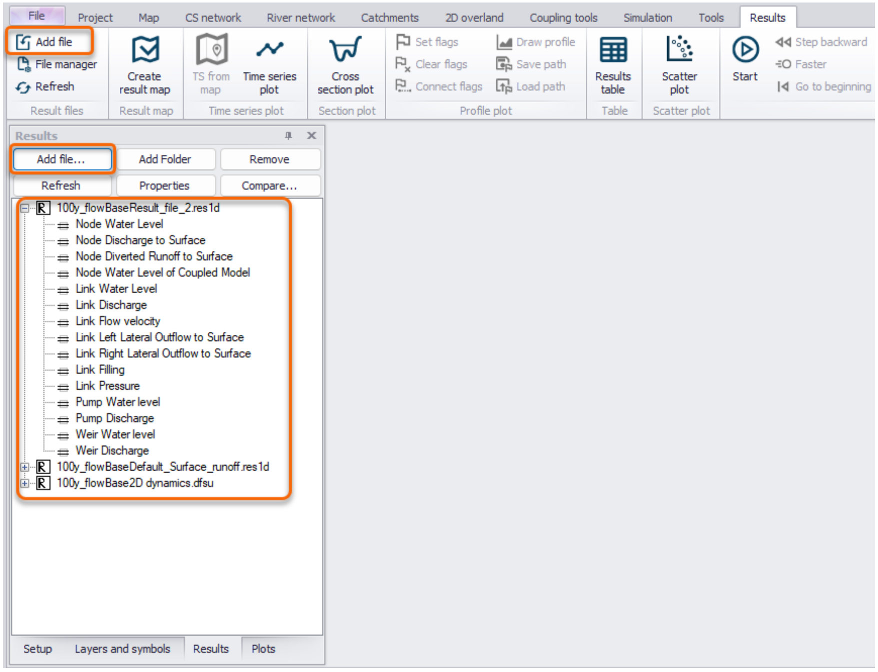


Figure 20.1 The list of loaded result files and the 'Add file...' buttons

When loading a result file, a dialog appears with options to control the content to load into memory:

- Time period to load: this can be used to ignore parts of the simulation period, e.g. to limit memory usage for large result files, or to ignore an initial period which is influenced by inaccurate initial conditions of the simulation. This is only available for 1D result files.
- Step every: this is the time step frequency to load, which can be used to skip some time steps during loading to limit memory usage for large result files. This is only available for 1D result files.
- Data items to load: selection of result items which should be loaded in the project. This list is a combination of result items saved in the result file (which can be selected from the 'Result Files' editor before running the simulation, for MIKE1D and 2D results), and derived results which are computed in memory. See 'Result Files' (in MIKE+ Collection System User Guide) for description of derived items for MIKE1D results, and ‘Output’ (in MIKE+ Water Distribution User Guide) for description of derived items for Water Distribution results. For 2D results, all result items in the file are always loaded.
- Items to plot on map: selection of result items to show on the model map while loading the file. This option is only available when a project database is opened. Result layers can alternatively be added later to the map from the 'Layers and symbols' tree.

Press 'OK' to finish loading result files items. Only the loaded data will later be available and visible in the results presentation windows.

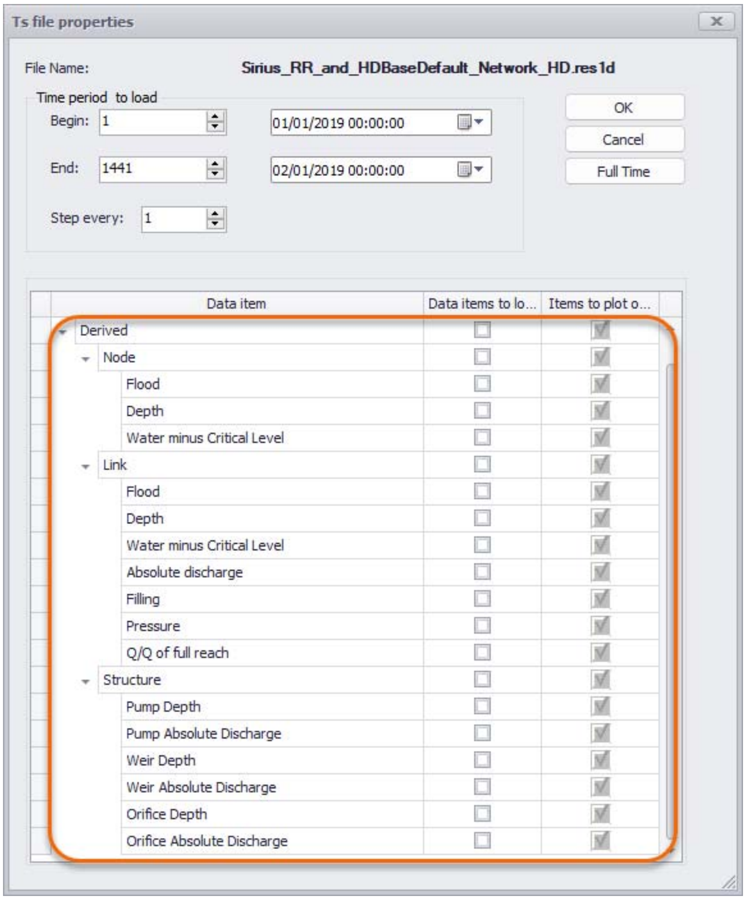


Figure 20.2 Example of the result file data selection dialog

The following result file types are supported in MIKE+:

- From Collection System and River simulations:
  - res1D files (network or catchment results varying in time and space)
  - dfs0 files (time series)
- From 2D Overland simulations:
  - dfs0 files (time series)
  - dfsu files (surface results varying in time and space on a flexible mesh)
  - dfs2 files (surface results varying in time and space on a rectangular grid)
- From SWMM simulations:

  - .out files (network and catchment results varying in time and space)

- From Water Distribution simulations:
  - .res files (standard hydraulic and water quality results)
  - .resx files (extended hydraulic results)
  - .whr files (water hammer results)
  - .whrx files (extended water hammer results)
  - .msxr files (multi-species water quality results)
  - .csv files (fire flow, network vulnerability and flushing analysis results). The *.csv file is a comma-separated text file which can also be opened in other editors like Microsoft Excel, if necessary.

Note that SWMM and Water Distribution result files don't store the network geometry (pipe connectivity, levels, etc.). When opening these results files in MIKE+ without opening a project (database), it is therefore recommended to keep the *.inp simulation file created with the result file, from which this network geometry can be retrieved, otherwise some results viewing functionalities are disabled (showing result map layers and profile plots).

Once result files are loaded, result presentation tools are accessed via the local context menu (i.e. right-click) on the file names in the 'Results' panel, as well as from the Results ribbon.

20.2 Displaying Results on a Map

Simulation results map be plotted on:

- Result Map. A map result document.
- Map View. The main Map view with the model layers.

Result Map

To create a new result map, right-click on the result file or one of its result items in the list of result files, or use the 'Create result map' button in the ribbon.

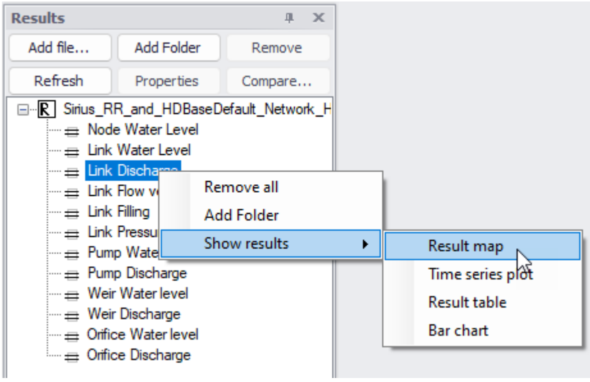


Figure 20.3 Creating a result map from list of result files

Figure 20.4 'Create result map' option on the Results ribbon

Both options will open the 'Create result map' window below.

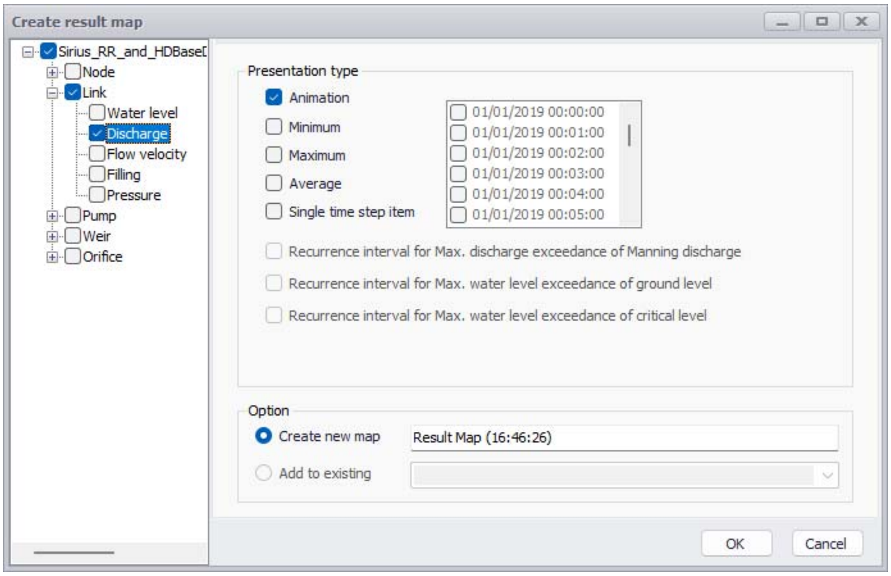


Figure 20.5 Choosing plot value type to display on the map

To create the result map:

- Choose among the data types to display on the map (e.g. water level in nodes, discharge in links, etc.) by ticking on the check boxes. The available data types are dependent on the result file and loaded result items.
- Select the presentation type:
  - Animation. Time-varying results.
  - Minimum. Minimum value statistics. Not available for 2D result files.
  - Maximum. Maximum value statistics. Not available for 2D result files.
  - Average. Mean value statistics. Not available for 2D result files.
  - Single time step. Option for plotting results at a particular time step. Select the date/time period from the input box on the right.
  - Single recurrence interval. For LTS extreme results. The left list contains computed recurrence intervals saved in the result file, whereas the right list allows specifying custom recurrence intervals for which the results are interpolated using a linear interpolation method from the recurrence intervals in the result file.
  - Recurrence interval for max. discharge exceedance of Manning discharge. For LTS results. Manning discharge refers to full-flowing discharge.
  - Recurrence interval for max. water level exceedance of ground level. For LTS results.
  - Recurrence interval for max. water level exceedance of critical level. For LTS results.
- Select between the two options:
  - Create new map: the new result layer will be shown in a new window, named with the specified title.
  - Add to existing: the new result layer will be shown in the selected existing result window.
- Click the ‘OK’ button to create the map.

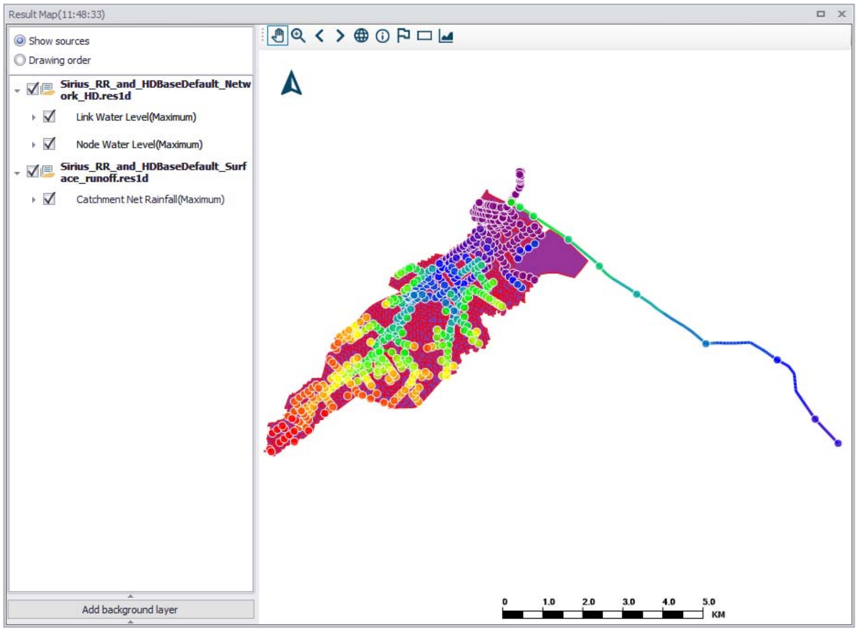


Figure 20.6 Example Map plot of rainfall, and node and link water levels.

The toolbar at the top of the map contains various tools, amongst which:

- Various tools to pan or zoom, to navigate on the map
- Identify: shows the properties and the result values of a network item in the property view of the result map, after selecting the item on the map.
- Add result items: opens a result selection window, to add extra result layers on the map.
- Flags and profile plots: adds flags on the result map to display a profile plot view along the network.
- Add time series from map: opens the 'TS from map' tool, to show time series from locations selected on the map. See ‘Time Series Plot’ chapter (page 403) for more information.
- Save to plots manager: saves the result map (with its list of result items) to the 'Plots' panel. The result map will initially be added to the active folder from this panel. See ‘Plots Management’ chapter (page 487) for more information on options to save and manage results windows.

Map View

It is also possible to directly visualize result items on the model Map (i.e. Map View).

Automatic loading of results into the project also loads several pre-selected result items into the Layers and Symbols tree view for visualizing on the main Map (Figure 20.7). When manually loading simulation results into the project, result items may also be chosen to be shown on the model map. See “Loading Results” on page 414. These result items are then also included in the Layers and Symbols tree view panel for showing on the main Map view.

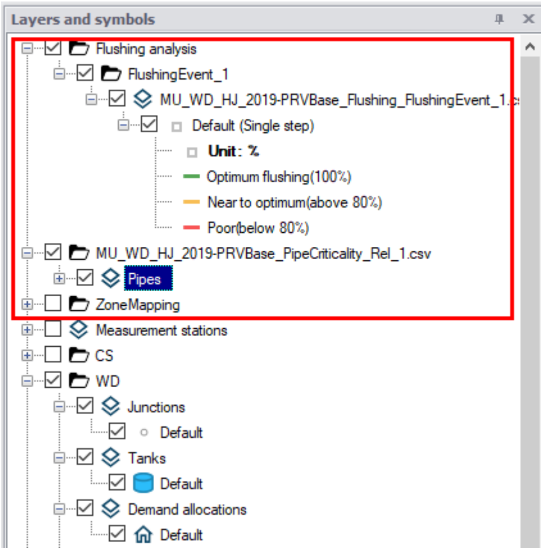


Figure 20.7 Example result items loaded into the Layers and Symbols tree view for showing on the main Map

Customization of the layer appearance is then performed via the Symbology Settings editor launched from the Layers and Symbols tree view by double clicking on a result item layer name.

Labelling and Symbology

To customize labelling and symbology of a result layer on the main Map view, edit the Symbology settings as for any other layer shown on the Map view.

To customize labelling and symbology on an extra result map layer:

- On the left panel of the result plot, right-click on the result item layer and choose ‘Show property panel’ from the local context menu. The property panel appears at the bottom left corner of the window (Figure 20.14).
- Click the ‘Edit style’ button in this property panel (Figure 20.14).

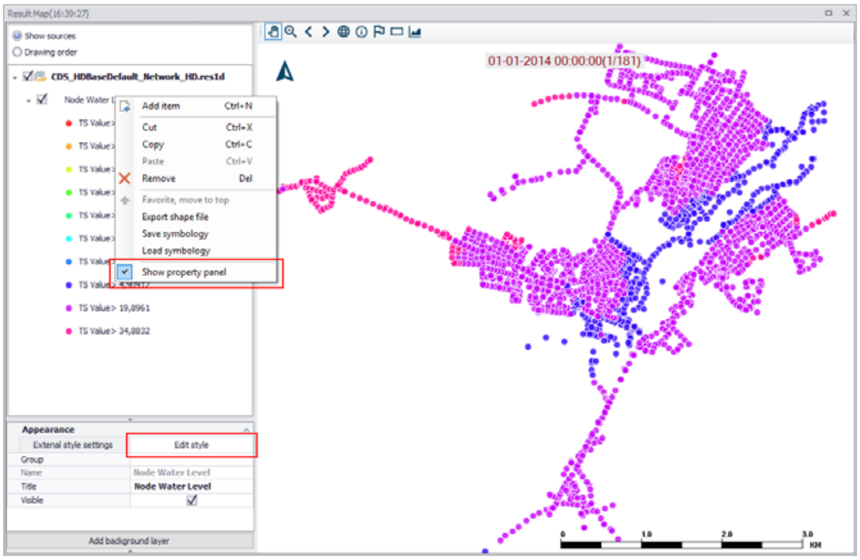


Figure 20.8 Edit a result layer symbology and label via the ‘Edit style’ button

This will open the ‘Symbology settings’ editor.

Flow Arrows

An option for showing direction arrows for line result layers is available in the Symbol tab.

Activate the ‘Draw direction arrow’ to display flow arrows along line features. Define arrow placement along features.

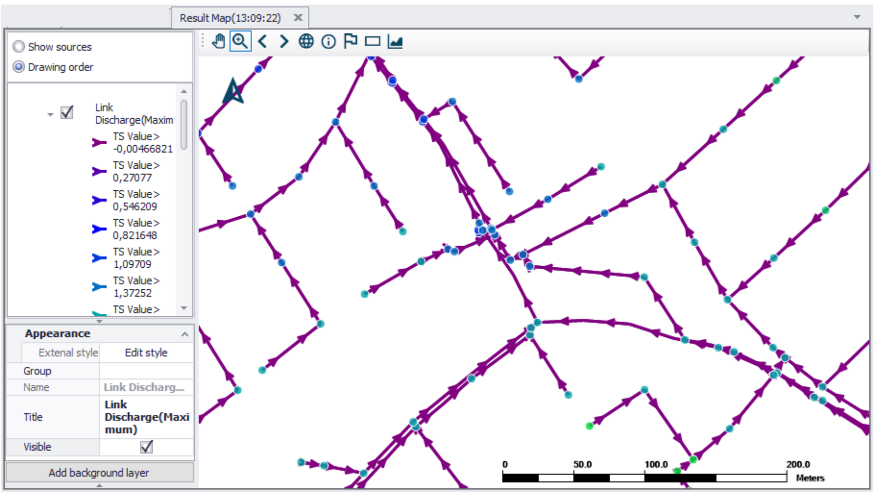


Figure 20.9 Example plot of link discharge results showing flow arrows

Save Symbology

It is possible to save the symbology and label settings for a result layer into a file, which may then be loaded and used for another result layer.

Right-click on a result layer and select “Save symbology” from the local context menu.

“Load symbology” allows loading a previously saved configuration file.

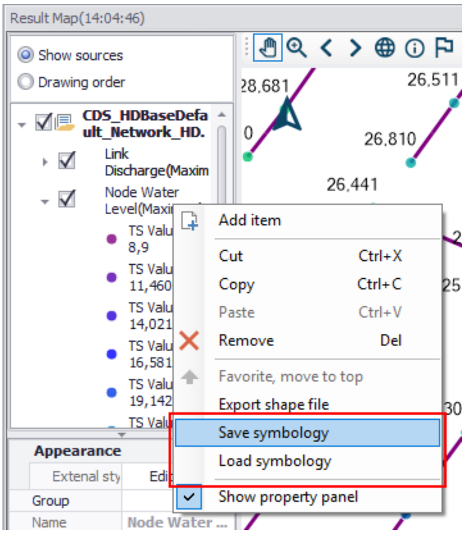


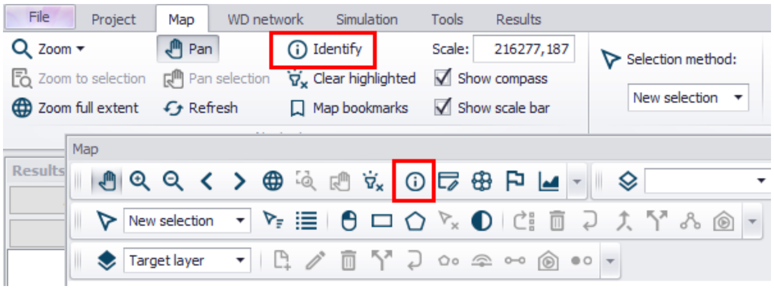
Figure 20.10 Options for saving or loading symbology settings from the left panel local context menu

20.3 Property and Result Explorer

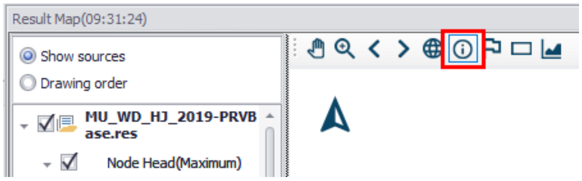
It is possible to quickly query associated result items by selecting model elements from maps via the Identify tool.

The tool is available for maps in several locations on the interface:

- **Map**: For the main Map view, the tools is available from the Map ribbon, and the Map view toolbar.



- **Result map**: For result maps, the tool is available from the result map toolbar.



Once the Identify tool is activated, you can click on any node (junction, reservoir, tank) or link (pipes, pump, valve) on the map and the Property and Result Explorer will display data attributes and result values.

Map View

On the Map view, use the Identify tool to view the Properties and Result Explorer (Figure 20.11). The Properties and Results Explorer displays result files that are currently loaded as a **Map layer**. It shows result values and also gives information about the element selected on the map.

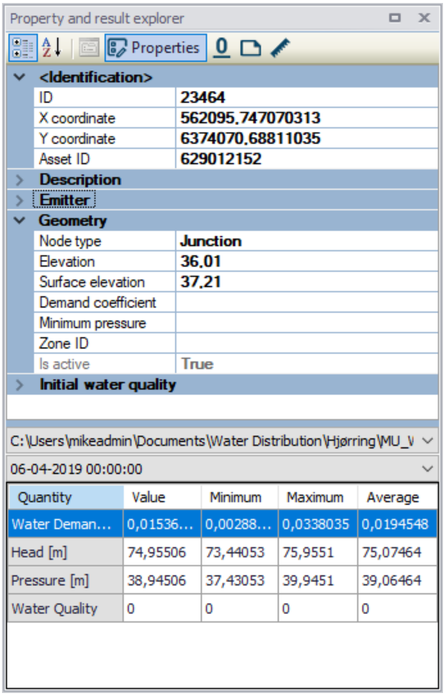
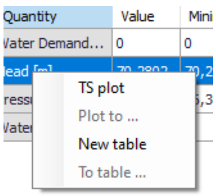


Figure 20.11 The Property and Results Explorer on the main Map view

In addition, it is also possible to create a Table or TS Plot (i.e. time series) result document from the Property and Result Explorer table's context menu (i.e. via right-click):



Result Map

On result map plots, use the Identify tool to view tabulated result values on the lower left panel of the plot (Figure 20.12). The table displays element properties, result files that are currently loaded in the **Results manager**, and result values.

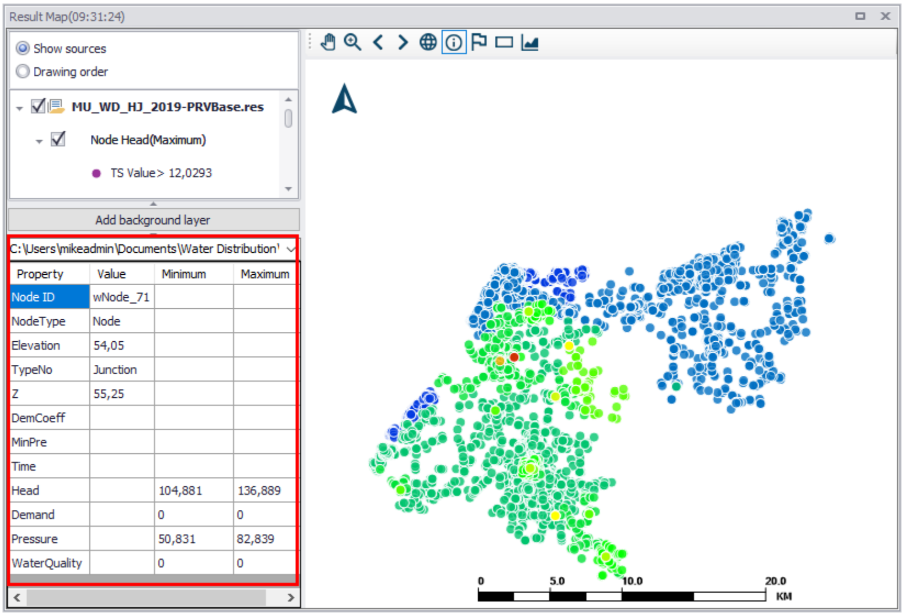


Figure 20.12 The results explorer on result map plots

20.4 Time Series Plot

Time series can both be input time series or time series taken from result files.

In this section, the focus is on displaying time series from result files. Start by loading a result file into MIKE+, see Chapter 20.3 Loading Results.

There are basically two ways to create new time series plots:

- **From a list of files and locations, using the ‘Create time series plot’ tool**. This tool is only relevant for 1D result files and dfs0 time series files. This tool is accessed from the 'Time series plot' button in the 'Results' tab in the ribbon, or from a right-click on a result file or one of its result items. In this dialog:
  – Choose the result file, result item and locations to plot from the list. Use the filters to search through the potentially long list of available locations, to filter on ID and/or chainages (chainage filtering being available only for link results from .res1d result files). One may also use a selection list (saved in the ‘Selection manager’) or the active selection, to select all items from the selected list.
  – Select between ‘Create new plot’ and ‘Add to existing’ options and click on the ‘OK’ button.

- **By picking a location on a map, using the 'TS from map' tool**. This tool is accessed from the 'TS from map' button in the 'Results' tab in the ribbon The 'Add time series from map' dialog appears, wherein one specifies:
  - Source file
  - Source group
  - Source item

  Specify whether to 'Create a new plot' or to 'Add to existing', click on the 'OK' button and finally select the corresponding model item on the map for which to plot time series results. To include multiple locations in the plot, either hold the 'Ctrl' key down while clicking on the map, or continue to click locations once the plot has been created. To stop adding more time series when clicking on the map, deactivate the 'TS from map' button in the ribbon.

Figure 20.13 Time series plot tools on the Results ribbon

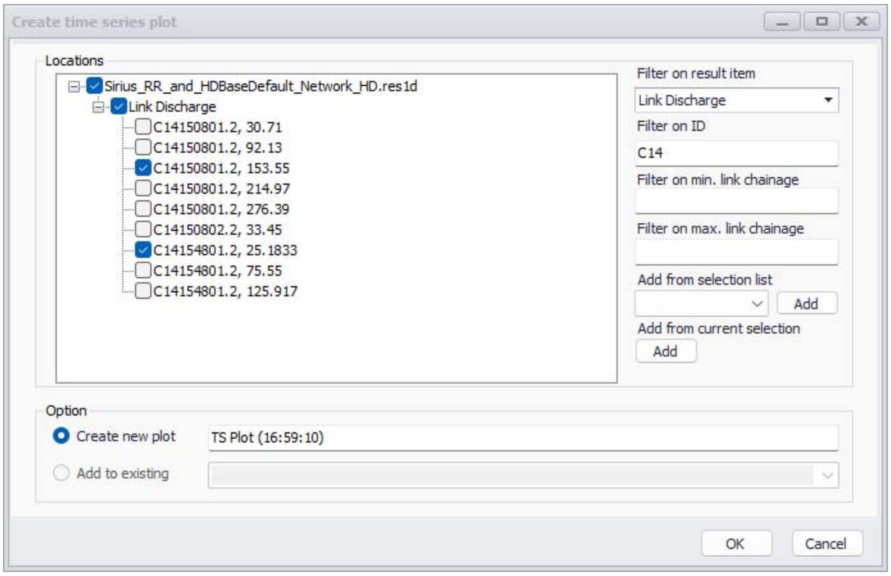


Figure 20.14 'Create time series plot' window, showing the full list of result locations

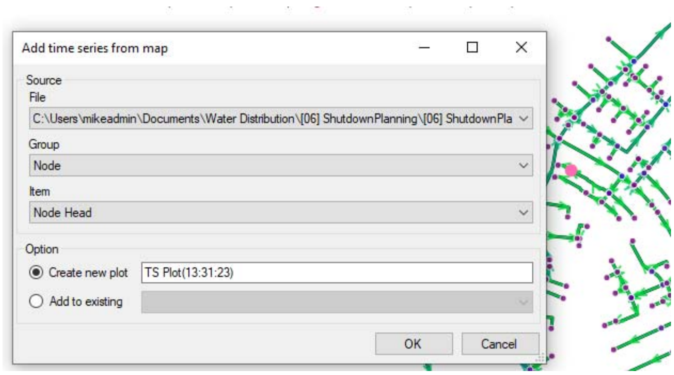


Figure 20.15 The 'TS from map' tool, to select result locations from the map

Once a time series plot is created, additional result time series can be added to the plot in various ways:

- Right-click in the table of content of the time series plot and select 'Add items
- Use the 'TS from map' tool with the option to add the new items to an existing plot
- Use the 'Create time series plot' tool with the option to add the new items to an existing plot
- Drag a result item from the 'Results' tree (click the '+' box to the left of a result file name to expand its list of loaded result items) and drop it in the table of content of the time series plot
- Add result items from another time series plot, using the Copy / Cut / Paste options in the context menu of the table of content.

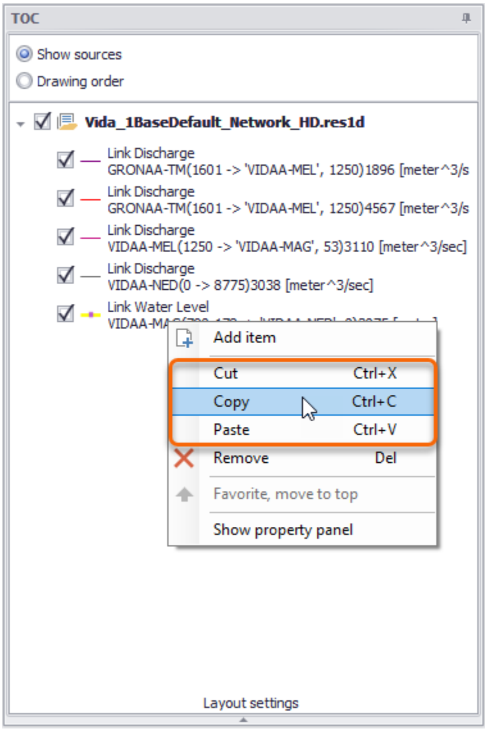


Figure 20.16 Options to copy and paste time series from one plot to another

Note: In the 'Create time series plot' window, filters are used to hide some items in the list but do not unselect them, i.e. a location can be selected and later become hidden using a filter, but will still be plotted when clicking 'OK'. The buttons 'Add from selection list' and 'Add from current selection' will tick all items from the selection, for all result files and result items which are visible, excluding those which are hidden due to filters.

Data series format

To customize the appearance of TS plot data series, right-click on a data series and activate the ‘Show property panel’ option from the local context menu (Figure 20.17).

Options for configuring data series appearance include customizing line color and style, adding markers, and changing marker styles and size.

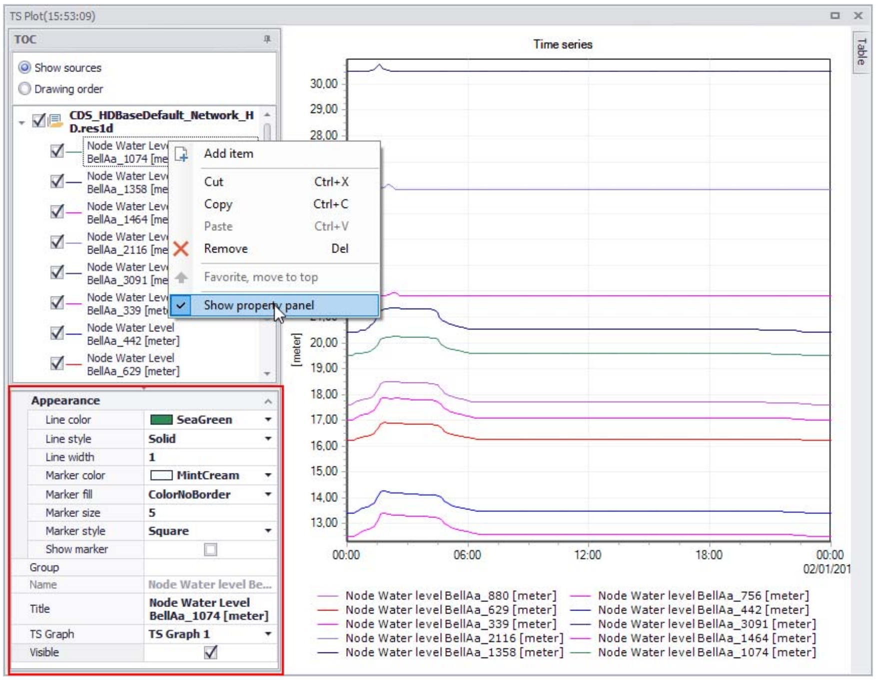


Figure 20.17 Customize the appearance of TS Plot data series via the Property Panel

Context menu

Right-click on the time series plot to access options to control the zoom level, save the time series plot, copy data or export to an image file.

'Zoom in' lets you draw a rectangle on the plot to select the area to zoom to. While drawing a rectangle, dragging to draw a horizontal line will display arrows to zoom along the horizontal axis only, keeping the vertical axis unchanged. Similarly, dragging to draw a vertical line will zoom along the vertical axis only. 'Zoom to full extent' brings you back to the full view of visible time series. Note that additional options are available to control the zoom options:

- Hold down the Shift key, to zoom in
- Scroll with the mouse wheel to zoom in or out
- Hold down the Ctrl key to pan.

The 'Bookmarks' menu opens the bookmarks manager which can save and restore zoom extents. It contains the following options:

- Add: adds a new bookmark, saving the current time extent shown on the plot

- Zoom to: zooms to the time period saved for the bookmark selected in the left list
- Rename: renames the bookmark selected in the left list
- Remove: removes the bookmark selected in the left list
- Close: closes the bookmarks manager.

Note that bookmarks only save the time extent (X axis), but not the extent on the vertical axis, and can therefore be applied to any plots even if they don't all have similar ranges of Y values.

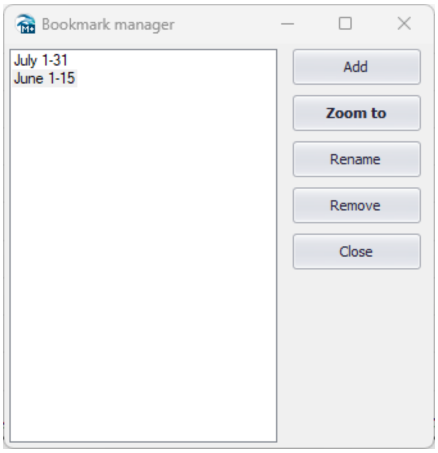


Figure 20.18 The bookmark manager from the time series plots

'Sync zoom in all plots' allows to apply the time extent (X axis) from one plot to all other opened time series plots. Two options are available:

- This time only: this option applies the time window from the active plot (from which the option is activated) to all other opened time series plots. Later changes to the zoom level in any plot have no automatic effect on the other plots.
- Always: this enables an automatic synchronization of the time window in all opened time series plots. Any change to the zoom level along the X axis in any plot also applies instantaneously to the other plots. Click again this 'Always' option to turn off this automatic synchronization.

'Save to plots manager' will save the time series content (list of time series locations and display settings) to the 'Plots' panel. The time series plot will initially be added to the active folder from this panel. See 'Plots Management' chapter (page 487) for more information on options to save and manage results windows.

'Copy plot to clipboard' will copy the plot as an image in memory, to be pasted in another program. 'Copy data to clipboard' will copy the time series values in memory, to be pasted in another program.

'Save data to .dfs0 file' will save the time series to an external .dfs0 file, which can later be e.g. used as boundary condition for a future simulation or loaded as a result file in the 'Results' tree.

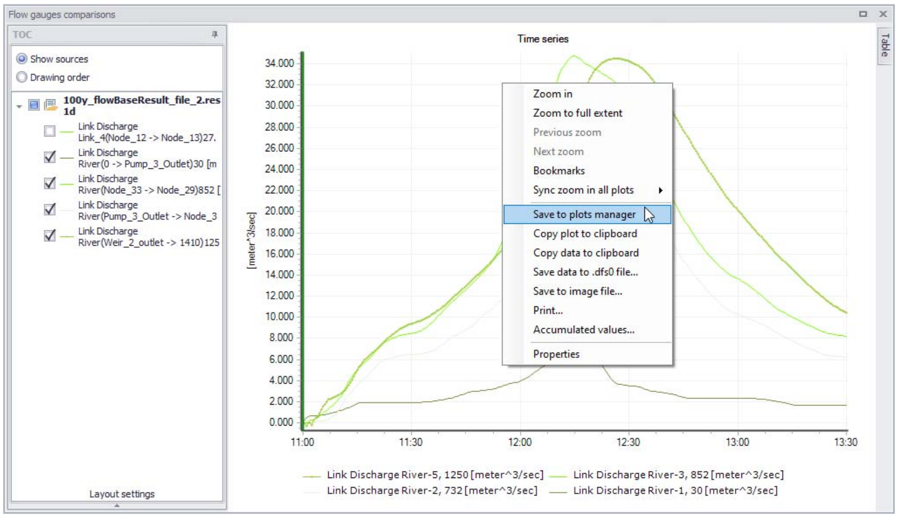


Figure 20.19 Context menu of the time series plot

When a time series has a unit which can be accumulated over time (e.g. discharge), the context menu offers an 'Accumulated values' option, which will compute the accumulated values over a given period of time. Three options are available to define this period: entire period, zoom extent, or custom period. In the latter case, the start and end date and time of the period can be customised.

After selecting the period, press the 'Compute' button to view the results of the accumulated values over the selected period, for each time series item.

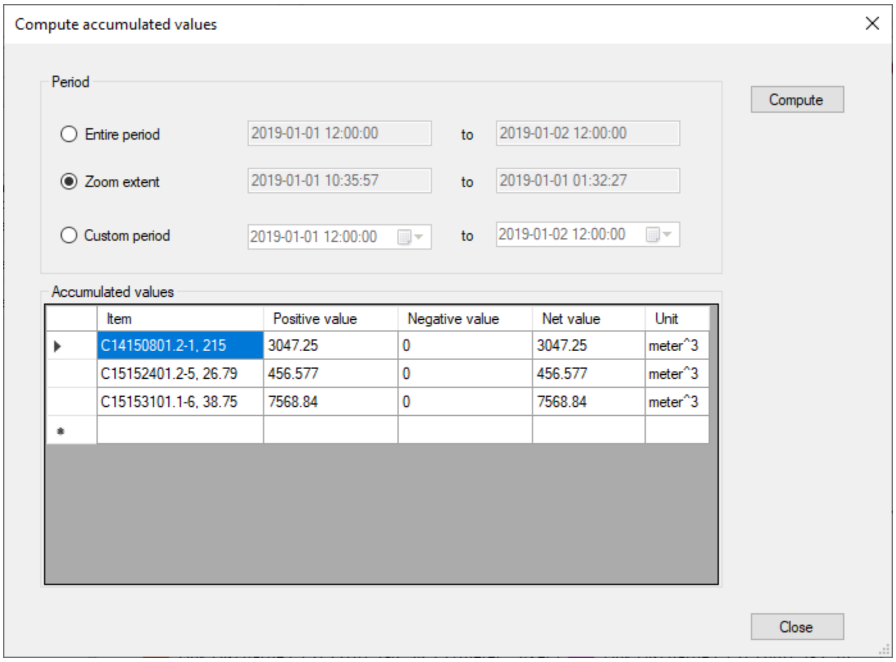


Figure 20.20 Compute accumulated values from time series

For each time series, the table shows :

- The positive value, which is the accumulated result of the positive values from the time series
- The negative value, which is the accumulated result of the negative values from the time series
- The net value, which is the accumulated result of all values from the time series.

Note: These accumulated values are computed from the time series stored in the result files, and therefore the accuracy of the accumulated values depends on the saving frequency of the results. If the saving frequency is low (i.e. long time step between two saved results), then the available results will not reflect significant variations of results between two saved time steps, and the calculated accumulated values may deviate from the actual simulated accumulated values.

The context menu also offers a 'Properties' option to control the layout and the symbology of the time series plot.

20.5 Results Table

The results table provides an overview of all or selected results in tabular form. This is only available for 1D network results.

To create a new result table, right-click on the result file or one of its result items in the list of result files, or use the 'Results table' button in the ribbon.

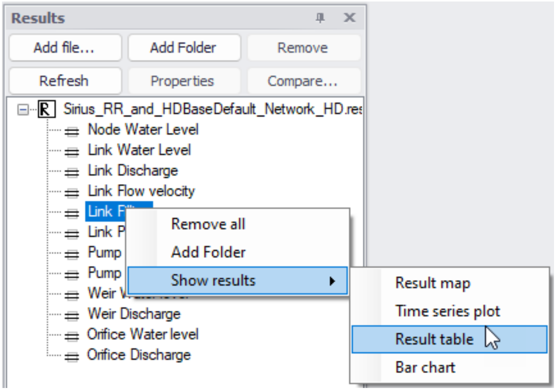


Figure 20.21 Creating a result table from the list of result files

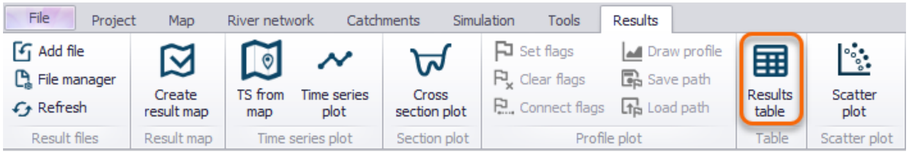


Figure 20.22 'Results table' option on the Results ribbon

Both options will open the 'Create result table' window below.

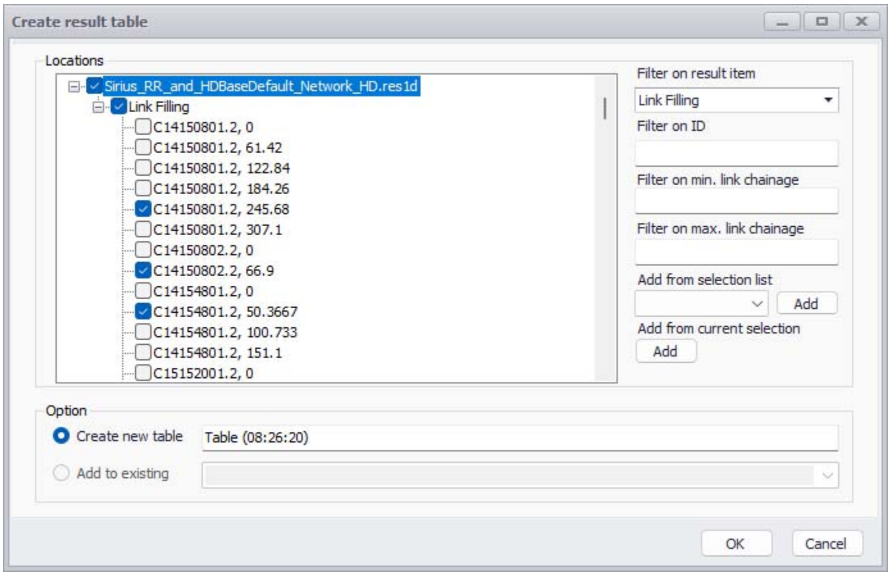


Figure 20.23 The 'Create result table' window

Choose the result file, result item and locations to plot from the list. Use the filters to search through the potentially long list of available locations, to filter on item type, ID and/or chainages (chainage filtering being available only for link results from .res1d result files). One may also use a selection list (saved in the 'Selection manager') or the active selection, to select all items from the selected list.

Select between the two options:

- 'Create new table': the new result data will be shown in a new window, named with the specified title.
- 'Add to existing' options: the new result data will be shown in the selected existing result table.

Then click on the 'OK' button to create the table. This will open the results table in a new window.

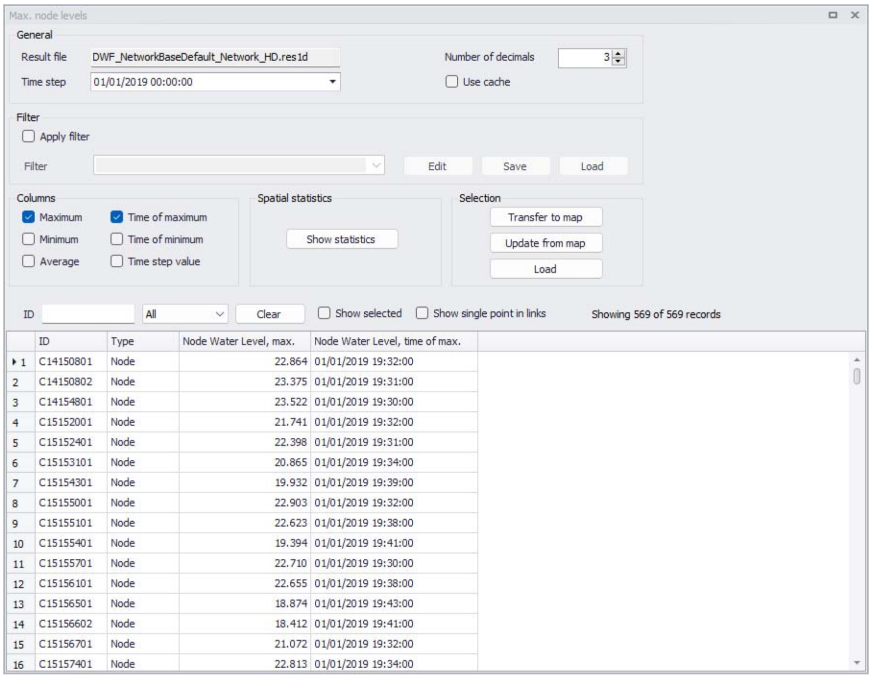


Figure 20.24 The Results table window

20.5.1 General

The 'General' group at the top shows the following items:

- Result file: the name of the result file, shown for information only (non editable).
- Time step: the date and time of the result time step. This is only relevant for result columns showing instantaneous results (when the 'Time step value' column is included).
- Number of decimals: controls the number of decimals to be shown in the table.
- Use cache: when this option is selected, displaying the table may be faster but will also use more memory on the computer.

20.5.2 Filter

The filter can be used to reduce the number of records shown in the table, using any type of filter criteria (e.g. based on item ID, item type, result value).

When 'Apply filter' is active, it is possible to create a new filter or edit the current one, using the 'Edit' button. Alternatively, it is possible to change to a previous filter, from the drop-down list showing all previous filters. This list of previously-applied filters can be saved to an external file using the 'Save' button, in order to be loaded later in other tables using the 'Load' button.

Clicking 'Edit' will open the filter editor. The 'Text' tab shows the filter's expression which can be manually edited. The 'Visual' tab offers an easy way to edit the filter. Add new criteria by clicking the '+' button, and click the various components to edit e.g. the column to be filtered or its filtered value.

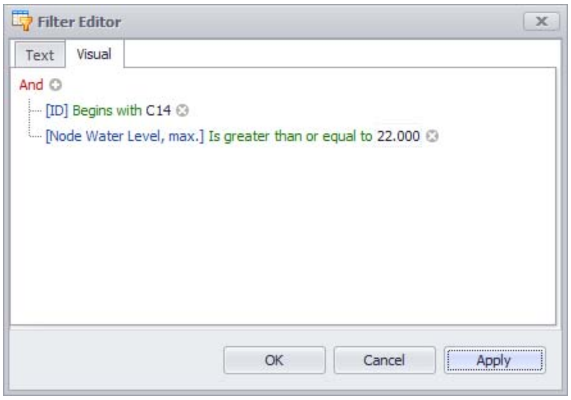


Figure 20.25 The filter editor

20.5.3 Columns

Select here which columns should be displayed in the table:

- Maximum: the maximum value during the simulation period, for each calculation point.
- Time of maximum: the time at which the maximum value is computed.
- Minimum: the minimum value during the simulation period, for each calculation point.
- Time of minimum: the time at which the minimum value is computed.
- Average: the average value during the simulation period, for each calculation point.
- Time step value: this column shows the instantaneous result values, corresponding to the date and time shown in the 'General' group and shown in the other result views (map, profile plots, etc.). Values in the 'Time step value' column therefore change when the time step of the results is changed.

If multiple result items are shown in the table (e.g. 'Node water level' and 'Node water depth'), the selected columns will be displayed for each of these result items.

Note: User defined columns, providing the result of a custom expression / equation, can also be added from the context menu. Refer to Section 20.5.6 Table *(p. 416)* for more information.

20.5.4 Spatial statistics

Click 'Show statistics' to obtain statistics computed throughout the entire network, either for the current time step only or for the entire simulation period.

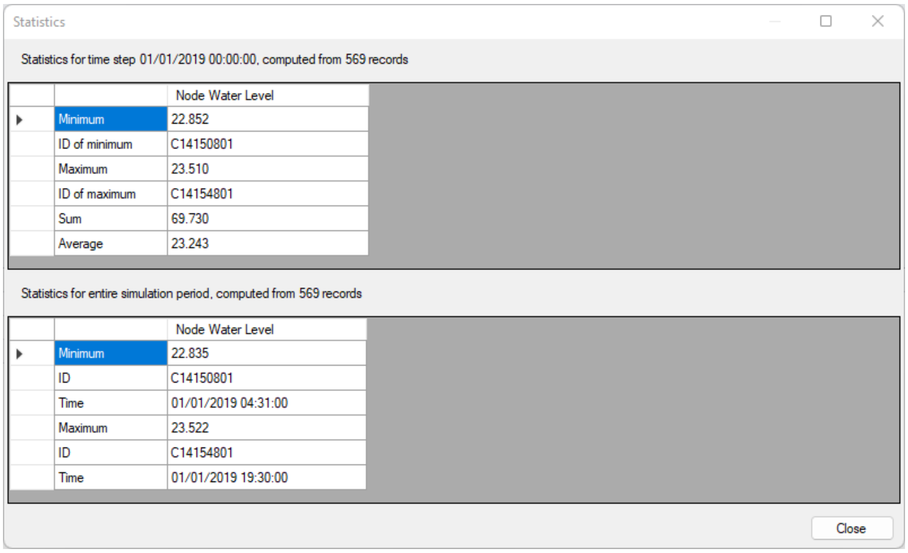


Figure 20.26 The spatial statistics tables

The 'Sum' statistics is not necessarily relevant for all result items. It is mainly relevant for 'Volume', 'Flooded area', 'Water flow rate to node volume about ground' or 'Mass error' result items.

20.5.5 Selection

Records in the results table can be selected independently of the selection of network items made on the map or in the model editors. The following buttons can be used to synchronize these two types of selection.

- Transfer to map: use the currently-selected records in the results table, and also select the corresponding items (nodes, pipes, rivers, etc.) in the model editors and on the map.
- Update from map: use the currently-selected records in the model editors and map, and select the corresponding items in the results table. If a selected item contains several result points (e.g. for rivers), then all records in the results table corresponding to the selected item will be selected.

- Load: use a selection saved to the 'Selection manager', and select the corresponding items in the results table. If an item from this selection contains several result points (e.g. for rivers), then all records in the results table corresponding to the selected item will be selected.

20.5.6 Table

The following controls are available above the main table:

- Filter: type some text to filter the list of shown item IDs, or use the drop-down list to show only some result types (nodes or links).
- Show selected: when this option is active, only the selected records in the table will be shown.
- Show single point in links: when this option is active, when some links contain several calculation points (i.e. several records for the same link), then only the first one will be shown in the table.

The context menu (right-click options) on the table offers the following options:

- Add item: Adds additional results for a specific location or a selection. To add additional result items for all locations (nodes or links), use instead the 'Results table' button in the ribbon to add more content to the existing table.
- Add user-defined column: Inserts additional columns to the table, with values derived from an expression. This expression can be a function of the result columns, as well as a function of the network properties (e.g. pipe diameter, node level, etc.).
- Save to plots manager: saves the result table (with its list of result items) to the 'Plots' panel. The result table will initially be added to the active folder from this panel. See 'Plots Management' chapter (page 487) for more information on options to save and manage results windows.
- Copy: Copies the values from the current selection.
- Select all: Selects all records in the table.
- Open selected in time series plot: Shows the time series from calculation points selected in the table. All result items added to the results table are shown in the time series plot.
- Export to text file: Exports the table to a text file in .csv format. This text file contains a table with columns separated by semicolons. All result items are listed in columns, whereas the various links and nodes are listed in rows.

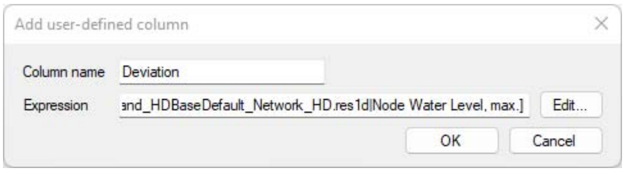


Figure 20.27 Defining the expression for a user-defined column

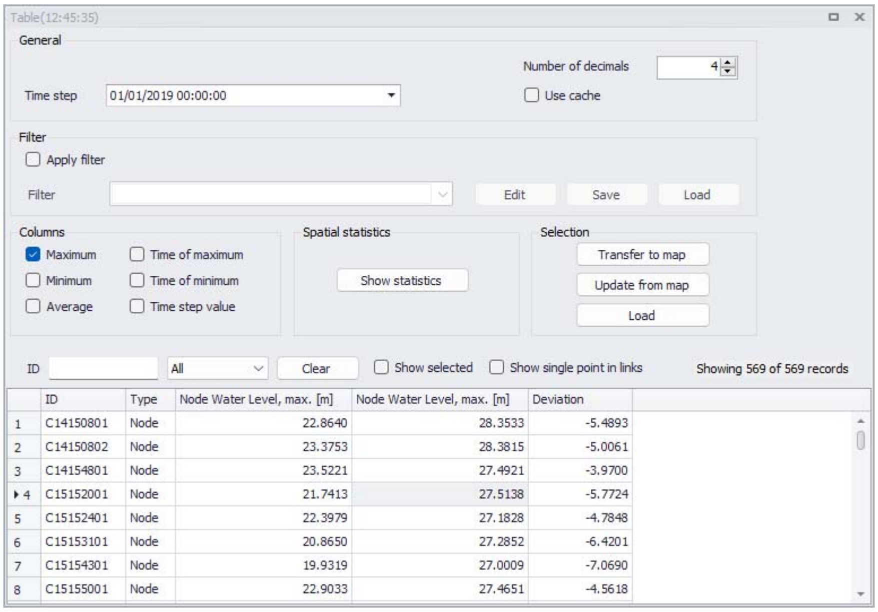


Figure 20.28 Example of user-defined column showing the difference of water level from two different result files

Note: When a table contains result columns from different result files, the file name can be seen in the fly-by text, while hovering over the column's header.

20.6 Profile Plots

Profile plots can be created from the main Map view, with or without results, or from a map result plot.

A profile plot is drawn between specified flags. If there are two or more possible paths between two flags, the path with the smallest number of links (smallest number of pipes, or smallest number of river reaches delimited by connections to other rivers or pipes) will be selected. Hence, in order to better control the path, more flags should be set until a unique path between flags can be identified. If multiple pipes exist between the same two nodes, the active pipe with the largest diameter or height will be selected.

The selected path can be seen on the map using the 'Connect flags' option. Using this option is however optional, and it is possible to draw the profile plot without connecting the flags.

20.6.1 Creating Profile Plots from the Map

Profile plots can be created from the model map (i.e. Map View) with or without simulation results.

- On the Map View, define path flags via ‘Set flags’ on the ‘Profile and Tracing’ toolbox on the Map ribbon (Figure 20.29).

Figure 20.29 The Profile and Tracing toolbox on the Map ribbon

- Click on the starting location for the profile path. Flags can be set on nodes, junctions, and rivers. This will place a small flag at the selected location along with the number of the flag (Figure 20.30).

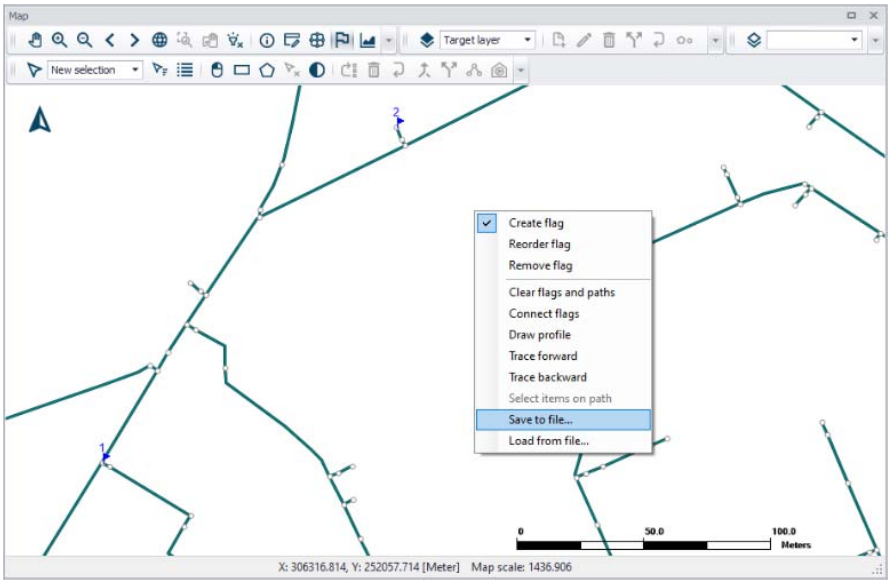


Figure 20.30 Set flags defining the path of the profile plot on the model map

- Continue setting flags on the map until the path is well-defined. The horizontal plan will then look as seen in Figure 20.30. You may save the set flag information using the 'Save to file...' option from the local context menu. The path information is saved to a *. PATH file, which can be loaded in another session via the 'Load from file...' option.
- Finally, click on 'Draw profile' on the Profile and Tracing toolbox. This will create a new profile plot (Figure 20.31).

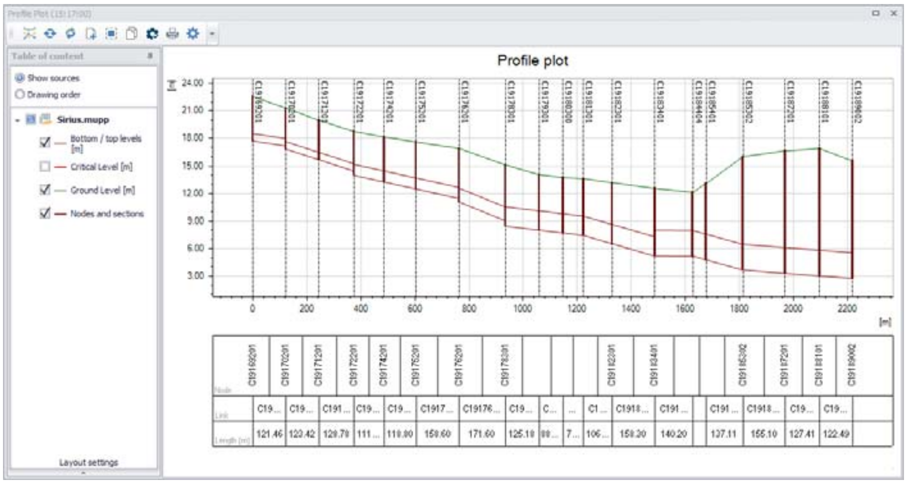


Figure 20.31 The Profile Plot form window showing an example longitudinal profile plot without simulation results

Profile Plot with Results

When result files are available, default results are plotted when a profile plot is created. Result items can also be added to a profile plot, following these steps.

- Add simulation results to a profile plot using the 'Add item' button on the Profile Plot window (Figure 20.32.). The option is also offered from the local context menu (i.e. right-click) on the left panel of the profile plot form. Alternatively, drag a result item (e.g. Link Water Level) from the list of result files in the 'Results' tree, and drop it in the table of content of the profile plot.

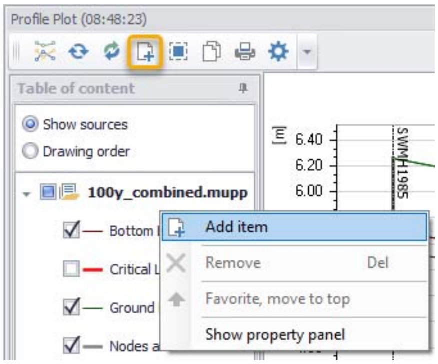


Figure 20.32 The 'Add item' tool on the Profile Plot window

- Specify the result file, item, and data type to add to the profile plot on the 'Add item' dialog that appears (Figure 20.33).

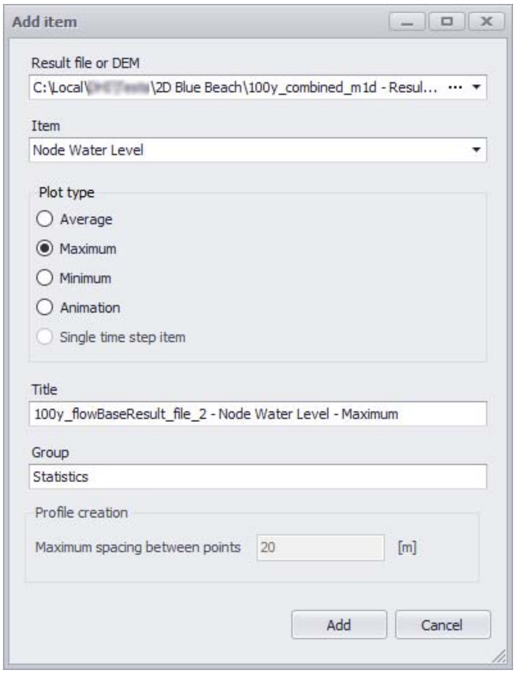


Figure 20.33 The 'Add item' dialog where the result file and items to add to the profile plot are defined

- If the selected result file is a 'LTS extreme statistics' type of file, the plot type is always a single time step representing a recurrence interval, which must be selected from the list which appears in the 'Add item' dialog. The left list contains computed recurrence intervals saved in the result file, whereas the right list allows specifying custom recurrence intervals for which the results are interpolated using a linear interpolation method from the recurrence intervals in the result file.
- If the selected result file is a 2D file from a 2D overland simulation, a maximum spacing must also be specified: it controls the spacing between points along the 2D profile.

The added result item then appears on the Profile Plot (Figure 20.34). Result items are plotted on the secondary (i.e. right) y-axis, when their item type differs from an elevation.

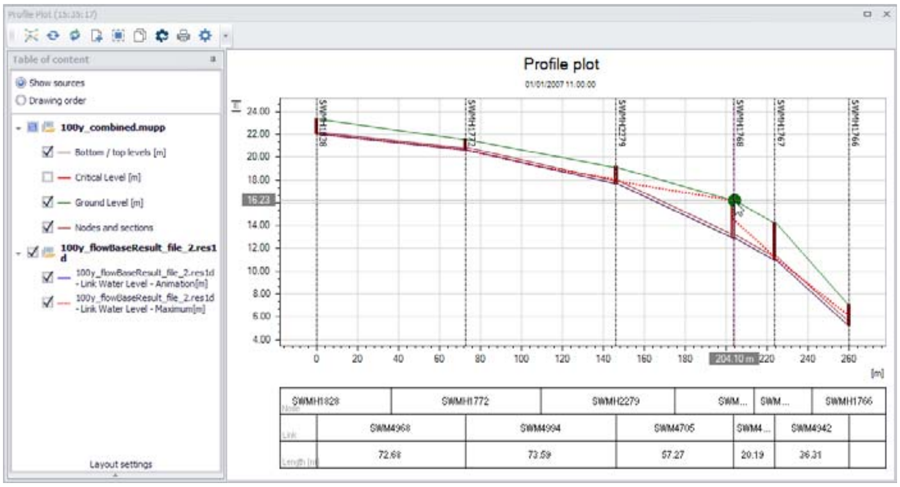


Figure 20.34 The Profile Plot form window showing an example longitudinal profile plot with max. link water level simulation results (red broken line)

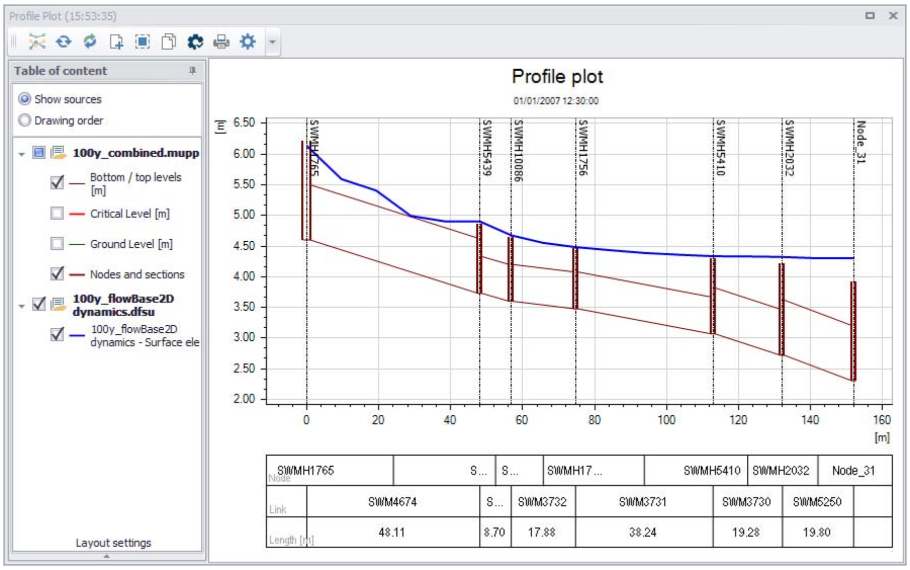


Figure 20.35 The Profile Plot form window showing an example longitudinal profile plot 2D water level simulation results (blue line)

When result items are added to the profile plot, it is possible to change the presentation mode (choice between Animation, Maximum, Minimum or Average type) afterwards. This option is offered from the local context menu, with a right-click on the related result item.

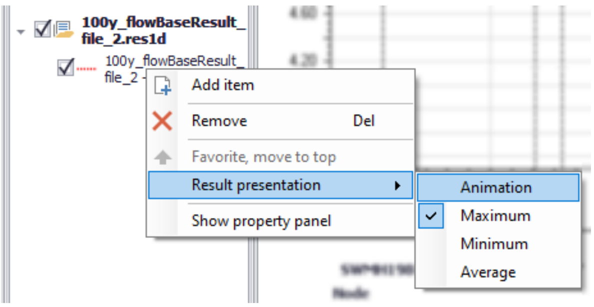


Figure 20.36 The selection of result presentation mode for a result item

Profile Plot with DEM

DEM profiles can also be added to a profile plot. DEM profiles can be obtained either from raster files or from 2D flexible mesh files, following these steps:

- From a profile plot window, use the 'Add item' button or the 'Add item' option in the local context menu (i.e. right-click) on the left panel.

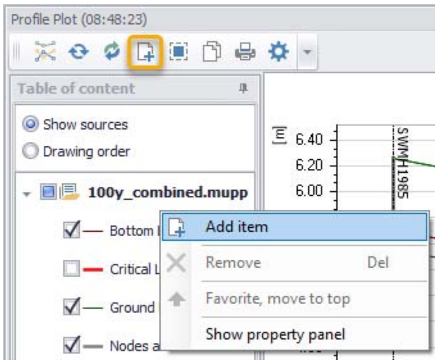


Figure 20.37 The 'Add item' tool on the Profile Plot window

- Select the DEM file, as well as the item if the file contains multiple items. A maximum spacing must also be specified: it controls the spacing between points along the DEM profile (Figure 20.38).

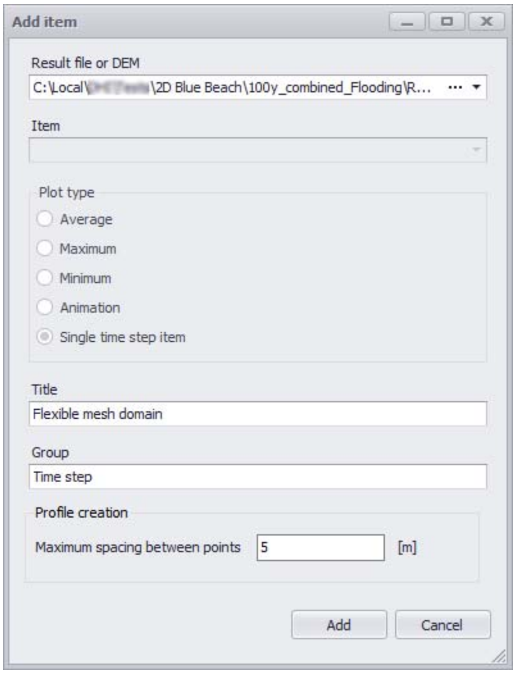


Figure 20.38 The 'Add item' dialog where the DEM file and settings are specified

The DEM item then appears on the Profile Plot (Figure 20.39). Result items are plotted on the secondary (i.e. right) y-axis.

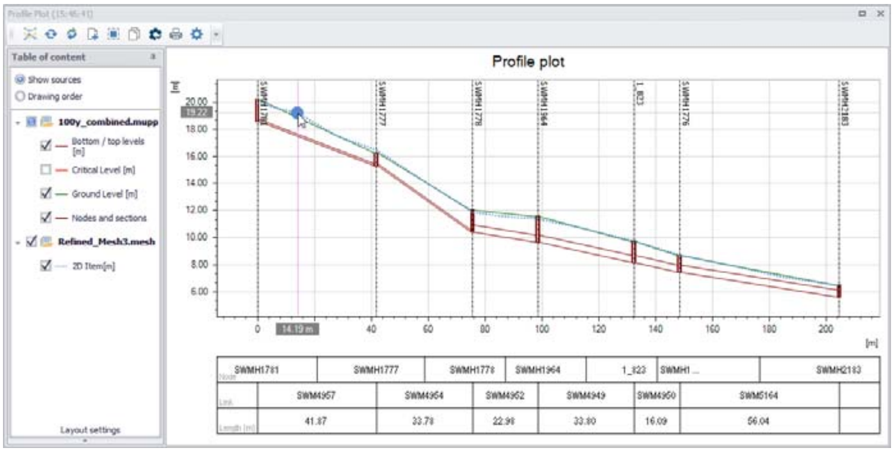


Figure 20.39 The Profile Plot form window showing an example longitudinal profile plot with a DEM profile (blue broken line)

The DEM profile is plotted with one point at each node's location, plus additional points in-between in order to fulfill the specified maximum spacing.

When a 2D overland model is defined, a default DEM profile is added to new profile plots, using the 2D domain as source of DEM.

20.6.2 Creating Profile Plots from a Result Map

Profile plots may also be created from extra result maps (obtained from the ‘Create result map’ button). The profile tools that can be used with this type of maps are located on the Results menu ribbon.

- First, create a result map plot as described in Chapter 20.2.
- Set path flags on the result map using the ‘Set flags’ tool from the Profile Plot toolbox on the Results ribbon (Figure 20.40).

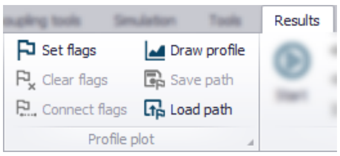


Figure 20.40 Set flags from the Profile Plot toolbox on the Results ribbon

- Click on the starting node for the profile path. This will place a small flag next to the node along with the number of the flag. Continue setting flags on the result map until the path is well-defined. Having both point (i.e. node) and line (i.e. link) result features on a result map helps with path-setting.

- Finally, click on ‘Draw profile’ on the Profile Plot toolbox. This will create a new profile plot on a profile plot form.
- Default items are added when creating the profile plot. Other result items may be added as described in the “Profile Plot with Results” on page 419.

20.6.3 The Profile Plot Window

The Profile Plot window displays longitudinal profile plots created in the project. Its various parts and components are described in succeeding sections.

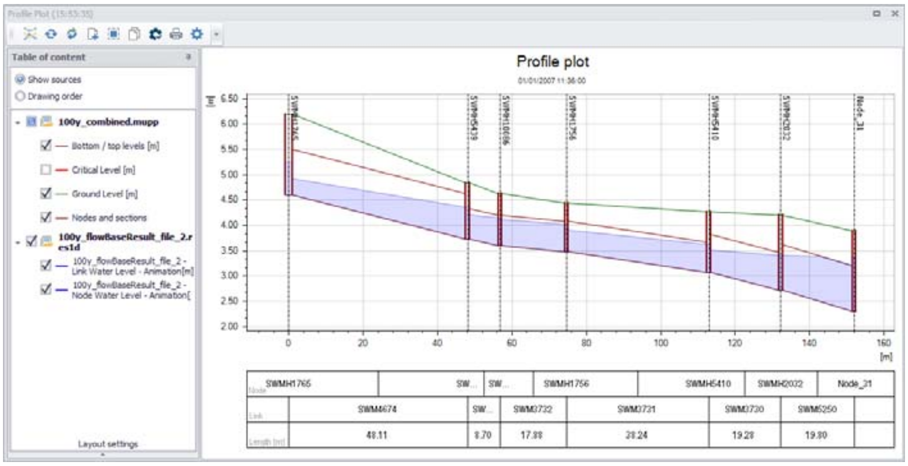


Figure 20.41 The Profile Plot window

Table of Contents

The Table of Contents (TOC) panel is located on the left side of the profile plot form. The panel lists information on the various information layers that are used for the profile plot (see Figure 20.42).

Show sources

Groups data layers according to data source and indicates the file from which the data are obtained.

Drawing order

Use this option to allow reordering/grouping of data layers on the plot. Reorder or group layers by dragging layer labels up or down the TOC list.

When you right-click on the TOC panel, the local context menu opens (see Figure 20.42).

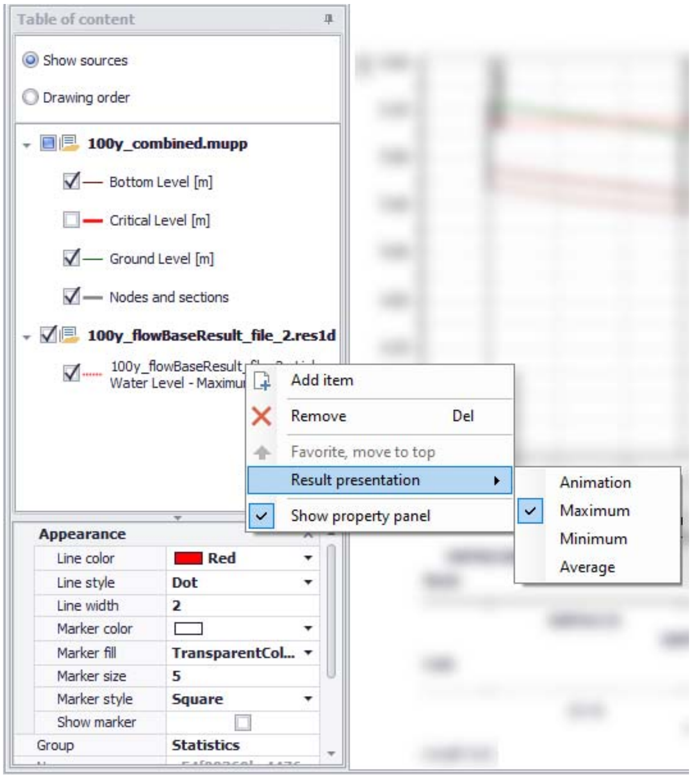


Figure 20.42 The local context menu on the longitudinal profile

Add item

Use this option to add result items or DEM items to an existing profile plot. See “Profile Plot with Results” on page 419 and “Profile Plot with DEM” on page 422.

Remove

Use this option to remove a layer from the profile plot.

Result presentation

Use this option to select the presentation mode of a result item. Possible modes are Animation (showing instantaneous results for the current date and time), Maximum, Minimum or Average.

Show property panel

Activate this option to view the Property Panel below the TOC. The Property Panel is used to customize the appearance of data layers on the profile plot.

Alternatively, click on the expand arrow icon at the bottom of the TOC to view the Property Panel.

Property Panel

The Property Panel is used to customize the appearance of data layers on the profile plot.

Select a layer from the TOC to view its properties on the Property Panel. Customize layer properties on the panel as needed. See Figure 20.43.

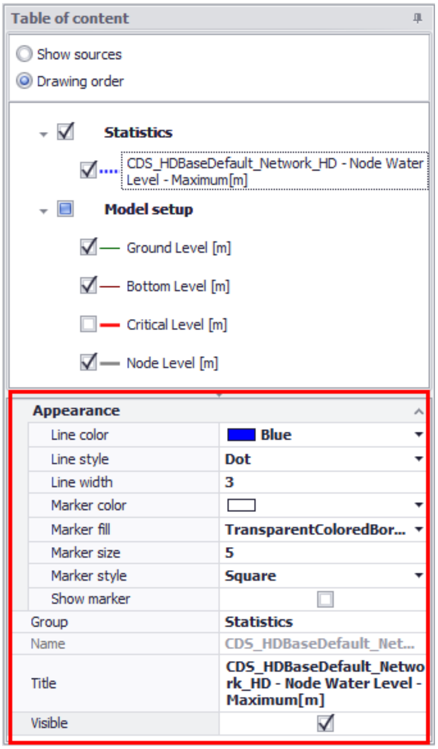


Figure 20.43 The Property Panel on the Profile Plot window

Plot Context Menu

Right-click on the profile plot to access the local context menu.

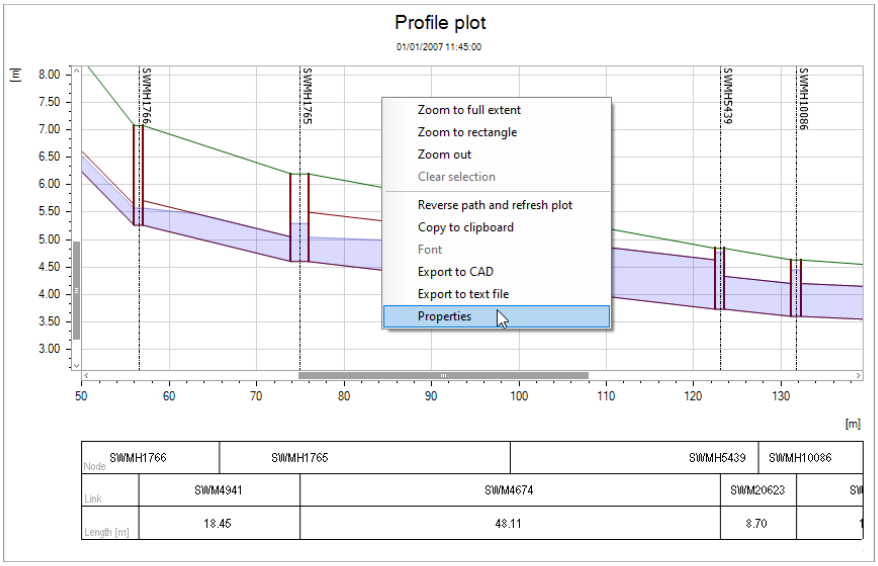


Figure 20.44 Right-click on the plot to access the local context menu

Zoom to full extent, Zoom to rectangle, Zoom out

Allows to zoom in and out on the plot. While zooming to a rectangle, dragging to draw a horizontal line will display arrows to zoom along the horizontal axis only, keeping the vertical axis unchanged. Similarly, dragging to draw a vertical line will zoom along the vertical axis only.

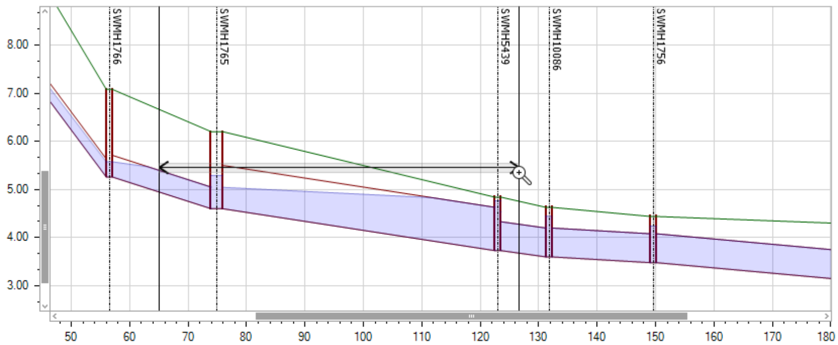


Figure 20.45 Use the 'Zoom to rectangle' option to zoom along one axis only

Zoom to full extent brings you back to the full view of visible data layers on the profile plot. Panning is also enabled upon activation of zoom options.

Note that additional options are available to control the zoom options:

- Hold down the Shift key, to enable 'Zoom to rectangle'
- Scroll with the mouse wheel to zoom in or out.

Clear selection

Deselects selected elements in the longitudinal profile.

Reverse path and refresh plot

Will swap the profile, e.g. swap profile from being drawn from node A to node B (from left to right on the plot) to being drawn from node B to node A.

Copy to clipboard

Copies the longitudinal profile displayed to the clipboard and allows it to be pasted into other applications.

Export to CAD

Opens the ‘CAD export options’ window to export the profile plot view to a CAD file.

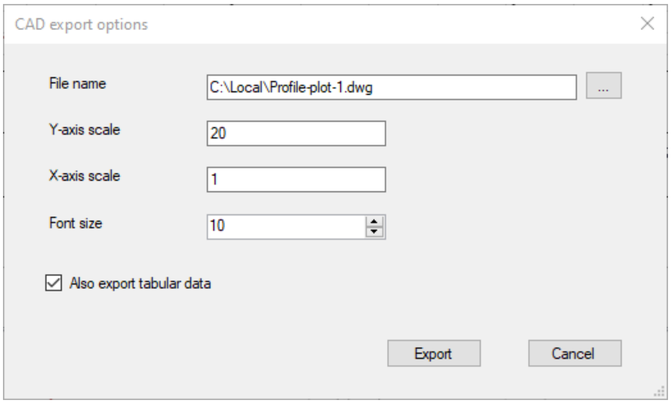


Figure 20.46 The ‘CAD export options’ window

From this window, it is possible to select a folder location as well as a file name for the created CAD file.

The X-axis and Y-axis scales control the lengths of the horizontal and vertical axes in the CAD file. They define scales relative to lengths in this CAD file.

The selected font size applies to all texts and values saved in the CAD file: select an appropriate font size that fits e.g. the exported axes and table.

Data in the table below the profile plot are exported to the CAD file only if ‘Also export tabular data’ option is selected.

Export to text file

Export the data from the profile plot view to a text file.

This text file contains a table with columns separated by semicolons. All data items (Link ID, Node ID, distance along the profile plot, elevations, etc.) are listed in columns, whereas the various links are listed in rows. When result items exist on the profile plot, they are also exported to extra columns in the file. The result value exported is the value shown on the plot (e.g. current time step for animated results, or maximum value, etc.).

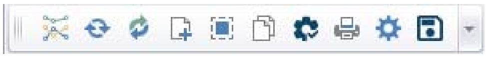
Note: When exporting data from a collection system network with results, some of the links may contain multiple calculation points depending on their length. In this case, all calculation points' values are listed in the same cell with a separator, for example like this: 6.96499 | 6.87148 | 6.78332.

Note: When exporting river networks, each river is exported as a link but extra information is exported at each cross section location. The exported table will also contain information at "virtual node" locations, a virtual node being the location of a flag controlling the profile plot locations. Virtual nodes therefore do not represent actual calculation points on the network, and will show interpolated values.

Properties

Activate this option to view the Profile Plot Properties dialog. See Chapter 20.6.5 - Profile Plot Properties.

Profile Plot Tools



The toolbar on top of the Profile Plot window offers several tools that may be used for working with profile plots.

New plot

Generate a new profile plot (on the existing profile plot window) from a new set of defined path flags.

Refresh

Update/refresh existing data layers on the plot. Ensures that changes to model data (e.g. node invert level via the Nodes editor) for elements included in the profile are reflected in the plot. The location of the profile plot is not changed even if flags have been moved on the map.

Reverse path and refresh plot

Swaps the left to right plot path orientation going from first to the last flag locations to last to first flag locations.

Add item

Use this option to add result items or DEM items to the profile plot. See “Profile Plot with Results” on page 419 and “Profile Plot with DEM” on page 422. Result items are plotted on the secondary (i.e. right) y-axis.

Selection mode

Allows for selecting model elements from the longitudinal profile. It uses ‘Select by rectangle’ option. Selected elements are also highlighted on the Map. The displayed selection in the profile is synchronized with both the map and the editors.

Copy to clipboard

Copies the longitudinal profile displayed to the clipboard and allows it to be pasted into other applications.

Set as default

Changes the default layout of the profile plot in the current MIKE+ project, to match the layout of the current profile plot. After pressing this button, the layout of the current profile plot will therefore apply to new plots, i.e. showing the same layers and with the same layer properties.

Print/Export

Option for formatting the plot for printing.Launches the print preview window. See Chapter 20.6.4 - Print/Export Preview.

Properties

Launches the Profile Plot Properties dialog. See Chapter 20.6.5 - Profile Plot Properties.

Save to plots manager

Saves the profile plot (path along the network, list of result items plotted and display settings) to the 'Plots' panel. The profile plot will initially be added to the active folder from this panel. See 'Plots Management' chapter (page 487) for more information on options to save and manage results windows.

20.6.4 Print/Export Preview

The Print/Export tool from the Profile Plot toolbar launches the Preview window (Figure 20.52), wherein print layouts for the plot may be configured.

It also allows for exporting the plot layout to various document file types for potential inclusion in reports or information dissemination.

File Menu

The File menu on the Preview window offers options for:

- **Export Document**. Export the layout to documents in the following format:
  - PDF
  - HTML
  - MHT
  - RTF
  - XLS

- XLSX
- Image File (e.g. PNG, JPG, etc.)

- **Send via E-mail**. Exports the layout to a document (as above) and then launches the email program including the document as attachment to an email for sending.

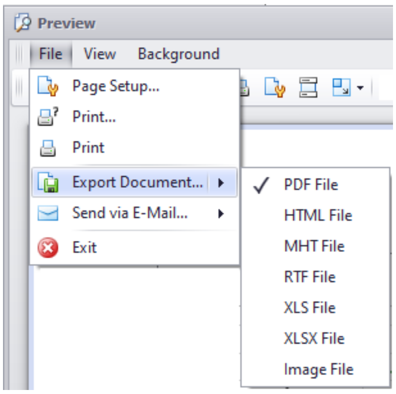


Figure 20.47 Preview window File menu

View Menu

The View menu offers options for modifying the appearance of the Preview window:

- **Page Layout.** Customize the layout display on the window. 'Facing' displays facing pages at once.
- **Toolbar and Status Bar.** Options for adding or removing the respective components from the window.
- **Customize.** Launches the Customization dialog, where various options for further modifying the window are available. These options include:
  - Activating/deactivating toolbars
  - Creating custom toolbars
  - Enlarging toolbar icons
  - Activating/deactivating tooltips

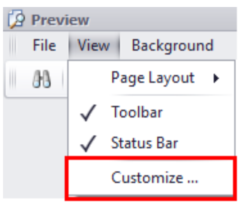


Figure 20.48 The View menu on the Preview window allows for customization of the window appearance and toolbars

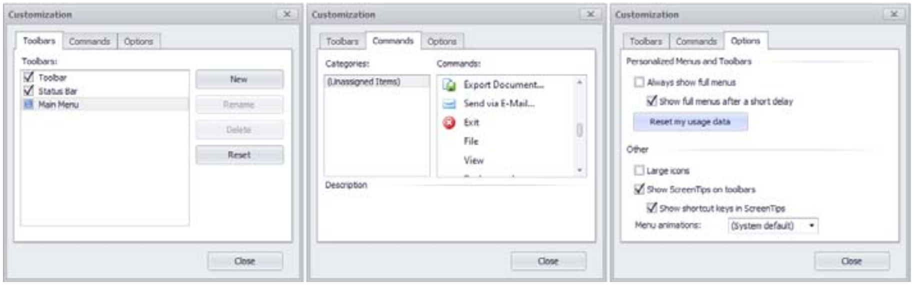


Figure 20.49 Various options for modifying the appearance of the Preview window on the Customization dialog

Background Menu

Customize the layout background via the Background menu. It offers options for:

- **Color**. Modifying the solid layout background color.
- **Watermark**. Launches the Watermark dialog, where text and/or image watermarks may be added to the layout.

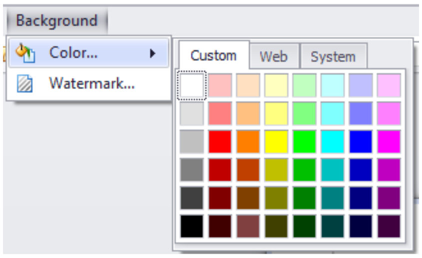


Figure 20.50 The Background menu on the Preview window

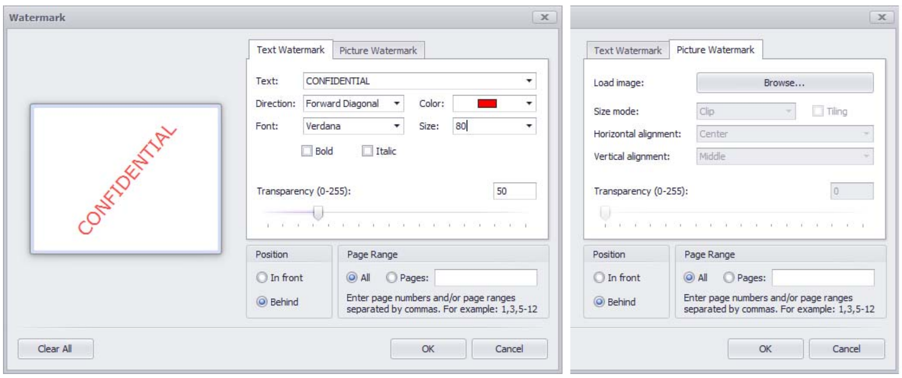


Figure 20.51 Add text or image watermarks to layouts via the Watermark dialog

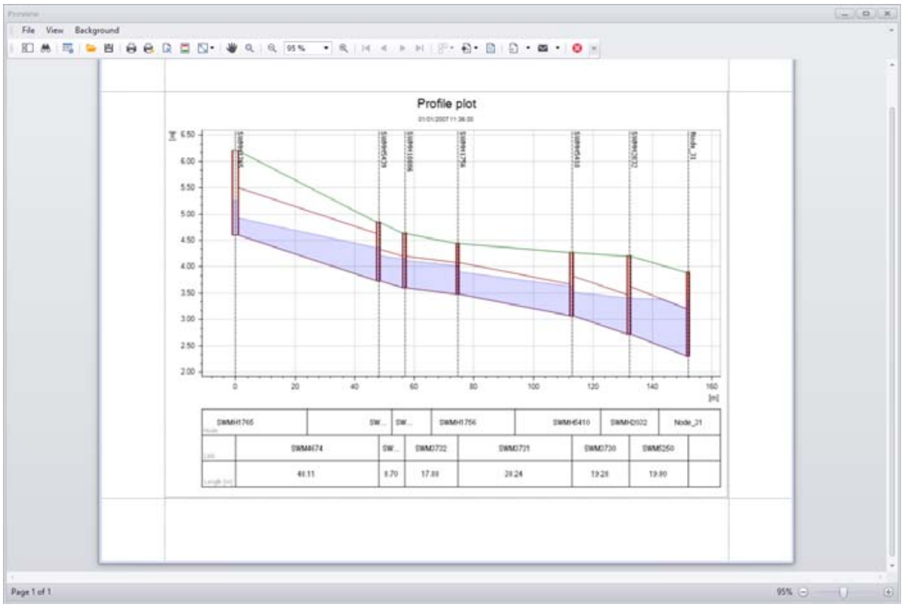
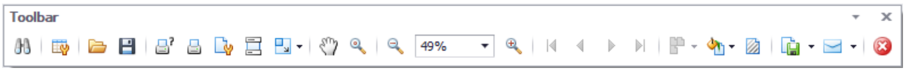


Figure 20.52 The Print/Export Preview window

Preview Toolbar



The toolbar on the Preview window offers various tools for working with the layout.

Search

Text search on the plot.

Customize

Offers options for plot resizing during printing: None, Stretch, or Zoom.

Open

Option for loading an existing preview document *.PRNX layout configuration file.

Save

Option for saving the layout configuration in a preview document *.PRNX file.

Print
Launches the Print dialog where printing options may be customized before actual printing.

Quick Print
Option for immediate printing of layout using current configuration.

Page Setup
Launches the Page Setup dialog for defining layout printing setup.

Header and Footer
Launches the Header and Footer dialog, where custom page headers and footers may be defined.

Hand Tool
Tool for panning around the layout.

Magnifier, Zoom Out, Zoom In, Zoom
Tools for zooming in and out on the layout.

Color
Customize the layout background color.

Watermark
Launches the Watermark dialog for adding text and/or image watermarks to the layout.

Export Document
Exports the layout to a document.

Send via Email
Exports the layout to a document and adds it as attachment to a new email for sending.

Exit
Closes the Preview window.

20.6.5 Profile Plot Properties

The properties of the longitudinal profile can be changed via the Properties dialog (Figure 20.53). The dialog is accessed in several ways:

- Choose ‘Properties’ on the local context menu on the profile plot area.
- Double-click on the profile plot area.
- Activate the Properties tool from the Profile Plot toolbar.

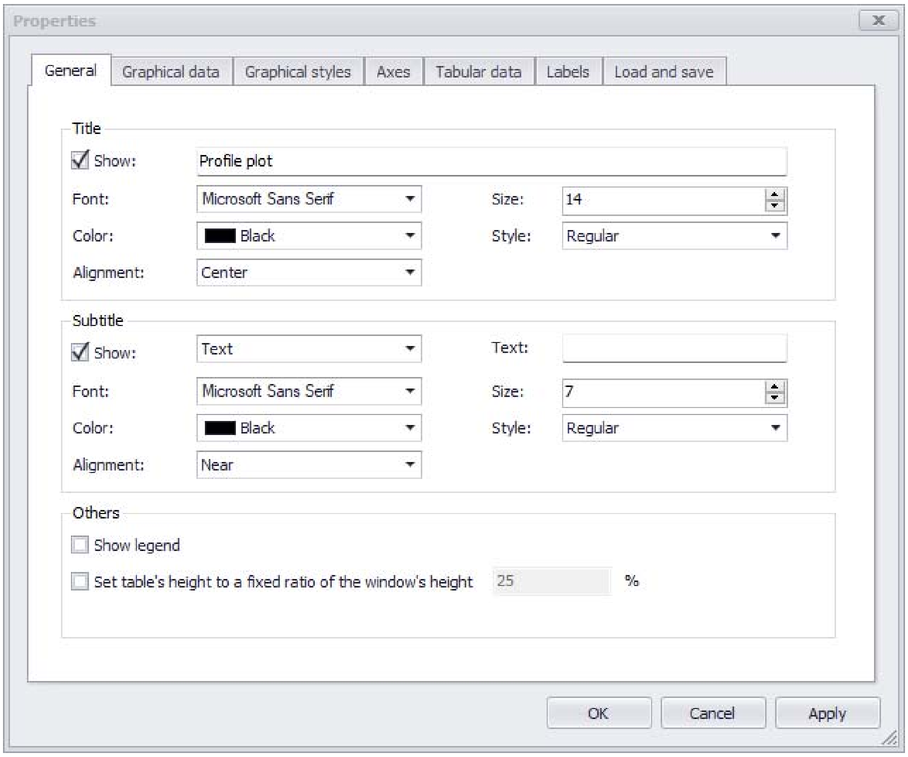


Figure 20.53 Setting the properties of the profile plot

The dialog has various tab pages wherein changes to the profile plot properties can be made. The following general button functionalities are available at the bottom of the dialog:

OK
Will apply the settings specified and close the properties dialog.

Cancel
Will cancel any changes made and close the properties dialog.

Apply

Will apply the settings specified, but leave the properties dialog open.

General

The General tab page (Figure 20.54) offers options for:

- Adding and formatting plot titles
- Adding and formatting subtitles
- Showing the data Legend
- Controlling the height of the table below the profile plot (height expressed as a percentage of the entire window's height)

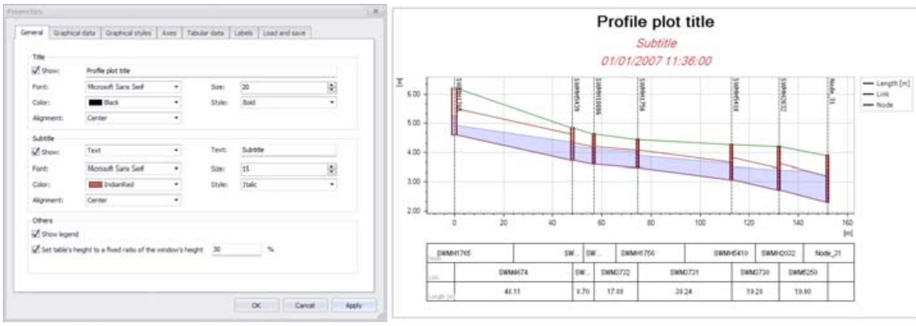


Figure 20.54 Customizing the plot title and subtitle via the General tab page

Graphical Data

On the Graphical Data page (Figure 20.55), it is possible to specify which general information is to be shown on the profile plot. The various display options available for the graphical items can be reviewed and changed on this page.

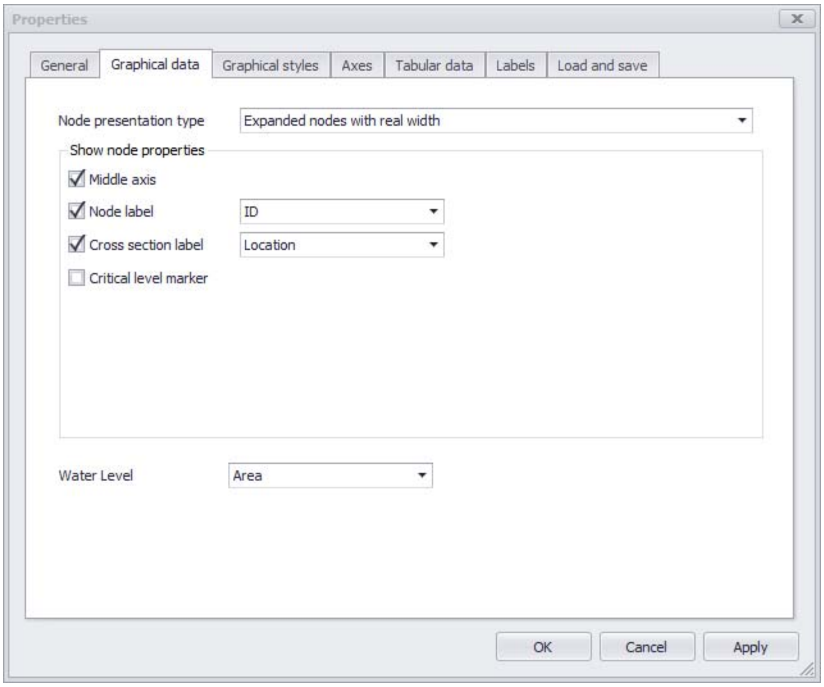


Figure 20.55 The Graphical Data tab page

- **Node presentation type**: this controls the type of presentation for nodes along collection system networks (in 'Rivers, collection system and overland flows' mode only). The available options are:
  - Collapsed nodes. To not show node lateral dimensions on the longitudinal profile. Useful for long profiles with many nodes.
  - Expanded nodes with real width. To show node lateral dimensions the longitudinal profile, using the actual nodes's width.
  - Expanded nodes with fixed width. To show node lateral dimensions on the longitudinal profile, using a fixed width regardless of the actual node's size. Useful to keep showing node results, while keeping their size limited for showing long network path. With this mode, when a node symbolic width becomes larger than half of the connected link length (e.g. when zooming out), the node will automatically collapse in order to keep showing long network paths.
- **Node label**: Toggles on/off labels for nodes above the node in the profile. Select the label parameter from the dropdown menu on the right.
- **Cross section label**: Toggles on/off labels for cross sections in the profile. Select the label parameter from the dropdown menu on the right.
- **Critical level**: Toggles on/off if critical levels (specified by the user) are drawn on the longitudinal profile.

- **Water level**: Option for showing either only the water surface line, or only the water level filling in the pipes, or both. Note: if both line and filling are selected, and if the profile plot includes both nodes and links water levels, the water surface line will only display the link water level line, in order to avoid overlapping lines.

Graphical Styles

Format profile plot data layer appearance on the Graphical Styles tab page (Figure 20.56). For each data item, line symbol style, width, and color may be customized. In addition, markers may be included, and marker appearance defined.

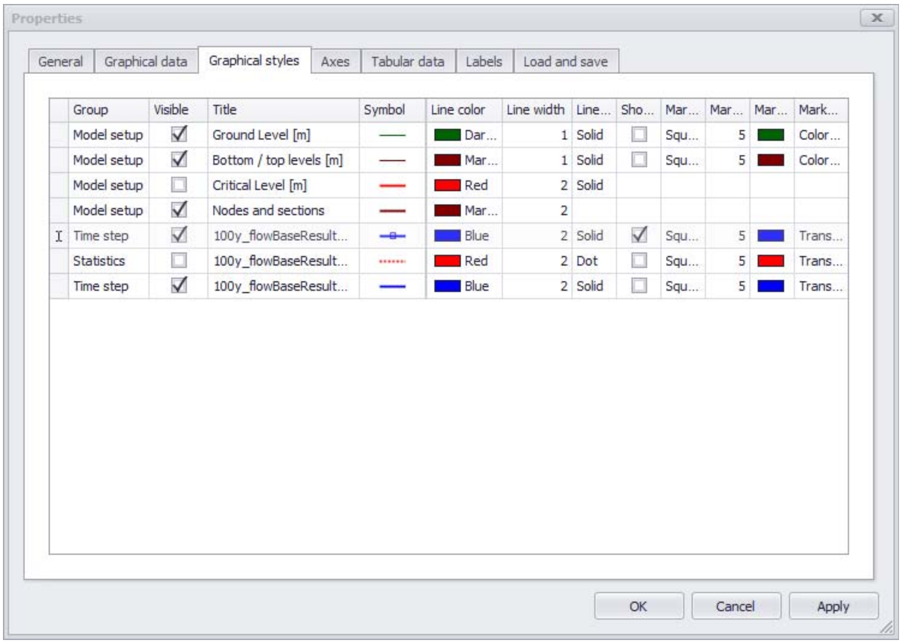


Figure 20.56 The Graphical Styles tab page

Axes

The Axes tab page offers options for setting axes properties, including axes labels and grid lines (Figure 20.57). It has options for:

- Customizing axes title and label fonts
- Modifying axes line appearance
- Formatting the title, scale, label, grid lines, and visual range for the x-axis
- Formatting the title, scale, label, grid lines, and visual range for the primary (i.e. left) y-axis

- Formatting the title, scale, label, grid lines, and visual range for items on the secondary (i.e. right) y-axis. Result items are plotted on the secondary (i.e. right) y-axis.

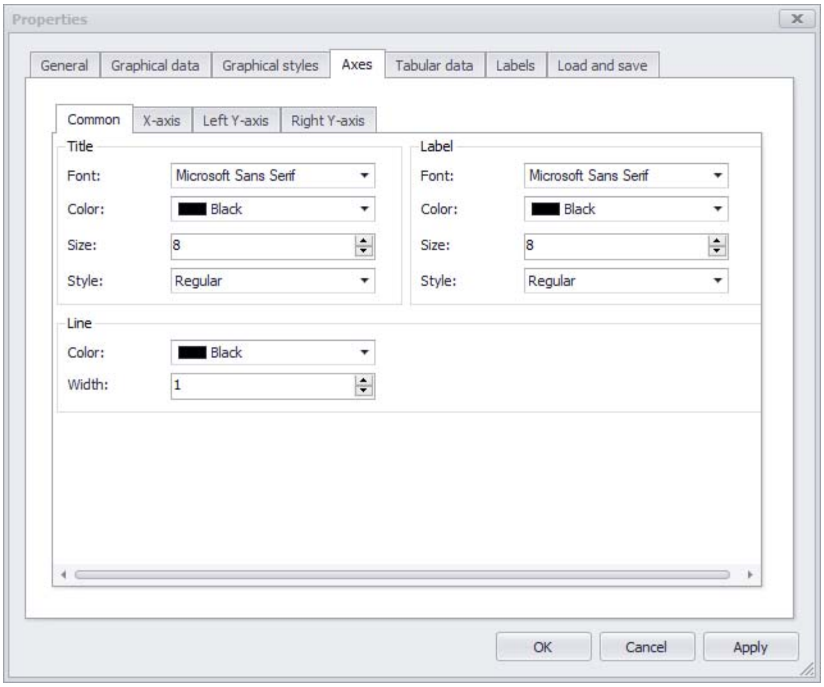


Figure 20.57 The Axes tab page

Tabular Data

The Tabular Data tab page offers an option for configuring the content of the table below the profile plot.

From the 'Common data' tab, select the attributes to include in the table which are common to all link layers (e.g. pipes, rivers, valves, weirs, etc.) or node layers (typically nodes or junctions). From the 'Additional data' tab, select the extra attributes to show, specifically for each individual table of the database. To add an item to the profile plot window, tick its 'Show in table' box.

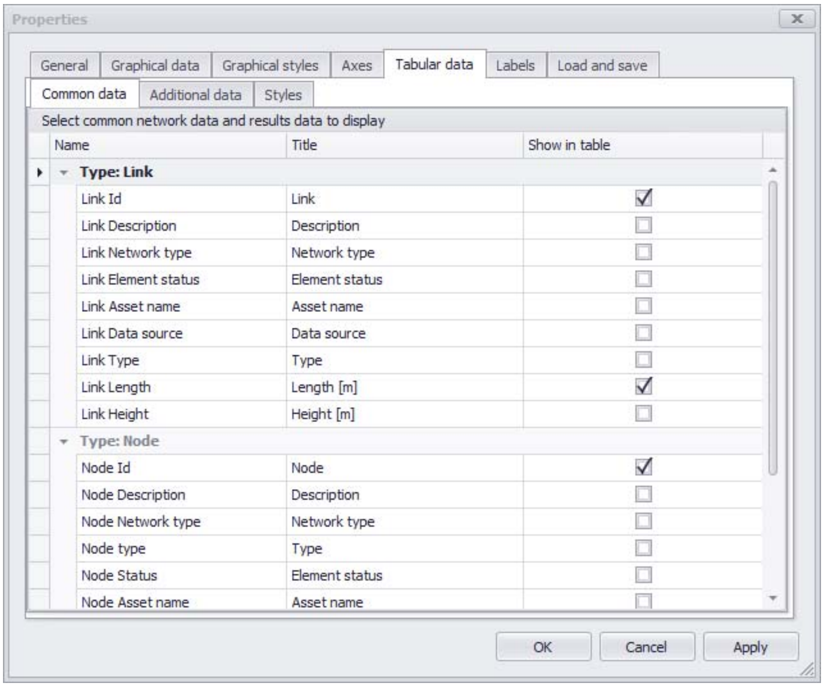


Figure 20.58 The Tabular Data tab is used to control the content of the table

The 'Styles' tab allows formatting the content of the table. It can especially be used to control the order of the displayed items (rows) in the table, customize their displayed names, control the font and its rotation, or adjust the heights of the rows.

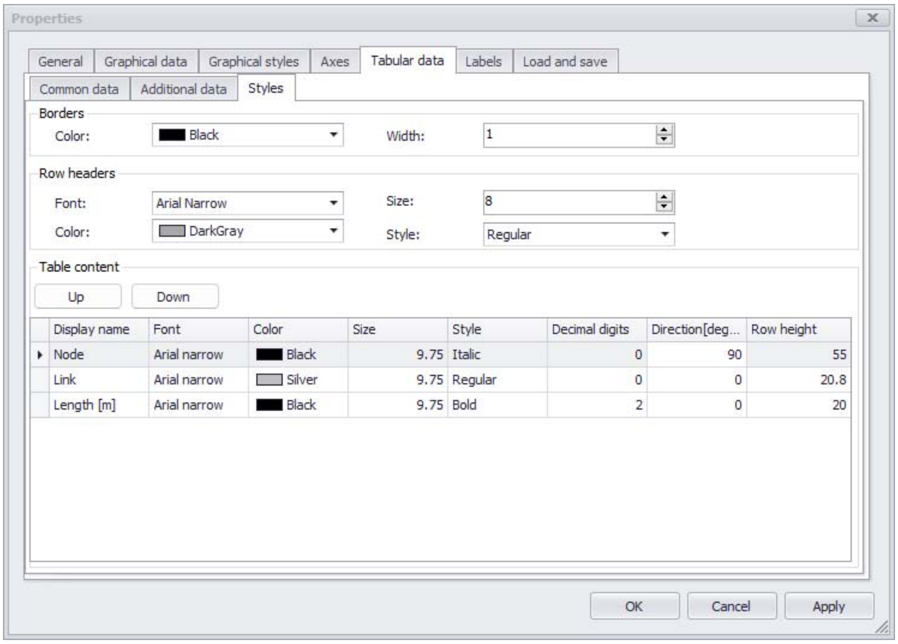


Figure 20.59 The Tabular Data tab allows customizing the styles of the table's data

Labels

Add custom text labels to the profile plot via the Labels tab page (Figure 20.60). ‘Add’ a new custom label and define it’s location and text annotation. Custom labels could be placed at locations relative to model object positions (e.g. nodes, links, etc.).

Save custom label configurations into profile plot label files via the ‘Save to file’ option. Existing label configurations may be loaded through ‘Load from file’.

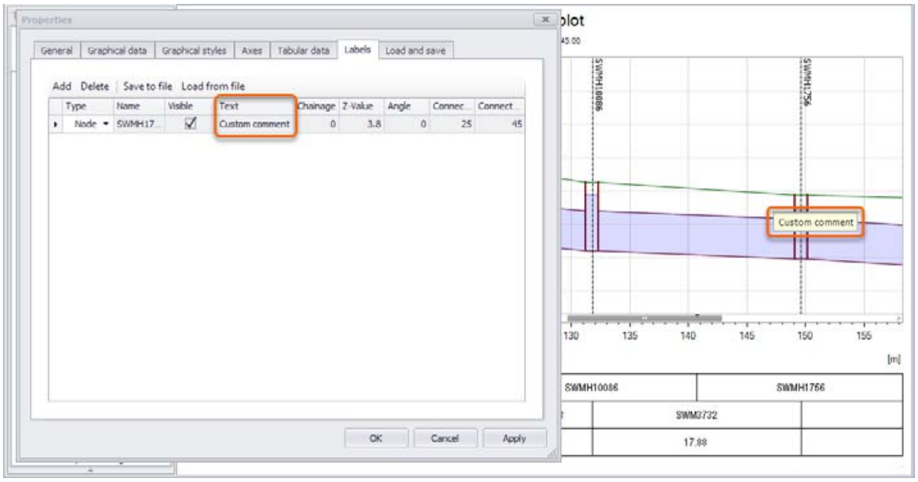


Figure 20.60 The Labels tab page showing an example custom label by a node

Load and Save

This tab contains options to re-use a custom layout and a path. The layout covers the list of layers shown on the profile plot (including result layers), their graphical styles, the axes style, the additional X-axis data, the plot's title, etc. The plot's path is defined by the location of the flags on the map, used to create the profile plot's path. This path is not saved in the layout definition.

The ‘Save to file’ option (in the ‘Layout’ group) saves the current profile’s layout to a profile plot file (*.profileplot) containing the layout definition.

The ‘Load from file’ option loads a profile layout file (*.profileplot) to update the current profile plot.

The 'Set as default' option changes the default layout of the profile plot in the current MIKE+ project, to match the layout of the current profile plot. After pressing this button, the layout of the current profile plot will therefore apply to new plots.

The 'Apply default' option updates the current profile plot, by applying the default layout from the current MIKE+ project. This default layout may be a customised layout, in case the 'Set as default' option has been applied beforehand from a customised profile plot.

The 'Save to file' option (in the 'Path on map' group) saves the location of the flags used to create the current profile plot, to a file (*.path). This file can later be used to create new profile plots using the same path (see Profile and tracing for more information).

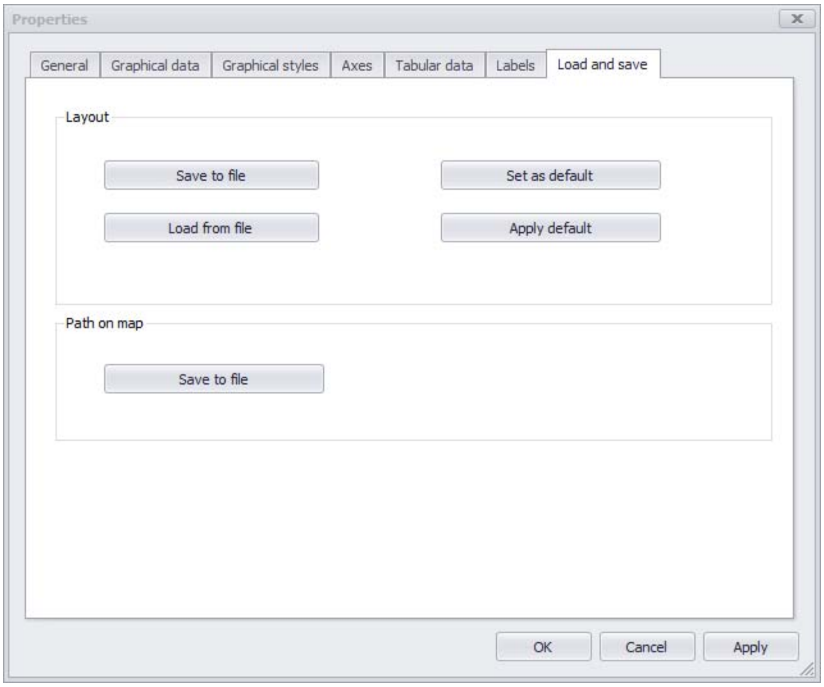


Figure 20.61 The Load and Save tab page

20.7 Bar Chart

Simulation results may be plotted as bar charts in MIKE+. These types of plots are especially relevant for visualizing LTS simulation results.

To create a new bar chart, right-click on the result file or one of its result items in the list of result files.

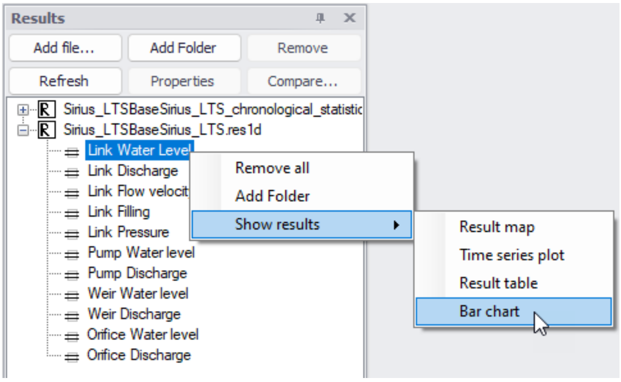


Figure 20.62 Creating a bar chart from the list of result files

This will open the 'Create bar chart' window below.

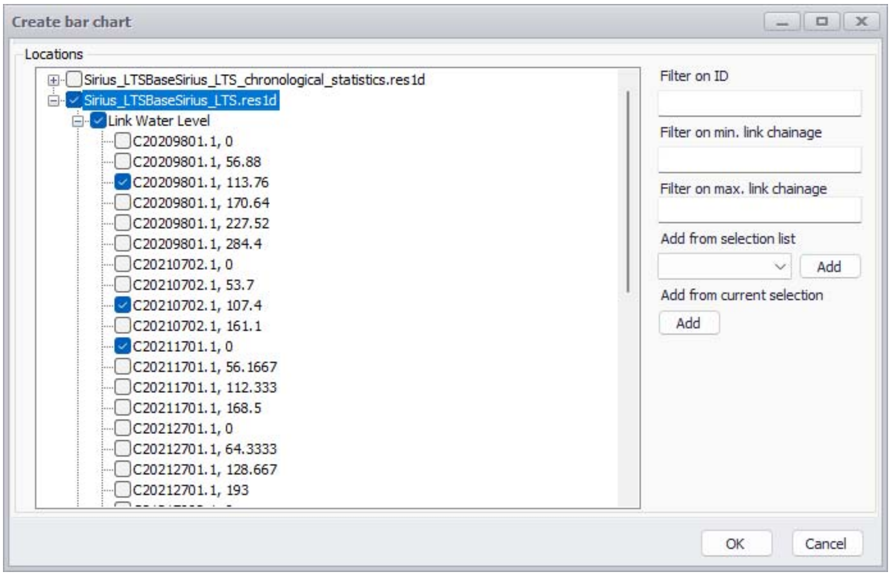


Figure 20.63 The 'Create bar chart' window

Choose the result file, result item and locations to plot from the list. Use the filters to search through the potentially long list of available locations, to filter on ID and/or chainages (chainage filtering being available only for link results from .res1d result files). One may also use a selection list (saved in the 'Selection manager') or the active selection, to select all items from the selected list.

Then click on the 'OK' button to create the chart.

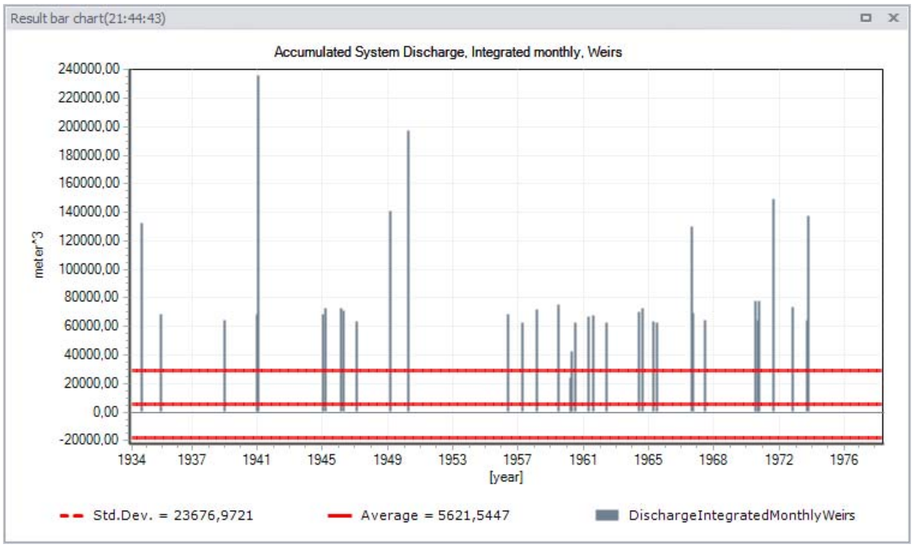


Figure 20.64 Example bar chart plot of monthly weir overflow volumes from LTS chronological statistics results

Customize the bar chart appearance via the Properties dialog accessed through the chart local context menu. The bar chart local context menu offers the following options:

- Zoom in, Zoom to full extent, Previous zoom, and Next zoom
- Save to plots manager: saves the bar chart (with its source result item) to the 'Plots' panel. The bar chart will initially be added to the active folder from this panel. See 'Plots Management' chapter (page 487) for more information on options to save and manage results windows
- Copy to clipboard
- Save to image file
- Print
- Properties

Bar Chart Properties

Format the bar chart through the Bar Chart Properties dialog. Set chart element properties in the various tab pages within the dialog.

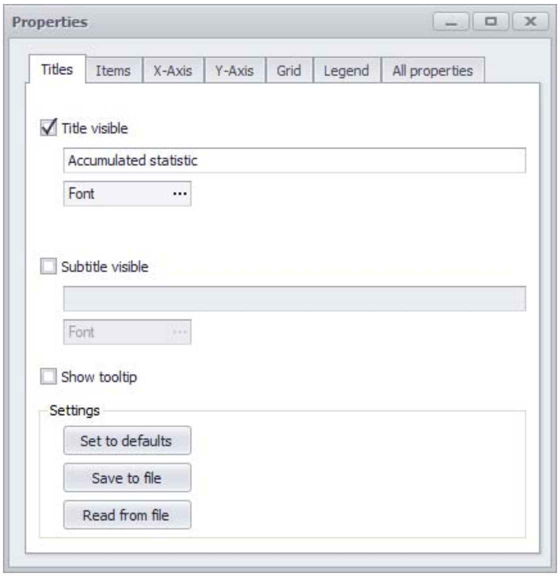


Figure 20.65 The Bar Chart Properties dialog

The sections below describe the various bar chart property options available on the Properties dialog.

Titles

- **Title visible, Subtitle visible, Font**. Options for adding and defining chart titles and subtitles, as well as customizing font types.
- **Show tooltip**. Option for showing a small "hover box" with information about a result item when hovering over it on the chart.
- **Set to defaults**. Reset chart properties to default values.
- **Save to file**. Option for saving bar chart configuration into a *.PFS file.
- **Read from file**. Option for loading bar chart configuration from an existing*.PFS file.

Items

- **Visible**. Activate/deactivate result items on the chart
- **Use title in legend** and **Title.** Customize Legend labels for result items on the chart
- **Vertical axis** (Left axis or Right axis). Option for defining the y-axis to use for result data on the chart.
- **Show bar**. Options for customizing the appearance of the result data series on the chart.

- **Show marks, Show average, Show median**. Option for showing annotation and additional information series on the chart, and customizing their appearance.

X-Axis

- **Interval**. For customizing x-axis bounds (i.e. interval).
- **Reverse.** Option for showing values in reverse order along the y-axis.
- **User-define tick marks**. For customizing the appearance of tick marks on the x-axis.
- **Labels**. Options for customizing labels along the x-axis, including font, orientation, and data format.
- **Title**. Option for defining x-axis title and font.

Y-Axis

- **Interval**. For customizing y-axis bounds (i.e. value range).
- **User-define tick marks**. For customizing the appearance of tick marks on the y-axis.
- **Labels**. Options for customizing labels along the y-axis, including font, and number of decimals.
- **Title**. Option for defining y-axis title and font.

Grid

- **Vertical grid lines**. Options for showing vertical grid lines on the chart area.
- **Horizontal grid lines**. Options for showing horizontal grid lines on the chart area, and defining the y-axis of reference.
- **Style**. For customizing the appearance of both vertical and horizontal grid lines on the chart area.

Legend

- **Visible**. Activate/deactivate the Legend on the chart
- **Colorized text**. Option to use the same color for label text as the data series in the Legend.
- **Alignment**. Option for defining the location of the Legend on the chart.

All properties

- This tab page collectively shows the various parameter configurations from the other tab pages in tabular format.

20.8 LTS Report

MIKE+ has facilities for generating LTS statistics reports. The reports can be viewed and exported in various document file formats for printing, editing, or information dissemination.

To create a new LTS report, right-click on the LTS result file or one of its result items in the list of result files. The following types of LTS Reports could be generated:

- Summary report on extreme events statistics
- Detailed report on extreme events statistics
- Report on annual/monthly statistics

See also MIKE+ Collection System User Guide Chapter 9.4 LTS Statistics Presentation for more details on LTS result statistics.

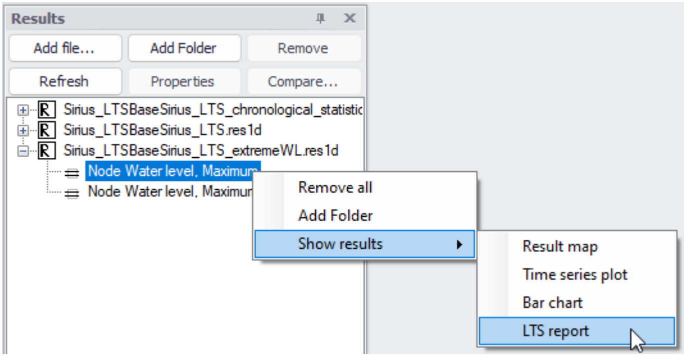


Figure 20.66   Creating a LTS report from the list or result files

20.8.1   Summary Report on Extreme Events Statistics

This type of LTS report contains tables with a summary of all calculated statistics, i.e. for all individual locations, variables and statistics types. It can be generated only for result files with LTS **extreme event statistics**.

To create a new LTS report, right-click on the extreme event LTS result file or one of its result items in the list of result files, and select 'LTS report'.

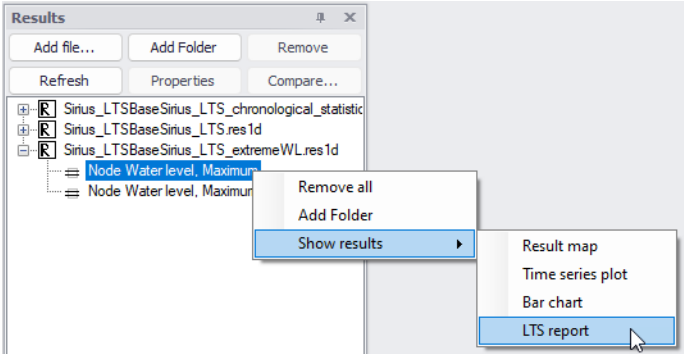


Figure 20.67 Creating a LTS report from the list of result files

This will open the 'Create LTS report' window below.

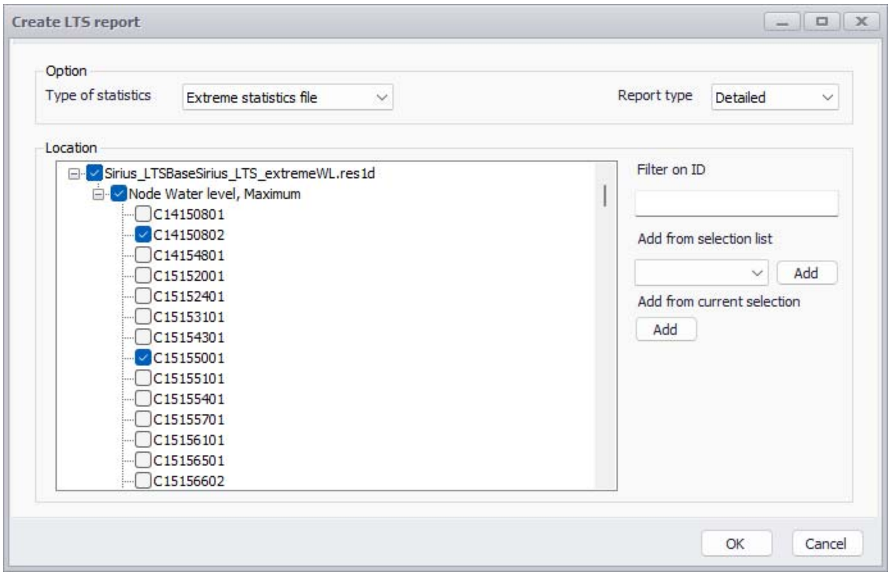


Figure 20.68 The 'Create LTS report' window

Choose to generate a report with the ‘Extreme statistics file’ from the drop-down list at the top, and select the ‘Summary’ report type. Choose the result file, result item and locations to include from the list. Use the filter on ID to search through the potentially long list of available locations. One may also use a selection list (saved in the 'Selection manager') or the active selection, to select all items from the selected list.

Click on the ‘OK’ button to create the new LTS Extreme Event Statistics Summary report. An example is shown in Figure 20.69.

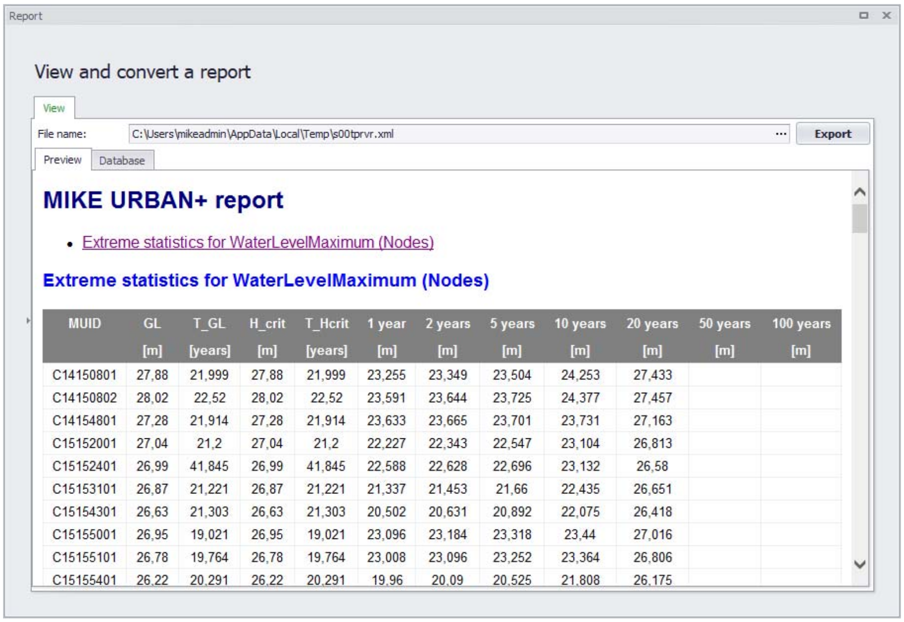


Figure 20.69 Example LTS extreme event statistics summary report

See also MIKE+ Collection System User Guide Chapter 9.4.7 Generating Reports on LTS Statistics for more details on LTS result statistics.

20.8.2 Detailed Report on Extreme Events Statistics

This type of LTS report contains tables with details of all calculated statistics in the file, i.e. for all individual locations, variables and statistics types. It can be generated only for result files with LTS **extreme event statistics**.

To create a new LTS report, right-click on the extreme event LTS result file or one of its result items in the list of result files, and select 'LTS report'.

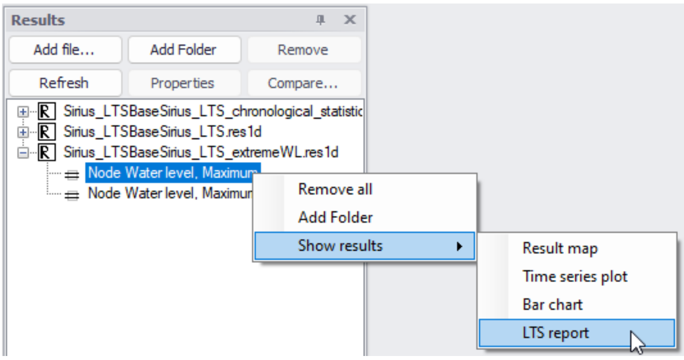


Figure 20.70 Creating a LTS report from the list of result files

This will open the 'Create LTS report' window below.

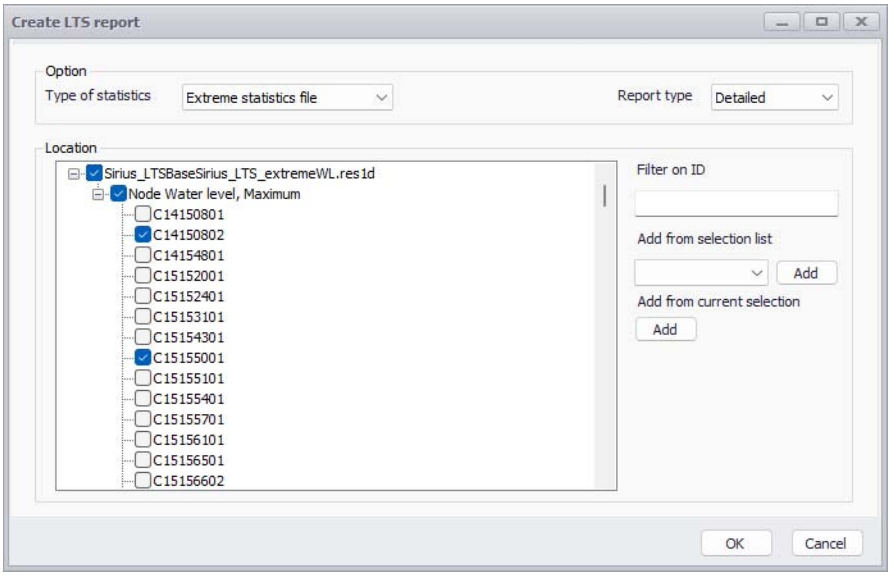
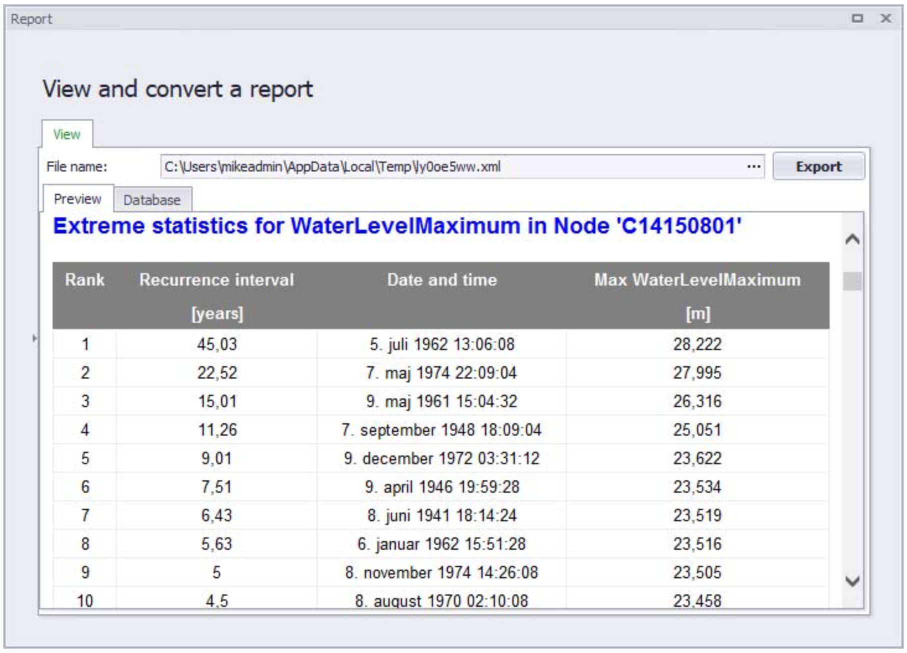


Figure 20.71 The 'Create LTS report' window

Choose to generate a report with the ‘Extreme statistics file’ from the drop-down list at the top, and select the ‘Detailed’ report type. Choose the result file, result item and locations to include from the list. Use the filter on ID to search through the potentially long list of available locations. One may also use a selection list (saved in the 'Selection manager') or the active selection, to select all items from the selected list.

Click on the ‘OK’ button to create the new Detailed LTS Extreme Event Statistics report. An example is shown in Figure 20.72.



Rank	Recurrence interval [years]	Date and time	Max WaterLevelMaximum [m]
1	45,03	5. juli 1962 13:06:08	28,222
2	22,52	7. maj 1974 22:09:04	27,995
3	15,01	9. maj 1961 15:04:32	26,316
4	11,26	7. september 1948 18:09:04	25,051
5	9,01	9. december 1972 03:31:12	23,622
6	7,51	9. april 1946 19:59:28	23,534
7	6,43	8. juni 1941 18:14:24	23,519
8	5,63	6. januar 1962 15:51:28	23,516
9	5	8. november 1974 14:26:08	23,505
10	4,5	8. august 1970 02:10:08	23,458

Figure 20.72 Example detailed LTS extreme event statistics report

See also MIKE+ Collection System User Guide Chapter 9.4.7 Generating Reports on LTS Statistics for more details on LTS result statistics.

20.8.3 Report on Annual/Monthly Statistics

This type of LTS report contains tables with all calculated annual/monthly statistics for all individual locations and variables (e.g. volumes, accumulated mass, durations, number of events). It can be generated only for result files with LTS **chronological (annual/monthly) statistics**.

To create a new LTS report, right-click on the chronological LTS result file or one of its result items in the list of result files, and select 'LTS report'.

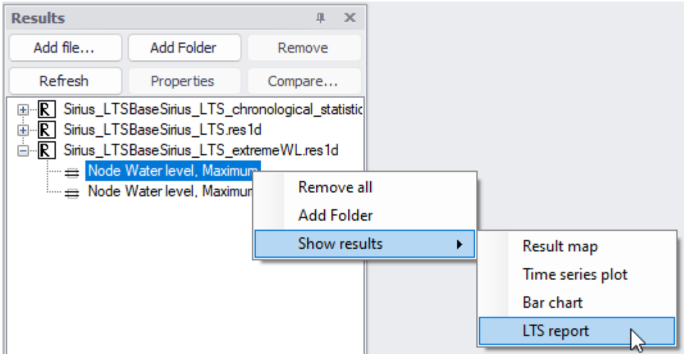


Figure 20.73 Creating a LTS report from the list of result files

This will open the 'Create LTS report' window below.

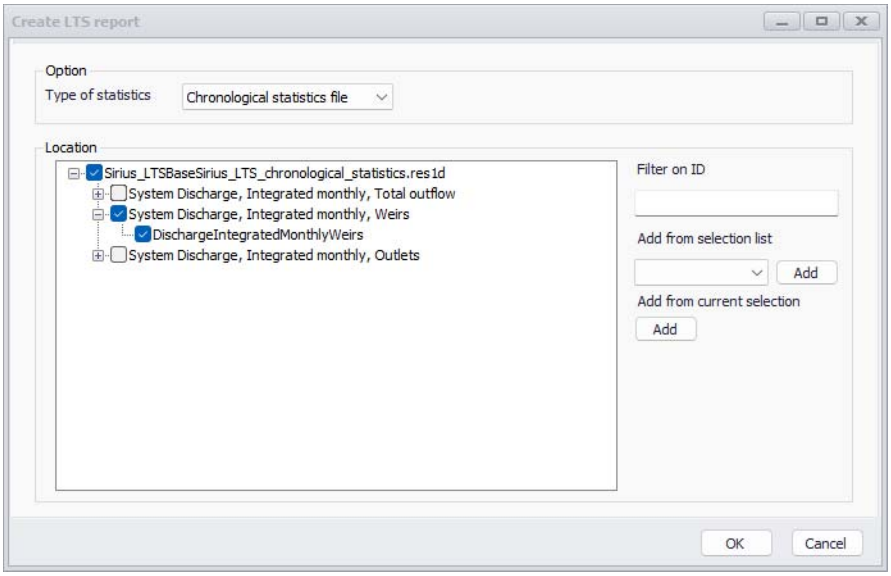


Figure 20.74 The 'Create LTS report' window

Choose to generate a report with the ‘Chronological statistics file’ from the dropdown menu at the top. Choose the result file, result item and locations to include from the list. Use the filter on ID to search through the potentially long list of available locations. One may also use a selection list (saved in the 'Selection manager') or the active selection, to select all items from the selected list.

Click on the ‘OK’ button to create the new LTS chronological statistics report. An example is shown in Figure 20.75.

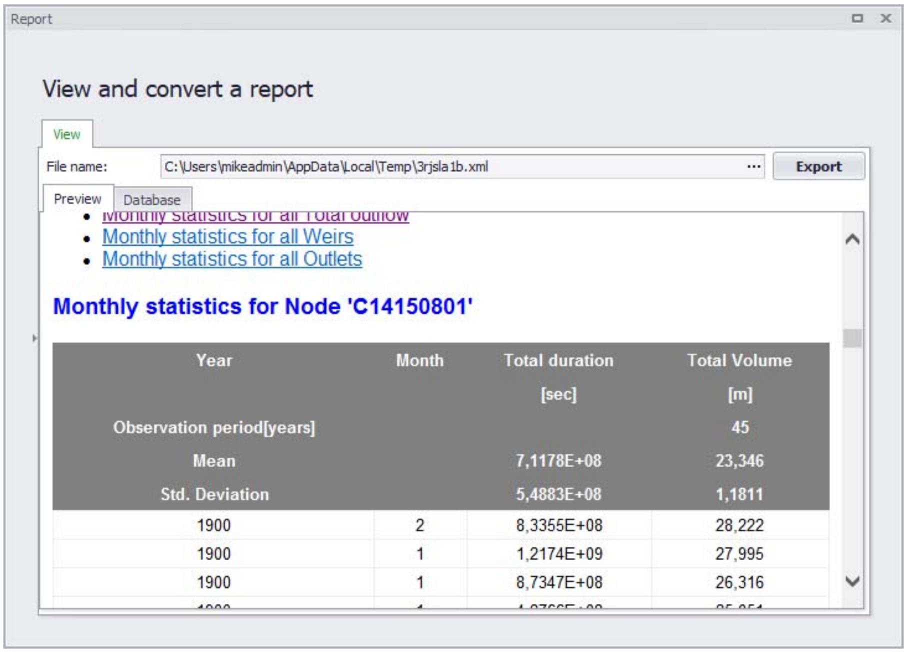


Figure 20.75 Example LTS chronological statistics report

See also MIKE+ Collection System User Guide Chapter 9.4.7 Generating Reports on LTS Statistics for more details on LTS result statistics.

20.8.4 The LTS Report Window

The LTS Report window presents the LTS report generated from the Result Items dialog (Figure 20.76). It has the following components and functionalities:

Preview Tab

Shows a preview of the report document, including content formatting.

Database Tab

Shows a tabular (unformatted) view of information included in the report.

Export

Button functionality allowing export of the generated report to various types of documents (e.g. *.DOCX, ‘.PDF, ‘.HTML, ‘.CSV, among others).
(Figure 20.76)

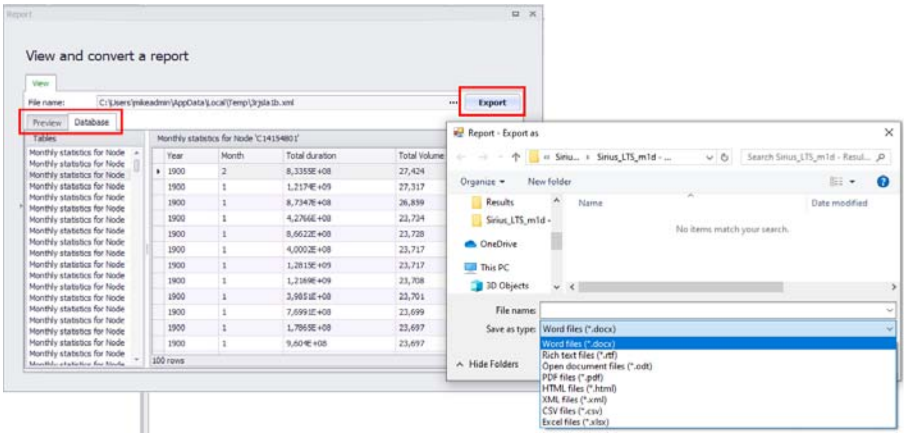


Figure 20.76 LTS report export functionality from the Report window

20.9 Cross section Plots

A cross section plot displays animated water level results from network and/or 2D overland result files.

Cross section plots are created from the maps (main Map view or any result map), and result files must be loaded beforehand. For 2D overland results, both flexible mesh results (.dfsu files) and rectangular grid results (.dfs2 files) are supported. New cross section views are opened using the 'Cross section plot' button.

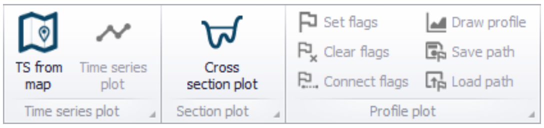


Figure 20.77 The Cross section plot tool in the Results ribbon

The 'Add cross section plot' window will show up, from where it is possible to select the files to plot the results from. When 2D overland files are available, a maximum spacing must also be specified, which controls the number of points to be plotted along the 2D cross section. Valid result files to be selected in this window are files containing a water level result item. If there is only one network result file available, this window will not show up, and the available result file will be used for the cross section plots.

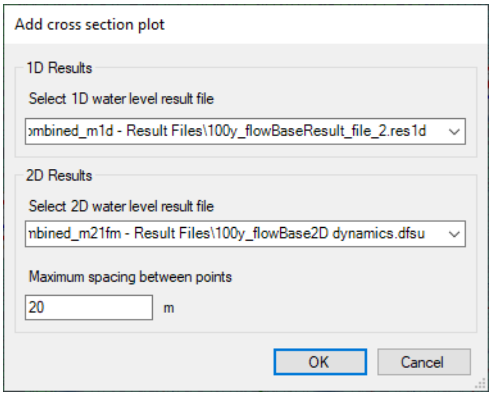


Figure 20.78 The settings for the Cross section plots

The date and time of the water level being shown can be selected from the Results ribbon, or by clicking along the time axis of a Time series plot.

The topography information in the 2D domain is obtained from the 2D domain file, selected in the '2D domain' editor, when a model database is opened. If MIKE+ is used for results presentation only (no model database opened), the 2D domain file or DEM to be used as source of the 2D topography line must also be selected in the 'Add cross section plot' window.

20.9.1 Creating cross section plots from river results

To open a cross section view from a river network result file, click on a location on the Map view with a cross section.

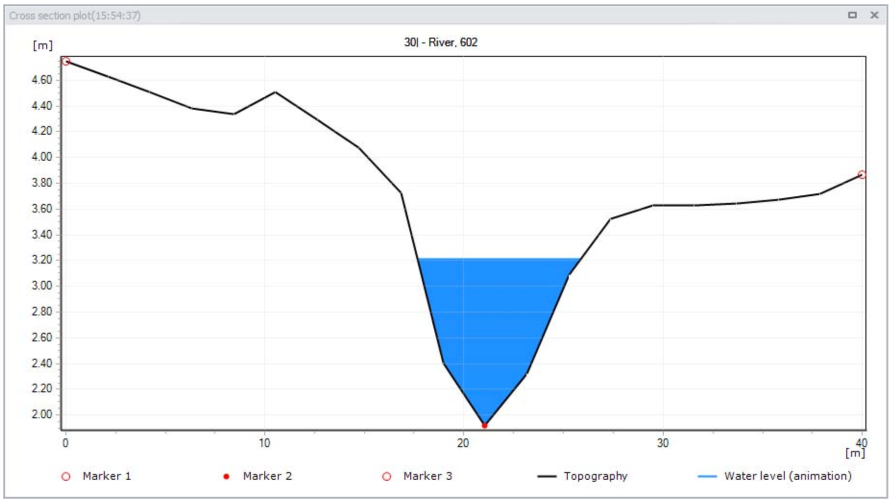


Figure 20.79 The Cross section plot window showing results from a river simulation

The topography line is obtained from the result file, and corresponds to the topography specified in the Cross sections editor. The water level is constant within this cross section, and is also obtained from the selected result point.

20.9.2 Creating cross section plots from 2D results

To open a cross section view from 2D overland result file, it is required to draw the horizontal location of the cross section on the Map view. To do so, hold the 'Control' key down and start digitizing a polyline on the map. It is possible to release the 'Control' key after clicking the first location.

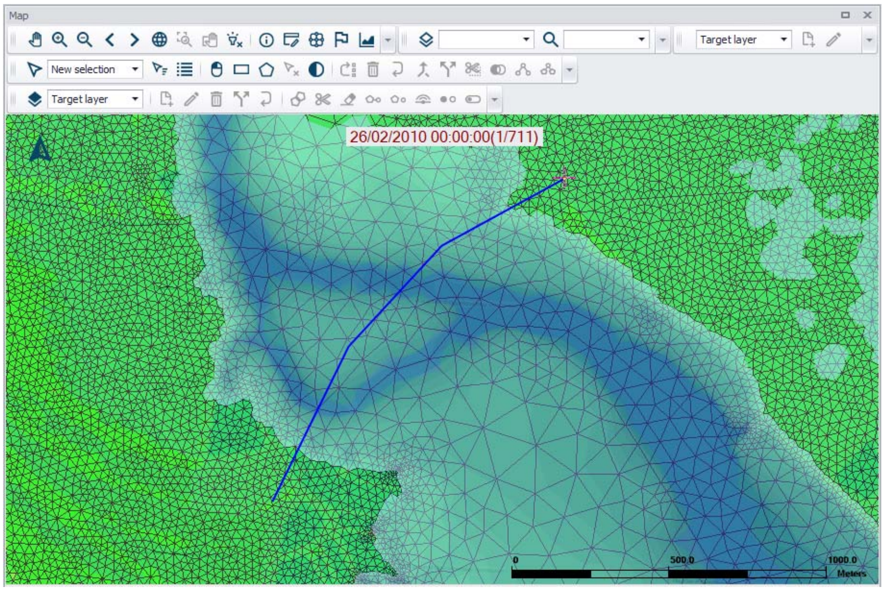


Figure 20.80 Digitising the location of a 2D overland cross section plot

The cross section plot shows a topography line with levels obtained from the 2D domain used in the 2D overland simulation, and a water level line obtained from the selected result file. Both lines are drawn with one point at each location clicked on the map, and adding intermediate points in-between with an equidistant interval controlled by the maximum spacing specified in the 'Add cross section plot' dialog.

For each point along the cross section, the plotted water level is the value of the 2D mesh element in which the point falls. For the topography line, the point's level is the average value from all the nodes defining the element.

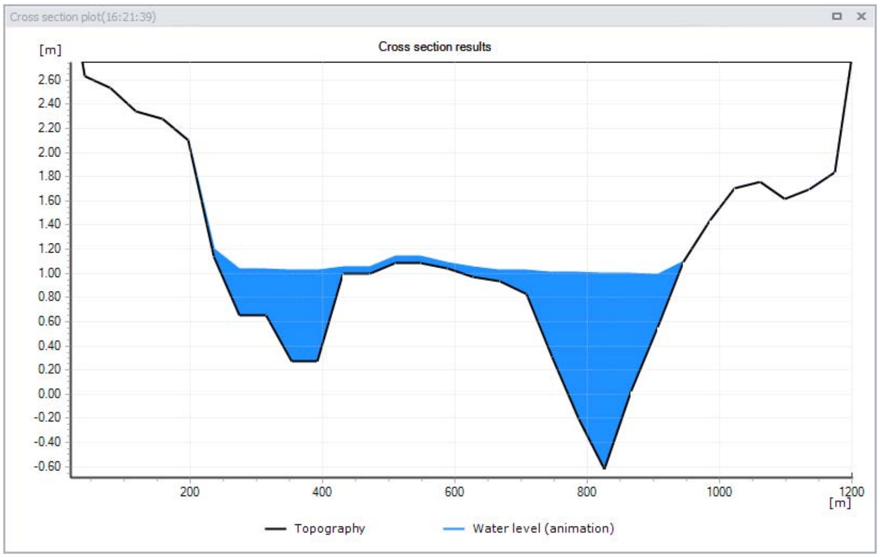


Figure 20.81 The Cross section plot window showing results from a 2D overland simulation

If there are gaps in the 2D domain (i.e. polygons excluded from the 2D domain), then the cross section plot will also show gaps at these locations.

20.9.3 Creating combined cross section plots

It is possible to combine river and 2D overland results in a common cross section view, in order to get an overview of the entire domain, regardless of the modelling technique. To open a cross section view with combined results, it is required to draw the horizontal location of the cross section on the Map view and to select (click) river cross sections during that process. To do so, hold the 'Control' key down and start digitizing a polyline on the map. It is possible to release the 'Control' key after clicking the first location.

During this digitisation process, it is possible to click river cross sections at any time, and the river cross section can therefore be added to the left, to the right or in the middle of the 2D cross section.

The resulting cross section plot shows the river cross section data first, and then 2D cross section data elsewhere. Therefore, in case the 2D results overlaps the extent of the river's cross section, the overlapping 2D data are not shown. On the contrary, there may be some gaps between the river cross section and the 2D cross section, in case there is a gap in the input 2D domain along the river. To reduce the risk of such gaps in the cross section plots, it is preferable to digitize the location perpendicular to the river.

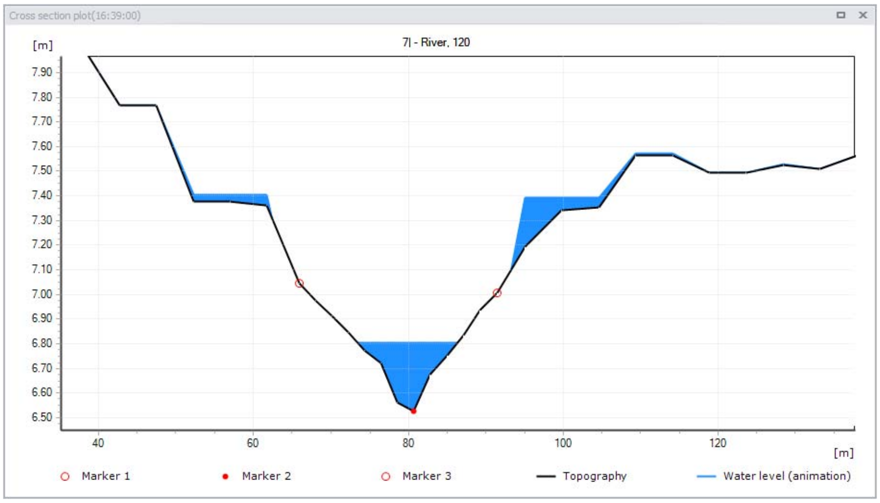


Figure 20.82 The Cross section plot window showing combined river and 2D overland results

The markers drawn on the cross section show the limits between the river and 2D overland data.

20.9.4 Plot Context Menu

Right-click on the profile plot to access the local context menu.

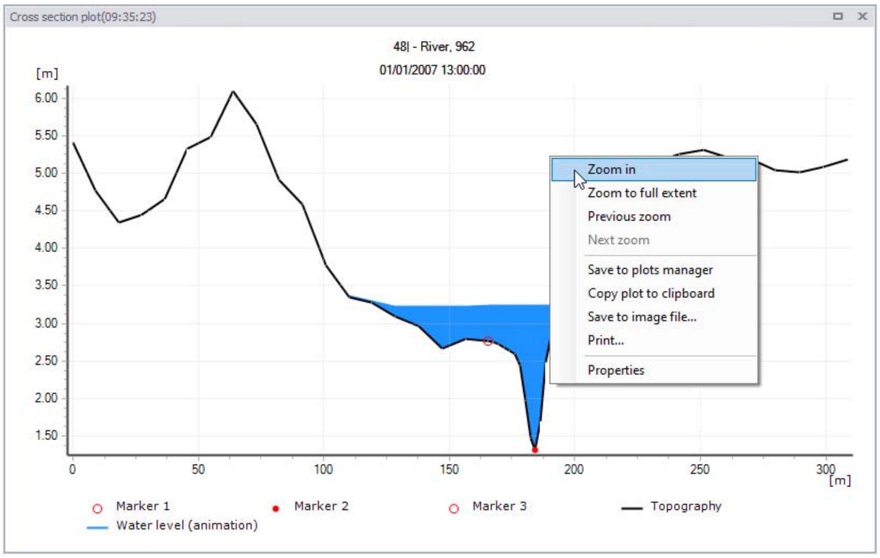


Figure 20.83 Right-click on the cross section plot to access the local context menu

Zoom to full extent, Zoom in, Next zoon, Previous zoom

Allows to zoom in and out on the plot. Zoom to full extent brings you back to the full view of visible cross section data on the plot. Panning is also enabled upon activation of zoom options, using the 'Shift' key.

Save to plots manager

Saves the cross section plot (location on the map and source result files) to the 'Plots' panel. The cross section plot will initially be added to the active folder from this panel. See 'Plots Management' chapter (page 487) for more information on options to save and manage results windows.

Copy to clipboard

Copies the cross section view displayed to the clipboard and allows it to be pasted into other applications.

Save to image file

Saves the cross section view displayed to an image file on the disk, using various supported image formats.

Print

Prints the cross section view displayed to the clipboard.

Properties

Activate this option to view the Cross section plot Properties dialog.

20.9.5 Cross section plot Properties

The properties of the cross section plot can be changed via the Properties dialog (Figure 20.84). The dialog is accessed from the 'Properties' option in the local context menu on the cross section plot area.

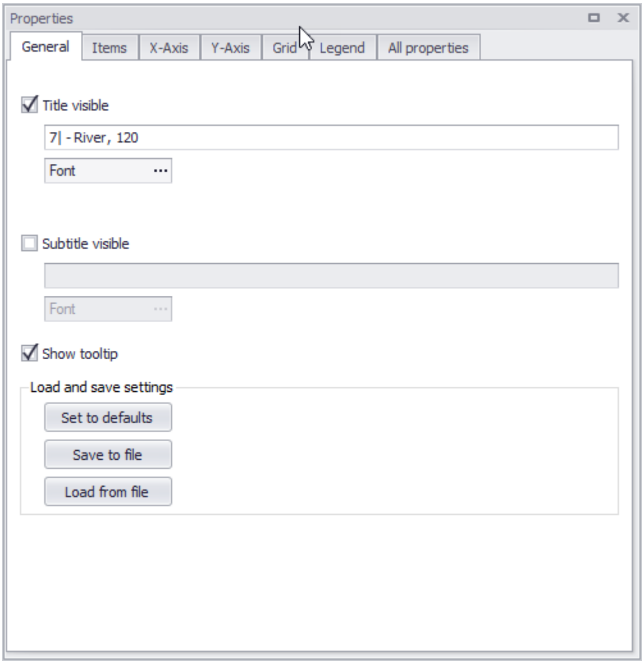


Figure 20.84 Setting the properties of the cross section plot

General

The dialog has various tab pages wherein changes to the cross section plot properties can be made.

The General tab page offers options for:

- Editing and formatting the plot title
- Adding and formatting a subtitle
- Showing the topography and water level results on the fly, in a tooltip
- Loading and saving these display settings to a file, to apply in other cross section plots

Items

On the Items page (Figure 20.85), it is possible to control which layers are drawn on the cross section plot. For each layer, it is also possible to customize its name shown in the legend, and change its symbology.

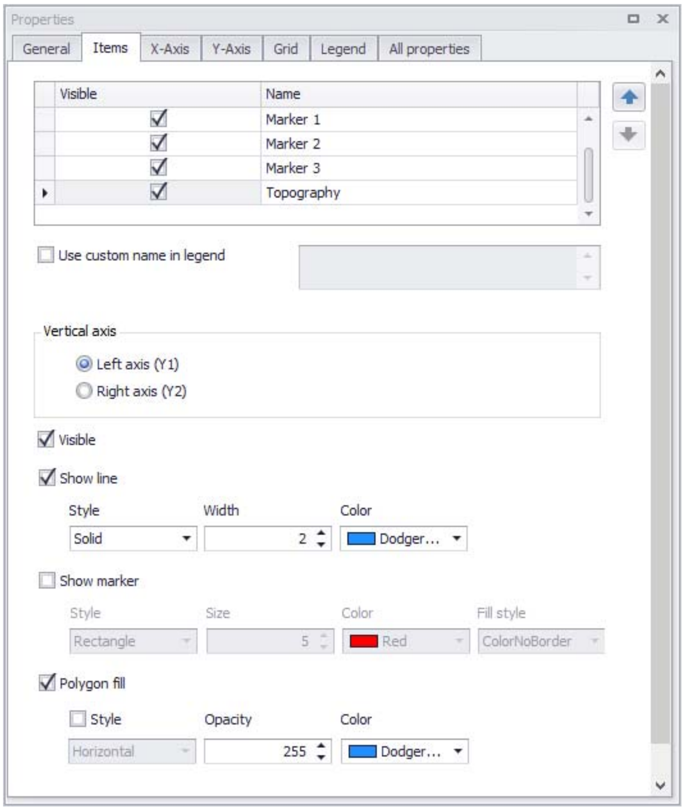


Figure 20.85 The Items tab page

X-axis

The X-axis tab offers options for customizing axis title and label fonts, as well as modifying the vertical line appearance.

Y-axis

The Y-axis tab offers options for customizing axis title and label fonts, formatting labels as well as modifying the horizontal line appearance.

Grid

The Grid tab offers options for customizing the style for the horizontal and vertical lines appearance.

Legend

The Legend tab offers options for showing or hiding the legend, and also controlling its location within the cross section window.

20.10 Scatter Plot

The scatter plot shows a graphical presentation of the relationship between any two result items from a common result file. It can e.g. plot Q/H relations in rivers or pipes, level-volume relations in tanks, flow/pressure relations in pumps, etc. In this plot, every point represents a different time step saved in the result file. This is only available for 1D network results.

To create a new scatter plot, right-click on the result file or one of its result items in the list of result files, or use the 'Scatter plot' button in the ribbon.

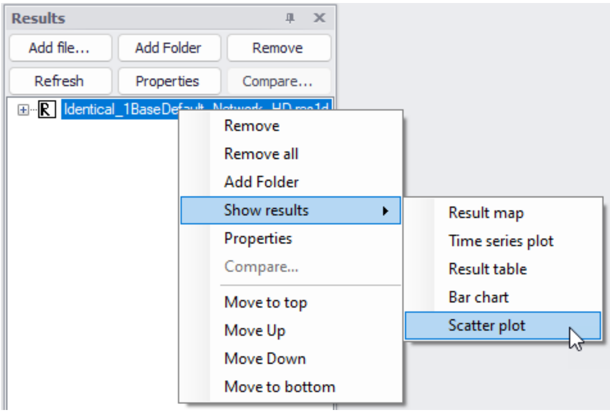


Figure 20.86 Creating a scatter plot from the list of result files

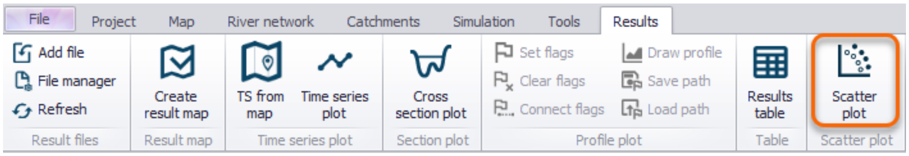


Figure 20.87 'Scatter plot' option on the Results ribbon

Both options will open the 'Add scatter plot' window below.

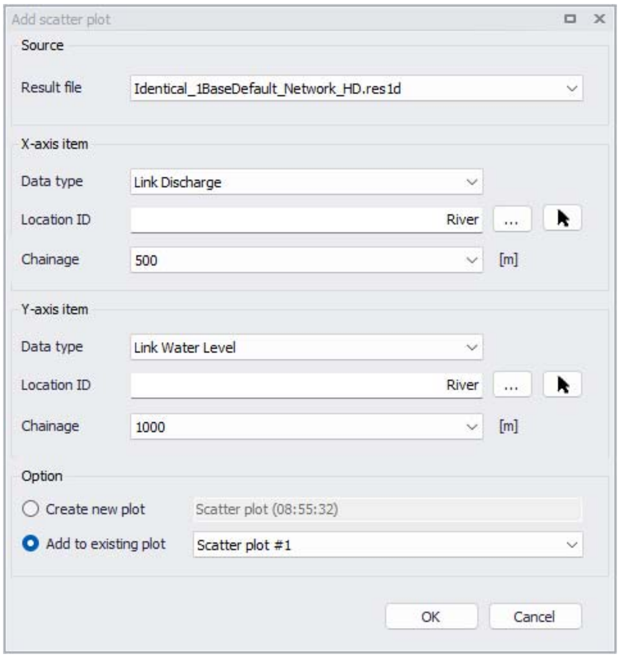


Figure 20.88 The 'Add scatter plot' window

Choose the result file result file to pick results from. Then for each axis, select the item type and location to plot.

Select between the two options:

- 'Create new plot': the new result data will be shown in a new window, named with the specified title.
- 'Add to existing plot' options: the new result data will be shown in the selected existing scatter plot.

Then click on the 'OK' button to create the scatter plot. This will plot the results in the scatter plot window.

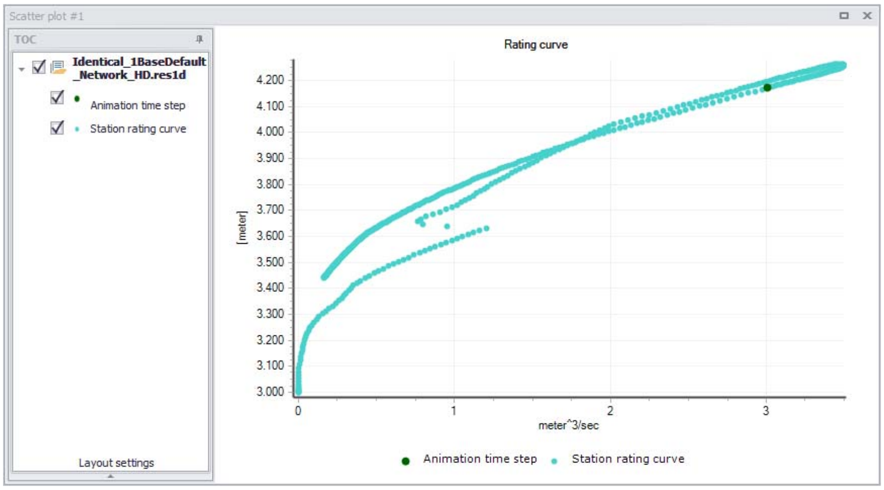


Figure 20.89 The scatter plot window

Once a scatter plot is created, additional result time series can be added to the plot in various ways:

- Right-click in the table of content of the scatter plot and select 'Add items'
- Use the 'Add scatter plot' tool again, with the option to add the new results to an existing plot
- Add result items from another scatter plot, using the Copy / Cut / Paste options in the context menu of the table of content.

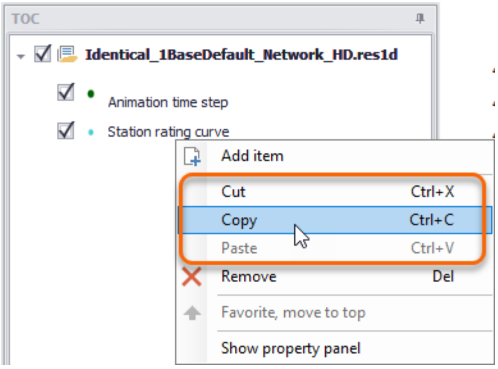


Figure 20.90 Options to copy and paste scatter plots from one plot to another

20.10.1 Data series format

To customize the appearance of a scatter plot series, right-click on a data series and activate the 'Show property panel' option from the local context menu (Figure 20.17).

Options for configuring data series appearance include customizing line color and style, adding markers, and changing marker styles and size.

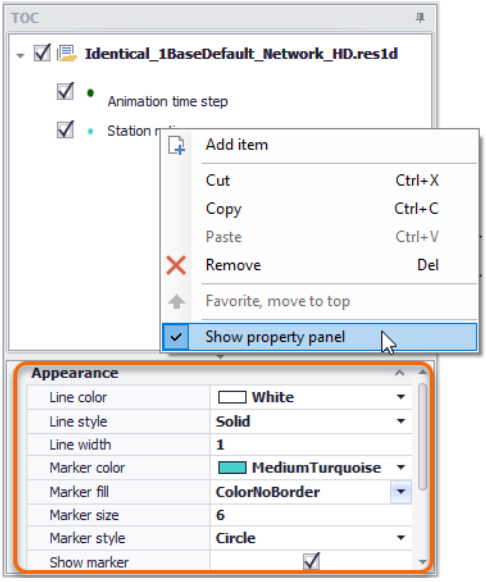


Figure 20.91 Customize the appearance of scatter plot data series via the Property Panel

20.10.2 Context menu

Right-click on the scatter plot to access options to control the zoom level, copy data or export to an image file.

'Zoom in' lets you draw a rectangle on the plot to select the area to zoom to. While drawing a rectangle, dragging to draw a horizontal line will display arrows to zoom along the horizontal axis only, keeping the vertical axis unchanged. Similarly, dragging to draw a vertical line will zoom along the vertical axis only. 'Zoom to full extent' brings you back to the full view of visible data. Note that additional options are available to control the zoom options:

- Hold down the Shift key, to zoom in
- Scroll with the mouse wheel to zoom in or out
- Hold down the Ctrl key to pan.

'Open time series…' opens a Time Series Plot showing two time series for each scatter plot series, respectively representing the items plotted on the X-axis and on the Y-axis of the scatter plot.

'Save to plots manager' will save the scatter plot's content (list of result items and locations, and display settings) to the 'Plots' panel. The scatter plot will initially be added to the active folder from this panel. See 'Plots Management' chapter (page 487) for more information on options to save and manage results windows

'Copy plot to clipboard' will copy the plot as an image in memory, to be pasted in another program. 'Copy data to clipboard' will copy the scatter plot values in memory, to be pasted in another program.

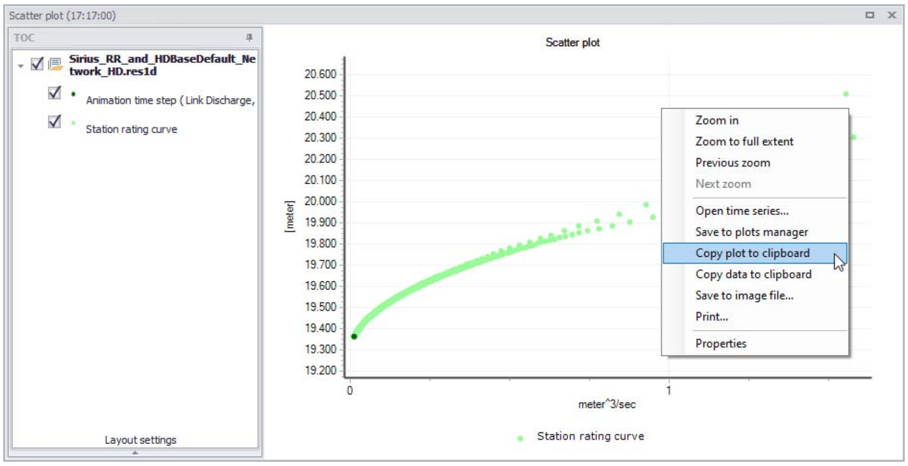


Figure 20.92 Context menu of the scatter plot

The context menu also offers a 'Properties' option to control the layout and the symbology of the scatter plot.

20.11 Pump Q-H Plot

This type of plot is only relevant to Water Distribution models. It displays the operating pump points computed during the simulation period, on top of the pump curve defined in the 'Pumps' editor. QH results are obtained from the 'Pump flow' and 'Pump headloss per 1000unit' result values at each time step saved in the result file.

To display these Q-H results, click the 'Pump QH plot' button in the 'Results' tab of the ribbon, and select the result file to get pump results from (this result file must be loaded in the MIKE+ project beforehand).

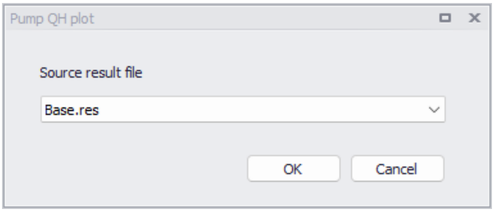


Figure 20.93 Selecting a result file to plot its pumps Q-H results

After selecting the result file, the 'Pumps' editor shows QH results from all corresponding pumps. Note that QH results from multiple result files may be superimposed by repeating the above step.

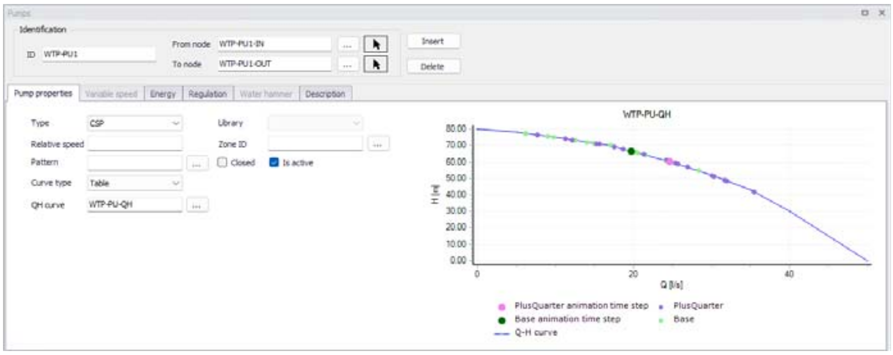


Figure 20.94 Displaying Q-H results from two simulations in the Pumps editor

For each result file added, two series are added to the plot: one showing the entire set of results (one QH point per time step in the result file), and one showing the current time steps as selected in the 'Results' tab of the ribbon.

The 'Properties' editor, available from the context menu of the plot, allows editing symbols for all series on the plot as well as removing results series.

20.12 Hydrant Q-H Plot

This type of plot is only relevant to Fire Flow analyses, when applying the method to compute a Q-H curve. In this case, the computed Q-H curve storing the relationship between the hydrant flow and the residual pressure is saved to a .csv file.

To display these Q-H data, the corresponding .csv files must first be loaded in the 'Results' tree view. New plots are created by right-clicking on the file, and selecting 'Show Q-H curve' in the context menu.

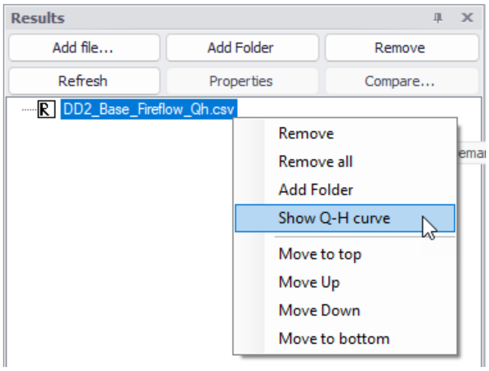


Figure 20.95 Creating a new Q-H plot

This opens a selection window, showing the list of .csv files with Q-H data and the list of junctions for which they contain a table. Select the list of junctions and files to plot, and then select between plotting data to a new window or to an existing one.

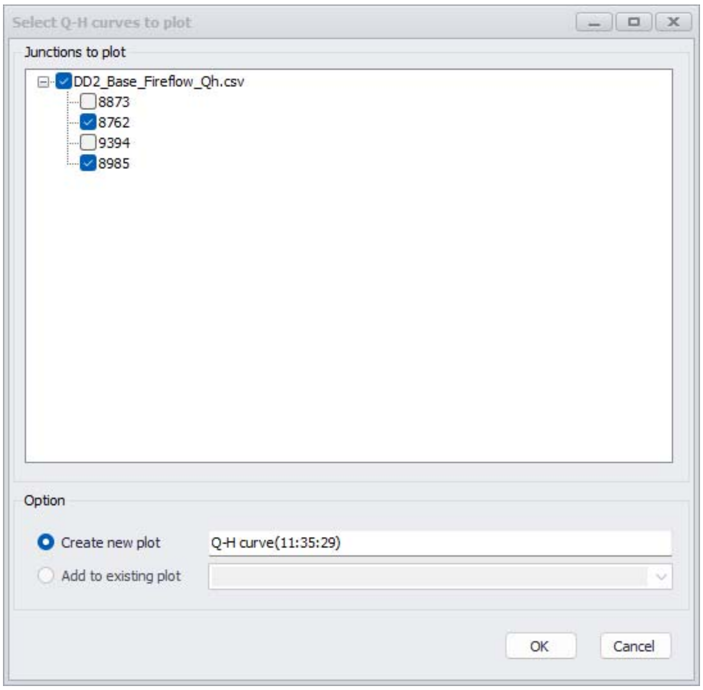


Figure 20.96 Selecting Q-H data to plot

Once opened, the Q-H plot window offers similar settings for controlling the layout as the Time Series Plot window (see page 403).

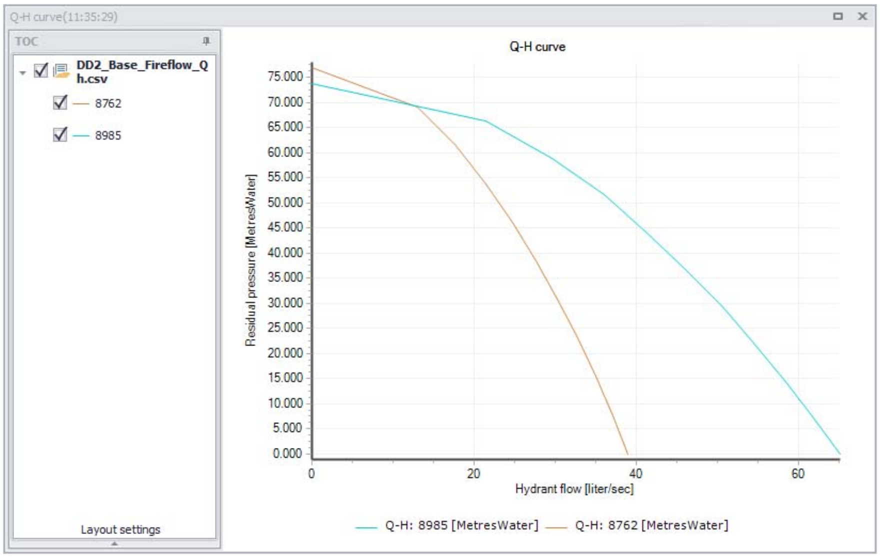


Figure 20.97 The Q-H plot window

20.13 Animations

After having loaded result items into the project (see Chapter 20.3) and plotted dynamic items on a map (see Chapter 20.2), on a profile plot or on a cross-section plot, it is possible to animate results. All result plots are synchronized, i.e. they show the same date and time. To animate results, go to the 'Results' ribbon and use the tools available in the 'Animation' toolbox:

- Start plays the animation.
- Step forward: moves to the next time step
- Step backward: moves to the previous time step
- Faster: speeds up the animation, when automatically animating the results using the 'Start' button
- Slower: slows down the animation, when automatically animating the results using the 'Start' button
- Go to beginning: moves to the first time step
- Go to end: moves to the last time step
- Track bar: move the cursor to quickly change the time step
- List: expand the list to see all date and times and pick a new time step.

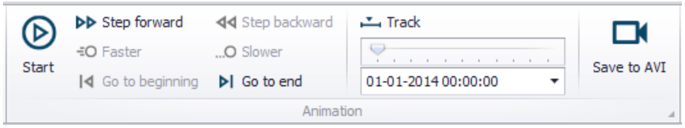


Figure 20.98 Animate dynamic results using tools from the Animation toolbox on the Results ribbon

The date and time of the results can also be changed by clicking in a time series plot: the selected date and time in the plot will be used for the animation in the other results views.

When multiple result files are loaded on the map, they may have different time spans and different storing frequencies (i.e. different time steps). Therefore, it is not possible to display results from all result files at the exact same date and times. So, the following approach applies:

- The time step of the animation is the smallest time step from all result files loaded on the map
- At a selected date and time of the animation, if a result file does not have data at this exact date and time, then the nearest date and time (which can be either before or after the selected date and time) is shown.

A video file of the result animation may be generated and saved using the 'Save to AVI' tool. On the 'Save animation to AVI file' dialog, one may specify:

- Target map. Whether to record the video with a result map or the model map.
- The starting and ending time steps for the recording. This allows for recording only part of available results.
- The file name of the video to be generated, and its compression quality.

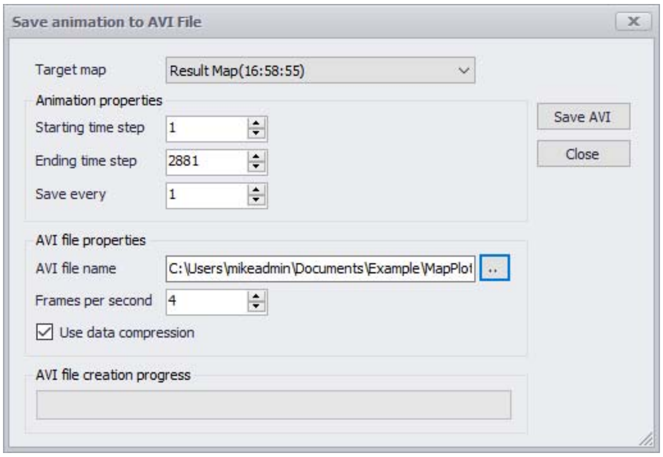
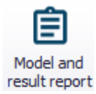


Figure 20.99 Setting the properties for saving the animation into a video file

20.14 Reports



MIKE+ has facilities for setting up reports based on information from model data as well as simulation results. It is also possible to join information between different tables in the report.

The ‘Model and Result Report’ tool is found under the Tools ribbon. The tool uses a wizard approach for configuring reports.

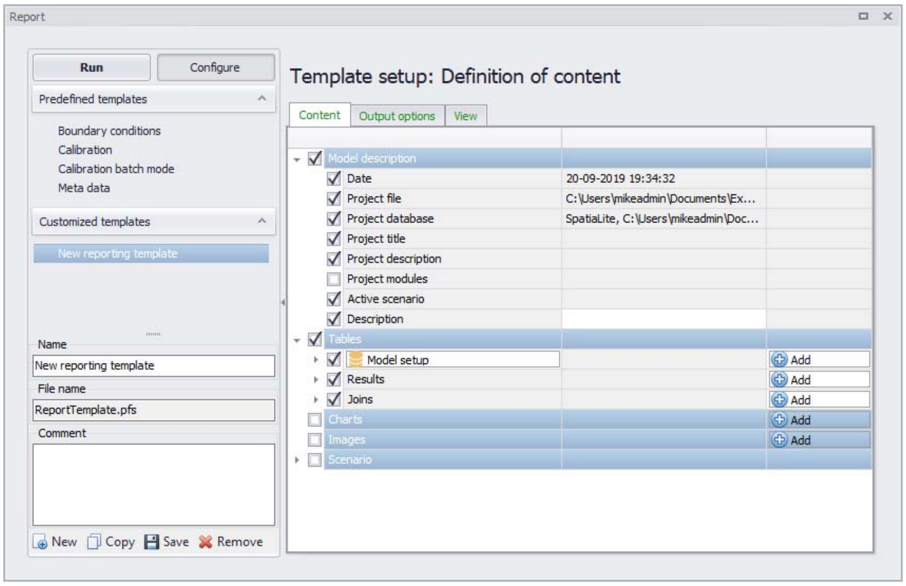


Figure 20.100 The MIKE+ Report Editor

20.14.1 Setting Up a Report

The report tool generates reports based on configured templates. A list of templates is shown on the template navigator on the left panel of the editor (Figure 20.101). Templates are organized as:

- **Predefined templates**. Preconfigured report templates designed around typical reporting themes, such as:
  - Boundary conditions
  - Calibration
  - Metadata
- **Customized templates**. User-configured report definitions.

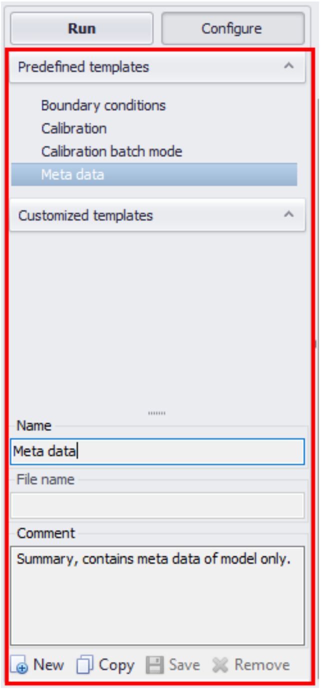


Figure 20.101 The template navigator on the Report editor

Set up a new custom report template with the ‘New’ button at the bottom of the panel. Specify a name for the custom template, and add a text description under ‘Comment’ (Figure 20.102).

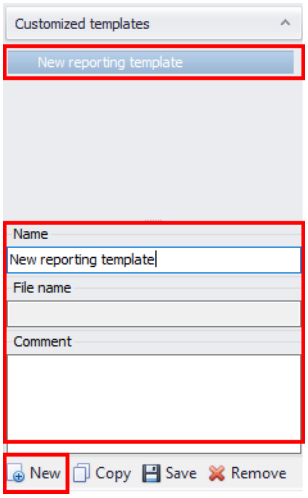


Figure 20.102 Defining a new custom report template

20.14.2 Content

On the Content tab page on the right panel (see Figure 20.103), it is possible to define contents for the report template setup:

- Navigate to the Content tab on the right panel
- Activate/deactivate items under each data group depending on content preferences for the report being set up. The available data groups are:
  - Model description
  - Tables
  - Charts
  - Images
  - Scenario
- The 'Add' button allows adding information from MIKE+ model setup tables and loaded results, as well as adding charts and images.
- The 'Remove' button will delete items from the report setup.

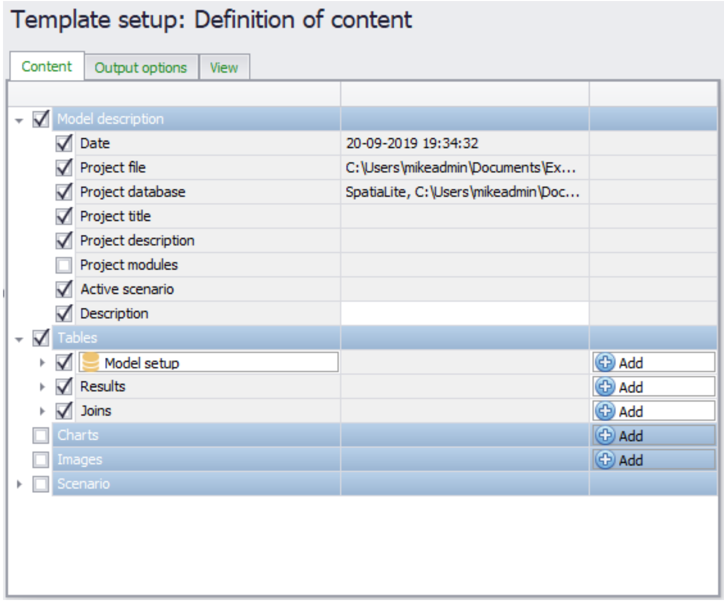


Figure 20.103 The report can contain information from the database tables as well as from result files

After choosing the tables to include, select which attributes of the data table you wish to add to the report (see Figure 20.104). The 'Columns' field provides a heading for columns in the report.

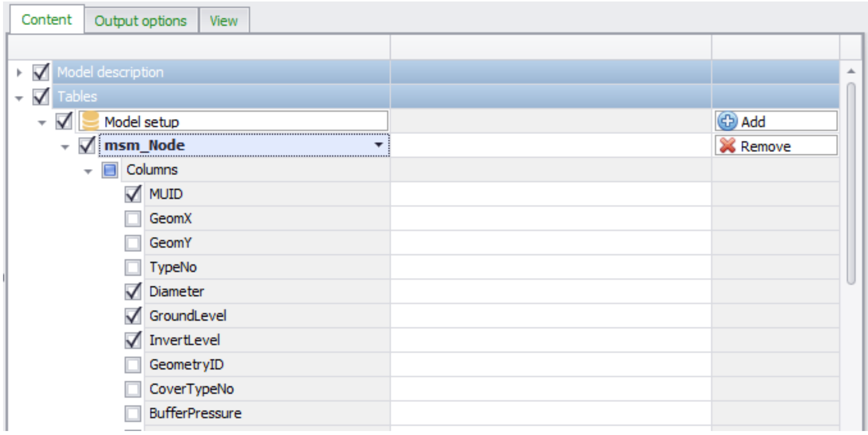


Figure 20.104 Specification of content from the table added to the report as well as the column title

Adding results information is done in a similar manner. It is necessary to have the result file loaded in the project beforehand (see Figure 20.105). When adding results you can choose to add summary statistics.

You can also specify reference values (i.e. ‘Values highlighted’) against which values will be compared in the report.

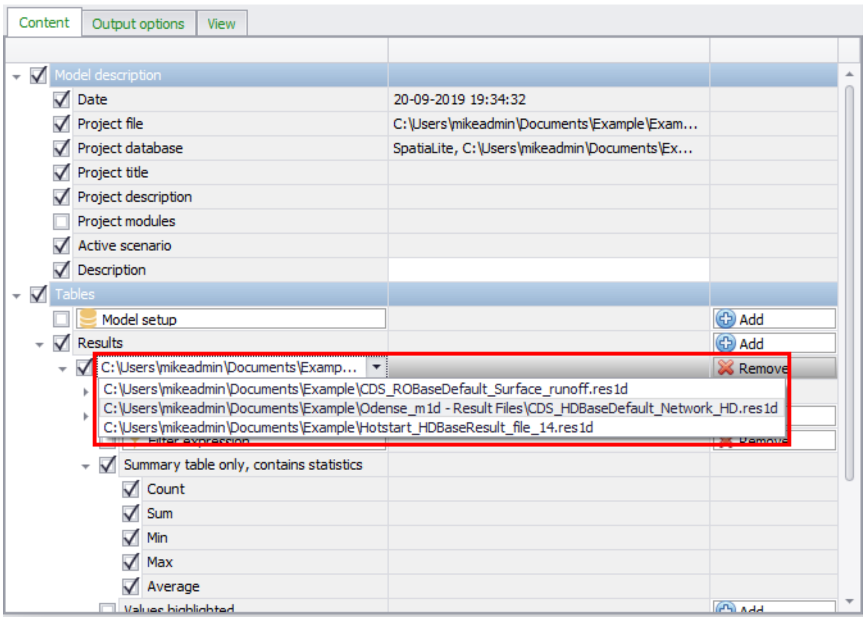


Figure 20.105 When adding result information to reports, you can choose to display summary statistics, e.g. average, maximum, etc.

Join of Tables and Results

It is also possible to combine data from two different tables into a joined table in the report.

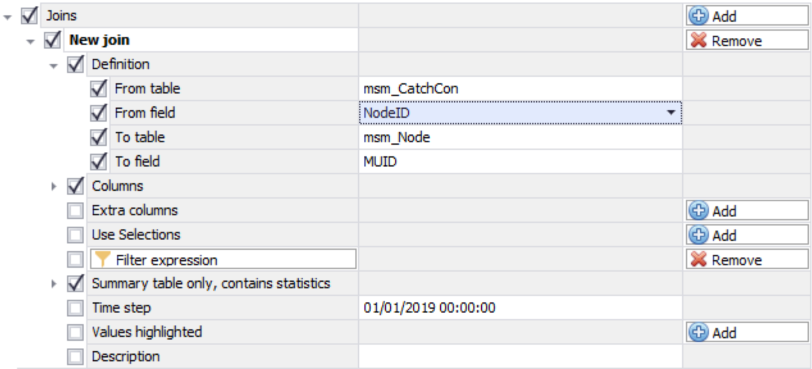


Figure 20.106 When joining tables you need to specify the common field in the two tables used for the join

Choosing the field with which to base the join: In the example shown in Figure 20.106, the join is based on the ‘MUID’ in the msm_node table. The table that we wish to join information from is the msm_CatchCon table (containing the catchment connections information for the network model) - the field in this table to base the join on is ‘NodeID’. This means that the report will list all the catchments connected to each node.

Unmatched records will contain no values for fields being appended from the join table, e.g. if no catchments are attached to a specific node then the joined columns in the report will be left empty.

In the 'To table' list, it is also possible to select a result file, amongst the result files previously loaded into MIKE+. This can create a joined table containing both results and model data.

Statistical results (average value, maximum value, etc.) can be added to this table by activating 'Summary table only, contains statistics'. Instantaneous results from a given date and time can also be included, by activating 'Time step' and selecting the expected time step of the results.

Using Filters

If you wish to set up the report only to show information from selected elements, you can do so by specifying a filter.

You can use existing selection lists, or filter expressions. In the example shown in Figure 20.107, a selection list (i.e. ‘Bell’) and a Field expression (i.e. [Diameter]>1) are used to qualify which nodes to include the report. If no filter is used, all elements are reported in the order they are extracted from the database.

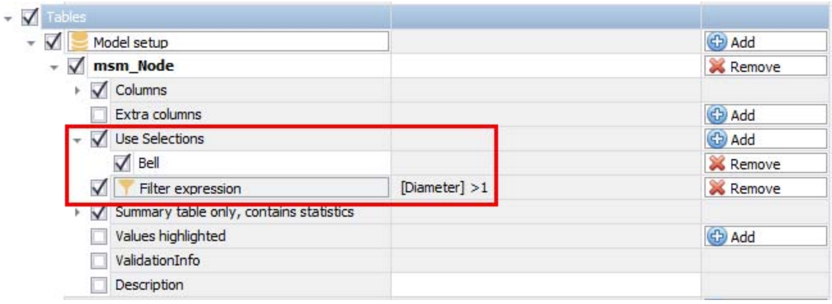


Figure 20.107 A report can be limited to only selected elements

20.14.3 Output Options

Configure report appearance and style in the Output Options tab page of the Report editor (Figure 20.108).

Reports can be generated in several formats, including HTML and CSV. Style sheets are used in generating the reports. MIKE+ comes with two different style sheets -- one for HTML format (MUReport.xsl) and one for CSV format (MUReportCSV.xsl). These default style sheets are installed in the Templates folder of your MIKE+ installation (i.e. the 'Templates' folder in the MIKE+ installation directory). It is also possible to use custom style sheets via the browse option in the tab page.

Title

Specify a title for the report in this input box.

Order

This panel lists the selected report items from the Content tab. The order of the items may be modified using the 'Up' and 'Down' buttons to the right of the panel to reorder the items on the list.

Style

Define the style sheet to use in generating the report. It can be selected from default styles sheets, or custom style sheets.

Target file name

File name and path for the XML file holding the report information that will be generated.

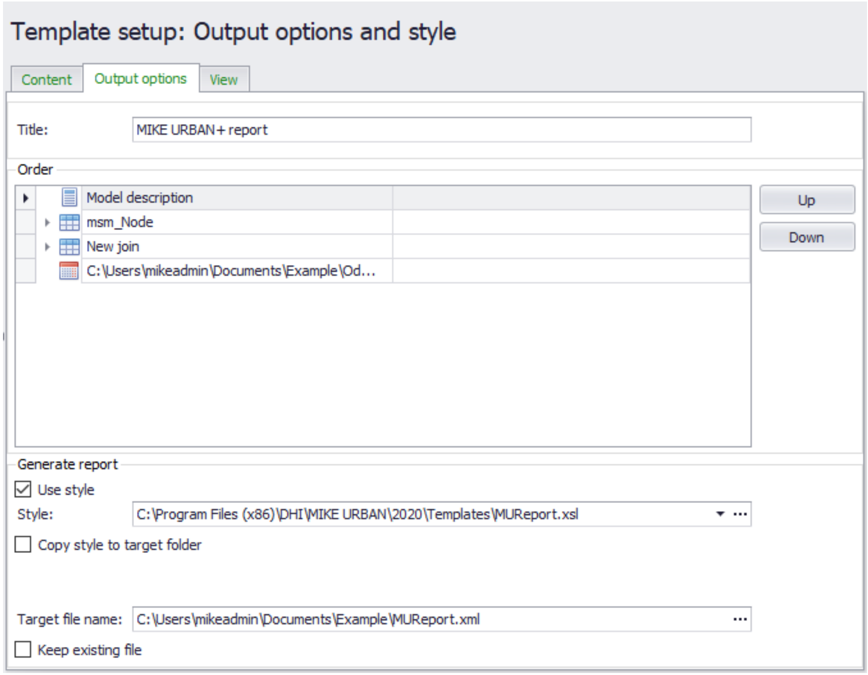


Figure 20.108 Specify the format of the generated report

20.14.4 Run the Report Setup

After defining the report content and format in the Content and Output Options tabs, respectively, execute the report configuration by clicking on the 'Run' button on the left panel of the Report editor (Figure 20.109).

A preview of the generated report is then displayed on the View tab of the editor.

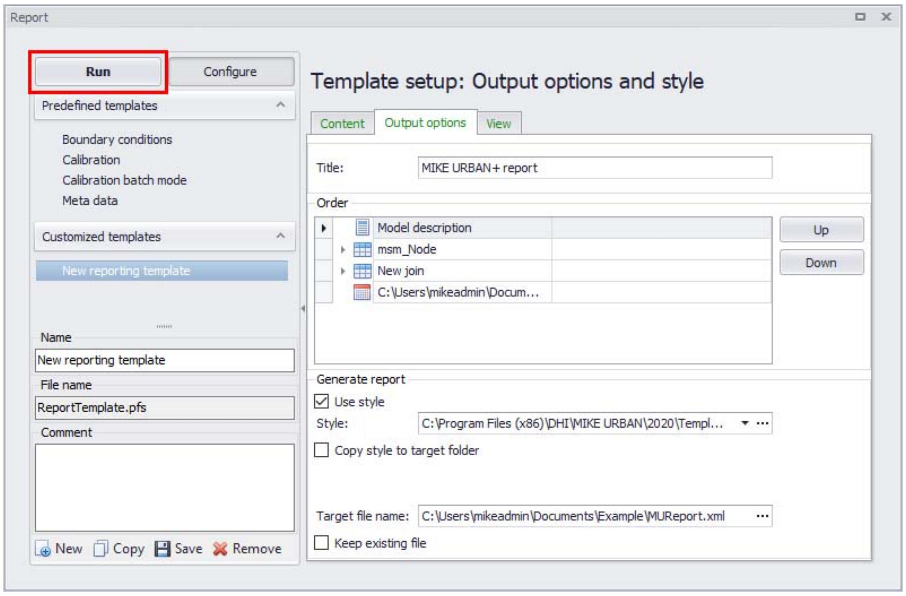


Figure 20.109 Run the report setup configuration

20.14.5 View

After running a report configuration, a preview of the generated report is displayed on the View tab of the Report editor (Figure 20.110).

Use the ‘Export’ button to save the generated report to various types of document formats(e.g. *.DOCX, ‘.PDF, ‘.HTML, ‘.CSV, among others).

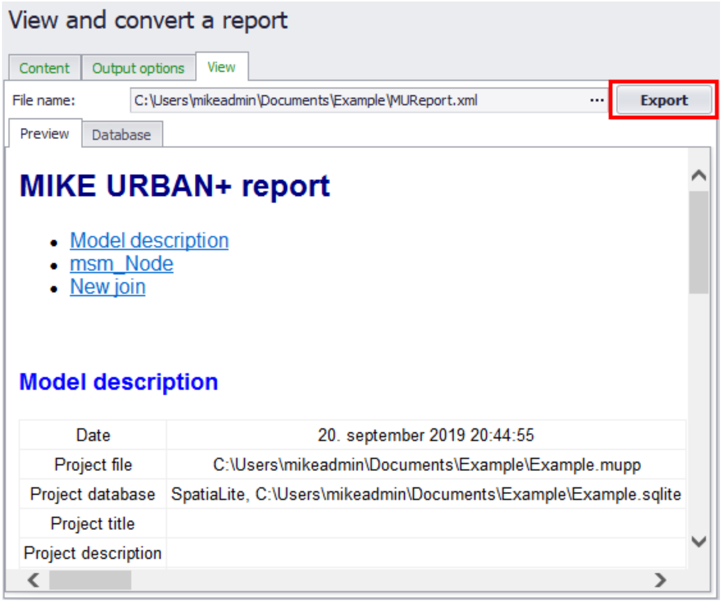


Figure 20.110 The View tab page presents a preview of the generated report

20.14.6 Save the Configuration File

Once the report layout is in place, it is recommended to save the custom report configuration so that it may be reused. Button functionalities on the template panel are described below:

Run

Executes a report template configuration.

Configure

Presents the Content and Output Options tab pages of the Report editor for modification. Predefined templates may not be edited.

New

Creates a new (custom) report template.

Copy

Makes a copy of an existing report template (e.g. predefined template) and adds it to customized templates list.

Save

Saves a custom template into a *.PFS file.

Remove

Removes an item from the customized templates list.

20.15 Result Comparison

It is possible to compare results that are computed on similar networks, but with different parameters. The comparison basically subtracts the results from one another. The comparison becomes a new result layer available for plotting on a result document (e.g. a result map, time series plot, table, profile plot, or bar chart).

The result comparison option supports the following result file types for comparison: .res1d, .dfsu, .dfs2, .res, .resx, .out. Other file types cannot be compared.

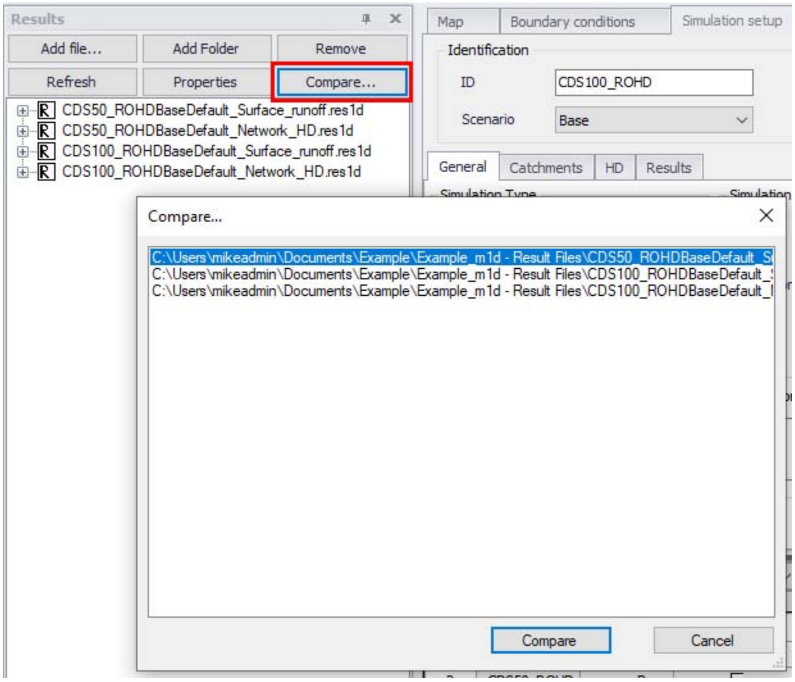


Figure 20.111 Loading result files for comparison of results

To compare results:

- Run two different instances of a simulation (e.g. network simulation) with different input parameters but on a similar network setup or on a 2D domain with unchanged grid / mesh.
- If not automatically loaded, load the simulation results to compare on the Results manager. (See Chapter 20.3 - “Loading Results” on page 414)
- Select one of the result file layers (i.e. File A) on the Results manager, and click on the ‘Compare...’ button at the top of the panel (Figure 20.111). This will launch a dialog listing other loaded result file layers available for comparison with the currently-selected layer.

- Alternatively, launch a file comparison via the results manager local context menu by right-clicking on a result file layer (Figure 20.112).

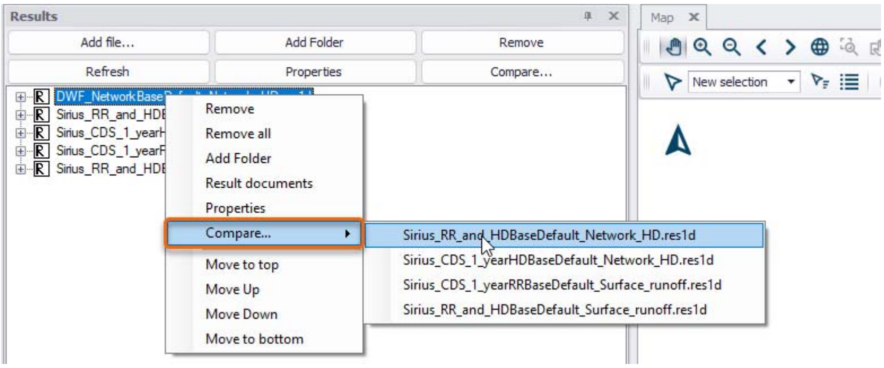


Figure 20.112 Compare results via the local context menu

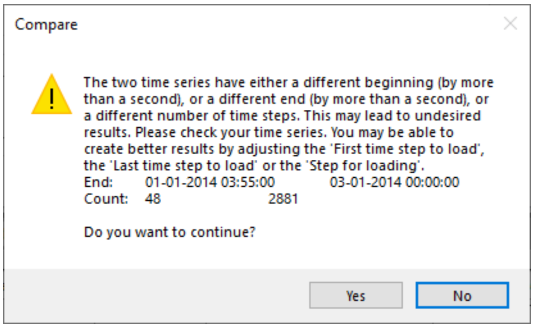
- Select the second (compatible) result file (i.e. File B) for the comparison and click on the 'Compare' button. The comparison takes values from the first result file and subtracts corresponding values from the second result file (i.e. File A minus File B).
- A new result layer based on the comparison (i.e. differences) is created and added to the Results manager.
- Result documents may be generated based on the various result items of the resulting comparison layer.
- Create a result document (i.e. map plot, time series plot, etc.) via the file manager local context menu.

Differences in result items

Result items not common in the files being compared will not be available in the comparison result.

Differences in Time Steps and Period

Comparison is done based mainly on the order of time series values. A mismatch in the number of time steps results in time series containing the smaller number of rows. With .res1d results, mismatch in time series periods and time step widths will return a warning about the mismatch; continuing with the comparison should be reconsidered.



For .res, .resx, .dfs2 and .dfsu files, the comparison supports different simulation periods but doesn't support different time step intervals.

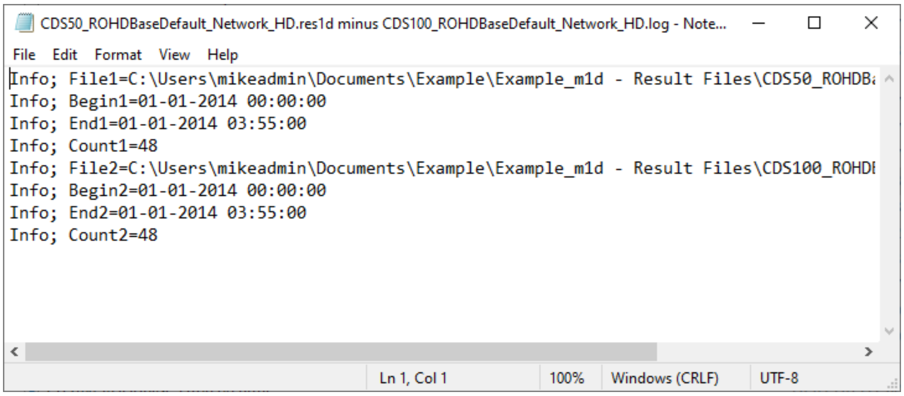
For .out files, the comparison requires that the two files have the same duration and same reporting frequency.

Differences in Network Geometry

If the two result files have geometry differences (e.g. link deleted), the result comparison skips non-existing items. Thus it is still possible to compare scenarios where e.g. network as been changed or extended.

Comparison Log File

The Comparison tool creates a log file in the project directory. The file is named similarly as the resulting comparison file but with a *.LOG extension. It contains information on the files being compared.



20.16 Export Results to Shapefiles

Export MIKE+ simulation result layers to shapefiles via the Layers and Symbols panel or the Result Map plot TOC panel.

20.16.1 From Map Layers and Symbols

Right-click on a result layer item on the Map via the Layers and Symbols panel to access the local context menu. Select 'Export...' and then 'Export layer to shapefile'. Specify the name of the shapefile and click on 'Save'.

This will export the selected results layer to a shapefile. The exported shapefile contains the layer geometry, the simulated result item value, and the unique MUID.

When only some items are selected on the map, the option to export only the selected items from the layer is also enabled.

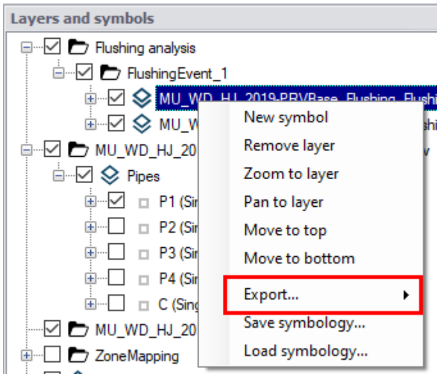


Figure 20.113 The export option available from result layers

For river models, it is also possible to export a shapefile with the cross sections layer, and with water level results in these cross sections. In the result file, cross sections are saved in the links result layer, therefore this option is only available from a link result layer and when displaying its water level result.

It is possible to export only some cross sections results using the option 'Export selected cross sections to Shapefile'. To enable this option, some rivers from the result layer must be selected first: all cross sections from these selected rivers will be exported.

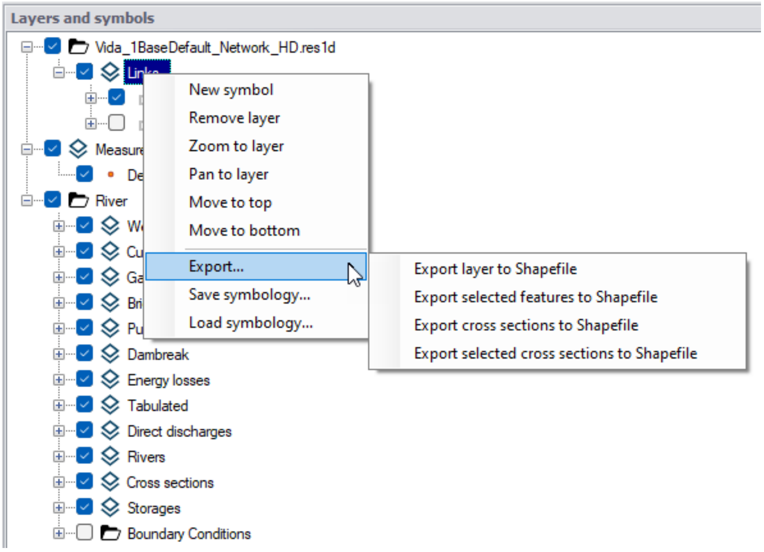


Figure 20.114 The options to export water level results from the river cross sections layer

For 2D overland models, it is possible to export vectors and/or isolines.

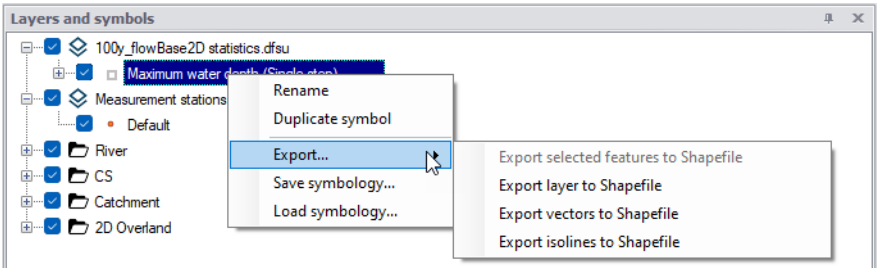
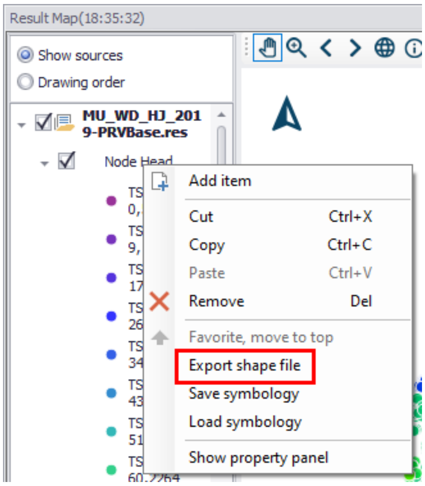


Figure 20.115 The options to export 2D overland vectors or isolines

20.16.2 From Result Map TOC

If you have created a result map plot from simulation results (See “Displaying Results on a Map” on page 394.), right-click on a result map layer on the left panel to access the context menu. Select ‘Export shapefile’. Specify the name of the shapefile and click on ‘Save’.



For river models, it is also possible to export a shapefile with the cross sections layer, and with water level results in these cross sections. In the result file, cross sections are saved in the links result layer, therefore this option is only available from a 'Link water level' result layer.

20.17 Plots Management

The 'Plots' panel allows saving and organizing all types of result presentation windows (hereby referred to as plots) for later reuse: time series plots, results tables, result maps, etc. This also applies to profile plots even when they do not include any result layer.

All plots can be organized in folders, created with the 'New folder' button at the top of the panel or from the context menu. Plots and folders can be dragged and dropped to re-organize the lists.

The plot type can be identified based on the plot's icon which is identical to buttons from the 'Results' tab in the ribbon.

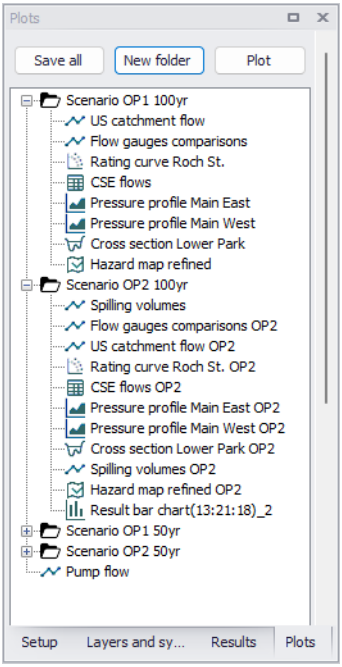


Figure 20.116 The Plots panel with several result plots saved in various folders

There are two ways to save a plot to the Plots panel:

- Use the 'Save to plots manager' option from the plot to be saved: if the plot already exists in the panel, it will be saved again with its updated settings (e.g. changes of result items, or changes to symbols). If it does not already exist, it will be added to the active folder.
- Use the 'Save all' button at the top of the panel: this will save all open plots. Plots which already exist will be saved with their updated settings whereas others will be added to the active folder.

The following options can be used to open a saved plot:

- Double-click on the plot name
- Use the 'Plot' button at the top of the panel to open the selected plot
- Press the Enter key to open the selected plot
- Use the 'Plot' option in the context menu, after right-clicking on the plot name.

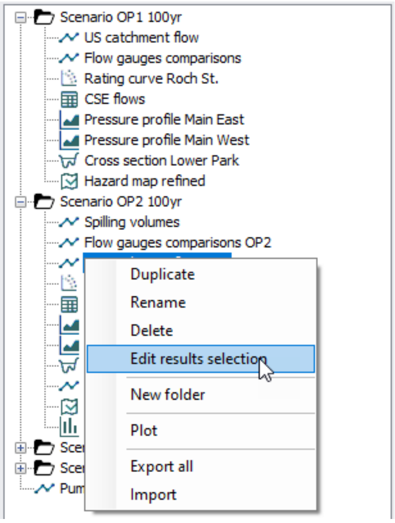


Figure 20.117 The context menu of the Plots panel

Additional options are available from the context menu (right-click on a plot or folder). These are:

- Duplicate: creates a copy of the selected plot, or the selected folder with all its content.
- Rename: renames the selected plot or folder.
- Delete: deletes the selected plot, or the selected folder with all its content. This operation can be canceled using the Ctrl+Z keys, right after the deletion.
- Edit results selection: this is only available for time series plots, profile plots and scatter plots. It opens an editor where the list of result items and locations can be edited prior to displaying the results. This e.g. allows to re-use existing plots with alternative result files, by duplicating the plot and then editing its input result file. See following chapters for more details.
- New folder: adds a new folder to the list. New folders are initially created at the bottom of the tree, but can later be moved by click-and-drag.
- Plot: opens the window corresponding to the selected plot. Using the 'Plot' action on a folder opens all plots in this folder.
- Export all: saves the content of the Plots panel (list of plots and folders) along with the plots' settings (result items, symbols, etc.) to a text file, which can be imported afterwards in another MIKE+ project.

- Import: imports the list of plots and folders from a text file saved from another MIKE+ project. This appends the content of the text file to the existing plots in the panel, when they have different plot names or folder names. If a plot in the file has the same name and the same folder location as an existing plot, then it is overwritten with the file's settings. Note that, in this text file, locations of result files are saved using paths relative to the location of the text file itself: it is therefore mandatory that the relative locations of the text file and result files are preserved, so that links to result files are restored while importing the text file again.

Note: Although two plots can be given the same name (which may e.g. be expected when similar plots are used in different folders), it is recommended to use unique names for each plot, because it is not possible to open multiple plots with the same name at the same time.

20.17.1 Editing time series plots

The 'Edit results selection' option opens the editor below for time series plots.

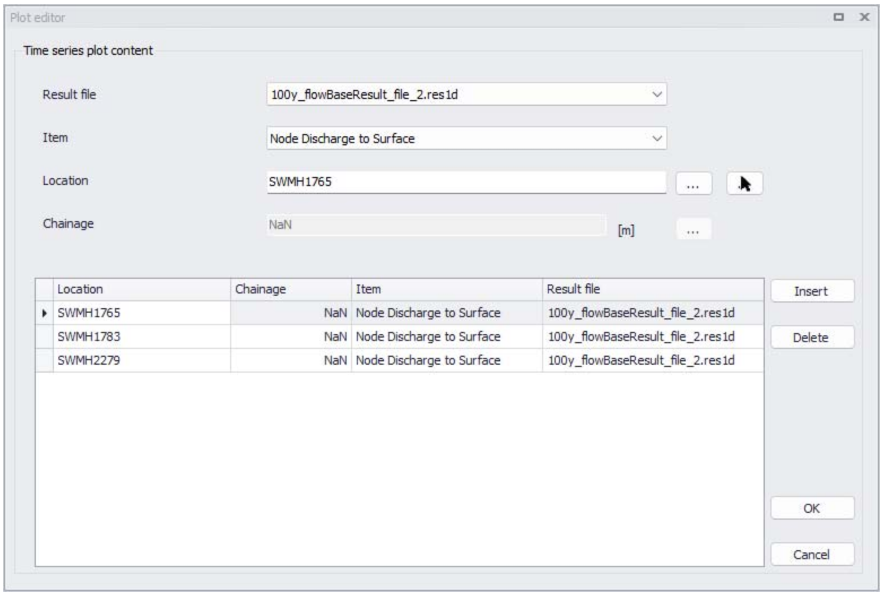


Figure 20.118 The Time Series Plot Editor

The overview table shows the list of time series included in the time series plot, while the above fields show the same properties for the selected record in the table. Time series can be added or removed from the plot using the 'Insert' and 'Delete' buttons on the right.

Each time series is defined with the following editable properties:

- Result file: this is the file name of the source result file. It is possible to select from the list of files currently loaded in the MIKE+ project from the drop-down list.
- Item: this is the type of result item. It can be selected from the drop-down list, which shows all items currently loaded in the MIKE+ project for the selected file.
- Location: this is the item ID where the time series is picked from, e.g. node ID, link ID, or catchment ID. This ID can be typed manually, selected from a list using the '…' button or picked from a map using the arrow button.
- Chainage: this is the chainage along the selected link where the time series is picked from. It can also be picked from a map using the arrow button This field is active only for relevant result types.

These properties can be edited for multiple records at once using the Field calculator which is available by right clicking on a column's header. An expression editor is then available where simple or complex expressions can be written to edit selected rows. If no rows are selected, the expression will be applied to the entire column. See Expression Editor (page 533) for more information.

Click OK to save the changes, and the time series plot will show the updated time series definitions the next time it is opened.

Note: The [FilePath] variable defining the source result file is a path, and not a simple file name. While the tool can in some cases succeed converting a simple file name to a path, it may be necessary to specify the full path to result files in the Field calculator.

20.17.2 Editing profile plots

The 'Edit results selection' option opens the editor below for profile plots.

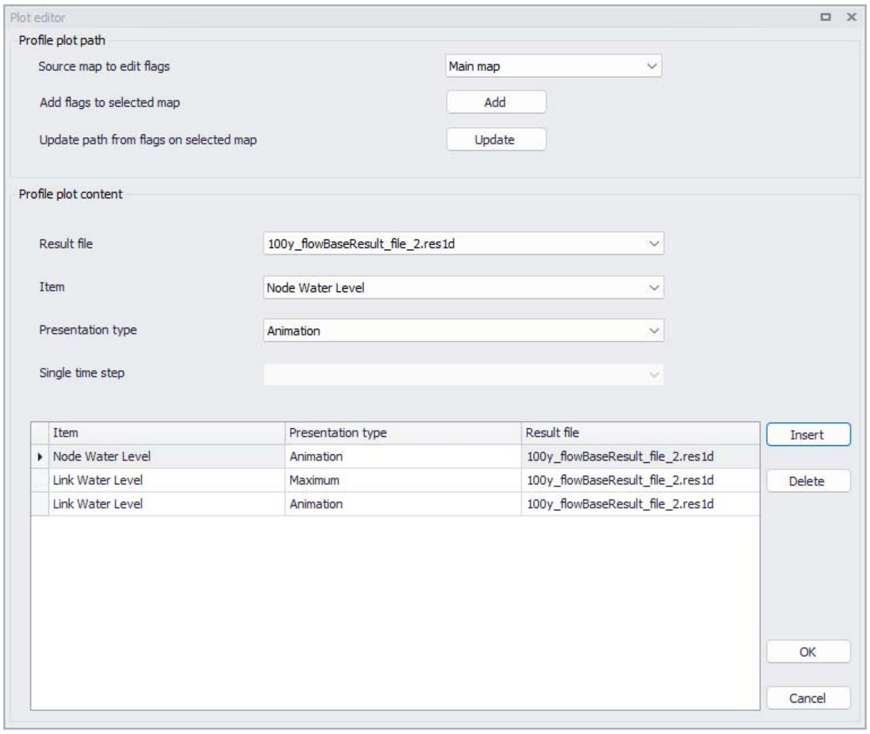


Figure 20.119 The Profile Plot Editor

The upper part of the editor allows editing the path of the profile plot on a map. The list 'Source map to edit flags' shows the map where the profile plot was defined and where flags can be edited. 'Main map' indicates that the plot was created on the model map, and this cannot be changed. When the profile plot was created from a result map, this list shows the name of this result map, and it is possible to select another result map to edit the location. It is not possible to change this selection from the main map to a result map or vice-versa.

To edit the path, click 'Add' to show the original flags on the selected map, where it is then possible to add, delete or re-order the flags. When changes to the flags definitions are done, click 'Update' to save the new flags in the edited profile plot.

The overview table shows the list of result items added to the profile plot, while the above fields show the same properties for the selected record in the table. Result items can be added or removed from the plot using the 'Insert' and 'Delete' buttons on the right.

Each result item is defined with the following editable properties:

- Result file: this is the file name of the source result file. It is possible to select from the list of files currently loaded in the MIKE+ project from the drop-down list.

- Item: this is the type of result item. It can be selected from the drop-down list, which shows all items currently loaded in the MIKE+ project for the selected file.
- Presentation type: this controls the type of presentation for the result item on the plot, which can be Average / Maximum / Minimum / Animation.
- Single time step: this shows the date and time of the result item. This field is active only for relevant result types.

These properties can be edited for multiple records at once using the Field calculator which is available by right clicking on a column's header. An expression editor is then available where simple or complex expressions can be written to edit selected rows. If no rows are selected, the expression will be applied to the entire column. See Expression Editor (page 533) for more information.

Click OK to save the changes, and the profile plot will show the updated results definitions the next time it is opened.

Note: The [FilePath] variable defining the source result file is a path, and not a simple file name. While the tool can in some cases succeed converting a simple file name to a path, it may be necessary to specify the full path to result files in the Field calculator.

20.17.3 Editing scatter plots

The 'Edit results selection' option opens the editor below for scatter plots.

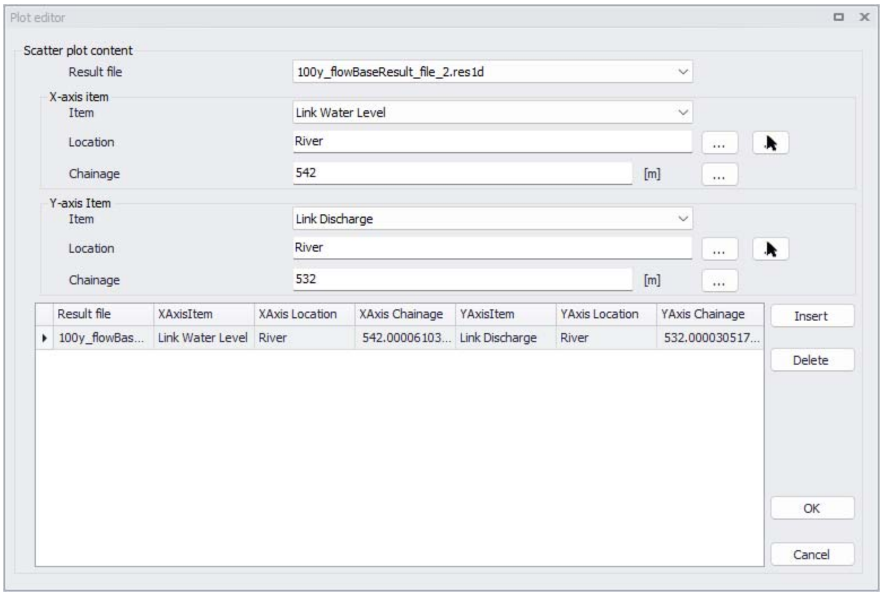


Figure 20.120 The Scatter Plot Editor

The overview table shows the list of scatter point series included in the scatter plot, while the above fields show the same properties for the selected record in the table. Series can be added or removed from the plot using the 'Insert' and 'Delete' buttons on the right.

Each scatter point series is defined with the following editable properties:

- Result file: this is the file name of the source result file. It is possible to select from the list of files currently loaded in the MIKE+ project from the drop-down list.
- Items: this is the type of result item, to be selected separately for the X and Y axes. It can be selected from the drop-down list, which shows all items currently loaded in the MIKE+ project for the selected file.
- Locations: this is the item ID where the axis data are picked from, e.g. node ID, link ID, or catchment ID, to be selected separately for the X and Y axes. This ID can be typed manually, selected from a list using the '…' button or picked from a map using the arrow button.
- Chainages: this is the chainage along the selected link where the axis data are picked from, to be selected separately for the X and Y axes. It can also be picked from a map using the arrow button. This field is active only for relevant result types.

The result file property can be edited for multiple records at once using the Field calculator which is available by right clicking on the column's header. An expression editor is then available where simple or complex expressions can be written to edit selected rows. If no rows are selected, the expression will be applied to the entire column. See Expression Editor (page 533) for more information. The other properties can only be edited from the upper form, but not from the overview table.

Click OK to save the changes, and the scatter plot will show the updated time series definitions the next time it is opened.

Note: The [FilePath] variable defining the source result file is a path, and not a simple file name. While the tool can in some cases succeed converting a simple file name to a path, it may be necessary to specify the full path to result files in the Field calculator.

21 Create Flood Maps Tool

21.1 Introduction

The purpose of the 'Create flood map' tool is to convert 2D result files into a polygon layer, where each polygon represents a group of neighboring cells with the same category of results (e.g. same class of water depth). All resulting polygons are saved in a unique shape file.

2D result files in both flexible mesh format (.dfsu file) and rectangular grid format (.dfs2 file) are supported, no matter if the latter format is obtained from a 2D overland simulation or 1D river simulation.

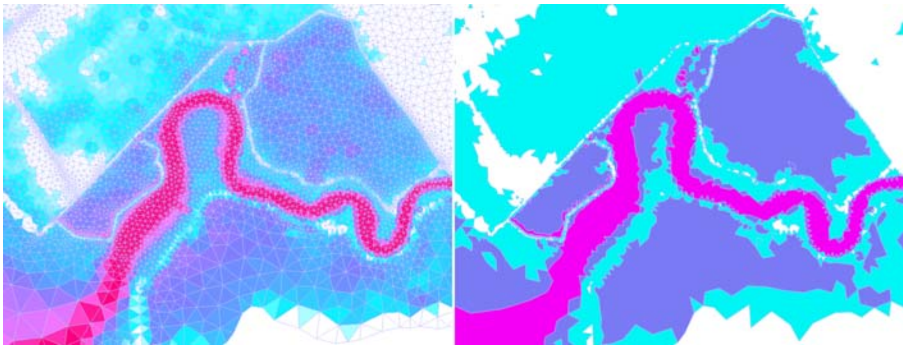


Figure 21.1 Comparison of original flood depth results (left) and created polygons with three depth categories (right)

The tool also offers some filters to remove or merge polygons which may be considered too small compared to the level of analysis expected from the output map.

21.2 Main settings

The tool processes a selected result item at a selected time step, to create the polygon layer based on user-defined categories.

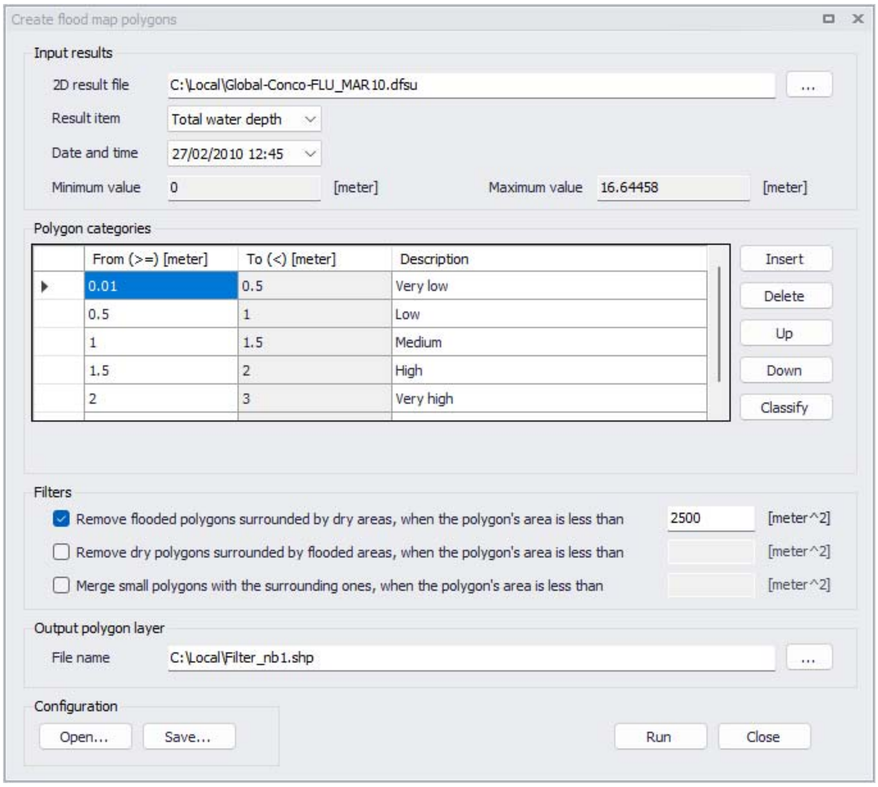


Figure 21.2 The Create flood map tool

21.2.1 Input results

Selection of results data is made in the 'Input results' group:

- 2D result file: the path to the input result file.
- Result item: the result item to be processed, from the selected result file.
- Date and time: time step at which the results should be processed, selected from the list of time steps available in the result file.

Minimum and maximum values in the result file for the selected result item and time step are shown to help defining the polygon categories.

Note: In case the polygons have to be created from statistical results (typically the maximum values), it is necessary to select a result file containing these statistical results with a single time step. Such result files can be saved during the simulation, or by post-processing an animated result file with instantaneous results.

21.2.2 Polygon categories

The categories are specified in the table, using the 'From' column. Values in the 'To' column are automatically filled in.

The first 'From' value can be used to exclude very small results from the polygons. For instance, for water depth results, if the first category starts at 1 cm, then all areas with depths lower than this threshold will not be mapped in the output shape file (no polygon will contain these areas).

There is however no upper limit for the last category, and its 'To' value therefore always shows the infinity symbol.

It is recommended to provide meaningful descriptions or names of the categories in the 'Description' column. All three columns will be saved as attributes of the polygons in the output shape file.

The buttons on the right side of the table can be used to add, remove or re-order the categories in the table.

The 'Classify' button opens a dialog to define the number of categories to use and the 'From' values for each class. In this dialog, the 'Distribute using min/max from data set' button distributes equidistantly the specified number of categories using the minimum and maximum values of the selected result item and time step. The 'Distribute using first/last from table' button distributes equidistantly the specified number of categories using the first and last custom values specified in the table in this classification window.

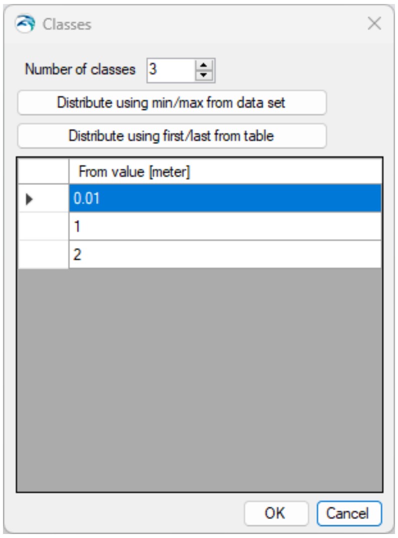


Figure 21.3 Customizing value classes for the categories

21.2.3 Output polygon layer

This group holds the path and name of the output shape file.

21.2.4 Configuration

Once the tool has been configured, it is possible to save its configuration to a file using the 'Save…' button for later re-use. This configuration can later be loaded again using the 'Open…' button, or can be used to execute the tool from a command line.

21.3 Filters

Filters can optionally be used to remove or merge polygons which may be considered too small for the expected analysis of the output map. Their goal is to get rid of very small local variations of categories, to only focus on larger-scale variations.

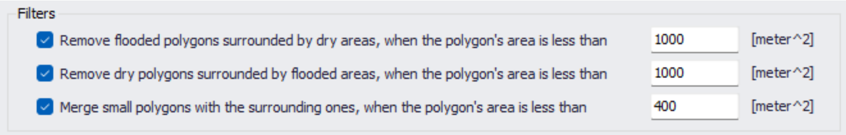


Figure 21.4 List of available filters

For each of the filters, polygons are filtered only when their area is smaller than the specified threshold.

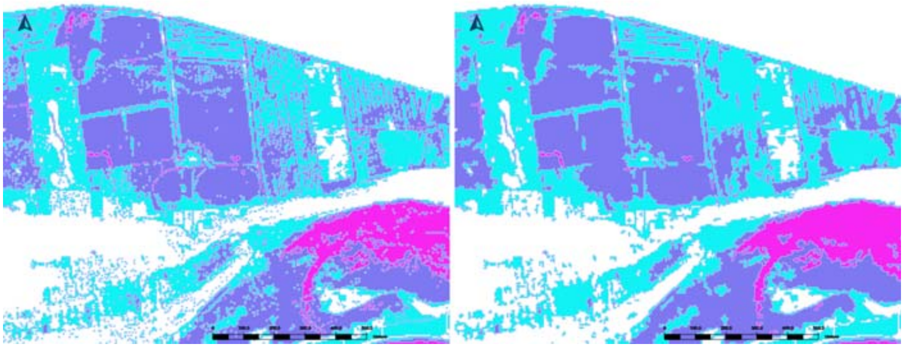


Figure 21.5 Comparison of flood hazard categories without filters (left) and with filters (right)

When several of these filters are active, the data processing associated to them is performed in the order of the filters shown in the tool (from top to bottom).

21.3.1 Remove flooded polygons surrounded by dry areas

This first filter helps removing small isolated flooded areas which are disconnected from a main flooded area.

Such small areas may arise e.g. due to rainfall accumulating in small ponds, in which case they are not of interest for a flood map, or due to artefacts of interpolation of river water levels on a DEM in which case they may not be realistic.

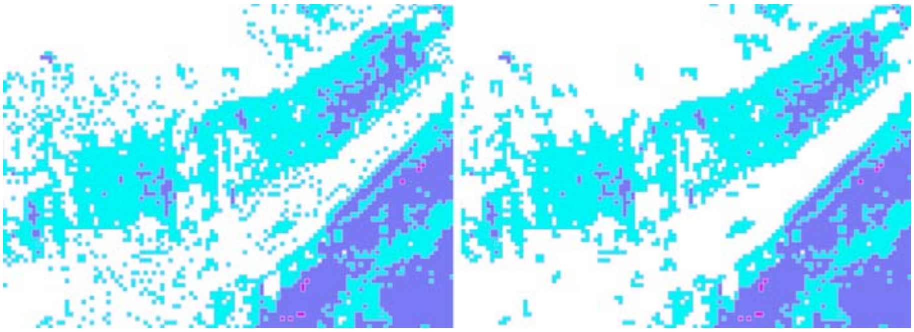


Figure 21.6 Comparison of flood hazard categories without filters (left) and with first filter (right)

The tool will first create category polygons by gathering neighboring cells / elements with result values in common categories. Once the polygons are created, and if this filter is active, the tool will search for isolated polygons and remove those which have their area smaller than the specified threshold.

A polygon is declared as isolated when it doesn't share any segment of its borders with another polygon. Note that two polygons will not be considered as sharing a border if they only share a single point location (this may especially occur with rectangular grid results, when flooded cells are connected only through their corner). A consequence of this approach is that a flooded area can only be removed if it is included in a single category.

Note: This filter is solely based on a spatial analysis and can remove polygons from different categories, although such isolated areas are usually areas with low depths / risks. An alternative way to skip areas with low depths / risks is to exclude low value results from the analysis by increasing the 'From' value of the first category.

21.3.2 Remove dry polygons surrounded by flooded areas

This second filter helps filling small gaps within flooded areas.

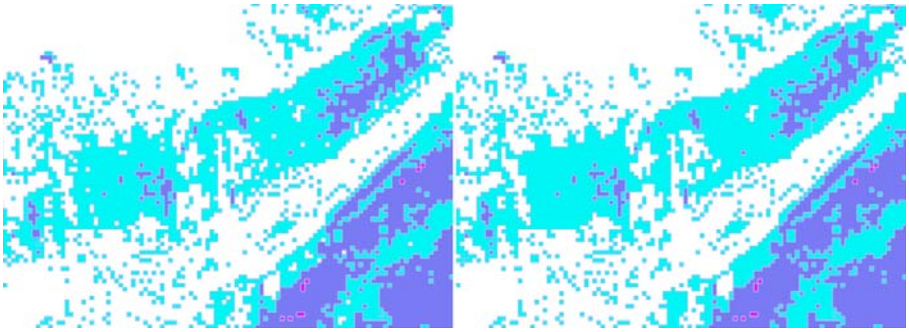


Figure 21.7 Comparison of flood hazard categories without filters (left) and with second filter (right)

If this filter is active, the tool will search for gaps / holes within the polygons (white holes in above figure) and remove those which have their area smaller than the specified threshold.

When the gap is contained within a single polygon / category, the area of the gap becomes included in this category, i.e. this area is given the category of the surrounding polygon. If the gap is at the junction between multiple categories, it is given the lowest category from all neighboring polygons (lowest 'From' value).

Note: Gaps are here defined as dry cells / elements, within a larger flooded area. When working with flexible mesh results, this filter therefore only acts on areas which are part of the mesh. Areas excluded from the mesh (typically buildings) are not filled.

21.3.3 Merge small polygons with the surrounding ones

This third filter merges small polygons having a local category with a neighboring (wider) polygon. This means that these small polygons are removed and replaced by the category of the neighboring polygon. The purpose is to ignore very local variations of the risk which may not be of interest for the flood map.

This filter acts only on polygons which are fully surrounded by other polygons. It does not merge polygons located on the edges of a flooded area.

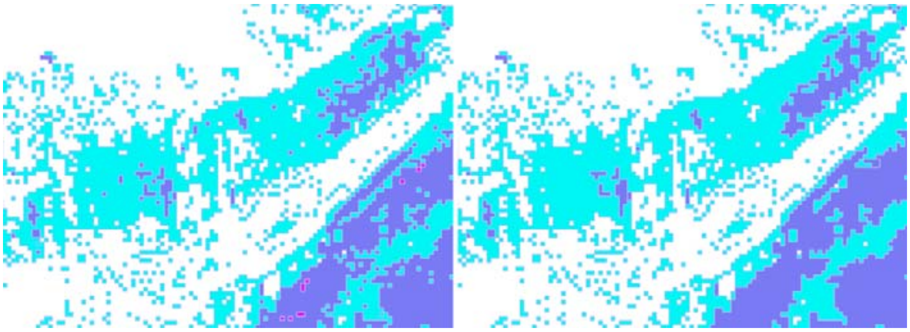


Figure 21.8 Comparison of flood hazard categories without filters (left) and with third filter (right)

The tool will first create category polygons by gathering neighboring cells / elements with result values in common categories. Once the polygons are created, and if this filter is active, the tool will search for small polygons which are completely surrounded by other polygons: those which have their area smaller than the specified threshold will be merged with a neighboring polygon.

When the polygon to be merged is contained within a single polygon / category, it is merged with this surrounding polygon. If the polygon to be merged is at the junction between multiple categories, it is given the lowest category from all neighboring polygons (lowest 'From' value).

21.4 Running the tool from command lines

MIKE+ user interface is usually the preferred way to execute this tool. However, there are times when it may be more convenient to execute the tool in an automated way without opening it in the user interface.

The MIKE+ executables enable you to execute some tools without opening their editor, through command lines. It is possible to run the 'Create flood map' tool in this manner, assuming you have prepared the configuration file beforehand in MIKE+.

Start by locating the MIKE+ executable named DHI.MIKEPlus.ToolShell.exe in the installation folder. From a command prompt, type the command below to access the list of supported tools, replacing the … characters by the actual path to the file:

"C:\...\DHI.MIKEPlus.ToolShell.exe" -h

The format of the command for running the 'Create flood map' tool is:

"C:\...\DHI.MIKEPlus.ToolShell.exe" CreateFilteredPolygonsTool -c [Configuration file]

Where [Configuration file] is the path to the *.xml configuration file.

22 Create Hazard Maps Tool

22.1 Introduction

The purpose of the 'Create hazard map' tool is to derive hazard values from 2D result files, where hazard values are typically derived from water depth and current speed results. Various definitions of the hazard variable may be applied. It can also derive additional result items like Time to start, Time to peak or Duration, for various result items.

2D result files in both flexible mesh format (.dfsu file) and rectangular grid format (.dfs2 file) are supported.

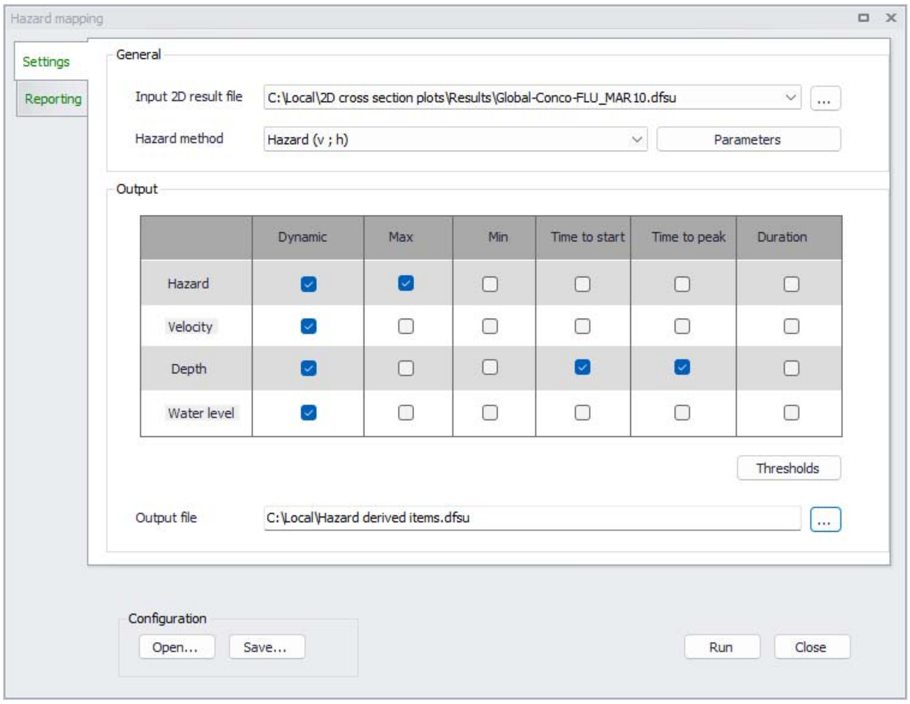


Figure 22.1 The Create hazard map tool

22.2 General settings

The input 2D result file is selected from the list of result files loaded in the MIKE+ project, or by selecting a new file with the '…' button. The selected file must at least contain the following result items:

- ‘Total water depth' or 'Surface elevation'
- and 'Current speed' or the combination of 'U velocity' and 'V velocity'.

The type of hazard calculation method is selected in the 'Hazard method' list. Four methods are available, and each of them uses user-defined settings specified using the 'Parameters' button. The available methods are:

- Hazard (v ; h)
- Hazard (v * h)
- UK Methods
- Italian method

Hazard (v ; h)

For each cell from the .dfs2 result file or element from the .dfsu result file, the tool reads the depth and the speed values and look up the corresponding hazard category (value) from the parameter table. In this table, the left column contains the input current speed values (in m/s or ft/s) and the top row contains the input water depth values (in meters or feet). Output hazard values are specified in the remaining cells of the table.

For this method, hazard is classified into categories, corresponding to given ranges of current speed and water depths. Values of current speed provided in the table (left column) and depth (top row) represent the lower limit of this range. For instance, for the example illustrated below, the hazard will be assigned the value 1 whenever the current speed will be between 0.2 and 0.5 m/s and the water depth between 0.5 and 1.0 m.

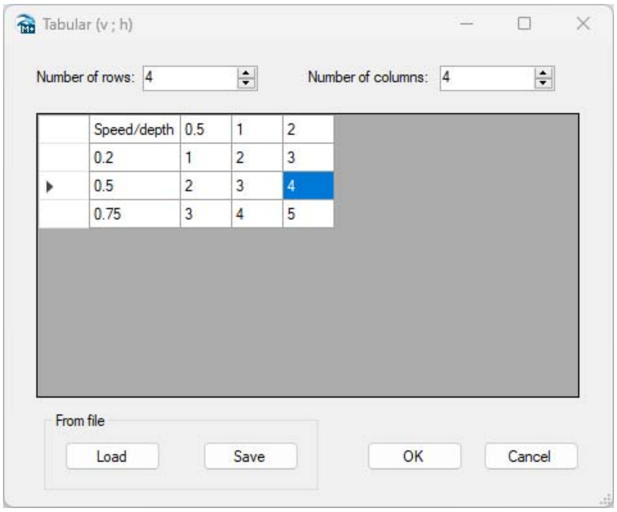


Figure 22.2 Example of parameter table for Hazard (v ; h) method

The 'Load' and 'Save' buttons may be used to save the content of the table in a text file, to be re-imported at a later stage.

Hazard (v * h)

For each cell from the .dfs2 result file or element from the .dfsu result file, the tool multiplies the depth and the speed values and look up the corresponding hazard category from the parameter table. In this table each column is a hazard category corresponding to a range of speed*depth values. Speed*depth values provided in the top row of the table represent the lower limit of this range.

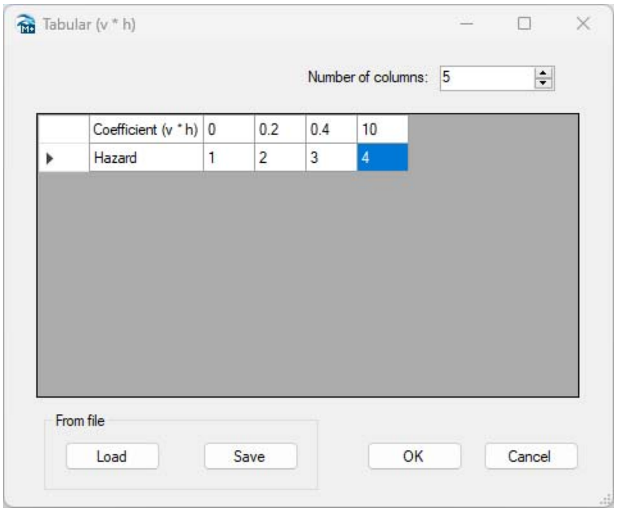


Figure 22.3 Example of parameter table for Hazard (v * h)

The 'Load' and 'Save' buttons may be used to save the content of the table in a text file, to be re-imported at a later stage.

UK Methods

Two UK methods are available, which both use a similar definition, simply differing in how a debris factor is estimated.

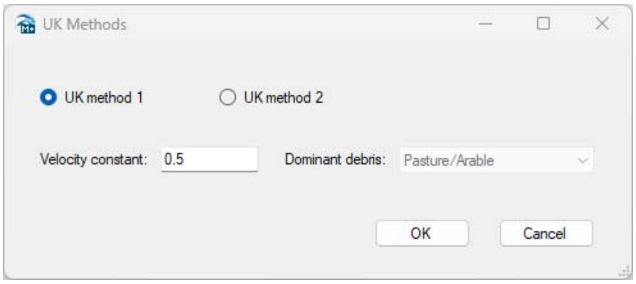


Figure 22.4 Example of input for UK Methods

With these methods, hazard is defined using the following formula:

$$HR = d \cdot (v + n) + DF \tag{22.1}$$

where:
HR : hazard rating
d : depth
v : current speed
n : a constant (typically = 0.5)
DF : debris factor

With UK method 1:

if *d* > 0.25 then *DF* = 1.0, otherwise *DF* = 0.5

With UK method 2:

DF = 0, 0.5 or 1, depending on water depth, current speed and dominant land use, with the relationship described in Table 22.1 below..

Table 22.1 Guidance on debris factors for different flood depths, velocities and dominant land uses

Depths (d) and speed (v)	Pasture/Arable	Woodland	Urban
d from 0 to 0.25 m	0	0	0
d from 0.25 to 0.75 m	0	0.5	1
d > 0.75 m and/or v > 2	0.5	1	1

Italian method

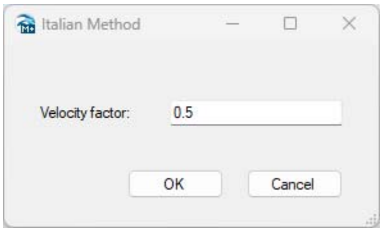


Figure 22.5 Example of input for Italian Method

With this method, hazard is defined using the following formula:

$$HR = d + Fac \cdot \left(\frac{v^2}{2g}\right) \tag{22.2}$$

Where *Fac* is the velocity factor specified in the tool, which is typically 1, 0.5 or 0.25.

22.3 Output settings

Two output files may be generated when pressing the 'Run' button:

- One file containing static output items: Maximum, Minimum, Time to start, Time to peak and Duration. This file will be created if at least one of these items is ticked. The file is named with the file name specified in the 'Output file' field underneath the table.
- One file containing the dynamic output items, i.e. instantaneous values, if a Dynamic item is ticked. This file is named with the file name specified in the 'Output file' field, but with the suffix '_dynamic'.

Each file may contain several variables, which are selected in the table: hazard (as defined in the Hazard method), velocity, water depth and water level.

Thresholds

Output values are controlled by threshold values, specified from the 'Thresholds' button. The threshold values are used to filter noise.

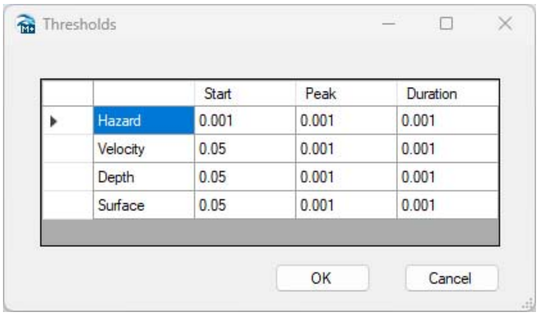


Figure 22.6 Example of threshold values

Thresholds values are only used when processing the variable they correspond to. Therefore, the calculation of hazard variable depends on hazard thresholds but not on velocity nor depth thresholds.

The Start, Peak and Duration thresholds are expressed in the same unit as the corresponding variable. The thresholds' units for hazard are expressed in hazard unit.

Maximum

The maximum value in a given cell is found by comparing the instantaneous values with the Peak threshold into account. Starting from the first value in this cell, the maximum value is updated with a new higher value from a following time step only if this new value is higher than the previous maximum value + the Peak threshold. If the values from the following time steps are higher but don't differ by more than the Peak threshold value, then the maximum value is not updated with these following values.

On the illustration below (Figure 22.7), if the difference between the value at location #1 and #3 is less than the Peak threshold, then the output maximum value is the value from location #1, which therefore slightly differs from the absolute maximum value. This may be useful in flooding applications, to catch the first value representative of the maximum, and therefore to also catch the corresponding time (see Time to peak below).

If the Peak threshold is reduced slightly, then the output maximum value will be value from location #2 and if the Peak threshold is reduced further, then the output maximum value will be the absolute maximum value from location #3.

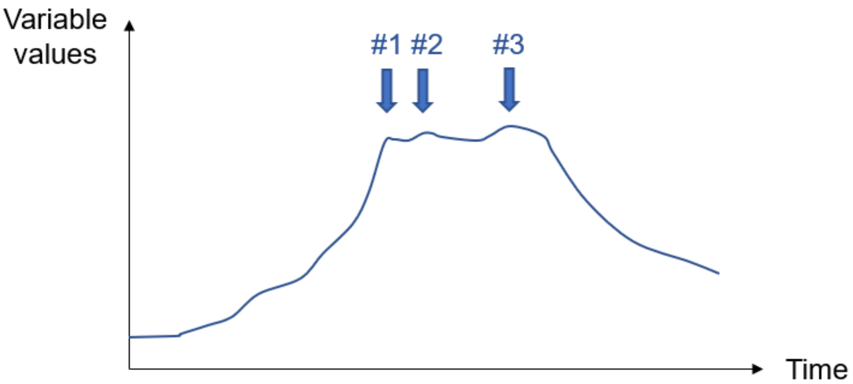


Figure 22.7 Example of captured maximum values depending on Peak threshold

Since the output maximum value may differ from the absolute maximum value, it is recommended to keep the Peak threshold relatively small. To ensure that the output maximum value is the absolute maximum value, the Peak threshold should be set to 0.

Minimum

The minimum value in a given cell is found with the same approach as for the maximum, but using instead the Start threshold. That is, the minimum value from previous time steps is updated with a new smaller value from a following time step only if this new value is smaller than the previous minimum value minus the Start threshold.

Since the output minimum value may differ from the absolute minimum value, it is recommended to keep the Start threshold relatively small. To ensure that the output minimum value is the absolute minimum value, the Start threshold should be set to 0.

Time to start

If the 2D cell is empty at the first time step, then the Time to start is the accumulated time spent before the cell is no longer empty. If the cell is not empty at the first time step, then the Time to start is the accumulated time until the variable's value differs from the value in the first time step by more than the Start threshold.

In the case where the cell does not change during the whole simulation (it remains empty, or it's not empty but its value does not change by more than the Start threshold), then the Time to start result is empty. In the special case where a non-empty cell keeps a constant value during the simulation period and where the Start threshold is set to 0, then the resulting Time to start is also 0.

The reported Time to start is expressed in hours.

Time to peak

The Time to peak is the accumulated time until the maximum value (as defined above, considering the Peak threshold) is reached.

The reported Time to peak is expressed in hours.

Duration

The Duration is the accumulated time while the variable's value is higher than the Duration threshold value.

The Duration thresholds are expressed in the same unit as the corresponding variable (for example, the Duration threshold for Depth is per default expressed in meters, etc.). The reported duration is expressed in hours.

22.4 Reporting

Once a hazard calculation is run, status and errors are reported in this tab of the tool. This report can be saved for further inspection at a later stage.

22.5 Configuration

Once the tool has been configured, it is possible to save its configuration to a file using the 'Save…' button for later re-use. This configuration can later be loaded again using the 'Open…' button, or can be used to execute the tool from a command line.

22.6 Running the tool from command lines

MIKE+ user interface is usually the preferred way to execute this tool. However, there are times when it may be more convenient to execute the tool in an automated way without opening it in the user interface.

The MIKE+ executables enable you to execute some tools without opening their editor, through command lines. It is possible to run the 'Create hazard map' tool in this manner, assuming you have prepared the configuration file beforehand in MIKE+.

Start by locating the MIKE+ executable named DHI.MIKEPlus.ToolShell.exe in the installation folder. From a command prompt, type the command below to access the list of supported tools, replacing the … characters by the actual path to the file:

"C:\...\DHI.MIKEPlus.ToolShell.exe" -h

The format of the command for running the 'Create hazard map' tool is:

"C:\...\DHI.MIKEPlus.ToolShell.exe" MappingHazard -c [Configuration file]

Where [Configuration file] is the path to the *.xml configuration file.

23 Discharge Through Section Tool

23.1 Introduction

The purpose of the 'Discharge through section' tool is to calculate the discharge on the surface through cross sections (i.e. polylines on the map), from both 1D and 2D result files.

If both 1D and 2D results are included, the tool provides one .dfs0 file for each section (polyline), each file containing the discharge time series from the 1D result file, the time series from the 2D result file, and the sum of the two. When a section only covers one of the two types of result file, or when only one type of result file is selected in the analysis, then the created .dfs0 file contains only one time series obtained either from the 1D or the 2D result file.

The tool is opened from the 'Results' tab in the ribbon.

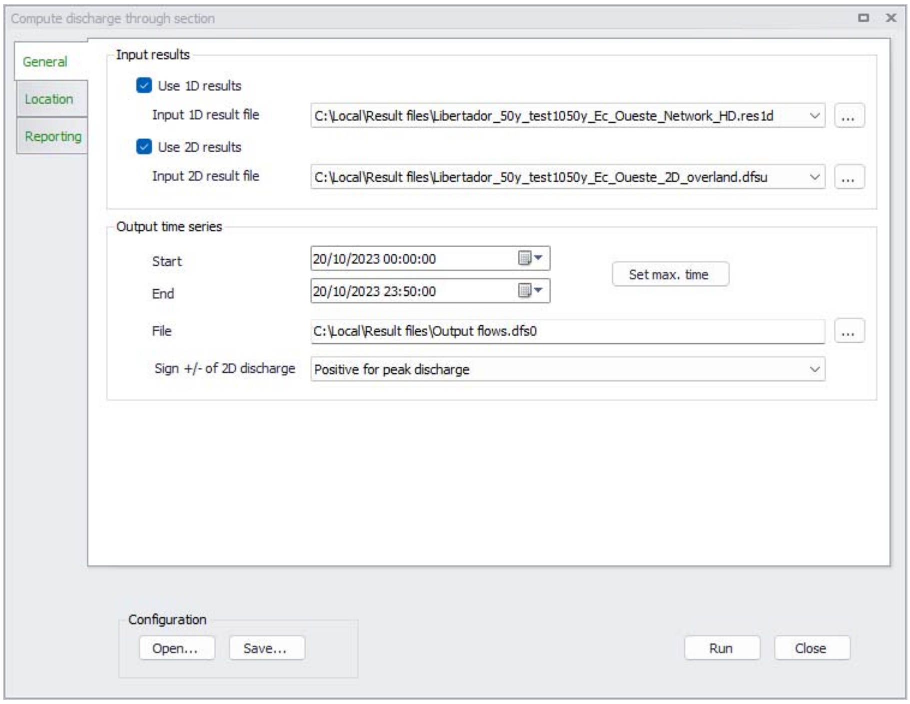


Figure 23.1 The Discharge through section tool

Time series values obtained from the 1D network are the exact discharge values obtained from the closest (to the section drawn on the map) calculation point on the network. Time series values obtained from the 2D results are computed from results in the various elements on the mesh along the polyline.

When processing only a 1D or only a 2D result file, the output time series gets the same time step as the input result file. When processing both 1D and 2D result files, values from the result file with the coarser time step are interpolated, so that the output time series gets the same time step as in the result file with the smaller time step.

The tool is primarily designed to estimate discharge on the surface and therefore to work with open channels for 1D results. It will anyway also work with closed pipes and get discharge results from these pipes, if the polylines cross any of them.

23.2 General settings

The 'General' tab holds selections of input result files to be processed, as well as output settings.

Use 1D results

Activate this option if the computed discharge time series should include discharge results simulated in a 1D model network. When active, a 1D result file must be selected either from the list of result files loaded in the MIKE+ project, or by selecting another file with the '…' button. Only 1D result files from MIKE 1D simulations (.res1d files) and from network simulations are supported.

If this option is unselected, the tool can be used to get discharge time series from 2D results only.

Use 2D results

Activate this option if the computed discharge time series should include discharge results simulated in a 2D overland model. When active, a 2D result file must be selected either from the list of result files loaded in the MIKE+ project, or by selecting another file with the '…' button.

The selected result file must contain at least one of the following combinations of result items:

- P flux, Q flux
- Total water depth, U velocity, V velocity
- Surface elevation, Still water depth, U velocity, V velocity
- Surface elevation, U velocity, V velocity.

Output start and end time

'Start' and 'End' date-times define the time period which will be saved in the output time series files. This period must match or be within the common period covered by the input result files.

The 'Set max time' button can be used to define the period so that it matches the common period of the input result files.

Output file

The specified output file name and location are used as base name for naming output time series. The actual output file names are based on this specified name, with the individual section names as suffix.

Sign +/- of 2D discharge

This option is only relevant for time series obtained only from 2D results. It has two options:

- Depends on direction of the section's digitization: with this option, the sign of the discharge values depends on the direction in which the section's polyline has been digitized. When looking in the direction of the digitization of the polyline, the discharge is considered positive when water flows from the left to the right, and negative when it flows from the right to the left.
- Positive for peak discharge: with this option, the output time series is corrected so that the maximum absolute value is always positive. When the maximum absolute value is found to be negative, the sign of all time step values is inverted. That means that the resulting corrected time series may still contain negative values at time steps where the water is flowing in the opposite direction than the direction found for the maximum value.

When the time series include 1D results, the direction is always based on the river direction, i.e. the discharge is assumed positive when water flows from upstream to downstream.

Notes on the discharge calculation

The following has to be noted:

- Flux on open boundaries will not be included.
- It is not possible to get correct calculations for partially wet elements since the result file has insufficient information for this. Flux on faces with neighbor cell that has delete values will be set to zero.
- The flux on a face in the input result file is an approximation to the 'real' flux used in the engine that produced the result file.

23.3 Location

This tab holds sections (i.e. polylines) through which the discharge is to be computed.

Sections are defined in the left table, with a name which is added as a suffix to the output file name. New sections can be added using the 'Add on map' button, to digitize them on a map, and can be removed using the 'Delete' button. Sections are processed only if their 'Include' check box in this table is selected, whereas unselected sections are ignored.

The table on the right gives coordinates of all vertices from the section selected in the left table. A polyline can therefore be edited from the table, by editing X and Y coordinate values, adding more vertices using the 'Insert' button or deleting vertices using the 'Delete' button. The 'Edit on map' button alternatively switches the polyline in edit mode on the map, after what it is possible to move vertices on the map, add new vertices by clicking on the line, or delete vertices by double-clicking. Right-click on the map to end editing the polyline.

When processing 2D results, the polyline must be within the extent of the 2D results. When processing 1D results, it cannot cross the same 1D channel more than once. Crossing different 1D channels is supported but in that case the line must cross all channels in the same direction (e.g. from the left bank to the right bank).

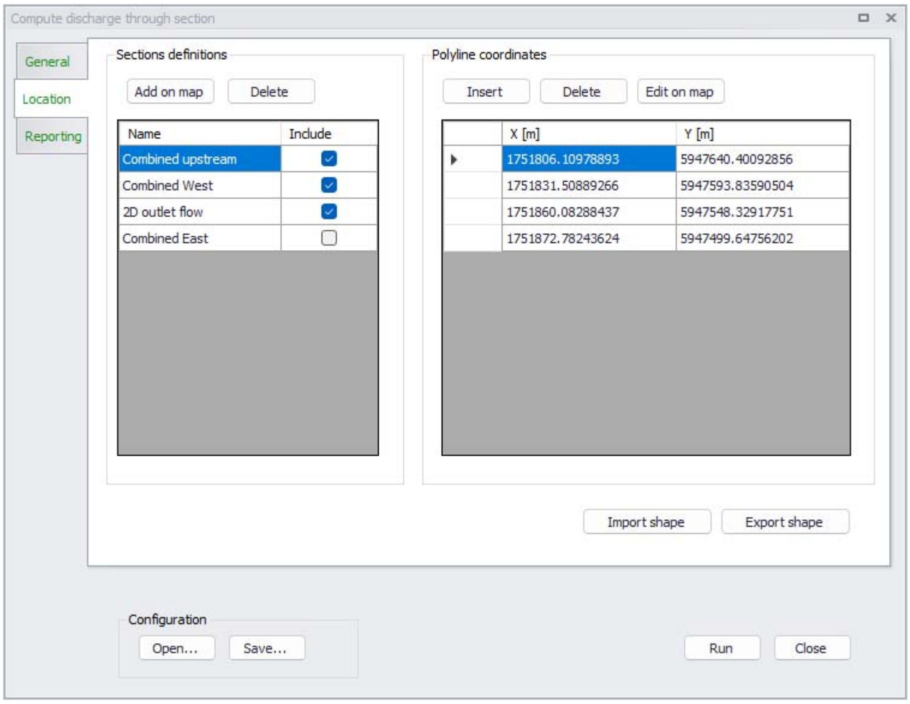


Figure 23.2 The definition of sections in the Discharge through section tool

Sections may be imported from a shape file using the 'Import shape' button. This action will add sections from the shape file to those already existing in the table. Sections defined in the tool can also be exported to a shape file using the 'Export shape' button.

23.4 Reporting

Once a discharge calculation is run, status and errors are reported in this tab of the tool. This report can be saved for further inspection at a later stage.

23.5 Configuration

Once the tool has been configured, it is possible to save its configuration to a file using the 'Save…' button for later re-use. This configuration can later be loaded again using the 'Open…' button, or can be used to execute the tool from a command line.

23.6 Running the tool from command lines

MIKE+ user interface is usually the preferred way to execute this tool. However, there are times when it may be more convenient to execute the tool in an automated way without opening it in the user interface.

The MIKE+ executables enable you to execute some tools without opening their editor, through command lines. It is possible to run the 'Discharge through section' tool in this manner, assuming you have prepared the configuration file beforehand in MIKE+.

Start by locating the MIKE+ executable named DHI.MIKEPlus.ToolShell.exe in the installation folder. From a command prompt, type the command below to access the list of supported tools, replacing the … characters by the actual path to the file::

"C:\...\DHI.MIKEPlus.ToolShell.exe" -h

The format of the command for running the 'Discharge through section' tool is:

"C:\...\DHI.MIKEPlus.ToolShell.exe" SectionDischarge -c [Configuration file]

Where [Configuration file] is the path to the *.xml configuration file.

24 Calibration Plots

Model calibration is important to ensure that model predictions represent the actual hydraulic and water quality conditions in the system. An attempt to calibrate a model should always be made when model results are used in decisions concerning possible remedial actions, augmentation works, forecasting etc.

Calibration is primarily focused on reproducing the observed hydraulics and water quality behaviour of the system in terms of flow depth/pressure, flow discharges, and velocities. The model calibration may include comparisons between model simulation results and field measurements for, but not limited to, the following data:

- Flow
- Water level / Pressure / Hydraulic head
- Velocity
- Water mass balance
- Contaminant concentrations
- Contaminant migration rates
- Degradation rates

These comparisons may be presented as maps, tables, or graphs. The calibration results need to be evaluated by the modeller using engineering professional judgement. There are no universally accepted "goodness-of-fit" criteria that can be applied in all cases. However it is important that the user makes every attempt to minimize the difference between model simulations and measured field data.

The model calibration tool in MIKE+ provides comparisons of measured and computed time series, scatter plot comparisons as well as a number of numerical calibration criteria, all provided at point locations were measurements are available.

24.1 Measurement Stations

Measurement stations representing locations of flow gauges, pressure meters etc. can be defined in MIKE+ for any model type.

The stations can be viewed on the main Map, providing the user with an overview of monitoring locations. Additional information such as image and description of the monitoring site can be defined for individual stations.

It is possible to insert a measurement station graphically on the map, as well as directly in the editor (Figure 24.1).

On the Map

The station can be inserted graphically using Create tool from the Edit Features toolbox, and first ensuring that the Target Layer for editing is the ‘Measurement stations’ layer. As with other point layers, the 'Create', 'Edit', and ‘Delete' tools are available for the Measurement stations layer.

Corresponding records for measurement stations graphically added on the map are added to the Measurement Stations editor (Figure 24.1).

In the Editor

Use the ‘Insert’ button on the Measurement Stations editor to define new measurement stations in the project.

Note that adding a feature from the editor locates by default the feature in the upper right corner of the current Map view.

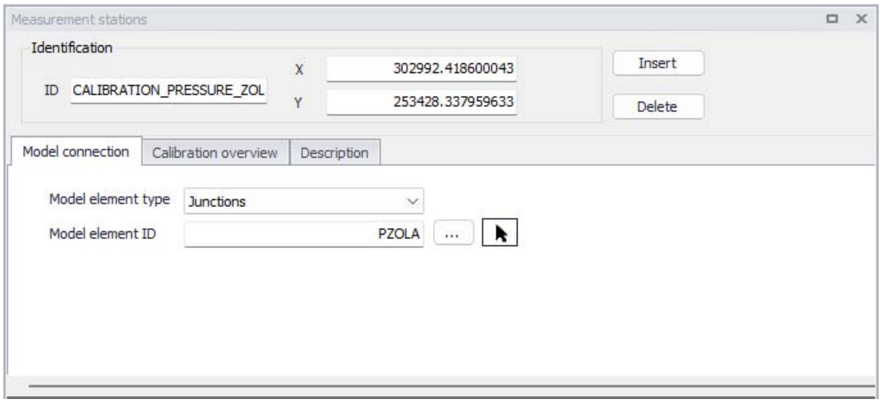


Figure 24.1 Measurement Stations editor

To relocate a feature, the X and Y coordinates may be modified in the editor, or the Edit tool from the Edit Feature toolbox may also be used to graphically move the feature on the Map.

Model Connection

In order to link the measurement station with a modelling location, the stations need to be connected to the model network.

Define the connection of the measurement stations in the Model Connection tab of the Measurement Stations Editor.

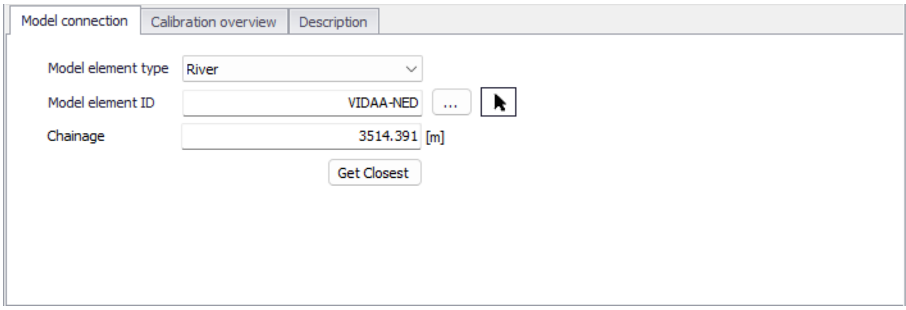


Figure 24.2 Model Connection tab on the Measurement Stations editor

The '…' button after 'Model Element ID' edit box opens a selection list depending on the selected 'Model Element Type'. The arrow button allows one to pick a Node/Junction/Link/Pipe/River on the map depending on the selected Model Element Type. For stations connected to CS links, specify the chainage or computational grid point of the connection as Upstream, Middle or Downstream of the link. For stations connected to rivers, specify the chainage value of the connection.

Note that connection lines between stations and the network are displayed on the Map after model connections are specified.

Also, on the Map View, the 'Connect station' tool from the 'Layer editing tools' toolbar may be used to connect stations to network elements. Activate the tool, click on a station feature to connect on the Map, and then select the network element to which to connect.

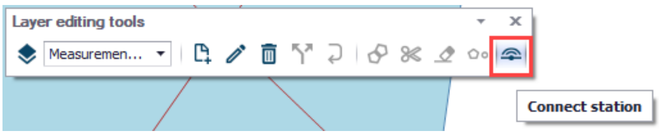


Figure 24.3 Layer editing tools on the Map

Connection Tool

For automatic (in-bulk) connection of measurement stations to the model network, use the Connection Tool accessed via the CS/WD Toolbox on the CS/WD Network menu ribbon.

This will launch the Connection Tool, which is also used for connecting catchments, load allocations and demands. The tool will generate station connections to the network.

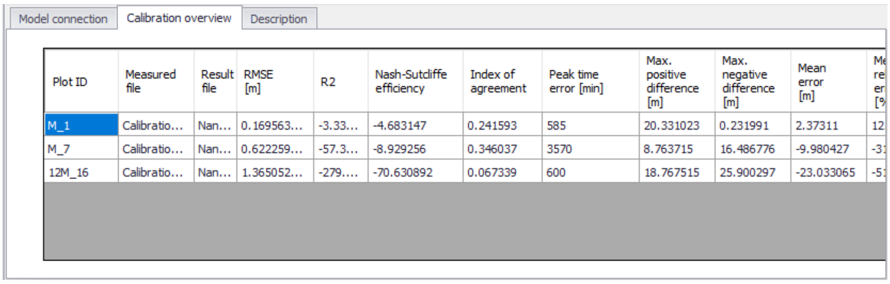
The tool works either on selected or all measurement stations. With the Connection Tool, the m_Station.Chainage will be set to 'Downstream' for CS link connections. The reason for this is that flow gauges are most likely located in the 'Downstream' end of pipe. The grid point location may still be modified in the editor afterwards.

Calibration overview

This tab shows a summary with the main calibration statistics, for all the calibration plots associated to the current station.

This provides an overview of the calibration results for each station, for example when different calibration plots need to be used in relation with different result files. So, this is especially relevant to compare the calibration results from different scenarios, or from different flood events.

Clicking the 'Edit' button in the last column of the table will open the selected calibration plot.



Plot ID	Measured file	Result file	RMSE [m]	R2	Nash-Sutcliffe efficiency	Index of agreement	Peak time error [min]	Max. positive difference [m]	Max. negative difference [m]	Mean error [m]	Me re er [%
M_1	Calibratio…	Nan…	0.169563…	-3.33…	-4.683147	0.241593	585	20.331023	0.231991	2.37311	12
M_7	Calibratio…	Nan…	0.622259…	-57.3…	-8.929256	0.346037	3570	8.763715	16.486776	-9.980427	-3
12M_16	Calibratio…	Nan…	1.365052…	-279.…	-70.630892	0.067339	600	18.767515	25.900297	-23.033065	-5

Figure 24.4 The 'Calibration overview' tab on the Measurement Stations editor

Description

In this tab page, optional information about the measurement station may be added (Figure 24.5). An image associated with the station record may also be uploaded.

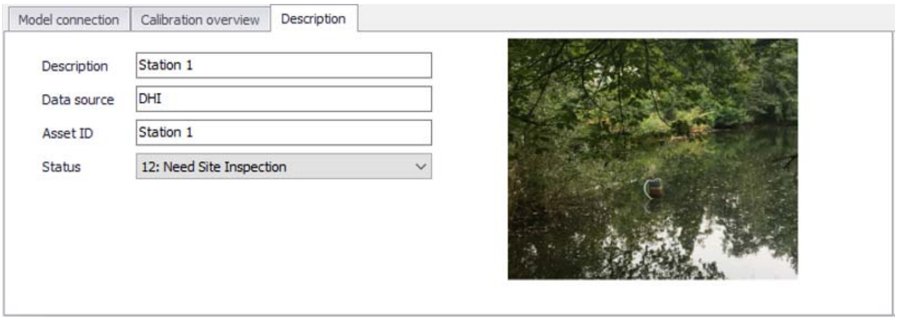
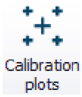


Figure 24.5 The Description tab on the Measurement Stations editor

24.2 Calibration Plots and Reports



The Calibration Plots and Reports dialog (Figure 24.6) is accessible from the 'Plots and statistics' menu in the Setup tree, or from the 'Calibration plots' button in the Results tab of the ribbon.

This functionality allows the user to define relations between externally measured data and simulation results, produce calibration plots, and specify the level of reporting for the calibration.

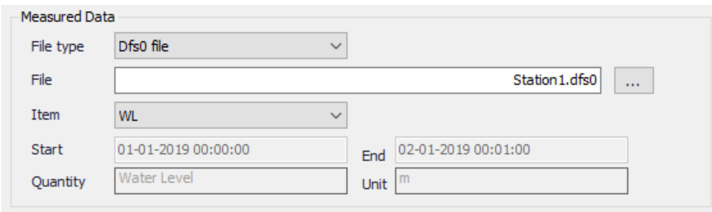
Identification

For each calibration plot, a unique 'ID' must be specified.

The associated measurement station must be specified in the 'Measurement station ID' field. Use the '…' button to pick an ID from a list, or use the arrow button to select the measurement station from the map.

Measured Data

The editor contains a Measured Data group where the external time series file (*.DFS0 or *.DAT) and item can be selected.



Dfs0 files can be created and edited using the '…' button. They can store multiple measured items in different columns and support various time axis formats. They also contain a definition of quantity type and unit, which is then shown in the editor.

Dat files are text files supporting two different formats.

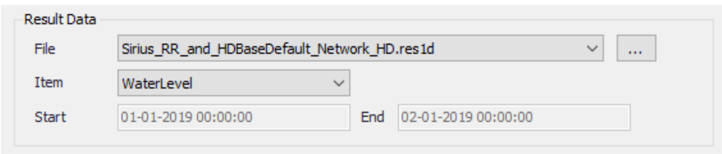
The first format consists in three columns separated by tabs or spaces: Item name, Time and Value. If multiple items are to be stored in the file, they must be provided in the same columns but in consecutive rows. The item name needs to be specified only for its first record and can be left empty for the remaining records. The time should contain a single numerical value with accumulated hours. Comment lines can be inserted and should start with a semicolon. An example is provided below:

;Item1	Time	Observed water level in reservoir
OL2-4	0	9.435275
	0.5	9.413775
	1	9.3955
	1.5	9.3783
	2	9.377225
	2.5	9.293375
	3	9.237475
	3.5	9.1423375
	4	9.0670875
	4.5	8.9595875
	5.5	8.779525
	6	8.7230875
;Item2	Time	Weir discharge
OD5-2	0	9.3353
	0.5	9.3143375
	1	9.2966
	1.5	9.2799375
	2	9.276175
	2.5	9.1907125
	3	9.1332
	3.5	9.0391375
	4	8.9649625

The second format for Dat files consists in two columns separated by tabs or spaces: Date-Time and Value. The Date-Time column should be formatted like this: dd/mm/yyy hh:mm.

Result Data

The Result Data section is where the result file and item to be compared with the measurement time series is specified.



The Result File field takes as default the loaded result file but it is also possible to select the result file manually.

Time series plot

In the ‘Time series’ tab in the right panel, the plot shows the superimposed time series of measured and computed data.

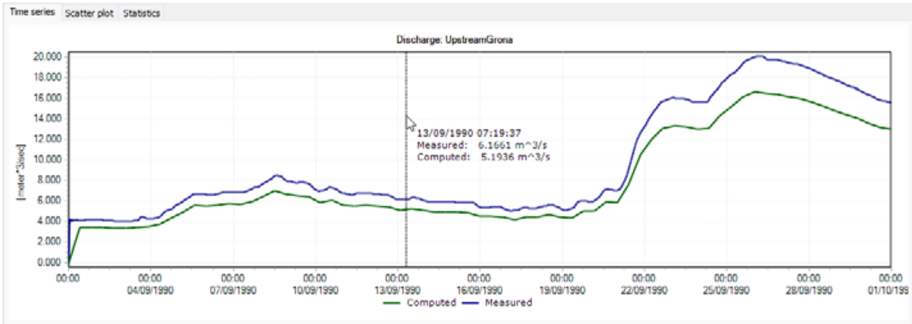


Figure 24.6 Comparison of measured and computed time series

The zoom extents can be saved and restored thanks to the use of bookmarks. The bookmarks manager is accessed from a right-click on the time series plot, and contains the following options:

- Add: adds a new bookmark, saving the current time extent shown on the plot
- Zoom to: zooms to the time period saved for the bookmark selected in the left list
- Rename: renames the bookmark selected in the left list
- Remove: removes the bookmark selected in the left list
- Close: closes the bookmarks manager.

Note that bookmarks only save the time extent (X axis), but not the extent on the vertical axis, and can therefore be applied to any calibration plots even if they don't all have similar ranges of Y values.

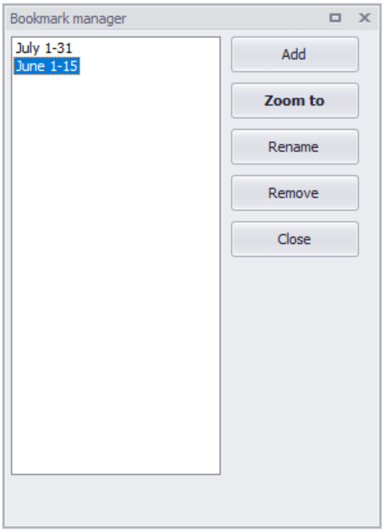


Figure 24.7 The bookmark manager from the calibration plots

Scatter plot

In the 'Scatter plot' tab in the right panel, the plot shows a scatter plot of the measured and computed values. Each point represents a specific date and time. The computed value of the point is shown on the Y axis, and its measured value on the X axis. The closer the points come to the 45-degree angle line (red line), the closer is the match between measured and computed values.

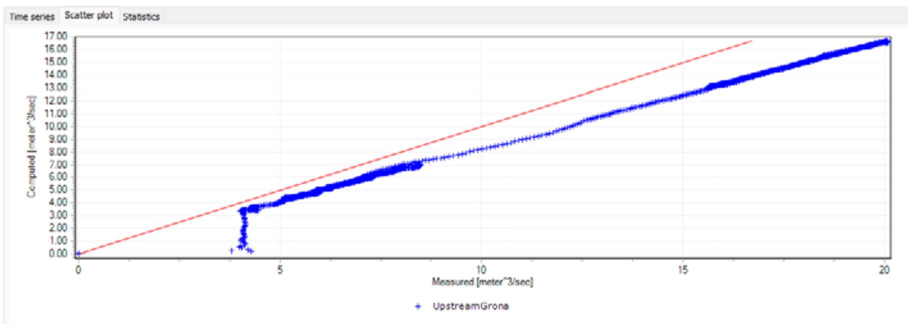


Figure 24.8 Scatter plot comparison

Statistics tab

In the 'Statistics' tab in the right panel, various statistical values are reported.

In the 'Period' group at the top, it is possible to control the time frame for which the statistics are computed. By default, the statistics are computed for the entire period, but it is possible to change to 'Zoom extent' which computes them for the time span visible in the 'Time series' plot, or to 'Custom period' which computes them for a user-defined period independent from the zoom level in the plot. For the 'Zoom extent' and 'Custom period' options, statistics are computed using values from all time steps within the specified time period. No matter which option is selected, statistics can only be computed for the period where both measured and computed data are available. When computing statistics for the active zoom extent, note that a facility in the Time series plot allows to save and re-use zoom extents (see the 'Bookmarks' option in the context menu of the plot).

In the 'Performance criteria' group, the quantities below are provided to evaluate how well the computed time series fits with the measured time series:

- RMSE (Root Mean Square Error): this criterion can be applied as a measure for the magnitude of the deviation between the two time series over the period being investigated.

$$"RMSE" = \sqrt{\frac{\sum_{i=1}^{n}(y_{1,i}-y_{2,i})^2}{n}} \tag{24.1}$$

  The values for the computed time series are linearly interpolated to get values at the date and times matching the measured data.

- Coefficient of determination R2: this is the coefficient of determination, also known as Nash-Sutcliffe efficiency, that measures how well the computed time series matches the measured time series. This criterion is widely used to evaluate model performance in hydrological modelling. It ranges from minus infinity to 1 with larger values indicating a better fit. An important special case is R2 = 0, which can be obtained if the mean measured value equals the mean computed value, indicating that the average of the measured values in this case is as good a predictor as the model. Thus, one would most likely require that R2 > 0 for the model to be fit. The R2 criterion measures the one-to-one relationship between measured and computed values, and hence it is sensitive to bias and proportional effects. It should be emphasized that R2 is based on the sum of squared residuals, and hence provides the same information on goodness-of-fit as the RMSE measure.

$$\mathrm{R2} = 1 - \frac{\sum_{i=1}^{N}(OBS_i - SIM_i)^2}{\sum_{i=1}^{N}(OBS_i - \overline{OBS})^2} \tag{24.2}$$

  Where $\overline{OBS}$ is the mean measured value.

The values for the measured time series are linearly interpolated to get values at the same date and times as in the computed data. The calculation of this criterion is valid only when the result file contains a constant time step.

- Index of agreement (d): this criterion is not as widely used as E and R2. It ranges from 0 to 1 with large values indicating a better fit. The d measure is also based on the sum of squared residuals, but standardised according to a potential error (the term in the summation in the denominator represents the largest error that each (Computed - Measured)^2 can reach throughout the analysed period). As is the case with E and R2, d is also sensitive to outliers.

$$d = 1 - \frac{\sum_{i=1}^{N} (OBS_i - SIM_i)^2}{\sum_{i=1}^{N} \left(\left|SIM_i - \overline{OBS}\right| + \left|OBS_i - \overline{OBS}\right|\right)^2} \tag{24.3}$$

Where $\overline{OBS}$ is the mean measured value.

The values for the measured time series are linearly interpolated to get values at the date and times matching the computed data. The calculation of this criterion is valid only when the result file contains a constant time step.

- Peak time error: this criterion indicates how far in time the two maximum values are located away from each other. This criterion can be used to evaluate if the reported peak values actually compare the same event.
- Maximum positive difference: this criterion computes a value indicating how much the computed time series is above the measured time series at the point in time where this difference has its maximum.

$$\text{"Max Positive Difference"} = \left|max(y_1 - y_2)\right| \tag{24.4}$$

The values for the computed time series are linearly interpolated to get values at the date and times matching the measured time series.

- Maximum negative difference: this criterion computes a value indicating how much the computed time series is below the measured time series at the point in time where this difference has its maximum.

$$\text{"Max Negative Difference"} = \left|min(y_1 - y_2)\right| \tag{24.5}$$

The values for the computed time series are linearly interpolated to get values at the date and times matching the measured time series.

In the 'General statistics' group, mean values and peak values can also be compared. For each of them, the following values are reported:

- Measured: the mean and peak values derived from the measured time series.
- Computed: the mean and peak values derived from the computed time series. For the calculation of the mean value, instantaneous values from the computed time series are linearly interpolated beforehand to get values at the date and times matching the measured time series.
- Error: the difference of mean and peak values reported for the two time series.
- Relative error: the relative difference of mean and peak values, expressed as a percentage of the measured value.

When a time series has a unit which can be accumulated over time (e.g. discharge), the same values are reported for the accumulated values. Typically for a discharge comparison, the table will also report the comparison of accumulated volume.

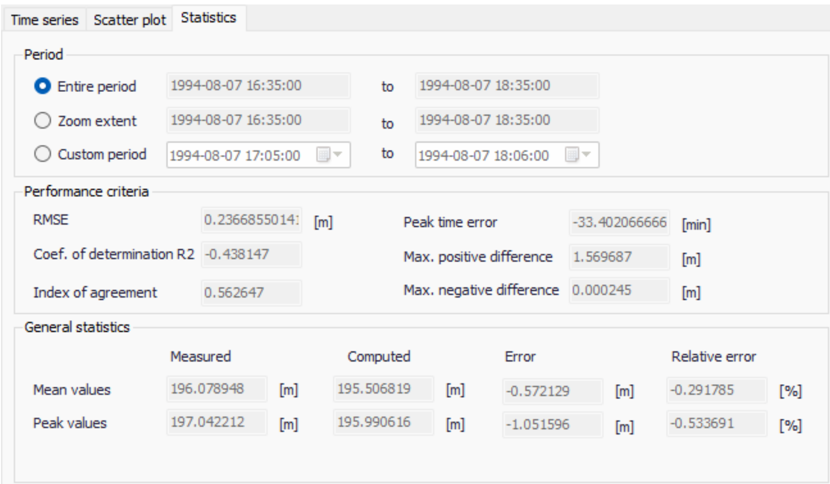


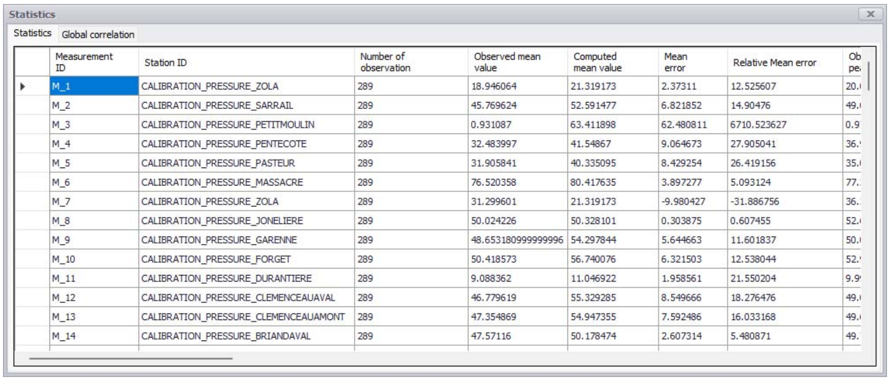
Figure 24.9 The calibration statistics results

Statistics button

Clicking the ‘Statistics’ button on the Plots and Statistics dialog will open a window providing the overview of results for all the calibration plots, in two different tabs.

The first tab 'Statistics' contains a table with all the statistical values, also provided for the individual plots in their 'Statistics' tab. Note that its columns with "accumulated" values contain data only for the rows associated with a time series with a unit defined "per unit of time" (e.g. discharge).

The second tab 'Global correlation' provides a scatter plot superimposing the sets of points from all calibration plots.



Statistics

Statistics | Global correlation

Measurement ID	Station ID	Number of observation	Observed mean value	Computed mean value	Mean error	Relative Mean error	Ob pe
M_1	CALIBRATION_PRESSURE_ZOLA	289	18.946064	21.319173	2.37311	12.525607	20.
M_2	CALIBRATION_PRESSURE_SARRAIL	289	45.769624	52.591477	6.821852	14.90476	49.
M_3	CALIBRATION_PRESSURE_PETITMOULIN	289	0.931087	63.411898	62.480811	6710.523627	0.9
M_4	CALIBRATION_PRESSURE_PENTECOTE	289	32.483997	41.54867	9.064673	27.905041	36.
M_5	CALIBRATION_PRESSURE_PASTEUR	289	31.905841	40.335095	8.429254	26.419156	35.
M_6	CALIBRATION_PRESSURE_MASSACRE	289	76.520358	80.417635	3.897277	5.093124	77.
M_7	CALIBRATION_PRESSURE_ZOLA	289	31.299601	21.319173	-9.980427	-31.886756	36.
M_8	CALIBRATION_PRESSURE_JONELIERE	289	50.024226	50.328101	0.303875	0.607455	52.
M_9	CALIBRATION_PRESSURE_GARENNE	289	48.653180999999996	54.297844	5.644663	11.601837	50.
M_10	CALIBRATION_PRESSURE_FORGET	289	50.418573	56.740076	6.321503	12.538044	52.
M_11	CALIBRATION_PRESSURE_DURANTIERE	289	9.088362	11.046922	1.958561	21.550204	9.9
M_12	CALIBRATION_PRESSURE_CLEMENCEAUAVAL	289	46.779619	55.329285	8.549666	18.276476	49.
M_13	CALIBRATION_PRESSURE_CLEMENCEAUAMONT	289	47.354869	54.947355	7.592486	16.033168	49.
M_14	CALIBRATION_PRESSURE_BRIANDAVAL	289	47.57116	50.178474	2.607314	5.480871	49.

Figure 24.10 The global Statistics table

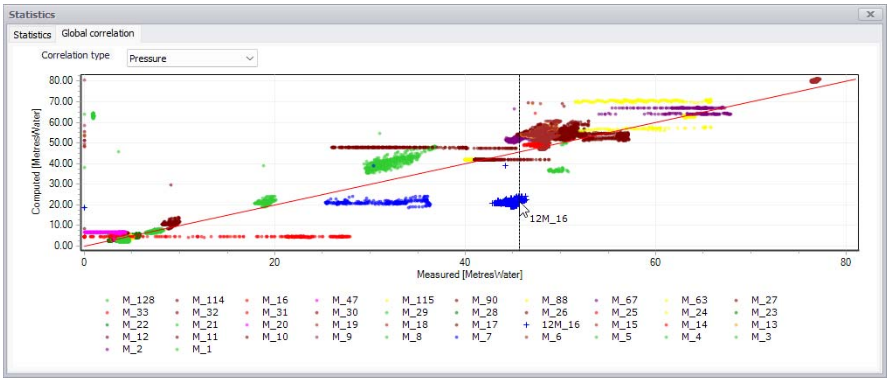


Figure 24.11 The global correlation plot

Report

The 'Report' button will generate an *.XML report for the currently active calibration plot. It uses a pre-set report template, which includes model description and calibration plots in the report.

The report document can then be exported into various types of document file formats for further use in reports and information dissemination.

Also see the Chapter 20.14 Reports *(p. 473)* for more details on generating Reports in MIKE+.

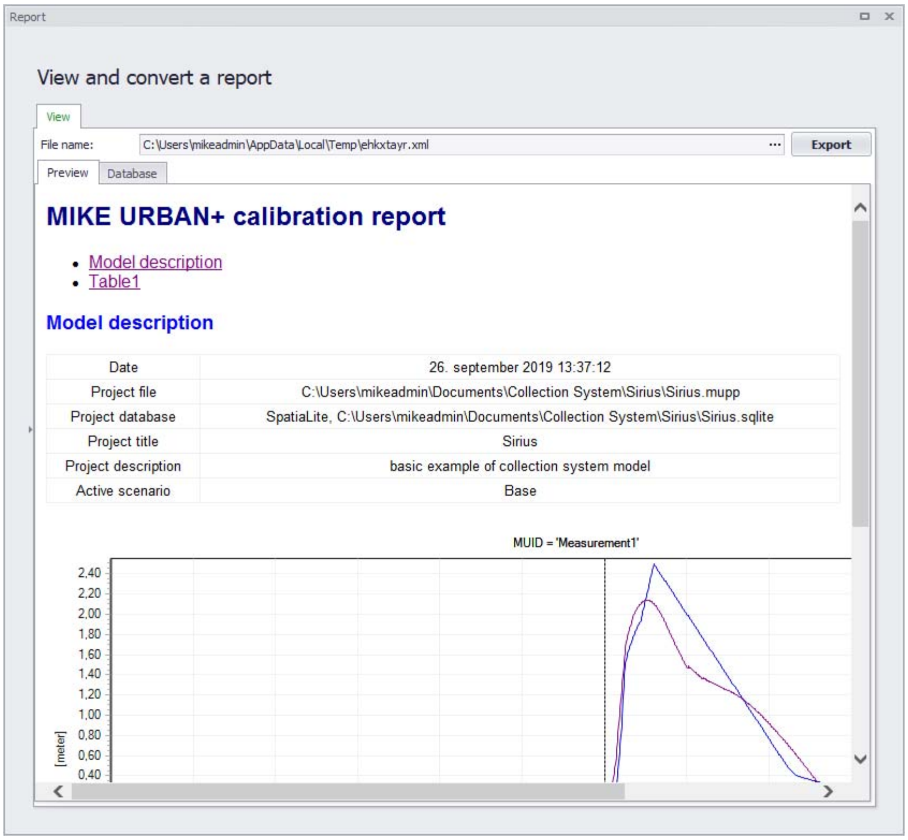


Figure 24.12 Example of a calibration report in MIKE+

25 Expression Editor

The Expression Editor supports creation of simple or complex assignment expressions.

An expression is a "sentence" involving variables, fixed values, functions and operators, designed to return a certain type of value, as e.g. a number or a date-time. It is required to build up the expression to return a value exactly of the type for which it is intended. As an example, when "x" is a variable being a number type, the expression "6+sqrt(x)" will also return a number type, and that expression can be used in a context where a double value is required.

The expression editor supports different types of values:

- Floating numbers, numbers which can include fractional part
- Integer numbers, numbers that does not have any fractional part
- Boolean values, which can either be True or False
- DateTime values, defining a date and a time
- TimeSpan values, defining a time span, as e.g. an hour.
- String values, containing some text.

The various operators and functions work on specific types and return specific types, so the expression must be composed such that types, operators and functions are compatible. As an example, the "-" operator can work on two number values and will then return a number, such as e.g. "6-1.3" returning the number 4.7. However, the "-" operator also works on other types, as e.g. a DateTime value minus a DateTime value will give a TimeSpan value, and a DateTime value plus a TimeSpan value will give a new DateTime value. However, only some combinations make sense, i.e. it is not possible to add a DateTime value and a Boolean value. The expression editor will help validate that the types are composed correctly and the expression returns the correct type.

The Expression Editor is used in several functionalities within MIKE+, such as:

- Field calculator
- For creating Import/Export assignments
- For creating control rules (for rivers and collection systems models)
- Creating data filters for Report configuration

For the Field calculator and Import/Export assignments, the expression return type matches the type of the field to update/import/export. For control rules, the return type for a condition is a Boolean, and for actions the return type is a number. For filters, the return type is a Boolean.

The Expression editor reduces the actual typing (hence the source of errors) to absolute minimum. Also, automatic expression validation is provided.

The Edit Expression Dialog

Expressions are created via the Edit Expression dialog (Figure 25.1). The dialog has three sections: History, Expression, and Error List sections.

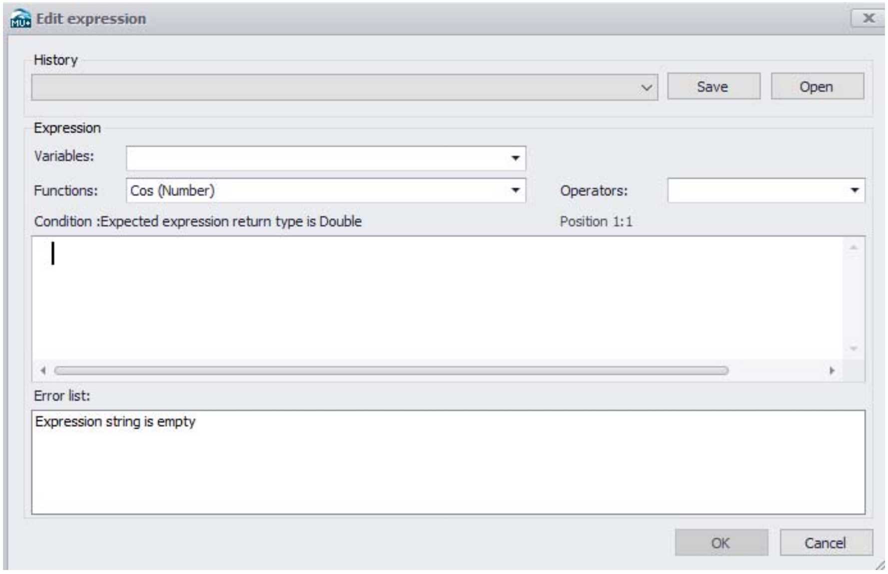


Figure 25.1 The Expression Editor in MIKE+

History

Provides a list of recently used expressions available for reuse in the current assignment.

Every new expression is automatically added to the history list. This allows for a very efficient reuse of similar assignments.

"History" can be saved into a simple text file (*.TXT) and reloaded (Open) in a future (relevant) expression editing session.

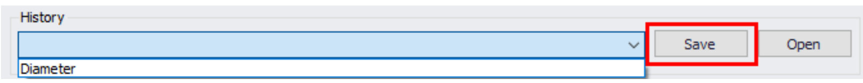


Figure 25.2 History section in Edit Expression dialog highlighting the Save button for saving previously-used expressions

Expression

This is the central part of the dialog. It is where expressions are built and value assignments defined using combinations of Variables, Domains, Functions, and Operators. It also lists the required return type.

More details on building expressions are provided in Chapter 25.1 Expressions *(p. 535)* below.

Error list

The "Error list" reports "on-the-fly" any syntactic errors in the expression and provides advice on how to complete the expression.

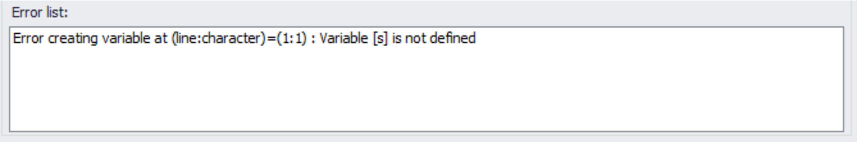


Figure 25.3 The Error list section reports on the real-time validation of expressions

25.1 Expressions

Expressions are built under the Expression section of the Edit Expression dialog (Figure 25.4) using combinations of Variables, Domains, Functions, and Operators.

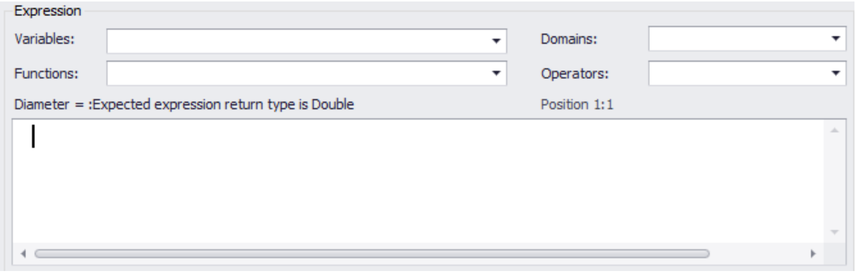


Figure 25.4 The Expression section

The left-hand side of the "equals to" sign of the expression is usually automatically provided. I.e. the user is expected to create only the right-hand side of the expression. This can be done either by direct typing, or by picking up the wanted variables, functions and operators from the respective drop-down lists. Typically, the process will involve both methods.

All variables in the expression should be embraced by square brackets ([]). This is good practice but not mandatory.

Strings should be embraced by double quotes ("").

“Domains” (for Field Calculator) lists the parameters in the data table being edited that use domain coded values.

"Variables" is a list including all attributes in the source table and any user-specified variable. A variable is included in the current expression by point & click. Square brackets are automatically provided.

"Functions" provides a list of available functions. A function is included in the current expression by point & click. Placeholders for the function's arguments are automatically provided.

"Operators" provide a list of available operators. An operator is included in the current expression by point & click.

25.1.1 Domains

This parameter is relevant for using the Expression Editor for the Field Calculator functionality.

It offers a list of parameters in the data table being edited that use domain coded values. For example, when editing a Node attribute in the node data table (i.e. msm_Node), the Domains dropdown shows the parameters that use domain code values (Figure 25.5). This information may then be used as a reference for defining the value in the expression.

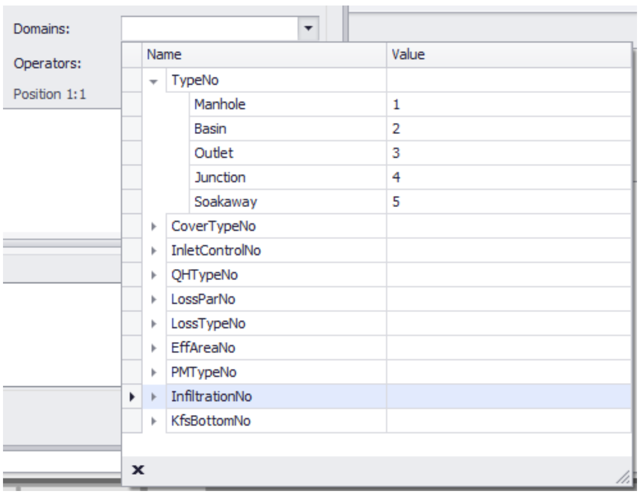


Figure 25.5 Domain coded values in the Nodes data table

25.1.2 Variables

The dropdown list shows all available attributes in the source table and any user-specified variable, which may be used in building the expression. Select a variable from the list to include in the current expression being built. Square brackets enclosing the variable are automatically provided.

Used with the Field Calculator, the variables are values of other columns in the table.

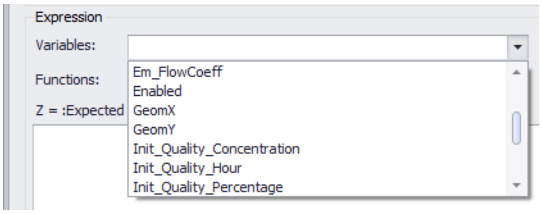


Figure 25.6 Example of variables offered when making edits in the Junctions data table (mw_Junction)

25.1.3 Operators

The dropdown shows the operators that may be used to create expressions.

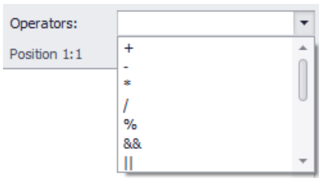


Figure 25.7 Operators in the Expression Editor

It contains Arithmetic operators that works on numbers, Comparison operators and Boolean operators.

Arithmetic Operators

Arithmetic operators work on numbers and always returns a number. To have an arithmetic operator returning an integer value, both operands must be integers. If one of the operands is a floating number value, the result will also be a floating number value.

Table 25.1 List of arithmetic operators

Item	Description
+	Adds two numbers
-	Subtracts two numbers
*	Multiplies two numbers
/	Divides the first number with second number
%	Remainder after integer division of the first number with second number

Some of the operators also work on other types. More details in the following sections.

Comparison operators

The result of a comparison is always a Boolean value. The two operands must be of the same type to be compared.

Table 25.2 List of comparison operators

Item	Description	Supported types
==	Equal operator. Checks if the values of two operands are equal or not, if yes then comparison returns true	Number, DateTime, TimeSpan, String, Boolean
<> !=	Not-equal operator. Checks if the values of two operands are equal or not, if values are not equal then comparison returns true	Number, DateTime, TimeSpan, String, Boolean
<	Less-than operator. Checks if the value of left operand is less than the value of right operand, if yes then comparison returns true	Number, DateTime, TimeSpan
<=	Less-than-or-equal operator. Checks if the value of left operand is less than the value of right operand, if yes then comparison returns true	Number, DateTime, TimeSpan

Table 25.2 List of comparison operators

Item	Description	Supported types
>	Greater-than operator. Checks if the value of left operand is greater than the value of right operand, if yes then comparison returns true	Number, DateTime, TimeSpan
>=	Greater-than-or-equal operator. Checks if the value of left operand is greater than or equal to the value of right operand, if yes then comparison returns true	Number, DateTime, TimeSpan

Logical Operators

Logical operators work on Booleans and always return a Boolean value.

Table 25.3 List of logical operators

Item	Description
&&	Logical AND operator. If both the operands are non-zero, then condition becomes true.
||	Logical OR Operator. If any of the two operands is non-zero, then condition becomes true.
!	Called Logical NOT Operator. Use to reverses the logical state of its operand. If a condition is true, then Logical NOT operator will make false.

25.1.4 Functions

The dropdown shows the functions that may be used in the current context. The set of functions available will depend on the context of the expressions.

25.1.5 Special functions for control flow

if statement

When it is necessary to return different values depending on some condition, the if statement comes in handy. It comes in two forms, but works the same:

if (boolExpression, trueExpression, falseExpression)

boolExpression ? trueExpression : falseExpression

Example if to return 5.6 or 6 depending on whether the a variable is larger than zero:

if ([a]>0, 5.6, 6)

ifs statement

If more than two values are to be chosen between, the ifs statement can help on this:

ifs(condition, value, [condition, value,]... defaultValue)

In the example below the expression will return the string *'a<0'* , *'a=0'*, *'a<3'* , *'a=...'* , depending on the value of the variable a

Ifs([a] < 0, 'a<0', [a] == 0, 'a=0', [a] < 3, 'a<3', 'a='+ToString([a]))

25.1.6 Expressions involving numbers

A fixed value can be specified directly in the expression editor, using a dot as the decimal separator.

Table 25.4 Syntax for defining numbers

6	Integer number value
6.12	Floating number value
6.1234e+2	Floating point number in scientific notation, same as 612.34

Standard math functions

The standard math functions work on floating numbers and returns floating numbers.

Table 25.5 List of math functions

Function	Description
Abs(x)	Absolute value of x
Acos(x)	Arccosine, or inverse cosine of x, x must be in the range [-1;1], the result is in radians
Asin(x)	Arcsine, or inverse sine of x, x must be in range [-1;1], the result is in radians
Atan(x)	Arctangent, or inverse tangent of x, the result is in radians
Atan2(y,x)	Arctangent calculated based on an (x,y) coordinate, the result is in radians
Ceiling(x)	Number rounded up, away from zero.
Cos(x)	Cosine of the angle x in radians
Cosh(x)	Hyperbolic cosine of x
Exp(x)	Exponential function, e raised to the power of x
Floor(x)	Rounds number down, toward zero

Table 25.5 List of math functions

Function	Description
Log(x)	Returns the logarithm of x to the base e
Log10(x)	Returns the base-10 logarithm of x
Max(x,y)	Largest of the two values x and y
Min(x,y)	Smallest of the two values x and y
Power(x,y)	Number x raised to a power y
Round(x)	Rounds x to nearest integer value
Round(x,y)	Rounds x to y number of digits, y must be an integer
Sign(x)	Determines the sign of x, returning -1, 0 or 1
Sin(x)	Sine of the given angle x in radians
Sinh(x)	Hyperbolic sine of x
Sqrt(x)	Calculates the square root of x, x must not be negative

Other functions related to numbers

ToInt(x)
Converts a floating number value to an integer number value. If the floating number contains a fractional part, the nearest integer is used (rounding).

ToString(x)
Converts a floating number value to a string representation.

ToString(x, ‘Fy’)
Converts a floating number value to a string representation, keeping only y digits after the decimal separator.

DoubleFromString(arg, sep)
Get double value from string representation arg, where sep is the decimal separator, either "." or ",".

25.1.7 Expressions involving DateTime and TimeSpan

A DateTime value can be specified directly in the expression editor in several ways.

Table 25.6 Syntax for defining date and times

#2004-03-15 19:30:00#	Fixed Date and time
DateTime(2004,03,15)	Only Date
DateTime(2004,03,15,19,30,00)	Date and time
DateTime(2004,03,15,19,30,00,123)	Including milliseconds

Table 25.6 Syntax for defining date and times

DateTimeFromTicks(x)	From integer ticks value
DateTimeFromYears(x)	From decimal year value
DateTimeFromString(x)	From string representation
Now()	Current date and time on this computer

The first "fixed date and time" is a constant value and cannot include expressions inside the ##. The other DateTime functions can include sub-expressions.

The DateTimeFromString(x) method supports both a local date-time representation which depends on the regional settings, and the standard date-time strings on the form: "2009-03-06 08:24:30" and "2009-03-06T08:24:30"

A Timespan value can be specified directly in the expression editor in several ways.

Table 25.7 Syntax for defining a Timespan

#+1085.12:54:30# #-1085.12:54:30#	Fixed TimeSpan, being 1085 days, 12 hours, 54 minutes and 30 seconds, either positive or negative
#+1085.12:54:30.020# #-1085.12:54:30.020#	Fixed Timespan, including milliseconds
#+12:54:30# #-12:54:30#	Fixed timespan, not including days value.
#+12:54# #-12:54#	Fixed timespan, not including days and seconds value
TimeSpan(1085,12,54,30)	TimeSpan function
TimeSpanFromDays(x)	Timespan from decimal days value
TimeSpanFromHours(x)	Timespan from decimal hours value
TimeSpanFromMinutes(x)	Timespan from decimal minutes value
TimeSpanFromSeconds(x)	Timespan from decimal seconds value
TimeSpanFromTicks (x)	Timespan from integer ticks value

The "fixed timespan" are constant values and cannot include expressions inside the ##. The other TimeSpan functions can include sub-expressions.

Operators involving DateTime and TimeSpan

Table 25.8 Operations with DateTime and TimeSpan

Name	Description	Return type
DateTime - DateTime	Subtract two date-times	TimeSpan
DateTime + TimeSpan	Add a time-span to a date-time	DateTime
DateTime - TimeSpan	Subtract a time-span from a date-time	DateTime
TimeSpan + TimeSpan	Add two time-spans	TimeSpan
TimeSpan - TimeSpan	Subtract two time-spans	TimeSpan

Functions involving DateTime

Table 25.9 List of functions for Date and Times

Name	Description	Return type
AddYears(dt, n)	Add n years to the dt datetime, n must be integer number	DateTime
AddMonths(dt, n)	Add n months to the dt datetime, n must be integer number	DateTime
AddDays(dt, x)	Add x days to the dt datetime	DateTime
AddHours(dt, x)	Add x hours to the dt datetime	DateTime
AddMinutes(dt, x)	Add x minutes to the dt datetime	DateTime
AddSeconds(dt, x)	Add x seconds to the dt datetime	DateTime
AddTicks(dt, n)	Add n ticks to the dt datetime, n must be integer number	DateTime
Year(dt)	Year component of date time	Integer
Month(dt)	Month component of date time	Integer
Day(dt)	Day component of date time	Integer
Hour(dt)	Hour component of date time	Integer
Minute(dt)	Minute component of date time	Integer
Second(dt)	Second component of date time	Integer
Millis(dt)	Millisecond component of date time	Integer
DayOfFraction(dt)	Fractional part of the day, between 0 and 1	Integer
DayOfWeek(dt)	Day of week, 1 being Monday and 7 being Sunday	Integer
DayOfWeek0(dt)	Day of week, 0 being Sunday and 6 being Saturday	Integer
DayOfYear(dt)	Day number of the year	Integer
Ticks(dt)	DateTime ticks value	Integer
TimeOfDay(dt)	Time of the day as a TimeSpan	TimeSpan

Table 25.9 List of functions for Date and Times

Name	Description	Return type
TotalYears(dt)	Whole and fractional year of date time	Number
YearFraction(dt)	Fractional part of the year, between 0 and 1	Number

Functions involving TimeSpan

Table 25.10 List of functions for Timespans

Name	Description	Return type
Abs(ts)	Absolute value of time span	TimeSpan
Days(ts)	Day component of time span	Integer
Hours(ts)	Hour component of time span	Integer
Minutes(ts)	Minute component of time span	Integer
Seconds(ts)	Second component of time span	Integer
Ticks(ts)	Time span ticks value	Integer
TotalDays(ts)	Whole and fractional days of time span	Number
TotalHours(ts)	Whole and fractional hours of time span	Number
TotalMinutes(ts)	Whole and fractional minutes of time span	Number
TotalSeconds(ts)	Whole and fractional seconds of time span	Number

25.1.8 Expressions involving strings

Strings are enclosed in either double or single quotes.

Table 25.11 Syntax for defining a text string

“MyString”	‘MyString’

If a quote character is required matching the enclosing character, it can be escaped using the backslash character.

Table 25.12 Syntax for defining a text string including quotes

Expression string	Result string
"my \"new\" value"	my "new" value
'my "new" value'	my "new" value
'my \'new\' value'	my "new" value

Operators involving string

Adding two strings together will concatenate the strings.

Functions involving string

Table 25.13 List of functions for strings

Item	Description	Return type
Concat(str1, str2 [,str3[…]])	Concatenates two or more strings.	String
Contains(str, substr)	Returns whether the specified substring occurs within a string.	Boolean
EndsWith(str, substr)	Returns whether a string ends with the specified substring.	Boolean
StartsWith(str, substr)	Returns whether a string starts with the specified substring.	Boolean
Substring(str, startIndex [,length])	Retrieves a substring from the str string. The substring starts at a specified index position and has at most the specified length. If length is not provided, the substring includes all characters till the end of the string.	String
Trim(str)	Remove leading/trailing white-space characters.	String
TrimStart (str, substr)	Removes the specified substring 'str' at the start of the 'str' string. The specified substring is removed only if placed at the start of the source string (any other occurrence of the same substring found at another location is not removed).	String
TrimEnd (str, substr)	Removes the specified substring 'str' at the end of the 'str' string. The specified substring is removed only if placed at the end of the source string (any other occurrence of the same substring found at another location is not removed).	String

Table 25.13 List of functions for strings

Item	Description	Return type
IndexOf (str, char [,int])	Returns the index (position) of a specified character within a string. This index may e.g. be used in the Substring function. Numbering starts at 0. The first argument is the source string, the second defines the character to get the index from. The last optional argument is an integer selecting the occurrence of the searched character, when the source string contains several times this character. This integer must be >= 1 and <= number of occurrences of the specified character in the input string.	Integer
Split (str, substr)	Splits a string using the specified substring as separator, to return a list of substrings. The first argument is the source string to split, the second defines the separator. This function is only meant to be used in the 'Import and export' tool, which can import each substring from the resulting list using the 'Iterate' action.	Array

25.1.9 Variables and functions for rivers and collection system control rules

Table 25.14 List of variables and functions for control rules

Item	Description	Return type
SimStartTime()	Returns the simulation start date and time.	DateTime
SimTime()	Returns the current time of the simulation.	DateTime
SimTimeStep() dt()	Returns the current time step size.	TimeSpan
SimTImeSpan()	Returns the elapsed simulation time, i.e. the time since simulation start time.	TmeSpan
TableLookup('table-id', input)	Based on an input value it will do lookup in a table and return the looked-up value. The table-id is a string that identifies the table to use. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used).	Number
TSLookup('ts-id', [phaselag])	Based on the current simulation time it will do lookup in a time series. The ts-id identifies the time series to use, by a file and an item name/number. The optional phaselag expression is a TimeSpan value used as offset from the current simulation time, to look forwards/backwards in time (if the phaselag is a negative time span, it will look backwards in time).	Number
TSTableLookup('table-id', [phaselag])	Based on the current simulation time it will do lookup in a table containing times in the input column. The table-id identifies the table. Values in the input column of the table must be strictly monotonically increasing. The input column can contain DateTimes values representing absolute time values, or double values representing the number of seconds since simulation start. The optional phaselag expression is a TimeSpan value used as offset from the simulation time, to look forwards/backwards in time (if the phaselag is a negative time span, it will look backwards in time).	Number

Table 25.14 List of variables and functions for control rules

Item	Description	Return type
PreviousIn-Time(input, time-Back)	Based on an input value it will return the value as it was some time back. The input expression must return a double value (it is often a sensor, but any ex-pression returning a double can be used). The timeBack expression must be either a double or a TimeSpan value (a double value is interpreted as a number of hours).	Number
TimeSin-ceChange(input)	Time since value changed. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used).	TimeSpan
MinInTime(input, startTime, end-Time)	Based on an input value it will return the minimum value within a specified time interval back in time from the current simulation time. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used). The startTime and endTime expressions must be either a double or a TimeSpan value (a double value is interpreted as a number of hours).	Number
MaxInTime(input, startTime, end-Time)	Based on an input value it will return the maximum value within a specified time interval back in time from the current simulation time. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used). The startTime and endTime expressions must be either a double or a TimeSpan value (a double value is interpreted as a number of hours).	Number
DiffInTime(input, startTime, end-Time)	Difference in value over time, from startTime to endTime. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used). The startTime and endTime expressions must be either a double or a TimeSpan value (a double value is interpreted as a number of hours).	Number

Table 25.14 List of variables and functions for control rules

Item	Description	Return type
TimeDerivative(input, startTime, endTime)	Based on an input value it will return the time derivative over a specified time interval back in time from the current simulation time. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used). The startTime and endTime expressions must be either a double or a TimeSpan value (a double value is interpreted as a number of hours).	Number
Average(input, startTime, endTime)	Based on an input value it will return the average value within a specified time interval back in time from the current simulation time. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used). The startTime and endTime expressions must be either a double or a TimeSpan value (a double value is interpreted as a number of hours).	Number
AverageIf(input, condition, startTime, endTime)	Based on an input value it will return the average value within a specified time interval back in time from the current simulation time, including the value in the average if some condition is fulfilled. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used). The condition expression must return a boolean value. The startTime and endTime expressions must be either a double or a TimeSpan value (a double value is interpreted as a number of hours). The condition is evaluated together with the input expression at every time step, and the input expression is only stored for processing if the condition at the current simulation time evaluates to true.	Number

Table 25.14 List of variables and functions for control rules

Item	Description	Return type
TimeIntegrate(input [, startTime, endTime]) Accumulate(input [, startTime, endTime])	Based on an input value it will accumulate/time integrate the value over time, either since start of simulation or within a specified time interval back in time from the current simulation time. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used). The startTime and endTime expressions must be either a double or a TimeSpan value (a double value is interpreted as a number of hours).	Number
TimeIntegrateIf(input, condition, startTime, endTime) AccumulateIf(input, condition, startTime, endTime)	Based on an input value it will accumulate/time integrate the value within a specified time interval back in time from the current simulation time, if some condition is fulfilled. The input expression must return a double value (it is usually a sensor, but any expression returning a double value can be used). The condition expression must return a boolean value. The startTime and endTime expressions must be either a double or a TimeSpan value (a double value is interpreted as a number of hours). The condition is evaluated together with the input expression at every time step, and the input value is only stored for processing if the condition at the current simulation time evaluates to true.	Number

25.2 Examples of Expressions

Below are some examples of expressions built in the Expression Editor used in MIKE+.

Table 25.15 Example expressions

Description	Variable	Expression
Time-Area model Imperviousness is equal to the Flat and Steep Kinematic Wave Impervious Areas	ModeAImpArea	[ModelBAIFlat] + [ModelBAISteep]
Kinematic Wave model Steep Impervious Area is equal to 80% of Time-Area model Imperviousness	ModelBAISteep	[ModeAImpArea]*0.8

Table 25.15 Example expressions

Description	Variable	Expression
Kinematic Wave model Flat Impervious Area is equal to 100% minus Kinematic Wave Steep Impervious Area	ModelBAIFlat	100-[ModelBAISteep]
An Action Active sensor senses a Rule for a valve (i.e. PID control) is Active.	(CS Model RTC Conditions)	([ActionActive_Tank_-Valve_Open])
A Level sensor at a node senses a level less than12.19	(CS Model RTC Conditions)	([Sensor_Col_OLS_-Suct] < 12.19)
A Level sensor at a node senses that level is less than 40.48 and another Level sensor at another node senses level is greater than 8.51	(CS Model RTC Conditions)	([Sensor_PCVRt199_Primary] < 40.48) && ([Sensor_WmbgPS_WW] > 8.51)
A Level sensor at a node senses that level is greater than 1.37 and an Action sensor senses a Rule for a valve (i.e.setting valve opening) is Active.	(Condition for Rivers and CS control rules)	([Sensor_NS_003] > 1.37) && ([ActionActive_Act_V003_OPEN_STOR])
If a Pump ON/OFF sensor is active (i.e. evaluated as ON) and a Discharge sensor at a link detects flows less than 0.005 m3/s	(Condition for Rivers and CS control rules)	([PumpIsActive_-BL_P10_fik_p1]) && ([BL_StopToem_Discharge] < 0.005)
If an Action Active sensor detects that a Rule is active	(Condition for Rivers and CS control rules)	([ActionActive_EK_P10_STOP_1])

Table 25.15 Example expressions

Description	Variable	Expression
Import a list of values to 'Curves and relations' (ms_TabD) from a single string storing all data in a row, using the 'Iterate' action in the 'Import and export' tool	An input string "List" with this value "12,2;43;27.9"	Split([List], ";") will return an array with three values: - 12,2 - 43 - 27.9 which can be imported one by one in each iteration.
Get the index of a given character with a variable position in the input strings, for use in the 'Substring' function	An input string "Str" with this value "224.9866;27.9;863.12"	IndexOf([Str], ";", 2) will return the index of the second semicolon. Substring ([Str], IndexOf([Str], ",", 2)) will return the last value 863.12

INDEX