MIKE HYDRO

Basin

User Guide

PLEASE NOTE

COPYRIGHT

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

LIMITED LIABILITY

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

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

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

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

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

CONTENTS

MIKE HYDRO BASIN USER GUIDE

1 Introduction

MIKE HYDRO Basin is the modelling package for water resources planning and management in river basins of MIKE Powered by DHI.

MIKE HYDRO Basin offers a map-centric user interface for intuitive model build, parameter definition and results presentation for water resources related applications.

It is a versatile and highly flexible model framework for a large variety of applications concerning Management and Planning aspects of water resources within a river basin.

Typical applications

- Integrated Water Resources Management (IWRM) studies
- Multi-sector solution alternatives to water allocation and water shortage problems
- Reservoir and hydropower operation optimisation
- Exploration of conjunctive use of groundwater and surface water
- Irrigation scheme performance improvements

Features

MIKE HYDRO Basin models utilise a river network and catchments within the specific river basin as the basic model-data. On top of this, a number of features can be applied depending on the type of application. Features include;

River routing, Water users (regular as well as Irrigation users), Hydro power and Reservoirs, Hydrology (rainfall-runoff simulations), Groundwater, Global ranking of water users, Reservoir sedimentation and Water quality options using MIKE ECO Lab.

MIKE HYDRO Basin is the successor of DHI's former product for Integrated water resources management and planning; 'MIKE BASIN'.

2 Working With MIKE HYDRO Basin

This section includes an overview of the different components in the MIKE HYDRO Basin graphical user interface and how to use them.

The general functionality of the individual windows is described in this section, whereas any specific functionality only available in special cases will be described later.

2.1 How to Start MIKE HYDRO Basin

During the installation of MIKE HYDRO Basin a shortcut for '*MIKE Zero*' is placed in the Start menu. When MIKE Zero is started the MIKE Zero shell is displayed.

To create a new MIKE HYDRO Basin setup go to the *File* menu and choose *New* and *File*, which will open the dialogue window shown in Figure 2.1. Select '*MIKE HYDRO Basin Model*' from the MIKE HYDRO Basin product type group and press the 'OK' button or double-click the icon to create a new MIKE HYDRO Basin document.
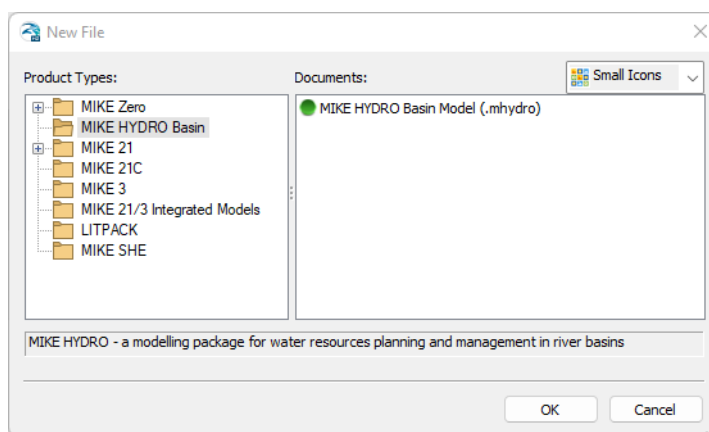
Figure 2.1   Create new MIKE HYDRO Basin document

Creating a new MIKE HYDRO Basin document automatically starts a MIKE HYDRO Basin setup wizard.

The wizard will guide you through defining key settings for the framework of your project. Pressing *Cancel* in the wizard will use default parameters.

Note that any settings specified in the setup wizard may later be changed from the individual dialogue boxes.

If the setup wizard will not be used in future projects, uncheck the checkmark on the first page of the wizard or change the settings in the *User Settings* see section User settings *(p. 29)*.

2.2   The MIKE HYDRO Basin User Interface

The MIKE HYDRO Basin User Interface consists of six main parts (see Figure 2.2):

- Tree view
- Map view and tabular view
- Property view
- Output windows
- Ribbon
- Menu bar
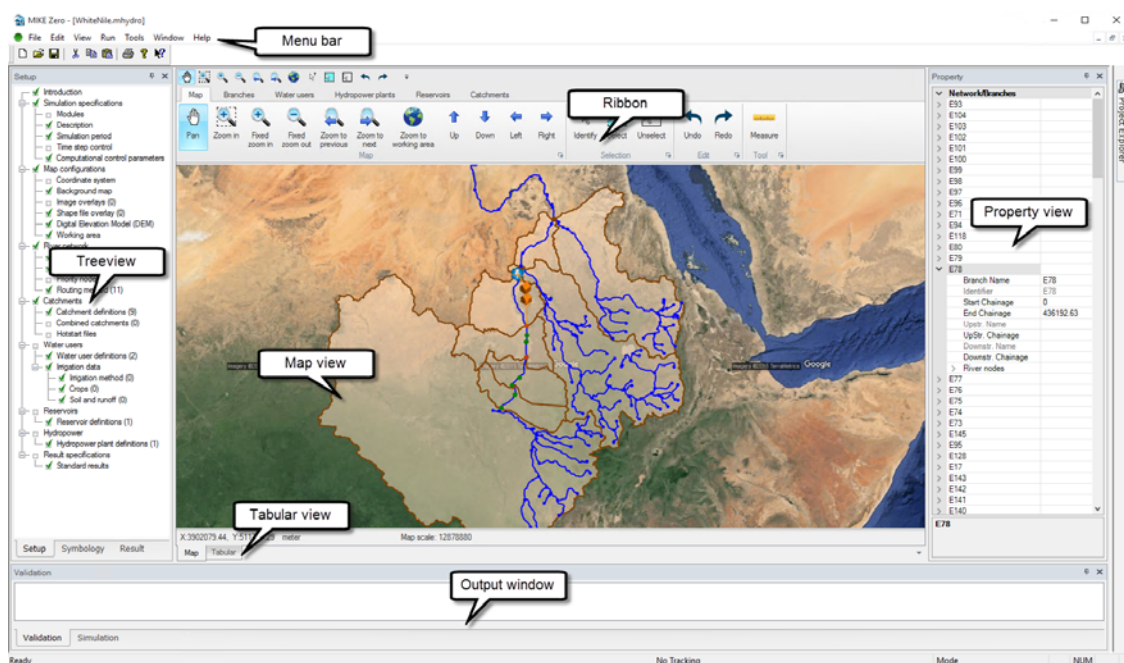
Figure 2.2   MIKE HYDRO Basin User Interface

2.2.1   Tree view

The tree view contains three tabs:

- - control of model parameters

- - control of symbology and labels
- - view simulation results

Setup tree view

The *Setup tree view* allows you to navigate easily between the different sets of model setup parameters. Each node in the tree corresponds to a particular set of parameters, and clicking the node will navigate to the particular set of parameters which will be displayed in the *Property view* as well as in the *Tabular view.*

The tree depends on the settings in the *Modules* page (see User Guide 4.1 Modes *(p. 41)*) and will only show relevant nodes.

The *Setup tree view* also displays the validation status of the parameters – if a green tick mark is displayed on the tree node, this group of setup parameters has been validated without errors. In case of warnings or errors, a red cross is displayed in the tree node and the messages are displayed in the *Validation* window.

Note: A green validation tick mark is displayed if the model setup parameter node has been opened. It is therefore possible to execute simulations without green tick marks in all tree nodes.

Symbology tree view

The *Symbology tree view* allows you to select the objects for which you want to edit the appearance in the *Map view*. The tree displayed under the *Symbology* tab matches the tree displayed in the *Setup* tab for all objects shown on the map.

The symbology of the *Map view* shown under the *Result* tab is specified under the 'Results' node in the tree view.

Symbology
For each object type it is possible to edit the symbology on *Symbol* tab. The symbology can be shown or hidden on the map using the setting 'Visible'.

Note that the 'Visible' setting can also be changed by double-clicking on the symbol in the tree view.

Changes in the symbology is reflected on the map when pressing 'Apply to map' at the bottom of the tabular view. The symbology is automatically applied when changing to the *Setup* or *Result* tree view.

Note that by right-clicking on an object in the tree view you can choose 'Reset symbology' and this will set the symbology for the selected object back to the default symbology.

Symbology settings are saved with the project.

The symbology settings will differ based on the object type e.g. line, point, symbol or polygon.

In general all elements will have 4 symbology types:

- Single symbol
- Graduated size
- Graduated color
- Unique values

Single symbol

Using the single symbol will display all elements on the map with a common symbology.

For point objects it is possible to define the symbol style as either 'Simple symbol' defining colour and size or as 'Picture symbol' using predefined pictures.

Graduated size

Using the graduated size will display the elements with different size or width depending on a user-specified parameter.

The general settings for the symbology is defined as for single symbols with the exception of the size which is defined with a minimum and maximum size. The parameter used for differentiating the size is defined in the 'Field' and can only be numeric values.

A grid is defined at the bottom for specifying the intervals used for different sizes. The number of intervals are changed by using the 'Classify' button. This will open a dialogue for defining the number of classes and the corresponding break values.

Note that the button 'Distribute equidistant' will set the break values according to the current values used in the setup.

Break values can be changed either in the 'Classify' dialogue or directly in the grid for existing classes. In the case where the break values does not cover the entire range of values in the setup a default size may be applied by setting the parameter 'Use default symbology'. If this is not specified elements not falling within the specified interval will be hidden on the map.

Graduated size is not available for polygons.

Graduated color

Using the graduated color will display the elements with different color depending on a user-specified parameter.

The general settings for the symbology is defined as for single symbols with the exception of the colour which is defined as a colour ramp. The parameter used for differentiating the size is defined in the 'Field'.

Colour ramps can be modified using the color palette wizard. Documentation of the wizard is available from the MIKE Zero General Documentation Index in "The Common DHI User Interface for Project Oriented Water Modelling, User Guide".

A grid is defined at the bottom for specifying the intervals used for different colours. The number of intervals are changed by using the 'Classify' button. This will open a dialogue for defining the number of classes and the corresponding break values.

Note that the button 'Distribute equidistant' will set the break values according to the current values used in the setup.

Break values can be changed either in the 'Classify' dialogue or directly in the grid for existing classes. In the case where the break values does not cover the entire range of values in the setup a default colour may be applied by setting the parameter 'Use default symbology'. If this is not specified elements not falling within the specified interval will be hidden on the map.

Unique Values

Using the unique values will display individual symbology depending on a user-specified parameter. The parameter used for differentiating the symbol is defined in the 'Field'.

A grid is defined at the bottom for specifying the relation between symbology and value. New entries in the grid is added using the either 'Add' for one value or 'Add all' for all values currently in the setup. The symbol for each value in the grid is changed by double-clicking on the symbol in the grid.

In the case where a value is not specified in the grid a default symbol may be applied by setting the parameter 'Use default symbology'. If this is not specified elements not falling within a specified value will be hidden on the map.

Labels

For each object type it is possible to edit the labels on *Label* tab. The label can be shown or hidden on the map using the setting 'Visible'.

Label settings are saved with the project.

Map Properties

Map properties are specified under 'Map configuration' in the tree view. Here options are provided for displaying a legend, scalebar and north arrow on the map.

The legend will only display elements shown in the map.

Results tree view

The *Results tree view* allows you to navigate easily between the different simulation results (model outputs). Each node in the tree corresponds to a particular set of simulation results, and clicking the node will navigate to that set of simulation results.

For more information on how to view results see section 2.6 Viewing Results *(p. 33)*.

2.2.2 Map view and tabular view

The central part of the user interface has the following two views:

- Map view
- Tabular view

Map view

The *Map view* is the key element of MIKE HYDRO Basin. Map view provides a graphical – or map – view of the model setup. In *Map view*, the different model objects (e.g. Branches) can be added and edited by selecting the appropriate model object tab in the ribbon.

Right-clicking on the map will provide quick access to zoom functions and exporting the map either to the clipboard or a file.

Selection tools

A set of useful tools is available to identify or select specific features defined in the map view. Tools are located in the '*Selection*' ribbon bar group with the following features:

- **Identity**: Identify button is used to identify and highlight specific model objects. Identify has two ways of operation; when activated at a location where only one single object is defined, it will automatically activate the specific object both in the Property view and activate the Tabular view for the specific object. If Identify is activated at a location where multiple objects are located, a list of objects are presented and the user can select which object should be activated.

- **Select**: Enable selection of model objects. Selections can be made through single click on the map or by drawing a rectangle. All objects positioned at the specified location will be selected and highlighted.

- **Unselect**: Removes all selections within the map

- **Delete selection:** Deletes all the selected objects

- **Zoom to selection:** Zooms to all selected objects on the map

Undo and redo

MIKE HYDRO Basin supports undo and redo on operations on the map and changes in the tabular view or property view.

If the cursor is placed over either undo or redo buttons the hint will list the change that will be either re- or undone.

In general all type of data for the network is covered by undo and redo functionality where as changes to the layout and display of data is not.

The following is not covered by undo and redo functionality:

- Symbology and labels
- Result presentations
- Windows layout
- Selections
- Changes to water quality definitions
- Pop-up windows

Measure tool

The measure tool allows to measure distances between points in the Map or along a path. This tool is located in the ribbon in the upper part of the Map view page.

When pressing the Measure icon, a dialogue pops up. In this dialogue it is possible to choose between two options:

- **Line**. When this option is selected, the distance is calculated along a straight line. The first click on the map locks the start point of the line. While the cursor moves on the map it is possible to see the line along which the distance is measured. At the same time, the length of the line is displayed in the dialogue.

- **Path**. This option must be selected to measure length along paths that are not straight. The first click on the map select the start point of the path. When the cursor moves further on the map and additional points are selected with a click, the distance from the start point are added to the measured length and displayed in the measure tool window. The specified path it is not visible on the map.

In the lower right corner of the Measure tool dialogue, the following two buttons are shown:

Clear. Click on this button to reset to 0 the distance displayed in the Measure tool dialogue.

Close. Click on this button to close the Measure tool dialogue.

Tabular view

The *Tabular view* provides a dialogue-style interface to the model parameter group selected in the tree view. In the *Tabular view* the properties of the model parameter groups are specified and edited. In some cases the *Tabular view* consists of one or more pages. Multiple pages will be shown as tabs which may be shown or hidden based on the individual parameter settings.

In general the tabular view is divided into an upper part with fields and a lower part with an overview table. When changing between records in the overview table the upper part will be updated with parameters for the active record.

Overview table

The overview table contains all attributes for the object group. The records can be sorted by clicking on the header.

One or more cells can be copied to the clipboard by selecting the relevant cells (they will turn blue) and either right-clicking and choose '*Copy*' or use Ctrl+C.

One or more cells can be pasted to the table by standing in the top left cell of the group of cells to be overwritten and either right-clicking and choose '*Paste*' or use Ctrl+V. Any value pasted to a read-only cell will be ignored.

Note that in tables related to some objects displayed on the map e.g. branches and catchments it is not possible to add new records - only existing records will be overwritten. For other tables new records are created.

In order to delete several objects in the table the entire row must be selected by clicking on the very left column in the grid (row will turn blue) and clicking 'Delete' on the keyboard.

Right-clicking in a table related to objects displayed on the map will give the option '*Zoom to active*'. This will zoom the *Map view* to the active record.

For tables related to objects displayed on the map the first column is a checkmark. The checkmark shows if the element is selected on the map. The selection status can be changed either in from the map or the overview table.

Selected attributes in the *Tabular view* has a corresponding line in the *Property window*. The *Tabular view* and the *Property window* are synchronised – a change in either will be reflected in the other. Thus, the same parameters can be edited in either of the two views.

2.2.3 Property view

The *Property view* provides another way of displaying and editing the properties of model parameter groups – including the model objects. The same properties can be edited through the *Tabular view*. The two places are syn-

chronised – any change made in the *Tabular view* is reflected in the *Property view* and vice versa.

The Property view will show properties for the active model object group. The data is grouped by a unique identifier (ID) and the ID is highlighted in dark grey for the active record.

The *Property window* may be hidden by unselecting *Property view* in the '*View*' drop-down menu in the MIKE Zero menu bar.

The content of *Property window'* may be defined to either show all model group objects or selected objects only by setting the *'Property view: Showselected'* in the '*View'* drop-down menu in the MIKE Zero menu bar.

2.2.4   Output windows

The *Output windows* can display a collection of different windows depending on the situation. Per default the windows are shown as tabs. The individual windows are described below.

All type of output windows may be shown by selecting the output window in the '*Project*' tab of the ribbon. If new information is shown in a hidden window the window will automatically be shown.

Validation

The *Validation* window shows validation status messages and error messages. Each time a model object is edited e.g. changing data in the *Tabular view* or the *Property view* the data is validated and any errors are shown in the *Validation* window.

Some extended validation checks are not performed on the fly, but can manually be run by selecting '*Extended validation*' from the '*Run*' drop-down menu in the MIKE Zero menu bar. Running the *Extended validation* will perform both the standard and extended validation checks.

Note that error messages from the extended validation checks are only updated when a new extended validation is performed. Hence extended validation messages may still appear even if the error has been corrected.

Running a simulation will automatically perform an extended validation.

Simulation

The *Simulation* window shows progress and output from the computational engine.

Time series

The *Time series* window is automatically shown when plotting either input or output times series.

To add an input time series to the window, click on the [...] button next to the time series in *Tabular view* and select '*Add to new plot*' or '*Add to current plot*'. An input time series may also be added to the window by clicking on the input time series grid cell in the *Property view*. A [...] button will appear in the right hand side of the cell and clicking on that button gives the options of '*Add to new plot*' or '*Add to current plot*'.

To add an output time series to the window, navigate to the relevant model parameter group in *Result tree view*, right click on the relevant model object in the *Property view* and select the time series to be added to a new/current plot in the *Time Series* window. See section 2.6 Viewing Results *(p. 33)* for more details.

The *Time Series* window can contain one or more plots which are shown as tabs. A plot can be renamed by right-clicking on the tab and selecting '*Rename*'.

A plot can contain one or more time series. The layout of a plot has 3 controls. The first control shown on the left contains a list of each time series in the plot grouped by file name and time series type. Time series can be removed from the plot by right-clicking and choosing '*Remove*' on either an individual time series or a group of time series.

The middle control shows the data of the time series. Depending on the focus in the left control either one or a group of time series are shown in the middle control. Data can be copied to the clipboard by selecting one or more cells and either right-clicking and choosing '*Copy*' or using 'Ctrl+C'.

The right control shows all time series added to the plot. Right-clicking on the plot provides a sub-menu for zooming, printing and user settings such as the use of secondary axis, position of legends, etc.

The *Symbology* dialogue from the sub-menu gives access to full control for specifying symbology, titles etc. for the plot.

Note on zoom in the window: Drawing a rectangle with the mouse from top left to bottom right will zoom in and drawing a rectangle from bottom right to top left will zoom out in the *Time series* window.

The configuration of plots is saved with the setup and will automatically appear when the setup is reopened. Any time series not available (either because they come from files that are not reloaded or because the files no longer contain the specified time series) are automatically removed from the plots.

2.2.5   Ribbon

The ribbon is located above the *Map view* and gives access to graphically editing of the geographically positioned objects in the model. Each model object will have a corresponding set of tools. The most common are described below.

Select

Using the function *Select* will select the model object on the map. Choose *Select* and then click on the element on the map. When an element is selected it will change colour to light blue. The selected elements will also be highlighted in the *Property view*.

For some model objects there are multiple types controlled by the same ribbon e.g. Structures. For these model objects the *Select* function will only select objects of the active type (determined by the combobox *Type* in the same ribbon), whereas the *Select any* function will select any object of the active group.

Add

Using the function *Add* will add the model object on the map. Choose *Add* and then click on the location on the map where the object should be inserted. Some model objects can only be inserted on a branch e.g. river nodes. The cursor will change to a cross when the cursor is located at a valid place on the map.

Branches and *Catchments* have multiple options for Adding data:

- **Branches**: 'Add' creates a branch polyline from user-defined digitisation on the map. 'Add trace' will add a delineated (traced) river branch when defining river branches through *River tracing and Catchment delineation* using DEM

- **Catchments**: 'Add digitise' creates a catchment polygon from user-defined digitisation on the map. 'Add delineate' add a delineated catchment when defining catchments through the *River tracing and Catchment delineation* using DEM

Note: the above functionality will add model objects one by one on the map. It is also possible to import multiple objects at once, from a shape file, in which case the location of objects is directly read from this shape file. To achieve this, please see section Import from Shapefile.

Move

Using the function *Move* will move a model object on the map. *Move* option is present for point data objects (e.g. Water users, Reservoirs, Hydro power etc.).

Choose *Move* and then click on the object that should be moved. The selected object will change colour to light blue and a small symbol 🔷 is shown. When clicking on the symbol the model object can be dragged to a new location.

For some model objects there are multiple types controlled by the same ribbon e.g. Structures. For these model objects the *Move* function will only move objects of the active type (determined by the combobox *Type* in the same ribbon), whereas the *Move any* function will move any object of the active group.

Edit

Using the function *Edit* will move a model object on the map. *Edit* option is present for polyline or polygon data objects (e.g. branches or catchments). *Edit* can be used to either edit the shape of a selected object or to move the entire object.

Choose *Edit* and then click on the object that should be edited. The selected object will change colour to light blue and a small symbol 🔷 is shown. When clicking on the symbol the entire model object can be dragged to a new location.

Edit polyline or polygon features has the following options for vertices (or points) in MIKE HYDRO:

- **Move vertices**: Individual vertices can be moved by click and drag when positioning the mouse cursor on top of an existing vertice.

- **Delete vertices**: Individual vertices can be deleted by double-click when positioning the mouse cursor on top of the point to be deleted

- **Add vertices**: Additional vertices can be added with a single mouse click by pointing to a location where the new vertice must be located.
  Note: It can be advantageous to click and hold the mouse button down for a second or so to insert a new point. This will avoid that a 'fast single-click' unselects the actual feature.

Delete

Using the function *Delete* will delete a model object on the map. Choose *Delete* and then click on the object that should be deleted. When the cursor is close to an object that can be deleted it will change symbol to an arrow 🔷. When clicking on the map the object is deleted.

For some model objects there are multiple types controlled by the same ribbon e.g. Structures. For these model objects the *Delete* function will only delete objects of the active type (determined by the combobox *Type* in the same ribbon), whereas the *Delete any* function will delete any object of the active group.

2.2.6 Menu bar

The menu bar contains functionality for viewing and working with your setup. For details about the individual functionality see section 2.4 MIKE Zero Menu Bar *(p. 28)*.

2.3 Layout

MIKE HYDRO has an IDE-style user interface (IDE=Integrated Development Environment) where all windows reside under a single parent window, referred to as the MIKE Zero shell.

The shell contains dockable and collapsible child windows, tabbed windows and splitters for resizing of child windows.

The default windows docking layout is shown in Figure 2.2, but the user may change this.

A window is undocked by either right-clicking in the heading and selecting '*Float*' or by dragging the heading away from its current position. If only a tab in a parent window containing multiple tabs should be undocked then you can either right-click or drag the individual tab.

A window is docked again by either right-clicking in the heading and selecting '*Dock*' or by double-clicking the heading. In both cases the window will be docked again at the last location it was docked.

A window can also be docked by manually placing the window at a new location. Select the heading of the window that should be docked and start dragging the window and a docking guide will appear (see Figure 2.3). While still dragging the window move the cursor to the new location using the docking guide, when the cursor is placed on the docking guide the location will be highlighted in blue. Once the correct location is highlighted the left mouse button is released and the window will be docked.

The docking guide will always relate to the active window, where the active window is the window located below the position of the cursor.

The docking guide will display small symbols where a docking location is possible. In the middle are 9 locations relative to the active window, where the new window can either be placed as a tab to the active window (middle symbol) or left/right/below/above the active window (symbols around the middle). On the edge of the shell are 4 other locations relative to the shell, where the new window can be placed to the left/right/top/bottom of the shell.
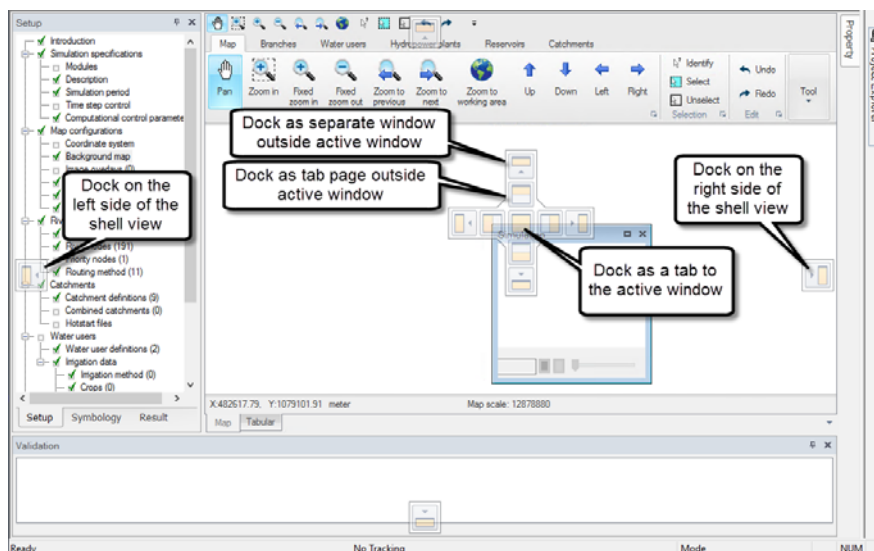
Figure 2.3   Docking guide

Right-clicking on a docked window gives the option of 'Auto Hide'. This will automatically hide the window when it is not in focus and only display a tab. Clicking the tab will show the full window again.

MIKE HYDRO has three type of windows available: *Map view*, *Tabular view* and standard views.

The *Map view* is fixed to the middle of the shell. It cannot be undocked, moved or resized. The size of the map will automatically fit any available space in the shell not used by other windows. Hence indirectly the map can be resized by resizing the other windows.

The *Tabular view* can be both undocked and docked in any location within the shell, however it needs to be docked with another window. The *Tabular view* differs from standard windows because it cannot be closed.

The standard windows which includes the *Tree view*, *Property view* and *Output windows* can be hidden, undocked or docked in any location within the shell.

Any changes made to the layout are automatically saved with the setup. It is further possible to save a specific layout to a file and re-use in other project. The layout can easily be reset to the default configuration through the *View* menu. See section 2.4.3 View menu *(p. 29)* for further details.

2.4   MIKE Zero Menu Bar

The MIKE Zero menu bar contains some menus that are generic to the common MIKE Zero framework and some that are specific to working with MIKE

HYDRO. For details about the common MIKE Zero framework see '*MIKE Zero, The Common DHI User Interface for Project Oriented Water Modelling, User Guide*' referred to as the 'MIKE Zero User Guide' below.

2.4.1 File menu

The menus *New*, *Open*, *Close*, *Close Project*, *Save*, *Save All*, *Save As*, *Save Project As Template*, *Print Setup*, *Print Preview*, *Print* and *Options* are all part of the MIKE Zero framework see 'MIKE Zero User Guide' for more details.

Print

All print functionality is related to the *Map View*.

Options

Install examples
Several examples are included in the MIKE HYDRO Basin installation. Select MIKE_HYDRO_Basin in the pop-up menu and click '*Install*'. The examples may then be opened from the folder named '...\MIKE Zero Projects\MIKE_HYDRO_Basin\'.

User settings
The '*User Settings'* dialogue contains general settings for MIKE Zero. For more details see 'MIKE Zero User Guide'.

The tab *MIKE HYDRO* relates specifically to MIKE HYDRO Basin and here it is possible to control if the setup wizard should be displayed when creating a new setup see .

The *MIKE HYDRO* tab also controls the language used in the dialogue windows.

2.4.2 Edit menu

For Undo and Redo options, see .

The other options are part of the MIKE Zero framework see 'MIKE Zero User Guide' for more details.

2.4.3 View menu

The menus *Project Explorer*, *Project Map*, *Start Page, Simulation History* and *Toolbar* are all part of the MIKE Zero framework see 'MIKE Zero User Guide' for more details.

Export graphics
Option for saving MIKE HYDRO graphics to either Clipboard or graphics file.

2.4.4   Run

The *Run* menu contains options for validation and simulation of model set-ups.

Extended validation

Before running the model, it is recommended to run an extended validation of the model setup. Any validation issues will be shown in the *Validation* window see .

Simulation

To execute the model engine, select '*Simulation*'. Output from the computational engine (status and error messages) are shown in the *Simulation* window see .

2.5   Creating Input Files

Most data is saved within the MIKE HYDRO file, but some input data is stored in external files. Input files will typical consist of time series saved in dfs0 files see section 2.5.1 Create a new dfs0 file *(p. 31)* for more details.

Where a link to an external file must be created a button with 3 dots is shown. Clicking on the button will show a sub-menu see Figure 2.4.
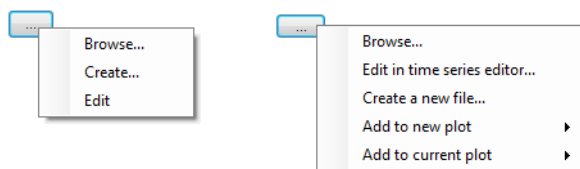
Figure 2.4   Buttons used for input files

The general functionality of the external file buttons has the options:

- **Browse**: Allows you to browse for an existing file.
- **Create**: Will create a file with default settings and automatically open the file for editing with the corresponding editor.
- **Edit**: Will open the file for editing with the corresponding editor.

For files that can be shown on a graph it will also be possible to plot the input file to the *Time Series* window.

2.5.1 Create a new dfs0 file

When creating a new dfs0 file using the button for external files, a dialogue window will be opened. This dialogue is used in different situations for creating a time series (dfs0) file or a table file. The model automatically detects the type of file needed and adjusts the dialogue accordingly.

Create a new time series file

To create a new time series file, the following properties must be specified:

- **Filename**: You may either browse for or manually type the name and location of the time series file.

- **Period**: You may choose the start and end date of the time series using the date-time pickers. The default values will be the simulation period for your setup.

- **Time axis type**: You may choose among non-equidistant calendar axis type and equidistant calendar axis type. For the non-equidistant type, the interval between data values may vary. For the equidistant type, equal time spacing between each time step will be enforced.

- **Time series interval**: The time series interval is the size of the time step.

  Example: Specifying the following:
  - Start date: 01-01-1980
  - End date: 31-12-1981
  - Time series interval: 1 month

  will cause a time series file with 12 time steps to be created.

- **Items**: Depending on the context in which the file is being created, the file will be created with one or more data items.

- **Item name**: The name of the data item

- **Uniform value**: All the time series files which are created with this dialogue will contain the value specified here in all time steps (uniform value). To create files with values varying over time (non-uniform), first create a uniform value time series and then edit it in the time series editor.

- **Time series type**: Contains information on how the data values are interpreted. The possible types are:
  - *Instantaneous*: Means that the values are representative at one precise instant. For example, a measured water level is an instantaneous value. Linear variation between time steps is assumed.

– **Accumulated**: Means that the values are representative of one successive accumulation over time from the start of the event (start time specified) to the current time step (.i.e. accumulated over the entire series). For example, the rainfall is accumulated over the year if we have recorded monthly rainfall values. Linear variation between time steps is assumed.

– **Step Accumulated**: Means values are representative of an accumulation over a time step. For example, rainfall is a step-accumulated value if the rain depth is recorded at different intervals. Values represent the time span between the previous time step and the current time step.

Example: *Rainfall measurement is started at 10:00:00. At 11:00:00 someone picks the recipient, registers the value of 10 mm and empties the recipient. At 12:00:00 the same process is repeated but with a value of 15 mm and so on. Thus, the time series will have the value of 10 mm at time step 11:00:00 and the value 15 mm at 12:00:00 etc.*

– **Mean step accumulated**: Means that values are representative of an average accumulation per time step. For example, the average rain rate is a mean step accumulated value. Values represent the time span between the previous time step and the current time step.

– **Reverse mean step accumulated**: Is equal to *Mean Step Accumulated* type, but values represent the time span between the current time step and the next time step. Used for forecasting purposes.

- **Item type**: The type of data item. The drop-down list can be used to choose one of the possible item types. The choices available will depend on the context.

- **Item Unit**: The unit type of the data item. The drop-down list can be used to choose one of the possible unit types. The choices available will depend on the item type.

Note: For more information on time serie types, item types and units see the '*DFS File User Guide*' in the MIKE Zero Documentation.

Create a new table file

To create a new table file the following properties must be specified:

- **Filename**: You may either browse for or manually type the name and location of the time series file.

- **No. of timesteps**: This defines the number of rows in the table (or the number of x-axis points on a curve).

- **X-axis unit**: The x-axis unit can be selected from the drop-down menu.

- **Items**: Depending on the context in which the file is being created, the file will be created with one or more data items.

- **Item name**: The name of the data item.

- **Values**: Here the pair-wise values of the x-axis and the item are entered. The table file can consecutively be edited in the time series editor.

- **Value series type**: Contains information on how the data values are interpreted. The possible types are the same as for a dfs0 file see section Create a new time series file *(p. 31)*.

- **Item type**: The type of data item. The drop-down list can be used to choose one of the possible item types. The choices available will depend on the context.

- **Item Unit**: The unit type of the data item. The drop-down list can be used to choose one of the possible unit types. The choices available will depend on the item type.

Note: For more information on time serie types, item types and units see the '*DFS File User Guide*' in the MIKE Zero Documentation.

2.6   Viewing Results

Results are viewed from the *Result* tab in the *Tree view*. It is possible to select results either from a list in the *Property view* or from the *Map view*. The results are viewed in the *Time Series* window see section Time series *(p. 24)*.

Result files and other time series are controlled from the *Tree view*. After a simulation the results are automatically added to the *Tree view*. Results from previous simulations, input files or measured data (dfs0 files) can be loaded through the '*Add file...*' button for comparison or calibration.

Files shown in the tree are automatically linked to the setup e.g. when re-opening the setup the same files will be available in the *Result* tree. If a setup contains links to files that are deleted or moved the link will automatically be removed. It is possible to manually remove the link to a file by right-clicking on the file name and choose '*Remove*'.

2.6.1   Basin result files

The output from a basin simulation is automatically loaded to the result view. However if results from a previous simulation should be loaded for comparison the corresponding mdh file should be chosen as this will load all related data.

Note: The result file from a rainfall runoff simulation not including a basin simulation is a dfs0 file which must be loaded manually using the '*Add file*' button. See section 2.6.2 Dfs0 files *(p. 34)*.

The *Tree view* will display 3 nodes for '*Nodes*', '*Reaches*' and '*Catchments*'. When selecting a node in the tree the related time series are shown in the *Property view*. The *Property view* is grouped by the model object identifier with the name in parenthesis. Expanding an identifier will show all result types related to that model object.

Right-clicking in the *Property view* shows a menu for either adding the result to a new plot or the current plot.

Selecting in the *Property view* highlights the model object on the map.

Selecting a model object in the *Map view* automatically selects the related time series in the *Property view*. Right-clicking on a model object gives access to the menu '*Result to plot*' where results can be plotted either on a new plot or the current plot.

Note: It is only possible to select results in the Map view, if the *Map* ribbon is active and set to '*Pan'*.

2.6.2 Dfs0 files

Dfs0 files can be used for input, output and measured time series. Dfs0 files does not contain information about the network and is therefore not connected to the *Map view*.

When adding a dfs0 file the *Property view* is automatically filled with a list of all time series available in the dfs0 file. A filter based on the item type is shown at the top of the *Property view*.

2.7 Getting Help

Online help is available in all dialogues of MIKE HYDRO Basin. The online help system utilises the Microsoft help technology known as HTML Help.

The online help can be accessed in two ways:

● Through the menu *Help* and choosing '*Help Topics*'

● By pressing F1 in any dialogue window, which automatically will direct you to the relevant section of the online help.

Note: In some cases the dialogue box/window must be activated (e.g. by clicking in a cell) to ensure correct links to the online help.

2.8 Examples

A useful way to quickly get started with MIKE HYDRO Basin is to open the examples that are included in the installation.

To install the examples go to the *File* drop-down menu, select '*Options*' and click '*Install Examples…*'. In the pop-up menu, select MIKE_HYDRO_Basin and click '*Install*'. The examples may then be opened from the folder named '...\MIKE Zero Projects\MIKE_HYDRO_Basin\'

2.8.1 Basin module examples

This section includes a brief overview of the Basin Modules examples included in the MIKE HYDRO Installation.

Demo

This example can be viewed and executed with a demo license of MIKE HYDRO. In the getting started section, steps are described to build a similar setup.

IrrigationFAO

This model shows an example of how to investigate the outcomes of different irrigation strategies in a field where different crops are cultivated.

The model includes three different crops: tree, tobacco and potatoes. The scope of such a model is to find the optimal irrigation strategy, which allows to obtain the maximum yield from each crop without spill of water and without incurring in water deficits.

Different irrigation strategies can be formulated by changing the trigger and application options for the crops, in particular the parameters TAW and RAW. This is done in the section Irrigation data → Irrigation method. The new irrigation methods should then be applied to the crops in the Water user → Irrigated Field section.

By changing these parameters, different irrigation scenarios can be run. A comparison of the results from the different simulations can be made to investigate how the different crops are affected by crop stress and in which scenario the maximum yield is achieved.

IrrigationRice

This model shows an example of rice crop irrigation. The model includes an irrigated field with three different rice crop shifts occurring during the year.

One of the applications of such a model is to investigate how the soil type properties affect the crop yield when growing rice. Different soils types have

different effects on percolation. For example, a sandy soil allows a larger amount of water to infiltrate, if compared to a clay soil, which has a higher hydraulic conductivity, thus offering a higher resistance to infiltration. Deep percolation is responsible for a part of the water losses and can therefore deeply affect the water demand of the rice crop.

Different scenarios can be set up by changing the soil models. This is done in the section Irrigation data → Soil and runoff. The key soil parameters to be varied are the three soil moisture content parameters and the saturated drainage coefficient. The different soils properties can then be assigned to the rice field in the section Water user → Irrigated Field.

After running the models with different soil types, it is interesting to investigate how the irrigation demand varies when the field has different soil properties. The differences in percolation loss between the different simulation results can also be compared.

Other factors also affect the irrigation demand on the rice field. As done in Example 2, different irrigation strategies can be formulated by changing the trigger and application options for the crops, in particular the parameters TAW and RAW. This is done in the section Irrigation data → Irrigation method. The new irrigation methods should then be applied to the crops in the Water user → Irrigated Field section.

WhiteNile

This example is a model of the White Nile river basin. It shows a more comprehensive river basin model as well as the utilisation of a coordinate system and background map.

Demo_Aqua-Republica

This setup is a water resources model included in the installation with the overall intension for illustrating the possibility for using a MIKE HYDRO Basin model in connection with Serious Gaming for which DHI has developed the 'Aqua Republica'.
The demonstration example includes a setup with 3 water users, 3 reservoirs and a number of catchments along river branches, and the setup forms the backbone of the demonstration model that can be accessed through the Aqua Republica webpage names as the 'Danida version'.

Additional information on Aqua Republica is available on
THE ACADEMY by DHI's homepage: http://theacademybydhi.com and on
the Aqua Republica homepage: http://aquarepublica.com/.

3 Tutorial: Setting Up a Simple Model

The current chapter includes a short description on how to operate MIKE HYDRO to set up simple examples of the Basin Module.

Setting up a Simple Basin Model

3.1 Setting up a Simple Basin Model

This short tutorial summarises the steps needed to set up a simple model (the demo example) in the Basin module.

3.1.1 Create a new document

To create a new document, start MIKE Zero and choose the MIKE HYDRO model document type from the MIKE HYDRO group.
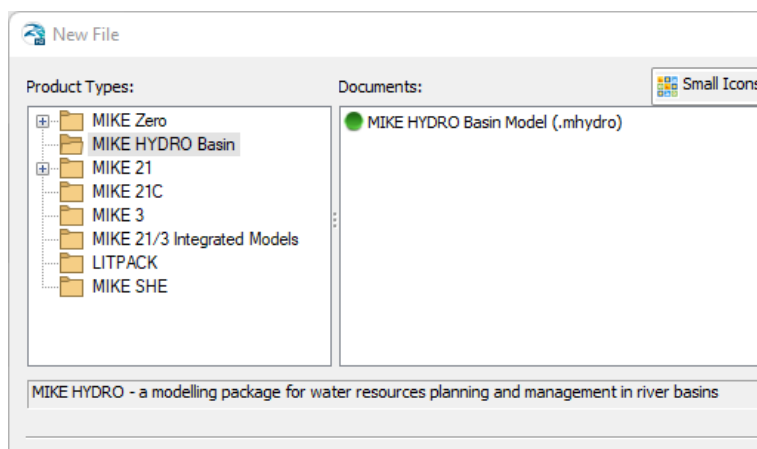
Figure 3.1   Create new MIKE HYDRO document

Press the 'OK' button and an empty document will be created and the setup wizard will automatically open.

3.1.2 Define simulation specifications

The setup wizard guides you through the following simulation specifications:

- Modes
- Description
- Coordinate System
- Background Map

- Digital Elevation Model (DEM)

- Working Area

The Simulation specifications may subsequently be edited by navigating to the relevant sections in the *Setup tree view* and editing the specifications/properties in the *Tabular view* or the P*roperty view*.

3.1.3 Create the River network

The Branches segments of a river network including associated River nodes and Branch connections can be created in three different ways:

- Method 1: Digitising a River network

- Method 2: Importing a River network

- Method 3: Derive the River network from a DEM

For more information see .

When the Branch has been completed, the properties can be inspected and edited in the *Tabular view* or the *Property view.*

3.1.4 Create a Catchment

In the demo example, the Catchment is a schematic catchment, i.e. the catchment is represented by a default shape. To create a schematic catchment, see .

When the catchment has been created, the catchment properties must be specified in the Catchments *Tabular view* or the P*roperty view* (see). A runoff time series must be provided. All runoff from the catchment is added to the River network in the Catchment node.

3.1.5 Insert a Reservoir

To insert a Reservoir, see .

When the Reservoir has been added, the reservoir properties must be specified in the reservoirs *Tabular view* or the *Property view* (cf.).

3.1.6 Insert two Water users

To insert a Water user, see .

Note that the location is for visual purposes only and has no impact on the model setup.

When the Water user and the associated supply and return flow Connections have been added, their properties must be specified in the Water users *Tabular view* or in the *Property view* (cf.).

3.1.7 Insert a Hydropower plant

To insert a Hydropower plant, see .

When the Hydropower plant and the associated supply and return flow connections have been added, their properties must be specified in the Hydropower plant *Tabular view* or in the *Property view* (cf.).

3.1.8 Validate the setup

The initial validation status of the model setup is indicated in the *Setup tree view*. If a green tick mark is displayed on the tree node, the group of setup parameters is valid. In case of errors, a red cross is displayed in the tree node and an error message is displayed in the *Validation* window.

Note: A validation tick mark is only displayed if the model setup parameter node has been opened.

Before running the model, it is recommended to run an extended validation of the model setup. This is done by selecting '*Extended validation*' in the '*Run*' drop down menu in the MIKE Zero menu bar. Any validation issues will be shown in the *Validation* window.

3.1.9 Running a simulation

To execute the model engine, select '*Simulation*' in the '*Run*' drop down menu in the MIKE Zero menu bar. Output from the computational engine (status and error messages) are shown in the *Simulation* window.

3.1.10 Inspecting the results

When the simulation has been run, the results may be viewed in the *Time series* window. To add an output time series to the window, navigate to the relevant model parameter group in *Results tree view*, right click on the relevant object in the *Property view* and select the time series to be added to a new/current plot in the *Time series* window.

Alternatively, output time series may be added to the *Output window* by right clicking on the relevant object in the *Map view*, selecting '*Result to plot*' and select the time series to be added to a new/current plot in the *Time series* window.

4 Simulation Specifications

The Simulation specifications includes the more overall control settings for model simulations such as; which sub-modules are included in the simulation, which simulation period and time step are to be applied and additionally, overall control parameters for the simulation performance are included.

The following pages are included for defining Simulation specification for each model type:

4.1 Modes

The first action to take in a MIKE HYDRO Basin project definition is to select modules to be included in the project and simulations.

Figure 4.1   Model type and Module selection page for MIKE HYDRO Basin.

The 'Modules' section of this page contains two columns of checkboxes. These can be activated and deactivated by the user depending on the preferred GUI appearance and requirement for the project simulations.

The checkbox functionality are as follows:

- **'Data'** :
  Controls whether the data related to the specific module or feature are visible in the tree view.
  Enable the specific '*Data*' checkbox for all features that you want be visible and editable from the GUI tree view.
  Important Notice: The Data checkbox activation does <u>not</u> enable a feature for simulation.

- **'Sim.'** :Control
  Controls whether the specific module is active in the simulation or not.
  '*Sim.*' checkbox <u>must</u> be enabled to include the feature in the simulation.
  Note: It is not possible to activate a 'Sim' checkbox if the equivalent 'Data' checkbox is no active as well.
  All '*Sim'* checkboxes are per default acitve

The MIKE HYDRO Basin module include specification of Basin specific features and Global parameters for the project:. Options are:

- Basin modules:

  – Rainfall runoff
  – Basin simulation
  – Include groundwater *(add-on for Basin Simulation)*
  – Include reservoir sedimentation *(add-on for Basin Simulation)*
  – Water quality *(add-on for Basin Simulation)*
  – Use global ranking *(add-on for Basin Simulation)*

- Global parameters

  – Unlimited resources in catchments without groundwater model
  – Subtract area of irrigation users and reservoirs from catchment area to calculate runoff

Rainfall runoff

Optionally, rainfall-runoff modelling can be included in the Basin model.

Inclusion of Rainfall runoff in a simulation requires that one or more Catchments are defined in the Basin model. Runoff from catchments to river nodes can then either be included through the use of fixed runoff time series files for individual catchments or by activating the calculation of runoff from Catchment characteristics and precipitation (rainfall).

Two types of rainfall-runoff models are available:

- **NAM:**
  A lumped, conceptual rainfall-runoff model, simulating the overland flow, interflow, and baseflow components as a function of the moisture contents in four storages.

- **UHM:**
  The Unit Hydrograph Model includes different loss models (constant, proportional) and the SCS method for estimating storm runoff.

The type of model and associated required input data are specified in Catchment definitions page (cf. Section 7.1.3).

Basin simulation

Basin simulation includes the options for calculating a large variety of different processes and model components taking part in the water consumption and influencing the water balance with a river basin. Basin simulation utilise the calculation engine from DHI's predecessor to MIKE HYDRO for water resources management simulations; MIKE BASIN, and most of the calculation methods for specific modelling features are therefore mature and validated through intensive project applications over a large period of time.

Include groundwater
Optionally, groundwater processes can be included in Basin module. The underlying conceptual model is the linear reservoir model concept (see Appendix A.1 *The Linear Reservoir model*) with one or two aquifers (fast/slow response).
The conceptual structure of the two-layer groundwater component is illustrated in Figure 4.2. Single-layer models only include the shallow (upper) aquifer.
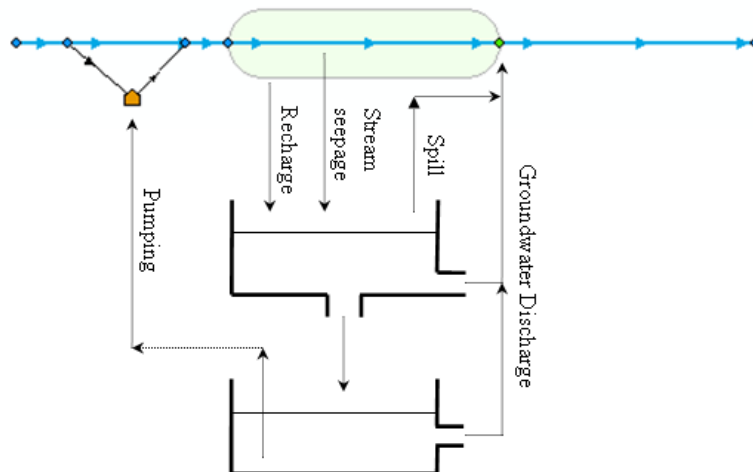
Figure 4.2   Conceptual structure of the groundwater component

As illustrated in Figure 4.2 groundwater interacts with the surface water via groundwater recharge, groundwater discharge and seepage from rivers, reservoirs and connections. Moreover, when the water table of the shallow

(upper) aquifer reaches the land-surface, it starts to spill directly into the river. Finally, groundwater from the deep aquifer can be pumped by water users.

Include reservoir sedimentation

The reservoir sedimentation module routes sediment through the river network and calculates the potential deposits of sediment in the Reservoirs. The reservoir sedimentation module is not a sediment transport model. Sediment is supplied by catchments and is routed through the river network under the assumption that no erosion takes place and deposition solely occurs in the Reservoirs. Any sediment not deposited in the Reservoir is routed to the river downstream and to users supplied by the Reservoir. Sediment is distributed according to the discharge distribution.

Water quality

The Water quality (WQ) module is based on MIKE ECO Lab, which is a highly flexible framework for defining water quality models. MIKE ECO Lab utilise a concept of templates where water quality models are defined with equations and variables, and the water quality option therefore, allows the usage of different water quality models/templates depending on the issue of concern. MIKE HYDRO Basin installation includes two predefined water quality templates that can be applied as is - or can be adjusted by the user to conform with the specific project requirements.

Use global ranking

Global ranking may be used to prioritise water allocation to Water users, Hydropower plants and Reservoirs independently of their geographical location in the river basin. When global ranking is selected, all other allocation rules/priorities are ignored. This includes local priorities and Reservoir reduction levels.

Global parameters

The Global parameters defines global characteristics in the model framework which will be globally applied (for all relevant features).

Unlimited resources in catchments without groundwater model

If this parameter is enabled, groundwater in Catchments without a groundwater model is assumed to be an unlimited resource with no feedback to the surface water system. If it is disabled, no groundwater is available in Catchments without groundwater model.

Subtract area of irrigation users and reservoirs from catchment area to calculate runoff

If this module/parameter is enabled, the area of irrigation users and reservoirs is subtracted from the catchment area prior to calculating catchment runoff. This is particularly important if irrigation users and/or reservoirs have significant surface areas compared to the catchment area, in which they are located.

4.2 Description

The *Description* dialogue includes the option for supplying a user specified Title and a Description for the simulation and the project in general.

Every simulation should be given a Title. Except from helping to describe the actual simulation performed the title will also be included as part of the default name for result files defined by MIKE HYDRO (see section 17.1 Standard Results *(p. 193)*).

Note: Simulation results with a title already used will overwrite the existing results unless a new result folder/file has been specified.

4.3 Simulation Period

The simulation period is generally defined by specifying the required Simulation start date and time and Simulation end date and time.

Date and time can be edited manually by clicking the start date or end date field and change the date (Year+Month+Day) and time (hour+minutes+seconds). In this case it is possible to use the right and left arrow keys to navigate between the date and time variables.

Alternatively, dates can be selected using the calendar icon; , which will open a calendar that allows to browse for the required date (see Figure 4.3). Note, that the simulation time will still have to entered manually after choosing a date from the calendar date selection.
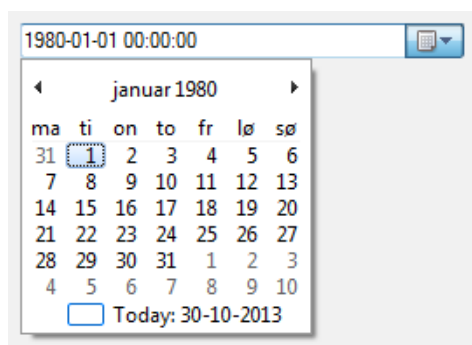
Figure 4.3   Calendar date selection option for simulation periods

Note that it is possible to use input time series for model input parameters, which have different length than the simulation period, and different temporal resolution than the simulation time step.

If an input time series does not cover the entire simulation period it must cover a period of at least one calendar year when the simulation period is longer than a year. If an input time series covers exactly one year it will be

recycled using the appropriate values for the simulation period, both for the period

If the input time series is longer than a year, input data from the first full year will be used for the simulation period before data exist, and input data from the last full year will be used for the simulation period after data exists.

The `*Apply default*' button can be used to extract the possible simulation start-time and end-time. Once the button is activated a search of time-intervals in all time series files included in the actual simulation takes place and the earliest possible start-date and latest possible end-date is automatically transferred to the date-fields.

Note: If no dates are proposed from activating the button the most likely reason will be that there are no overlapping time series in the setup.

4.4 Time Step Control

Time step control features includes options for defining simulation period and time step amongst other parameters for the different modules.

River basin management

Any constant time step can be used in the simulation. Note that the temporal resolution of the input time series can be with any variable temporal resolution different from the simulation time step. Depending on the value type of the time series different interpolation method is used. For more details, see 2.5 Creating Input Files *(p. 30)*.

Note that if the monthly time step is selected, the time step length must be one month, i.e. it is not possible to select a 2 or 3-month time steps.

The results will be written for every time step. However in order to reduce the size of the result file without compromising the accuracy of the computation it is possible to choose a time step sub multiplier. Each time step will be divided by equal sub time step for computing the results, however the results will be written at the end of every time step. For example, if the time step is 1 day, and the time step multiplier is 4, then the computation will be performed every 6 hours, and the results will be written daily.

It is possible to run a simulation with the Basin module in a simple stochastic type of mode, where the initial conditions are reset every year. This is convenient if you want to analyse e.g. drought management scenario for a number of months in the future for likely climatic/inflow conditions. Running in the stochastic mode for say a 20 years period, a 20 years output file of e.g. water level will be produced. It can be interpreted as running 20 times one-year simulations using the same initial condition, but for different inflows and climatic conditions.

If reservoir sedimentation is included in the simulation *(*cf. Section 4.1), the frequency at which the level area volume (LAV) curve is updated (number of time steps between updates) must be specified.

Rainfall runoff time step length

If Rainfall-runoff has been enabled in the simulation *(*cf. Section 4.1), the time step length to be used by the Rainfall-runoff model must be specified.

4.5 Computational Control Parameters

Computational Control Parameters includes options for specifying parameter values for variables used to control the behaviour of the computational engine within MIKE HYDRO.

Computational control parameters are used in the Basin simulation as criteria for solving the water quantity distribution.

The calculation of the flow in the river network for each time step requires an iterative procedure. The computation of the time step will finish when the convergence criteria have been reached. A convergence criterion is given for the flow on the branches and one for the remote flow control nodes. A maximum number of iterations is defined to ensure that each time step will be completed within an acceptable amount of time. The default value is 150 iterations. A warning message will be issued when the computation of the time step cannot be achieved successfully within the maximum number of time steps.

- **Maximum number of iterations**. Maximum number of iterations in solving water quantity distribution. This corresponds to the maximum number each time step will be computed. This number will affect the computation time of Basin simulation. Please note that every time step is computed by at least 4 iterations.

- **Flow convergence criteria**. Maximum change allowed in flow at each computation node between two successive iterations.

- **Control flow convergence criteria**. Maximum difference allowed in flow between the calculated and target flow at remote flow control nodes. This parameter applies for both maximum and minimum remote control flow rules.

5 Map Configurations

The *Map view* is the focal point of each MIKE HYDRO model session, and the Map configurations option enables a user-defined customisation of the *Map view* appearance to best suit the user's preferences and requirements.

Map configurations include the options:

- Coordinate System (Map view and Feature coordinate systems)
- Background Map (Map view background)
- Background layers (Shapefile, raster, and image definitions)
- Digital Elevation Model (DEM) (Background and processing)
- Working Area (Map view working area definition)

5.1 Coordinate System

The user can select a coordinate system for the *Map view* in the Setup tree view. The *Map view* coordinate system is selected using the drop-down menu labeled "Map view coordinate system".

If the user would like for features of the model (i.e., branches, water users, catchments, etc.) to use a different coordinate system than the *Map view* coordinate system, then it is possible to select a different coordinate system using the drop-down menu labelled "Coordinate system for features stored in setup file".

If two different coordinate systems are selected, the coordinates of the model features are automatically converted into the map coordinate system when displayed in the *Map view*. However, model features are saved in the model feature coordinate system.

In some cases, the coordinate system used for model features may be based on a different datum than the datum used in the *Map view* coordinate system. In these cases, a datum shift can be applied to locate model features correctly in the *Map view*. Two approaches are available: a three-parameter Molodensky transformation, and a seven-parameter 3D Helmert transformation. Details of the two approaches are documented in the geodesy manual supplied with the installation (MIKE_Zero_Geodesy.pdf).

See also: http://www.arcwebservices.com/arcwebonline/services/dat-trans.htm.

A different coordinate system for model features should be considered if the study area is located in an area where distances are distorted significantly in the *Map view* coordinate system. For example, using a Sphere Mercator coordinate system in the north of Alaska would result in significant errors in calculated river reach lengths.

If the model working area is located in a region where the risk of significant errors due to distortion is large, the user will receive a validation warning. Validation messages will not be issued unless a working area has been defined.

5.2 Background Map

A background map can be selected in the Setup tree view. Three options are available: None, a Google map background, and a Countries/Coastline shapefile. If the Google map background is selected, the user is prompted to select from a drop-down list of available map types. Both the Google map and Countries/Coastline shapefile can be disabled in the *Map view* by unchecking the Visible check box.

Note: In order to use Google map background, an internet connection must be available. Google maps images are automatically rectified to the *Map view* coordinate system.

5.3 Background Layers

Background layers including shape files, images, and rasters can be added in the Setup tree view. To add a new background layer, click on the **+** icon and select the layer type.

Image area coordinates can be read from a world file or else specified by the user. If using a world file, the world file must be located in the same directory as the image file, with the same name. Otherwise, image coordinates can be entered by hand.

Shape file background layers require specification of a projection type. If a projection file (PRJ file) exists, the projection type is automatically read. Otherwise, a user-specified map projection must be selected.

The raster background layer is used when the user would like to use a digital elevation model (DEM) or other raster as a background layer in the Map view. Four raster formats can be selected:

- **ESRI ASCII**: a text file format, which requires that the coordinate system as well as the elevation unit are specified manually.
- **dfs2**: for a dfs2 raster file, the coordinate system and units are obtained from the file's metadata.
- **GeoTIFF**: a special format of tiff file, which also stores the coordinate system and units as metadata.
- **ESRI binary grid**: a binary format of raster, composed of multiple files with extension .adf. The raster is actually the folder containing the .adf files.

Background layers are displayed in the *Map view* in the order given in the list of layers found at the bottom of the tabular view (when Background layers is selected in the Setup tree view). The appearance of background layers follows the order in the list of images (i.e. the image in line 1 is on top, then image 2 is below image 1, then image 3 is below image 2, etc.). It is possible to rearrange the order of images by selecting an image and using the arrows to move the selected image up or down in the list.

Display settings for background layers can be modified in the Symbology tree view. In this view, layer transparency settings can be changed so that underlying background layers are visible in the *Map view*. It is also possible to remove layers from the map view by unchecking the Visible box. Other aspects of symbology can also be modified, such as colour schemes and feature labels. For more details about symbology options, please see the discussion of symbology in Section 2 (Working With MIKE HYDRO Basin).

Note: The layers' data are loaded to the map when the layer is selected or when the MIKE HYDRO project file is opened. If the content of the layer is changed in another software product while the layer is opened in MIKE HYDRO, it is possible to refresh the map view using the 'Update' function available when clicking on the '…' button next to the file name.

5.4 Digital Elevation Model (DEM)

A Digital Elevation Model (DEM) may be included in the model setup for river tracing and catchment delineation.

When a DEM is to be used, it must be selected amongst the list of raster files loaded as background layers. Therefore, any DEM must first be added in the Background layers menu before it can be used as a DEM for e.g. river tracing or catchment delineation.

5.4.1 River tracing and catchment delineation

If the checkbox, 'Use DEM for river tracing and catchment delineation' is selected, the DEM can be used to delineate rivers and catchments based on the elevations in the DEM. Delineation is carried out using the 'Add trace' and 'Add delineate' tools in 'Branch' and 'Catchment' ribbons. More details about these tools are available in Section 6.4 (Alignment lines) and Section 7 (Catchments). Before these tools can be used, however, the DEM must be processed.

Processing elevation data

The DEM is preprocessed to create a raster file with flow direction information that is used to delineate river branches and catchments.

Note: It is recommended that the size of the DEM should not exceed 25 million pixels. A warning message will be shown if the size of the DEM is above this limit.

The size of the DEM can be reduced by changing the spatial extent or by resampling.

Spatial extent

The spatial extent of the DEM can be limited to the defined working area (see below), thereby reducing the number of pixels.

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.

Preprocess digital elevation data

In order to use the river tracing and catchment delineation on the map, the digital elevation data must be preprocessed by clicking the button 'Preprocess digital elevation data'. After changing the spatial extent or resampling the elevation data, the DEM must be preprocessed again.

The processed data is saved in a ppdd-file. If no location is specified, then the ppdd file is saved by default in the same location as the model setup. Click the '…' button to change the location of the ppdd file, or to export its data to a .dfs2 file format. The latter can be used to view its content for general information about the preprocessing results.

Many DEMs contain inaccuracies because of their resolution or artifacts resulting from the method of creation. When delineating catchments and rivers from the DEM, you may find some unexpected results. This can happen in flat areas where DEMs may not have the vertical resolution to correctly delineate the rivers and catchments.

Note: In many digital elevation models, all pixels in lakes have undefined (or blank) pixel values.

If the option '*Assume internal undefined areas as local depressions*' is set, all internal undefined areas are handled as a local depression. When a traced branch is added, an artificial channel will be traced through the depression along the shortest path between the upstream and downstream ends. For example, if a lake is represented in a DEM as an undefined area, then an artificial flow path will be created through the lake along the shortest path to the outlet.

If undefined areas are not set as local depressions, a traced branch will stop upon reaching an undefined area.

5.5   Working Area

A working area may be defined by clicking '*Draw rectangle on the map*' and dragging a rectangle on the map. Once defined, the working area may be edited (and edits must be saved) but it cannot be deleted.

If no working area is defined, the maximum extent of features is used as default working area.

6   River Network

The River network forms the basis of all Basin applications. The river network is defined as a combination of connected river segments and computational nodes, and module specific features are then generally added to the model using the river network as a base (river network example illustrated in Figure 6.1).

Figure 6.1   Example of river network in MIKE HYDRO.

Additional river network details:

- **Computational points** comprise a number of different features for which points are inserted in the network when these are defined (e.g. catchment nodes for catchment inflow and river nodes where branches are connected.

- Inflow to the River network is provided by Catchments (see Catchments).

This chapter describes the following topics in further detail:

- Creating a River Network
- Branches
- River Nodes
- Priority Nodes
- Expression Editor

6.1   Creating a River Network

The Branches segments of a River network including associated River nodes and Branch connections can be created in three different ways:

- Method 1: Digitising a River network
- Method 2: Importing a River network
- Method 3: Derive the River network from a DEM

In either way, it is recommended to reference your network geographically, so that features are located at their correct geographical coordinates (cf. Section 5.1 Coordinate System *(p. 49)*).

6.1.1   Method 1: Digitising a River network

For schematic models and small river networks, the network can be digitised directly on the map. A background map (cf. Section 5.2 Background Map *(p. 50)*) can be selected to assist when digitising.
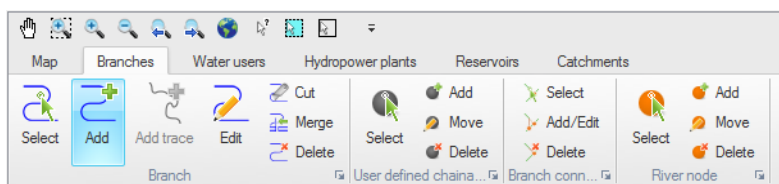
Figure 6.2   Add Branches and Branch Connections

To digitise a Branch, go to *Map view*, select the *Branches* ribbon and click on the '*Add*' button. Use the left mouse button and click at each point along the river. To finish the Branch, double-click on the last point.

To edit the shape of an existing branch click on the 'Edit Branch' button and click on the branch to be edited. All points on the branch are shown with a small square. To add a new point click on branch at the location of the new point. To move an existing point click on the square and drag to the new location. To remove an existing point double-click on the point.

Branches may be connected by using the '*Add/Edit Branch Connection*' button.

Note that the snap distance has been hard-coded as 5 pixels.

The Basin module requires that Branches and Connections are always digitise from upstream to downstream.

If you make a mistake while you are still digitising you can use the *Esc* button to cancel the operation.

6.1.2 Method 2: Importing a River network

The River network may be imported from an existing shape file by selecting '*Import*' in the *File* drop-down menu and clicking in '*Network from shape file*'. This opens the '*Import Branches*' dialogue.

Click on the [...] button and browse to the existing shape file. Branches can be auto-connected by enabling '*Auto connect branches*' and specifying a maximum distance for auto-connection.

Note: Existing MIKE 11 models can be imported by selecting '*Import*' in the *File* drop-down menu and clicking in '*Network from MIKE 11*'.

6.1.3 Method 3: Derive the River network from a DEM

A branch may be added using a DEM see Section 5.4 (Digital Elevation Model (DEM).

When a DEM has been loaded the 'Add trace' button is enabled on the *Branch* ribbon. Clicking on the upstream location of the river will trace down to the outlet of the river based of the DEM data. Subsequent branches will be traced down to the point where they flow into existing river branches.

Note: In local depressions (areas with local minimum values, e.g. lakes or flat areas) the tool connects the 'inflow' and the 'outflow' of the depression using an algorithm that ensures the river is traced within the depression. This line may not represent the actual location of the river.

Moving the cursor over the map will automatically display the traced river according to the cursor position.

6.2 Branches

The *Branches* properties dialogue contains the following tabs:

- General
- User defined chainage points

6.2.1 General

The properties of Branches are specified under the *General* tab. The lower part of the dialogue window contains a list of all Branches in the River network. Select a Branch from the list and specify the properties.

Definitions

Branch name
A Branch can be given any name.

Identifier

The branch identifier is an automatically generated - and unique - parameter string used internally in MIKE HYDRO Basin to link Basin features to specific river branches. The identifier is a fixed, non-editable variable.

Start chainage

The chainage of the first point in the branch. To change this chainage, it is required to insert a User defined chainage point in the first point of a branch. The chainage can then be specified under the *User defined chainage points* tab (see below).

End chainage

The chainage of the last point in the branch. To modify this chainage, a User defined chainage point must be inserted in this last point of the branch. The chainage can then be specified under the *User defined chainage points* tab (see below).

Connections

Information about Branches and their upstream and downstream linkages are provided.

Edit coordinates of vertices

In some cases it might be necessary to update the coordinates on the map view of the vertices of one or more branches.

When a specific branch is selected in the table and the "Edit coordinates of vertices…" button is pressed, a table-dialogue pops up. This table contains the X and Y coordinates of the points of the selected branch, which can be edited (copy and paste is also supported). No new rows can be added, nor deleted. When the update is done and OK is pressed, the map view is updated.

Load shape

It is possible to import/load the shape of an existing branch directly. Select/activate the actual branch in the tabular view and Click '*Load shape*', select the relevant shape file and choose the shape item that represents the branch.

Note: this functionality will only edit the shape of the active branch on the map. It will not create any new branch. Instead, it is possible to import and create one or multiple branches at once from a shape file. To achieve this, please see section Import from Shapefile, page 195.

6.2.2   User defined chainage points

A user defined chainage point is a specific location with a controlled chainage.

If a branch does not contain any user defined chainage points the start chainage is per default 0 and the end chainage equivalent to the length of the branch calculated based on the geographical length of the branch shape. Both the start and end chainage can be controlled by inserted user defined chainage points.

The chainage along a branch between two user defined chainage points are linearly calculated between the two defined chainage values. The chainage along a branch between a user defined chainage point and a branch end is calculated is calculated as the geographical distance from the defined chainage value.

User defined chainage points are added to the River network by using the *'Add'* button in the '*User defined chainage point*' section under the *Branches* ribbon in the *Map view*.

If user defined chainage points have been added in *Map view* their chainage may be edited under the *User defined chainage points* tab.

Note: If a user defined chainage point is moved the defined chainage is fixed and hence moving the point will update all chainages along the branch.

Example: *A branch without user defined chainage values are created with a geographical length of 1000 m. The start chainage will be 0 and the end chainage will be 1000.*

A user defined chainage point is added at 500 m (geographically) along the branch and given the chainage 750. The start chainage is now automatically updated to 250 and the end chainage to 1250.

Another user defined chainage point is added at 250 m (geographically) along the branch and given the chainage 100. The start chainage is now automatically updated to -150 and the end chainage is unchanged at 1250.

6.3   River Nodes

River nodes are automatically inserted, but may also be added manually by using the *'Add'* button under the *River nodes* ribbon in the *Map view*.

There are several types of River nodes in the Basin module:

- *Regular river nodes* are inserted automatically in all confluences and diversions (including to/from 'Connections').

- *Catchment nodes* and *Reservoir nodes* are inserted automatically when Catchments and Reservoirs, respectively, are added to the River network.

- *ReservoirCatchment* **nodes** are inserted automatically when Reservoirs are located on Catchment nodes.

Other River node types are created when a regular river node is given additional properties such as Priority or Routing River nodes.

In the *River nodes* properties dialogue, three types of information can be specified:

- Flow loss time series

- Flow capacity time series

- Bifurcation type and associated data

If one or more Water users are connected to a River node, the node is automatically defined as a Priority node and additional information must be specified in the *Priority node* properties dialogue (cf. Section 6.4).

6.3.1   Flow loss time series

River Branches may lose water due to seepage and /or evaporation. If these processes are considered to be of importance they may be included in the model as a time series that specifies the losses. Flow loss is applied to the flow leaving the river node. To model flow loss through an entire river Branch, flow loss must be specified for all River nodes in that Branch. Both seepage and evaporation can be specified as a fraction of the actual flow (dimensionless), or as flux (volume per time).

To add a flow loss time series, enable '*has flow loss*' and click on the [...] button and click on *Browse...* to select an existing flow loss time series or click on *Create a new file…* to open the *Create a new dfs0 file* dialogue.

If groundwater is defined in the Catchment where the River node is located, seepage loss is added to this groundwater storage. Seepage loss from a Catchment node, however, is added to the immediate downstream Catchment, if present. Otherwise, seepage flow is lost.

6.3.2   Flow capacity time series

Branches may be assigned a flow capacity [Volume per time] that can never be exceeded (relevant for pipes and closed channels). The flow capacity is restricting the flow leaving a River node. To model flow capacity in an entire river Branch, flow capacity must be specified for all River nodes in that Branch.

To add a flow capacity time series, enable '*has flow capacity*' and click on the [...] button and click on *Browse...* to select an existing flow capacity time series or click on *Create a new file…* to open the *Create a new dfs0 file* dialogue.

If the flow capacity of a Branch is reached, the model will attempt to force the water in an alternative direction. If this is not possible, the simulation will terminate with a message.

6.3.3 Bifurcation type and associated data

(Only for Bifurcation nodes).

A river Branch can be divided into maximum two downstream river Branches at a node. Such a node is interpreted as a Bifurcation node and allows bifurcation properties to be specified.

Note that a Bifurcation node cannot be used as an extraction point.

There are two ways of specifying bifurcation:

1. Specifying bifurcation in a *Bifurcation time series* file, where the flow to the minor Branch (the diverted water) is a function of time (though limited by the available net flow to the bifurcation node). When choosing this option, you must supply a time series of bifurcation 'demand'.

2. Specifying a *Bifurcation table*, where the flow to the minor Branch (the diverted water) is a function of the net flow to the Bifurcation node. When choosing this option, you must supply a table that defines this relationship.

Note to avoid an error due to the need for extrapolation, the table should have (0,0) as its first row and a large value in the first column of the last row

To add a *Bifurcation time series* or *Table* click on the [...] button and click on *Browse...* to select an existing Time series/Table or click on *Create a new file…* to open the *Create a new dfs0 file* dialogue.

Note that in order to specify properties for the minor Branch (e.g. flow loss, flow capacity and routing) a River node must be inserted at the beginning of the minor Branch.

6.4 Priority Nodes

A Priority node is a river node from which water is extracted to one or more water users. Priority nodes are automatically added to the *Priority nodes* properties dialogue.

In the *Priority nodes* properties dialogue two types of attribute data is specified:

- Minimum flow
- Priorities

To specify properties for a particular Priority node, select the node from the list of priority nodes.

6.4.1 Minimum flow

A Minimum flow time series can be specified for a Priority node. Minimum flow is the river discharge that must be allowed to pass the node (passing flow) before any water is extracted for Water users connected to the node. To add a Minimum flow time series, enable *Minimum flow (passing flow)* at *priority node;* click on the [...] button and click *Browse...* to select an existing minimum flow time series or click *Create a new file…* to open the '*Create a new file*' dialogue.

Note that the Minimum flow rule is a local rule. It is not to be used to control the discharge in remote locations. It cannot be specified for the outlet node.

6.4.2 Priorities

When more than one Water user is connected to the Priority node, a set of rules that defines how the available water is allocated in case of water shortage is required. Two different options are available:

1. **Supply by priority**: Each of the Water users is getting their demand fulfilled in order of priority, i.e. each water user is assigned a priority and the demands will be fulfilled according to the assigned priorities.

2. **Supply by fraction of flow**: Each Water user is assigned a fraction of the available water at the priority node. The sum of the assigned fractions must be equal to one.

If the *Supply by priority rule* is selected, a priority number has to be specified for each Water user. The number 1 specifies the highest priority, the number 2 the second highest priority and so forth. By default, Water users are given a priority according to the sequence the connections are digitised. The priority can be changed in the priority column.

If the *Supply by fraction of flow* is selected, a time series file needs to be specified for each water user. The time series must contain the fraction of the flow to be supplied to the Water user.

It is possible to use a combination of *Supply by priority rule* and *Supply by fraction of flow* by assigning the same priority to one or more groups of water

supply priorities and then specifying the fraction of flow required from each water supply within the 'shared priority' group. The fractions for all water supplies within a 'shared priority' group must not exceed one at all times.

Example

In a Priority node, water is distributed to three Water user nodes. The available flow in the river is constant equal to 10 m3/s at the river node. The three water user nodes have the following water requirements: the City (W4) requires 5 m3/s, and the Village (W5) and the Industry (W6) each require 4 m3/s. The following rule was specified for the Priority node:

1st priority: City,

2nd priority: Village and Industry, sharing 50/50 (described by a time series).

The simulation yields the following supply to the water user nodes:

City: 5 m3/s, Village and industry: each 2.5 m3/s.

Note that the City is not restricted, whereas the Village and the Industry both experience an equal deficit.

6.5 Routing

In the *Routing method properties* dialogue, routing of flow as well as water level calculations can be included.

Routing can be used to model the hydraulics of upstream tributaries and secondary river branches, where the main concern is to route water to the main river system.

Routing is a simplified hydraulic calculation. Normally, simulation of hydrograph mitigation along a branch is based on the shallow water equations, which require cross section information. However, routing doesn't require such cross sections. The concept of routing is basically transformation of a hydrograph, i.e. calculating the modified hydrograph (different peak value and adjusted shape) at a given location based on the hydrograph at the upstream point and simple parameters.

6.5.1 Routing method

Routing is assigned to a river node and applied to flow leaving that river node. To add routing to a river node click on the append button $^+$ at the top of the table.

Location

Then select the branch on which the river node is located and select the river node. To enable routing through an entire river branch, routing must be specified for all river nodes in that branch.

Flow routing method

A number of routing options are available from a selection list. The below listed three routing options are supported by the Basin module:

- Linear reservoir routing
- Muskingum routing
- Wave translation

Linear reservoir routing

Linear reservoir routing distributes flow leaving the river node over all time steps. When linear reservoir routing is selected a delay parameter K (the linear routing time lag), must be specified.

Delay parameter
The delay parameter K specifies the time for the incremental flood wave to traverse the river between the selected river node and the next downstream node. Its value may be estimated as the observed travel time of the flood peak between the nodes.

For a pulse inflow, outflow peaks after a specified time given by the time lag, and then decays exponentially. The formula used is:

$$qo = (1 - T/(dt/K)) \cdot qi + T \cdot s \quad \text{with} \quad T = 1 - e^{(-dt/K)} \tag{6.1}$$

where qo is outflow from the node, dt is time step length, qi is inflow to the node, s is storage in the subsurface, and K is the linear routing time lag (or delay parameter). T [-] is an intermediate result.

This algorithm includes damping. For a given time step, river storage is updated based on the following formula:. Variables dt, K and T are as explained previously.

$$Volume_{Outflow} = \left(1.0 - T/\left(\frac{dt}{K}\right)\right) Volume_{Inflow} + T \cdot Storage \tag{6.2}$$

$$\Delta Storage = Volume_{Inflow} - Volume_{Outflow} \tag{6.3}$$

Note that for linear reservoir routing, river storage is a virtual quantity that can be negative.

Muskingum routing

When selecting Muskingum routing a delay parameter K and a shape parameter X must be specified.

Delay parameter

The K parameter specifies the time for the incremental flood wave to traverse the river between the selected river node and the next downstream node. Its value may also be estimated as the observed travel time of the flood peak between the nodes.

Shape parameter

The shape parameter, X (dimensionless) depends on the shape of the modelled wedge storage. In natural rivers, X has a value between 0 (reservoir-type storage) and 0.3 with a mean value of 0.2 (X must always be less than 0.5 (full wedge storage)). Greater accuracy in determining X may not be necessary because the results are relatively insensitive to the value of this parameter.

Note that Muskingum routing can only be used when simulation time step length (dt) is within the following range: $2Kx < dt < 2K(1-x)$. If simulation time step length is outside this range Muskingum routing becomes unstable and is automatically replaced with linear reservoir routing. Linear reservoir routing is a special case of Muskingum routing with shape parameter $x = 0$. Unlike general Muskingum routing, linear routing is defined for all combinations of time step lengths and K values.

The Muskingum method is a commonly used hydrologic routing method for handling a variable discharge-storage relationship. This method models the storage volume of flooding in a river channel by a combination of wedge and prism storage. During the advance of a flood wave, inflow exceeds outflow, producing a wedge of storage. During the recession, outflow exceeds inflow, resulting in a negative wedge. In addition, there is a prism of storage, which is formed by a volume of constant cross section along the length of the prismatic channel.

Assuming that the cross-sectional area of the flood flow is directly proportional to the discharge of the section, the volume of prism storage is equal to KQ, where K is a proportionality coefficient, and the volume of wedge storage is equal to $KX(I - Q)$, where X is the shape parameter. The total storage S is therefore the sum of two components.

$$S = KQ + KX(I - Q) \tag{6.4}$$

This expression can be rearranged to give the storage function for the Muskingum method:

$$S = K[XI + (1 - X)Q]$$ (6.5)

which represents a linear model for routing flow in streams.

The values of storage at time *j* and *j+1* can be written, respectively, as

$$S_j = K[XI_j + (1 - X)Q_j]$$ (6.6)

$$S_{j+1} = K[XI_{j+1} + (1 - X)Q_{j+1}]$$ (6.7)

Using equations (6.6) and (6.7), the change in storage over time interval is:

$$S_{j+1} - S_j = K[XI_{j+1} + (1 - X)Q_{j+1}] - K[XI_j + (1 - X)Q_j]$$ (6.8)

Wave translation

The Wave translation algorithm basically uses a cyclical buffer with 'slots' for every time step. The inflow at a time step is put into the current slot, and the corresponding earlier inflow that was stored in that slot is pulled out. The index of the current slot cycles within the buffer, such that a new inflow always replaces the 'oldest' previous inflow. The number of slots in the buffer is equal to the number of time steps that a flow gets delayed with. The number of slots is computed as floor dt/K, where *K* can vary among reaches, and dt [time] is the simulation time step. The latter must be constant during a simulation; for months, a standard month length (30 days) is assumed.

Delay parameter

The *K* parameter specifies the time for the incremental flood wave to traverse the river between the selected river node and the next downstream node. Its value may also be estimated as the observed travel time of a distinct hydrograph peak between two nodes.

6.5.2 Water level Calculation

In most applications, the Basin module is used only to calculate flows, not water levels. The Basin module is not a hydraulic model and thus cannot be used for proper flood modelling. The River module is more suited for such purposes. However, Water level calculations can be done in the Basin module. If the Water Quality module has been enabled water level calculations are required.

Two methods are available for Water level calculations:

- Rating curve approach (*Q-h* table)

- Manning formula.

Both apply for steady-state flow and are thus approximations, but they are usually reasonable estimates as long as the water level does not change rapidly (e.g. during a flood). The items in the dialogue window change depending on the option chosen.

Rating curve approach

When the Rating Curve approach is used, it looks up the water level in a user specified *Q-h* table. The table must cover the range of discharges encountered during a simulation.

Manning formula

The Manning formula is:

$$Q = 1/n \cdot A \cdot R^{(2/3)} \cdot \sqrt{S} \tag{6.9}$$

Where *n* is the Manning's number n, *A* is the cross-sectional area, *S* is the water surface slope, *R* is the resistance radius:

$$R = \left(1/A \cdot \int_0^B (h(b)^{(3/2)} db) \right)^2 = (1/A \cdot h^{3/2} \cdot B)^2 \tag{6.10}$$

assuming a rectangular cross-section with width *B*, and *h(b)=h*

so *R* can be written as:

$$R = (1/h \cdot h^{3/2})^2 \quad \text{as} \quad B/A = 1/h$$

and

$$Q = 1/n \cdot h \cdot B \cdot h^{(2/3)} \cdot \sqrt{S} = 1/n \cdot h^{(5/3)} \cdot B \cdot \sqrt{S} \tag{6.11}$$

Thus water level can be computed as:

$$h = ((Q \cdot n)/(B \cdot \sqrt{S}))^{(3/5)} \tag{6.12}$$

The parameters to be specified for the Manning formula are already specified elsewhere for the River Module, but must be specified here for the Basin Module:

Manning's number
Here the Manning number must be specified.

Slope

Here you can enter the slope. Generally, it is a good approximation to equate surface and channel slope.

Max level (optional)

Optionally, you can also specify a maximum water depth up to which results from the Manning formula are accepted. If you specify such a (non-zero) value, water levels will never exceed it.

Width

Cross section width B, used in the water level calculations. It is assumed that the cross-section is rectangular.

7 Catchments

Catchments are generally included in a model to provide catchment runoff as an inflow to the River network. A model may contain any number of catchments.

Catchments may be represented schematically, or by their delineated boundaries as illustrated in figure below. The primary difference between schematic and delineated catchments in a model setup is, that the catchment surface area is directly derived from the delineation of catchments whereas the surface area must be specified manually for the schematic catchments.

Figure 7.1   Illustration of catchments in a MIKE HYDRO model

Runoff from catchments are either user-defined through a predefined runoff time series file or it can be calculated using one of several rainfall runoff models available. Runoff from a catchment is added to a river network in catchment nodes.

Additional features and calculation options related to catchments include groundwater and definition of sediment load as input to a reservoir sedimentation calculation.

Creating a catchment

There are several ways of creating catchments, each of them being related to a specific *Add* button under the *Catchments* ribbon in *Map view*.
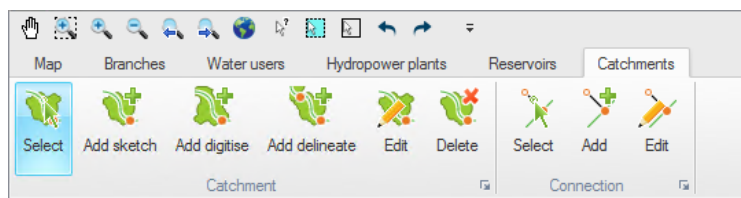
Figure 7.2   Catchment ribbon in Map view

MIKE HYDRO offer multiple options for catchments definitions

- **Sketched Catchments** are created by selecting the '*Add sketch*' button and clicking on the branch at the downstream end of a catchment. Here a catchment node will be automatically inserted. The upstream end of a catchment is given by the next upstream catchment node or the end of the branch.

- **Delineated catchments** may be created by using a DEM (cf. Section 5.4 (Digital Elevation Model (DEM)). This type of catchment is added by pressing the '*Add delineate*' button and clicking on the branch at the downstream end of a catchment. The catchment is automatically deline-ated based on the elevations from the DEM, and a catchment node is automatically inserted.

- **Digitized catchments** may be drawn on the map using the '*Add digitize*' button. For this type of catchment, the connection to the branch has to be manually defined with a connection. The catchment connections can be created and edited using the buttons from the *Connection* group in the ribbon.

Note that a catchment created from the map, regardless of its type (sketched, delineated or digitized), can have its shape corrected from a shape file using the '*Load shape*' button in the *Tabular* tab. This however does not change the catchment connection or the catchment node.

Note that delineated catchments starting in a local depression (area with local minimum values) can have an unexpected result as an artificial flow pattern in the lake is used for defining the catchment boundary.

If '*Ctrl*'-button is pressed when inserting the catchment outlet the catchment will include both the entire lake and the catchment to the lake.

Editing a catchment

The '*Edit*' button from the *Catchment* group in the ribbon may be used to edit the geometry of a catchment on the map. After selecting this button and click-ing on the desired catchment, all the vertices defining the polygon of the catchment become visible.

To move a vertex, click and drag the selected vertex.

To add a new vertex, click on the desired location along the edge of the catchment.

To delete a vertex, double click the selected vertex.

Note that to delete multiple vertices at the same time, zoom out in such a way that these vertices appear almost superimposed. After double clicking, all the vertices in the vicinity of the selected location will be deleted.

Deleting a catchment

To delete a catchment from the map, select the '*Delete*' button in the *Catchment* group in the ribbon and click on the desired catchment.

To delete one or more catchments from the *Tabular* tab, highlight the catchment(s) to be deleted in the overview table and press the '*Delete*' button above the table.

Connecting a catchment

The three buttons from the *Connection* group in the ribbon are described below:

Select. This button can be used to select one or more catchment connections. To select multiple connections, press the '*Ctrl*' key on the keyboard while selecting the items on the map.

Add. Select the '*Add*' button to connect a catchment to a branch, or to connect it to a new location. After pressing the '*Add*' button, click on the border of a catchment and drag to the branch. A catchment node is added at the connected location on the branch. **Edit**. This button may be used to move the connection to another location. To edit the connection, first click the '*Edit*' button and click once on the connection, then drag the white symbol to its new location. Click once again to stop editing. When a catchment is connected to a new location on a branch, a new catchment node is added to this branch.

7.1   Catchment definitions

Depending on the selected options, the *Catchment definitions* dialogue may contain the following tabs:

- General
- Groundwater
- Rainfall-Runoff
- Sediment Load

7.1.1 General

Under the *General tab* the following catchment properties are specified:

- Area
- Groundwater model
- Rainfall-runoff model
- Runoff time series
- Calibration plot
- Load shape

Area

When catchment runoff is specified as specific runoff rates (e.g. litres/sec/m^2) an accurate specification of the catchment area is required in order to calculate the runoff volume (area * specific runoff). When the runoff volumes are specified as discharges (e.g. m^3/s), the catchment area is not used.

Catchment areas can either be assigned manually, or calculated automatically based on the catchment shapes on the map by enabling '*Use catchment shape for area calculation*'. By default the calculated catchment areas are used. For sketched catchments a manual specification of the catchment area is normally required, whereas for delineated catchments the calculated area can normally be used. To manually assign catchment area, the option '*Use catchment shape for area calculation*' must be disabled.

Groundwater model

Three choices are available for the groundwater modelling of the catchment:

- No groundwater model included
- Single-layer groundwater model
- Two-layer groundwater model

A groundwater model can only be selected if groundwater has been included in the simulation. By default the menu assumes that there is no groundwater model for the catchment. In such a case groundwater can only be used if the option '*Unlimited groundwater in catchments without groundwater model*' is selected/enabled.

If a groundwater model is selected, groundwater parameters must be specified under the groundwater tab.

Rainfall-runoff model

By default a rainfall-runoff model is not selected. A rainfall-runoff model can only be selected if the rainfall-runoff module has been included in the *Modules* menu.

The following types of rainfall-runoff models are offered:

- NAM
- UHM

Runoff time series

If a rainfall-runoff model has not been selected a runoff time series must be specified for the catchment. Catchment runoff can be specified either as *Specific runoff* (volume per time per unit area) or as *Discharge* (volume per time).

To specify the runoff time series click on the [...] button and click *Browse...* to select an existing time series or click *Create a new file…* to open the '*Create a new file*' dialogue.

When a groundwater model has been included, total catchment runoff is calculated by adding baseflow to the user-specified runoff time series. The user specified runoff is not routed through the groundwater aquifers, but represents the fast catchment response. In this case it is important that user-specified runoff time series only include the surface/interflow components of the hydrograph and not the baseflow component. If baseflow is included in the user-specified runoff time series, baseflow will be overestimated.

When a rainfall-runoff model is selected, catchment runoff is automatically calculated and the baseflow component is only included if no groundwater model is present in the catchment.

If the specified time series does not cover the entire simulation period, it will automatically be recycled (**cf. 'Time Series' in MIKE Zero**).

Calibration plot

It is possible to save the calibration plot by enabling '*Calibration plot*'. The calibration plot shows observed and simulated runoff for the catchment, cf. Section 7.1.3.

Load shape

It is possible to import/load the shape of a catchment directly. Click '*Load shape*', select the relevant shape file and choose the shape item that represents the catchment.

Note: this functionality will only edit the shape of the active catchment on the map. It will not create any new catchment. Instead, it is possible to import and

create one or multiple catchments at once from a shape file. To achieve this, please see section Import from Shapefile.

7.1.2 Groundwater

It is assumed that the lateral boundaries of subsurface (groundwater) and surface catchments are the same. The groundwater storage (aquifer) is conceptualised as a linear reservoir model (cf. "The Linear Reservoir Model" on page 211) with one or two layers. The conceptual structure of the two-layer groundwater component is shown in Figure 7.3. A single layer model only include the shallow (upper) aquifer.
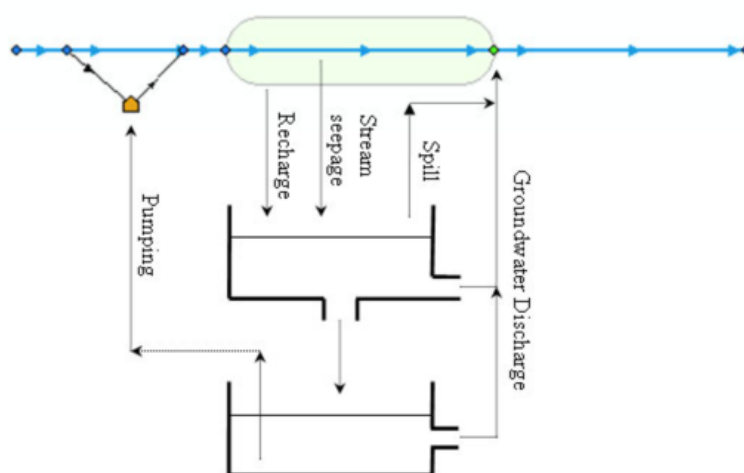
Figure 7.3   Conceptual structure of the groundwater component

As illustrated in Figure 7.3 groundwater interacts with the surface water via groundwater recharge, groundwater discharge and seepage from river branches, reservoirs and connections. Moreover, when the water table of the shallow (upper) aquifer reaches the land-surface, it starts to spill directly into the river. Finally, groundwater from the deep aquifer can be pumped by water users.

Note that groundwater pumping is assumed to take place from the deep aquifer unless a one-layer aquifer has been specified (then pumping takes place from the shallow aquifer).

Under the *Groundwater tab* three types of groundwater properties/parameters are specified:

- Depths
- Time constants

- Groundwater recharge time series

Properties can only be specified if a groundwater model has been selected under the general tab.

Depths

Initial water table depth

The initial water table depth (relative to ground surface) determines the magnitude of the groundwater discharge and the available water for pumping in the initial period of simulation. Depending on the time constant specified the initial depth may influence the results during a few days or up to several months of the simulation.

Outlet depth

The outlet depth (relative to ground surface) determines the storage capacity of the shallow aquifer and the storage capacity available for baseflow generation in the deep aquifer. For shallow aquifers the water table can vary between the outlet depth and ground surface (depth = 0 metres) at which time overflow of groundwater occurs to the river. For deep aquifers (only relevant if a two-layer groundwater model is selected) with no pumping, the water table (used for calculating baseflow) can vary between the outlet depth of the deep aquifer and the outlet depth of the shallow aquifer, at which time overflow of groundwater from the deep to the shallow aquifer occurs.

Bottom level (only relevant for deep aquifer)

Bottom level is the lowest level (largest depth relative to ground surface) from which water can be pumped from the deep aquifer. This indicates the total storage available for usage from the deep aquifer.

Time constants

A time constant (cf. 'The Linear Reservoir model' in the Appendix) is used to calculate the rate at which groundwater discharges into the river (as baseflow). The larger the time constant the lower the rate of discharge. A high time constant reflects a more constant baseflow contribution.

When a two-layer model is selected, separate time constants must be specified for the *Shallow aquifer* and the *Deep aquifer*. Additionally, a time constant must be specified for the *interface between shallow and deep aquifers.* This constant is used to calculate the rate at which water percolates from the shallow to the deep aquifer.

Groundwater recharge time series

A groundwater recharge time series must be specified if no rainfall-runoff model is selected under the *General tab*.

Click on the [...] button and click *Browse...* to select an existing time series or click *Create a new file…* to open the '*Create a new file*' dialogue.

Groundwater recharge is automatically calculated when a rainfall-runoff model is selected.

7.1.3 Rainfall-Runoff

Rainfall-runoff modelling is a first stage of the simulation that creates the run-off time series for the individual catchments. The simulation of the flow in the river branches is performed at a second stage.

- There are several rainfall-runoff models available:

- **NAM**. A lumped, conceptual rainfall-runoff model, simulating the overland flow, interflow, and baseflow components as a function of the moisture contents in four storages.

- **UHM**. The Unit Hydrograph Model includes different loss models (constant, proportional) and the SCS method for estimating storm runoff.

Note: The rainfall-runoff tab is available only if the rainfall-runoff module is activated in the *Modules* menu (4.1) and if a rainfall-runoff model has been selected for the catchment in the *General* tab.

7.1.4 NAM

The NAM model is a deterministic, lumped and conceptual rainfall-runoff model accounting for the water content in up to four different storages. NAM can be prepared in a number of different modes depending on the requirement. As default, NAM is prepared with nine parameters representing the *surface zone*, *root zone* and the *groundwater* storages.

In addition, NAM contains provision for:

- Extended description of the groundwater component.
- Two different degree day approaches for *snow melt*.

The NAM Rainfall-runoff component is accessed once the NAM model is selected under the *General* tab.

More details on the NAM model can be found in the 'MIKE 1D Reference' manual.

In the NAM model, the following sections can be found:

- Surface-rootzone
- Groundwater
- Snow melt
- Elevation zones

- Irrigation

- Initial conditions

- Autocalibration

- Seasonal variation

- Time series

The parameters of each section are specified for each representative catchment.

The NAM Rainfall-runoff simulation covers the period as specified in the *Simulation period* dialogue ("Simulation Period" on page 45).

Surface-rootzone

Storages

- **Maximum water content in surface storage (Umax)**. It represents the cumulative total water content of the interception storage (on vegetation), surface depression storage and storage in the uppermost layers (a few cm) of the soil. Typical values are between 10-20 mm.

- **Maximum water content in root zone storage (Lmax)**. It represents the maximum soil moisture content in the root zone, which is available for transpiration by vegetation. Typical values are between 50-300 mm.

Runoff parameters

- **Overland flow runoff coefficient (CQOF)**. It determines the division of excess rainfall between overland flow and infiltration. Values range between 0.0 and 1.0

- **Time constant for routing interflow (CKIF)**. It determines the amount of interflow, which decreases with larger time constants. Values in the range of 500-1000 hours are common.

- **Time constants for routing overland flow (CK1, CK2)**. They determine the shape of hydrograph peaks. The routing takes place through two linear reservoirs (serially connected) with different time constants, expressed in [hours]. High, sharp peaks are simulated with small time constants, whereas low peaks, at a later time, are simulated with large values of these parameters. Values in the range of 3-48 hours are common.

- **Root zone threshold value for overland flow (TOF)**. It determines the relative value of the moisture content in the root zone (L/Lmax) above which overland flow is generated. The main impact of TOF is seen at the beginning of a wet season, where an increase of the parameter value will delay the start of runoff as overland flow. Threshold value range between 0 and 70% of Lmax, and the maximum value allowed is 0.99.

- **Root zone threshold value for interflow (TIF)**. It determines the relative value of the moisture content in the root zone (L/Lmax) above which interflow is generated.

Groundwater

For most NAM applications only the *Time constant for routing baseflow* (CKBF) and possibly the *Root zone threshold value for groundwater recharge* (TG) need to be specified and calibrated. However, to cover also a range of special cases, such as groundwater storages influenced by river level variations, a number of additional parameters can be modified.

The groundwater parameters are:

- **Root zone threshold value for GW recharge (TG)**. It determines the relative value of the moisture content in the root zone (L/Lmax) above which groundwater (GW) recharge is generated. The main impact of increasing TG is less recharge to the groundwater storage. Threshold values range between 0 and 70% of Lmax and the maximum value allowed is 0.99.

- **Time constant for routing baseflow (CKBF)**. It can be determined from the hydrograph recession in dry periods. In rare cases, the shape of the measured recession changes to a slower recession after some time. To simulate this, a second groundwater reservoir may be included, see the extended components below.

- **Ratio of GW-area to catchment area (Carea)**. This parameter describes the ratio of the groundwater catchment area to the topographical surface water catchment area, which is specified under the General tab. Local geological condition may cause part of the infiltrating water to drain to another catchment. This loss of water is described by a Carea less than one. Usual value: 1.0.

- **Specific yield of groundwater reservoir (Sy)**. This parameter should be kept at the default value except for the special cases in which the groundwater level is used for NAM calibration. This may be required in riparian areas, for example, where the outflow of groundwater strongly influences the seasonal variation of the levels in the surrounding rivers. Simulation of groundwater level variation requires values of the specific yield Sy and of the groundwater outflow level GWLBFO, which may vary in time. The value of Sy depends on the soil type and may often be assessed from hydro-geological data, e.g. test pumping. Typically values of 0.01-0.10 for clay and 0.10-0.30 for sand are used.

- **Maximum GW-depth causing baseflow (GWLBF0)**. It represents the distance in meters between the average catchment surface level and the minimum water level in the river. This parameter should be kept at the default value except for the special cases in which the groundwater level is used for NAM calibration, cf. Sy above.

- **Seasonal variation of GW-depth causing baseflow**. In low-lying catchments the annual variation of the maximum groundwater depth may be of importance. This variation relative to the difference between the maximum and minimum groundwater depth can be entered if the checkbox is ticked. The monthly values are given relative to the difference between GWLBF0 and GWLBF_min [0-1]. This is done in the *Seasonal variation* page, in the column *Variation of groundwater maximum water depth*.

- **Minimum GW-depth for seasonal variation (GWLBF_min)**. If the Seasonal variation of GW-depth causing baseflow is selected, the minimum GW-depth level [m] has also to be entered for the calculations.

- **Capillary flux, depth for unit flux (GWLBF1)**. It is defined as the depth of the groundwater table generating an upward capillary flux of 1 mm/day when the upper soil layers are dry corresponding to wilting point. The effect of capillary flux is negligible for most NAM applications. Keep the default value of 0.0 to disregard capillary flux.

- **Use abstraction**. If this option is selected, it is possible to specify the groundwater abstraction depth from the catchment.

  The input type of this variable can be:

  – **Time series**. If this option is selected, a time series of abstraction depth has to be entered in the *Time series* page.
  – **Seasonal variation**. This option allows to specify monthly values of abstraction depth in the *Seasonal variation* page.

- **Use lower baseflow, recharge to groundwater (Cqlow)**. The groundwater recession is sometimes best described using two linear reservoirs, with the lower usually having a larger time constant. In such cases, the recharge to the lower groundwater is specified here as a percentage of the total recharge.

- **Time constant for routing lower baseflow (Cklow)**. It is specified for *Cqlow > 0* as a baseflow time constant, which is usually larger than the CKBF.

Snow melt

The snow module simulates the accumulation and melting of snow in a NAM catchment. It is included in the model when the '*Include snow melt*' checkbox is ticked.

Two degree day approaches can be applied: a simple lumped calculation or a more advanced distributed approach. The simple degree-day approach uses only two overall parameters: a *Constant Degree day coefficient* and a *Base temperature (snow/rain)*. The distributed approach allows the user to specify a number of elevation zones within a catchment with separate snow melt parameters, temperature and precipitation input for each zone.

The snow melt module uses a temperature input time series, usually mean daily temperature. This has to be specified in the *Time series* tab.

The snow melt parameters are:

- **Constant Degree day coefficient (Csnow)**. The content of the snow storage melts at a rate defined by the degree-day coefficient multiplied by the temperature difference above the *Base temperature*. Typical values for Csnow are 2-4 mm/day/C.

- **Base temperature (snow/rain) (To)**. The precipitation is retained in the snow storage only if the temperature is below the base temperature, whereas it is by-passed to the surface storage (U) in situations with higher temperatures. The base temperature is usually at or near zero degree C.

It is also possible to specify other input parameters for the snow melt module, if the relative checkboxes are ticked. These are:

- **Variation of degree day coefficient**. This option is selected when the melting coefficient is varying in time, instead of being constant. The input for this model can be specified in two ways:
  - **Specify as time series file**. In this case the melting coefficient is specified in a time series that has to be loaded in the *Time series* section.
  - **Specify as seasonal variation**. If this option is selected, monthly values of the degree day coefficient for snow melt must be specified in the *Seasonal variation* section.

- **Radiation coefficient (Radiation file on time series page)**. This option may be introduced when time series data for incoming radiation is available. The time series input file is specified separately on the *Time series* section. The total snow melt is calculated as a contribution from the traditional snow melt approach based on Csnow (representing the convective term) plus a term based on the radiation.

- **Rainfall degree day coefficient**. This option may be introduced when the melting effect from the advective heat transferred to the snow pack by rainfall is significant. This effect is represented in the snow module as a linear function of the precipitation multiplied by the rainfall degree coefficient and the temperature deviation above the base temperature.

Elevation zones

The *Elevation zones* tab becomes available when snow melt is included in the model. When the '*Delineate catchment into elevation zones for snow modelling*' option is selected, the snow melt distributed approach is applied. Following this approach, the catchment is divided into a certain number of

elevation zones, each with specific snow melt parameters, temperature and precipitation inputs.

The general parameters that have to be specified for the snow melt model are:

- **Number of elevation zones**

- **Reference level for temperature station**. This parameter defines the altitude [m] of the reference temperature station. This station is used as a reference for calculating the temperature and precipitation within each elevation zone (the file with temperature data is specified on the Time series page).

- **Dry temperature lapse rate**. It specifies the vertical gradient [C/100 m] for adjustment of temperature under dry conditions. The temperature in the actual elevation zone is calculated based on a linear transformation of the temperature from the reference station to the actual zone, the correction being defined as the dry temperature lapse rate multiplied by the difference in elevations between the reference station and the actual zone.

- **Wet temperature lapse rate**. This parameter specifies the vertical gradient [C/100 m] for adjustment of temperature under wet conditions defined as days with precipitation higher than 10 [mm]. The temperature in the actual elevation zone is calculated based on a linear transformation of the temperature from the reference station to the actual zone, the correction being defined as the wet temperature lapse rate multiplied by the difference in elevations between the reference station and the actual zone.

- **Reference level for precipitation station**. This parameter defines the altitude, expressed in [m], at the reference precipitation station (the file with precipitation data is specified on the *Time series* page).

- **Correction of precipitation rate**. This parameter specifies the vertical gradient for adjustment of precipitation and is expressed in [percent/100 m]. The precipitation in the actual elevation zone is calculated based on a linear transformation of the precipitation from the reference station to the actual zone, the correction being defined as correction of precipitation rate [percent/100 m] multiplied by the difference in elevation between the reference station and the actual zone.

The specific parameters for each elevation zone are entered in the table at the bottom of the **page**. These are:

- **Zone**. Zone number ID automatically assigned by the programme.

- **Elevation**. The elevation of each zone is specified in the table as the average elevation of the zone. The elevation must increase from zone (i) to zone (i+1).

- **Area**. The area of each zone is specified in the table. The total area of the elevation zones must equal the area of the catchment.

- **Min storage for full coverage**. This parameter defines the required amount of snow to ensure that the zone area is fully covered with snow. When the water equivalent of the snow pack falls under this value, the area coverage (and the snow melt) will be reduced linearly with the snow storage in the zone.

- **Max storage in zone**. This value defines the upper limit for snow storage in a zone. Snow above this value will be automatically redistributed to the neighbouring lower zone.

- **Max water retained in snow**. It defines the maximum water content in the snow pack of the zone. Generated snow melt is retained in the snow storage as liquid water until the total amount of liquid water exceeds this water retention capacity. When the air temperature is below the base temperature To, the liquid water of the snow re-freezes with rate Csnow.

- **Dry temperature correction**. The actual temperature correction for dry conditions to estimate actual temperature for the specific zone.

- **Wet temperature correction**. The actual temperature correction for wet conditions to estimate actual temperature for the specific zone.

- **Precipitation correction**. The relative correction for precipitation, expressed in percent, to estimate the precipitation for the specific zone.

The '*Calculate*' button above the table fills the three columns in the table for *Dry temperature correction*, *Wet temperature correction* and *Precipitation correction*, based on the specified rates above.

Irrigation

An irrigation module is included in the rainfall-runoff model when the '*Include irrigation*' box is ticked. This irrigation module has a different function than the one defined for the irrigation water user. While the latter is used to calculate water balances at the catchment scale, the irrigation module in the rainfall-runoff model is only used for adjusting the water balance in the NAM model. The purpose of the NAM irrigation module is that of simulating the runoff and groundwater recharge/baseflow from the irrigated areas, so that NAM can be calibrated for the non-irrigated part of the catchment.

Minor irrigation schemes within a catchment will normally have negligible influence on the catchment hydrology, unless transfer of water over catchment boundaries is involved. Large schemes, however, may significantly

affect the runoff and the groundwater recharge through local increases in evapotranspiration and infiltration as well as through operational and field losses.

The irrigation module of NAM may be applied to describe the effect of irrigation on the following aspects:

- **The overall water balance of the catchment**. This will be affected mainly by the increased evapotranspiration and by possible external water sources for irrigation.

- **Local infiltration and groundwater recharge in irrigated areas.**

- **The distribution of catchment runoff amongst different runoff components (overland flow, interflow, baseflow)**. This may be influenced by the increased infiltration in irrigated areas as well as by local abstraction of irrigation water from groundwater or streams.

The irrigation parameter is:

- **Infiltration rate at field capacity (K0-inf)**. This parameter defines the rate of infiltration at field capacity [mm/h].

The *Irrigation sources in percent* are:

- **Local groundwater (PC_LGW)**. Percentage of water for irrigation supplied by groundwater sources.

- **Local river (PC_LR)**. Percentage of water used for irrigation supplied by a local river.

- **External river (PC_EXR)**. Percentage of water for irrigation supplied by a river external from the catchment defined in NAM.

Another irrigation option is:

- **Include crop coefficients and operational losses**. If this checkbox is ticked, additional irrigation module parameters can be specified as monthly values in the Seasonal variation page.

  These are:

  - **Irrigation crop coefficient**. This parameter, also defined as Kc, allows to quantify the amount of water required (and transpired) by the crop. Different crops have different crop coefficient. Kc is multiplied by the reference ET of a standard crop (grass) to calculate the water demand of the crop of interest.

- **Irrigation operational and conveyance losses in percent of abstract water**. These represent the system losses of irrigation water for the different components/processes:
Groundwater
Overland flow
Evaporation

Initial conditions

The initial relative water contents of surface and root zone storage must be specified as well as the initial values of overland flow and interflow. Initial values for baseflow must always be specified. When lower baseflow is included, a value for the initial lower baseflow must also be specified.

Initials values of the snow storage are specified when the snow melt routine is used. When the catchments are delineated into elevation zones, the snow storage and the water content in each elevation zone are specified.

The parameters of the different subsections are:

Surface and rootzone

- **Relative water content in surface storage [0, 1] (U/UMax)**. This value ranges between zero and one, where one indicates wet initial conditions and zero dry initial conditions.

- **Relative water content in root zone storage [0, 1] (L/LMax)**. This value ranges between zero and one.

- **Overland flow (QOF)**. The overland flow at the beginning of the simulation, which is normally estimated from the hydrograph $[m^3 \ s^{-1}]$.

- **Interflow (QIF)**. The interflow at the beginning of the simulation, which is normally based estimated from the hydrograph $[m^3 \ s^{-1}]$.

Groundwater

- **Baseflow (BF)**. The baseflow at the beginning of the simulation, which is normally estimated from the hydrograph $[m^3 \ s^{-1}]$.

- **Lower baseflow (BF - low)**. The lower baseflow at the beginning of the simulation, which is normally estimated from the hydrograph $[m^3 \ s^{-1}]$.

Snow storage

If the *Elevation zones* option is not selected:

- **Global value**. This parameter represents the actual initial snow storage in [mm] over the entire catchment.

If the *Elevation zones* option is selected:

- **Zone, Snow storage, and Water in snow**. If the snow model is defined using elevation zones, it is required to enter for each zone the initial *Snow storage* in [mm] and the *Water in snow*, also in [mm], defining the water content in the snow pack.

Autocalibration

It is possible to use an automatic optimisation procedure to calibrate the 12 most important parameters in the NAM model. The calibration routine used in NAM is based on a multi-objective optimisation strategy, the SCE algorithm. The procedure implemented in NAM allows to simultaneously optimise four different calibration objectives or a combination of them. A description of the SCE algorithm is given in the '*Autocal User Guide*'.

To include the Autocalibration routine in NAM, tick the *'Include autocalibration'* checkbox.

Calibration parameters

The parameters which can be included in the autocalibration routine are listed in the *Calibration parameters* table:

- Maximum water content in surface storage (Umax).
- Maximum water content in root zone storage (Lmax)
- Overland flow runoff coefficient (CQOF)
- Time constant for interflow (CKIF)
- Time constant 1 for routing overland flow (CK1)
- Time constant 2 for routing overland flow (CK2)
- Root zone threshold value for overland flow (TOF)
- Root zone threshold value for interflow (TIF)
- Root zone threshold value for groundwater recharge (TG)
- Time constant for routing baseflow (CKBF)
- Lower baseflow, recharge to lower reservoir (Cqlow)
- Time constant for routing lower baseflow (Cklow).

If the '*Fit*' checkbox is ticked, the parameter is included in the autocalibration. The table also contains the following columns:

- **Initial value**. It is the value for the parameter specified in the Surface-rootzone or Groundwater page and used in the first model simulation.

- **Lower Bound** and **Upper Bound** are the minimum and maximum values that the parameter can assume and therefore define its range of variation.

Objective functions

In automatic calibration, the calibration objectives have to be formulated as numerical goodness-of-fit measures that are optimised.

Four calibration objectives are defined as numerical performance measures in the Autocalibration routine of NAM. These are selected and used by ticking the relative checkboxes.

The objective functions are:

- **Overall water balance**. This defines the agreement between the average simulated and observed catchment runoff overall volume error.

- **Overall root mean squared error**. This measure defines the overall agreement of the shape of the simulated hydrograph with the observed one.

- **Peak flow RMSE**. This optimisation measure defines the agreement of simulated and observed peak flows events. If this measure is selected, the minimum river discharge value above which the flow is defined as peak flow has to be specified in the *Peak flow* field.

- **Low flow RMSE**. This optimisation measure defines the agreement of simulated and observed low flows events. If this measure is selected, the maximum river discharge value below which the flow is defined as low flow has to be specified in the *Low flow* field.

Other input parameters are:

- **Maximum number of evaluations**. Here one of the stopping criteria of the auto calibration routine is specified. The automatic calibration will stop either when the optimisation algorithm ceases to give an improvement in the calibration objective or when the maximum number of model evaluations is reached.

- **Initial number of day excluded**. This is a warm up period that will be disregarded when calculating the objecting function.

Running the autocalibration

After having entered the autocalibration parameters, the autocalibration procedure starts when launching a normal simulation. When the optimisation routine is completed, the list of the parameters generated during the optimisation and the relative objective functions are made available in the *Catchment-Name*-Autocal.txt file. If the Calibration plot option has been selected in the *General* tab, a *CatchmentName*.plc file is also produced. This file shows the differences between the observed and the best simulated runoff data, and can be opened in the Plot Composer.

Note: The various NAM parameters shown in the *Rainfall-runoff* tab will not be automatically updated with the optimal parameter set. To visualize the

optimized parameters, either close (without saving) the MIKE HYDRO file and re-open it, or check the created text file *CatchmentName*-Autocal.txt.

Seasonal variation

In this section it is possible to enter the values of some selected time varying parameters. Monthly values have to be assigned to the parameters. Whether a parameter has a constant or a seasonal varying value is determined in the different sections of the NAM model.

It is possible to specify a seasonal variation for the parameters:

- **Variation of groundwater maximum water depth**. Seasonal variation of GW-depth causing baseflow. From *Groundwater* tab.

- **Abstraction**. Groundwater abstraction depth. From *Groundwater* tab.

- **Degree day coefficient for snow melt**. Melting coefficient for snow melt. From *Snow melt* tab.

- **Irrigation crop coefficient**. Parameter to quantify the amount of water required (and transpired) by the crop. From *Irrigation* tab.

- **Irrigation operational and conveyance losses in percent of abstracted water**. System losses of irrigation water for the different components/processes: **Groundwater**, **Overland flow** and **Evaporation**. From *Irrigation* tab.

Time series

In this section the input time series of the rainfall-runoff model are entered. Depending on which module of the model is used, these are:

- **Rainfall**. Always to be entered. A time series, representing the average catchment rainfall. The time interval between values may vary through the input series. The rainfall specified at a given time should be the rainfall volume accumulated since the previous value.

- **Evaporation**. Always to be entered. The potential evaporation is typically given as monthly values. Like rainfall, the time for each potential evaporation value should be the accumulated volume at the end of the period it represents. The monthly potential evaporation in June should be dated 30 June or 1 July.

- **Observed discharge**. This time series has to be entered if the *Autocalibration* module is used or the '*Calibration plot*' option has been selected in the *General* tab. The inclusion of the observed discharge will automatically enable additional outputs which include a calibration plot with comparison of observed and simulated discharge and calculation of statistical values.

- **Temperature**. A time series of temperature, usually mean daily values, is required only if snow melt calculations are included in the simulations.

- **Irrigation**. To be entered when the irrigation module of the rainfall-runoff module is included.

- **Abstraction depth**. To be entered when the 'Use abstraction' option in the Groundwater tab is selected and 'Time series file' is specified.

- **Radiation**. To be entered when the 'Radiation coefficient' option in the *Snow melt* tab is selected.

- **Degree day coefficient**. To be entered when the 'Variation of degree day coefficient' option in the Snow melt tab is selected.

Weighted time series may be used for some time series by enabling '*Use weighted time series*'. This adds a new tab 'TS weighted rainfall/evaporation/temperature' to the window (see below).

Weighted time series

To use weighted time series, '*Use weighted time series*' must be enabled under the Time series tab - see above. Under the TS weighted tab the following may be defined:

Allow gap in time series
If '*Allow gap in time series*' is enabled, simulation will proceed despite missing data (gap) in the time series. If '*Allow gap in time series*' is disabled, simulation will terminate with an error message if missing data (gaps) are encountered in the time series.

Distribution in time
If data is available from stations reporting at different frequencies, e.g. both daily and hourly stations, the *Distribution in time* of the average catchment rainfall may be determined using a weighted average of the high-frequency stations. You may, for example, use all daily and hourly stations to determine the daily mean rainfall over the catchment and subsequently use the hourly stations to distribute (desegregate) this daily rainfall in time.

Weighted average combinations
To specify a weighted average combination, add time series by clicking the Append '+' button above the table to add a new row, then click the browse '...' button to select a time series or create a new file and enter the weight to be assigned to that time series. Repeat the steps for all relevant time series/stations.

The weighted average values $T_{weighted}$ are computed using the following formula:

$$T_{weighted} = \frac{T_0 \cdot w_0 + T_1 \cdot w_1 + \ldots + T_n \cdot w_n}{w_0 + w_1 \cdot w_1 + \ldots + w_n} \tag{7.1}$$

where T_n is the value from time series n and w_n the corresponding weight.

If data are missing from one or more time series/stations, and '*Allow gap in time series*' is enabled, the Mean Area Weighting algorithm will ignore the time series with missing data.

In the present version it is possible to specify only one weighted average combination.

7.1.5 UHM

The UHM (*Unit Hydrograph*) model constitutes an alternative to the NAM model for flood simulation in areas where no streamflow records are available or where unit hydrograph techniques are already well established.

The model includes a number of simple unit hydrograph models to estimate the runoff from single storm events by the use of the well-known unit hydrograph technique. The models divide the storm rainfall in excess rainfall (or runoff) and water loss (or infiltration).

More details on the UHM model can be found in the 'MIKE 1D Reference' manual.

UHM parameters

The following parameters are used for all types of UHM models:

- Adjustment and baseflow
- Hydrograph
- Enlargement and loss model
- Lag time

Adjustment and baseflow

- **Area adjustment factor**. This adjustment factor (different from 1.0) may be applied if the catchment rainfall intensity is assumed to differ from the input rainfall data series by a proportional factor.

- **Baseflow**. If different than zero, the value specified represents a constant baseflow that is added to the runoff.

Hydrograph

The distribution of the runoff in time can be described using different methods.

Three different options can be selected for the hydrograph t*ype*:

- **SCS triangular**. This is a standard hydrograph in which the time to peak is assumed to be half the duration of the excess rainfall plus the lag time t_l.

- **SCS dimensionless**. This type of hydrograph has been derived from a large number of natural unit hydrographs from catchments of varying size and location. The flow values are expressed in Q/Qp, where Qp is the peak discharge, and the time in T/Tp, where Tp is the time from the start of the hydrograph rise to the peak.

- **User defined**. This hydrograph should be specified in its dimensionless form, i.e. Q/Qp as a function of T/Tp, as for the SCS dimensionless hydrograph above. But in this case this information has to be provided in the **Hydrograph table (Dimensionless from Q/Qp as a function of T/Tp). The hydrograph table must be provided in a time series file, where the axis type is either 'Equidistant Relative Axis' or 'Non-Equidistant Relative Axis' and containing one item with the type 'Dimensionless factor'.**

Enlargement and loss model

Four different options are available to represent the loss model, each of them requiring different inputs.

These types of loss models are:

- **Constant loss**. In this case the infiltration is described as an *Initial Loss* at the beginning of the storm followed by a *Constant Loss* term caused by infiltration.

- **Proportional loss**. If this model is selected, a *Runoff coefficient* should be specified as the ratio of runoff to the rainfall.

- **SCS method**. The SCS Loss Model uses a *Curve number* that characterises the catchment in terms of soil type and land use characteristics. The model also requires an *Initial AMC* (antecedent moisture conditions) parameter.

- **SCS generalised**. The SCS generalised loss model does not make use of the concept of an antecedent moisture content (AMC) but applies a storage *Initial abstraction depth*. Like in the SCS method, the *Curve number* should be specified for the SCS generalised.

Lag time

The Lag time type for the UHM model can be specified in two different ways:

- **User specified**. In this case the lag time is specified directly in hours.

- **Curve number method**. In this case the lag time is calculated by the standard SCS formula and will appear in the *Derived Lag Time* box. When this method is selected, three parameters need to be specified: *Hydraulic length*, the *Slope* of the catchment and the *Curve number*.

Time series

In this section the input time series of the rainfall-runoff model are entered. The UHM only requires two input time series:

- **Rainfall**. This time series represents the average catchment rainfall. The time interval between values, may vary through the input series. The rainfall specified at a given time should be the rainfall volume accumulated since the previous value.

- **Evaporation**. The potential evaporation is typically given as monthly values. Like rainfall, the time for each potential evaporation value should be the accumulated volume at the end of the period it represents. The monthly potential evaporation in June should be dated 30 June or 1 July.

Weighted time series may be used by enabling '*Use weighted time series*'. This adds a new tab 'TS weighted rainfall/evaporation' where time series, their corresponding weights and distribution in time may be defined (see 'Weighted time series' previous to this section).

7.1.6   Sediment Load

Important: The Sediment load tab is only available if 'Include reservoir sedimentation' is selected in the model (See "Include reservoir sedimentation" on page 44.).

Under the *Sediment load tab* the total load and the grain size class distribution of sediment supplied by the catchment is specified. This allows the calculation of load for each sediment grain size class.

To add a total sediment load time series, click the [...] button and click *Browse...* to select an existing time series or click on *Create a new file…* to open the '*Create a new file*' dialogue.

To specify the sediment grain size class distribution add a fraction time series to each grain size class. This is done by selecting a sediment grain

size class and clicking on the [...] button and clicking *Browse...* to select an existing time series or clicking *Create a new file…* to open the '*Create a new file*' dialogue. The fraction time series can be given as percentage or as fraction. The sum of fractions must equal one at each time step.

Note: The sediment grain size classes must be globally defined by their settling velocities under **13.1 Sediment Properties** *(p. 160)*.

7.2 Combined Catchments

A combined catchment is a catchment made up with a number of sub-catchments, which are defined in the *Catchment definitions* menu.

The runoff from the combined catchment is found by simple addition of the simulated runoff from each of the subcatchments. This type of catchment can especially be used during the calibration process to quickly get the total runoff at a gauge location, based on all upstream sub-catchments.

Combined catchments are not shown on the map, and can only be edited from the *Tabular* tab. Use the *Append* '+' or *Delete* '-' buttons above the overview table to add or remove a combined catchment.

The various properties specified for a combined catchment are described below.

- **Combined catchment name**. The combined catchment can be given any name.

- **Total area**. The total area of the catchment is automatically computed, and is equal to the sum of the areas from all its subcatchments.

- **Subcatchments**. All subcatchments have to be added in this table using the *Append* '+' button. They can be deleted using the *Delete* '-' button. Each subcatchment has to be selected from the drop-down list amongst the catchments defined in the *Catchment definitions* menu. The area considered for each subcatchment is always its total area.

- **Calibration plot**. It is possible to save the calibration plot by enabling 'Calibration plot'. The calibration plot shows observed and simulated runoff for the catchment.

- **Observed discharge**. When the 'Calibration plot' is enabled, a time series of observed runoff has to be specified here with the Browse '…' button.

7.3 Hotstart files

The initial conditions for a NAM simulation may be loaded from an existing rainfall-runoff result file by enabling 'Use hotstart files' and clicking on the 'Append' button. This will add a new row to the list of *Hotstart files*. Click on the 'Browse' button and select the required hotstart file.

Important: Initial conditions are read in the additional result files having the extension _AddNAM.res1d. Therefore, only this type of file can be selected

here. The file has to be created using the 'Create runoff hotstart files' in the Standard results menu.

Note: It is possible to define multiple hotstart files covering different or over-lapping areas. If some catchments are covered by more than one hotstart file the first hotstart file will be used. The 'Move up' and 'Move down' arrows may be used to modify the order of the hotstart files. If a catchment is not covered by any of the specified hotstart files, the initial conditions are those specified in the *Catchment definitions* menu.

Use simulation start time

If 'Use simulation start time' is enabled, the initial condition will automatically be extracted from the *Hotstart file* using the simulation start date and time as specified in the *Simulation period* page.

Date and Time

The date and time at which the initial conditions are loaded from the hotstart file. If 'Use simulation start time' is enabled the hotstart date and time will be taken as the simulation start.

8 Water Users

Water users represent water consuming activities withdrawing water from the river or a reservoir.

A model setup may contain any number of Water users where MIKE HYDRO overall supports two types of water users; Irrigation water users and Regular water user (e.g. municipal, industrial and any other type of Water user).

Water users a connected to the river network through Supply flow connections and Return flow connections, where supply flow is being extracted from the supply connection point and the Return flow is the water returned from the water user once the final consumption has been calculated. Water user definitions and options for connections are illustrated in the figure below.
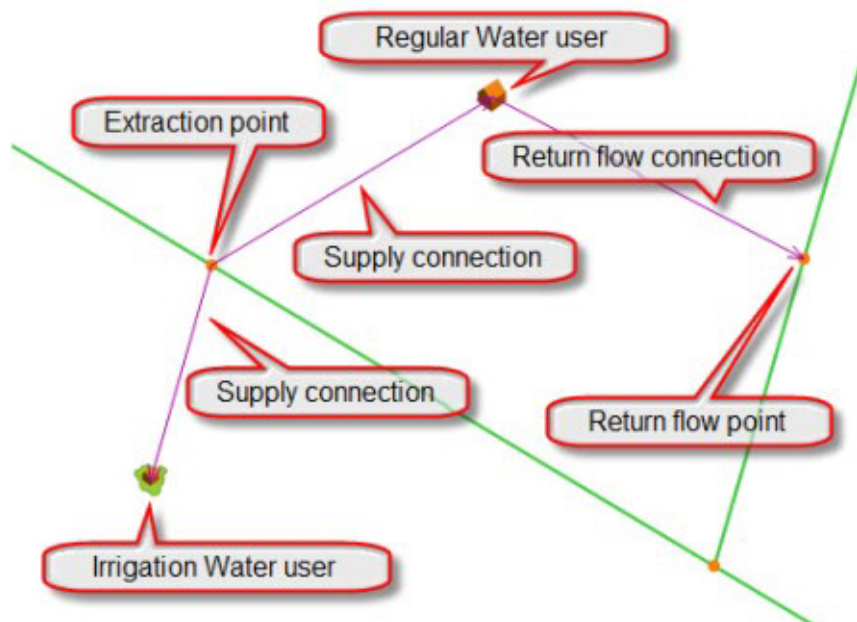
Figure 8.1   Illustration of Water users connected to the River network through Supply connections and Return flow connections

Two different entries are available for defining Water Users:

● Water User definitions (General parameters, Supply and Return flow connection details)

● Irrigation data definitions (Irrigation method applied, Crop definition as well as Soil and Runoff model specifications)

Inserting a water user

To insert a Water user (regular and irrigation) in the model select the '*Add*' button under the *Water user* ribbon in *Map view* and click on the desired location of the Water user in the map.
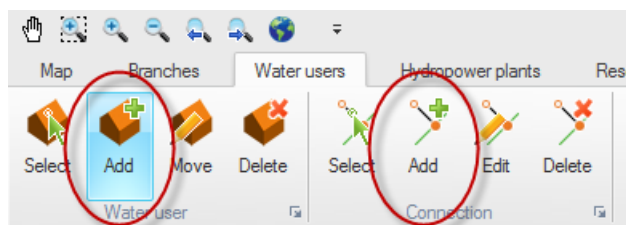

Figure 8.2   Insert a Water user

Note that the location is for visual purposes only and has no impact on the model setup.

A Water user can extract water from one or several Branches and/or Reservoirs, as well as from groundwater storage. A Water User can return unconsumed water to one or more Branches (return flow).

Use the '*Add Connection*' button under the *Water user* ribbon in *Map view* to connect a Water user to its water extraction points (Supply connections) and to downstream return flow points (Return flow connections) on Branches or Reservoirs. The digitisation of a Connection must always be done in the direction of the flow, i.e. from extraction point to Water user, or from Water user to return flow point. Single-click on the starting point of the Connection and drag the Connection to the desired end point.

Note that the cursor changes to a cross when properly placed. To complete the Connection release the mouse button.

A River node is automatically inserted on the extraction point/return flow point.

Note that the flow direction of the connection is shown with an arrow in the *Map view*.

Specifying water user properties

The *Water user properties* dialogueue contains the following tabs:

- General
- Supply Connections
- Return Flow Connections

- Irrigation Scheme (for irrigation users only)

- Irrigated Field **(for irrigation users only)**

Use the table in the lower part of the properties dialogue to navigate to a particular Water user.

8.1 General

Under the *General tab* three types of information can be specified.

1. Type of water user
2. Water demand time series and demand carry-over fraction
3. Groundwater properties

8.1.1 Type of water user

Specify the type of Water user: regular or irrigation.

Important notice for Irrigation users:
If irrigation users have a significant surface area compared to the area of the Catchment, in which they are located, it is important to subtract the irrigated area from the Catchment area. This is done by enabling the option; '*Subtract area of irrigation users and reservoirs from catchment area to calculate runoff*' in the '*Simulation specifications -> Modules*' page of MIKE HYDRO.

Note that an irrigation Water user with a user specified water demand series (e.g. not using the irrigation module to calculate water demand) must be treated as a regular Water user.

8.1.2 Water demand time series and demand carry-over fraction

For a regular Water user, a water demand time series must be specified.

This is done by clicking on the [...] button and clicking *Browse...* to select an existing time series file or by clicking *Create a new file…* to open the '*Create a new file'* dialogue.

The *Water demand time series* dialogue contains two items:

- **Water demand**: The total amount of water that is required to be extracted to fulfil the water demand of the Water user.

- **Demand carry-over fraction** (from one time step to next). Normally the allocation solution is only with respect to demands in the current time step. For some water use schemes it may be appropriate to allow deficits to carry over from one time step to the next, resulting in a larger demand in that next time step (larger than the "original" demand given in the demand time series). The 'Demand carry-over fraction' is used to specify the fraction of any deficit that should carry over to the following time step.

  Mathematically, this is described by:

  $$D'(t+1) \ = \ D(t+1) + f(t) \cdot X(t)$$

  Where *D'(t+1)* is effective water demand at time *t+1*, *D(t+1)* is water demand as given in the input time series for time *t+1*, *X(t)* is calculated deficit, and *f(t)* is the carry-over fraction. The latter two are calculated at the previous time step t. The *D', D*, and *X* have units of volume, while *f* is dimensionless.

  If *f = 0* the Deficit carry-over fraction option is ignored.

  Note: Results from monthly simulations may appear to 'oscillate'. This is not an error, but rather because the deficits in the above formula are absolute volumes, whereas demands in the input time series must be given as flows (volume per time). Due to the variable lengths of the months, a constant deficit in terms of flow means a varying deficit in terms of volume.

8.1.3 Groundwater properties

If groundwater is included in the model (cf. Section 4.1), three groundwater properties must be specified: groundwater extraction method (Groundwater options), the catchment from which groundwater is extracted (Supply catchment), and the groundwater use time series.

Groundwater options

There are three different options for using groundwater to fulfil the Water user demand:

- **Fraction of total demand**. This method will attempt to fulfil the Water user demand by taking a fraction of the demand from the groundwater. This option corresponds to a specified sharing of the surface water and the groundwater resources. If there is limited water available either in the surface water or the groundwater the demand will not be fulfilled.

- **Fraction of remaining demand**. This method will attempt to use groundwater to fulfil a fraction of the total Water user demand that cannot be fulfilled by surface water. If the fraction is set to 1.0 it means that the Water user first extracts surface water and then fulfils the remaining demand using groundwater. This option corresponds to specified priority between surface water (first priority) and groundwater.

- **Absolute demand**. This method will subtract absolute groundwater demand from the total water user demand to calculate surface water demand. If absolute groundwater demand is greater than total water user demand, surface water demand will be zero. If there is limited water available either in surface water or in groundwater, the water user demand will not be fulfilled.

Supply catchment

In this field you can specify the Catchment from which the groundwater is supplied. Only valid Catchments for which groundwater aquifers exist will be displayed in the drop down menu.

Groundwater use time series

Depending on the groundwater option selected, either a groundwater fraction use time series or a groundwater absolute use time series must be specified.

This is done by clicking on the [...] button and clicking *Browse...* to select an existing time series file or clicking *Create a new file…* to open the '*Create a new file'* dialogue.

Note that the groundwater use time series may optionally include two additional items: *Lower groundwater extraction limit* and *Upper groundwater extraction limit*, where the upper groundwater depth limit specifies the depth at which groundwater extraction may start to be reduced, and the lower groundwater depth limit specifies the groundwater depth at which no more extraction is allowed.
If the groundwater depth (at the beginning of the time step) is smaller than the upper limit, there is no reduction of the extraction (a reduction factor equal to 1 is applied).
If the groundwater depth is between the upper and the lower limit, a linear interpolation is performed on the reduction factor.
Complete reduction (reduction fraction equal to 0) takes place when the groundwater depth is larger than the lower limit. If only one of the two time series is given, the change in reduction factor goes abruptly from 1 to 0, when the groundwater depth crosses this limit.

Note: The Upper groundwater depth limit should be given a value, which is smaller than the lower groundwater depth limit.

8.2 Supply Connections

Under the *Supply connections* tab three types of attribute data can be specified:

- Priority of supply connections
- Flow loss time series
- Flow capacity time series

8.2.1 Priority of supply connections

Here, the priority of supply connections and the supply rules are specified.

By default, supply connections are given a priority according to the sequence in which they are digitised. The priority can be changed in the priority column. Number 1 has first priority; number 2 has second priority, and so forth.

Two types of **supply rules** are available:

- **Call by priority:** Water is supplied to satisfy the demand in the order of priority beginning with the supply connection with the lowest 'priority number'.

- **Fraction of demand:** Water supply is based on the fraction of the demand required from each supply connection. If a supply connection cannot fulfil its fraction there is no attempt to extract water from another supply connection.

  A fraction file, containing the fraction of demand requested from each supply connection must be specified.

  This is done by clicking on the [...] button and clicking *Browse...* to select an existing fraction file or clicking *Create a new file…* to open the '*Create a new file*' dialogue.

  Note: The sum of fractions must not exceed 1. It is possible, however, to have a sum less than 1, with the remainder showing in the results as "Water Demand Deficit". Be careful, however, as it can be hard to distinguish this deficit from a "true" deficit due to lack of water.

It is possible to use a combination of *Call by priority* and *Fraction of demand* by assigning the same priority to one or more groups of supply connections and then specifying the fraction of demand required from each supply connection within the 'shared priority' group. The fractions for all supply connections within a 'shared priority' group must add up to one at all times.

8.2.2 Flow loss time series

Supply connections may lose water due to seepage and /or evaporation. If these processes are considered to be of importance they may be included in the model as a time series that specifies the losses. Both seepage and evaporation can be specified as a fraction of the actual flow (dimensionless), or as flux (volume per time).

To add a flow loss time series, enable '*has flow loss',* click on the ⬜… button and click *Browse...* to select an existing flow loss time series or click *Create a new file…* to open the '*Create a new file'* dialogue.

The user-specified water demand *D* of the connected user is automatically adjusted to take the loss into account. The adjusted water demand *D** is calculated as follows:

$$D^* = \frac{D}{(1 - \text{loss factor})} \tag{8.1}$$

This means that the Water user demands will still be fulfilled, if sufficient water is available at the supply source. This also applies, if the flow loss is given as a flux.

If groundwater is defined in the Catchment where the supply connection begins (where the extraction point is located), seepage loss is added to this groundwater storage. However, with the current implementation seepage flow is lost, if the supply connection begins in a catchment node.

8.2.3 Flow capacity time series

Supply connections may be assigned a flow capacity [Volume per time] that can never be exceeded.

To add a flow capacity time series, enable '*has flow capacity*', click the ⬜… button and click *Browse...* to select an existing flow capacity time series or click *Create a new file…* to open the '*Create a new file'* dialogue.

The flow capacity overrules all other rules that may try to force more water through the supply connection.

Example

A water user may call for 5 m^3/s. If the flow capacity of the supply connection has been set to 4 m^3/s for example, the user will only receive maximum 4 m^3/s and hence suffer a deficit, even though there may be plenty of water at the source.

If the flow capacity of a supply connection is reached, the model will attempt to force the water in an alternative direction. If this is not possible, the simulation will terminate with a message.

8.3 Return Flow Connections

If return flow occurs from the Water user, a connection must be created between the Water user and the return flow point (cf. Inserting a water user in this Section).

Under the *Return flows tab* three types of attribute data can be specified:

- Return flow time series
- Flow loss time series
- Flow capacity time series

8.3.1 Return flow time series

A return flow time series must be specified either by clicking the [...] button and clicking *Browse...* to select an existing time series file or clicking *Create a new file…* to open the '*Create a new file*' dialogue. The return flow time series may contain the item type Return flow fraction (the fraction of the delivered water) or the item type Discharge (a specified return flow).

If the return flow is given as a discharge, this discharge cannot exceed the amount of water delivered to the user. A warning message will be issued if the return flow must be reduced.

Multiple return flow points

The total return flow cannot exceed the delivered amount to the water user. If there are multiple return flow points all given as a fraction of the delivered water, the sum of the return flow fractions given in the return flow time series should not exceed 1.0. It is possible, however, to have a sum less than 1, with the remainder showing in the results as "Used Water". The same apply if the sum of return flow as discharge is lower than the delivered water.

If the sum of the return flow should exceed the amount of water available at the user, the individual return flows will be reduced in the opposite order of the list in the return flow connections table. In other words, the last return flow listed, will be the first one being reduced in order to match the amount of water available at the node.

8.3.2 Flow loss time series

Return flow connections may lose water due to seepage and /or evaporation. If these processes are considered to be of importance they may be included in the model as a time series that specifies the losses. Both seepage and evaporation can be specified as a fraction of the actual flow (dimensionless), or as flux (volume per time).

To add a flow loss time series, enable *'has flow loss',* click the [...] button and click *Browse...* to select an existing flow loss time series or click on *Create a new file…* to open the '*Create a new file'* dialogue.

If groundwater is defined in the catchment where the return flow point is located (where the return flow connection terminates) seepage loss is added to this groundwater storage.

8.3.3   Flow capacity time series

Return flow connections may be assigned a flow capacity [Volume per time] that can never be exceeded.

To add a flow capacity time series, enable *'has flow capacity'*, click the [...] button and click *Browse...* to select an existing flow capacity time series or click on *Create a new file…* to open the Create a new file dialogue.

The flow capacity overrules all other rules that may try to force more water through the connection.

If the flow capacity of a return flow connection is reached, the model will attempt to force the water in alternative direction. If this is not possible, the simulation will terminate with a message.

8.4   Irrigation Scheme

The *Irrigation scheme* tab includes definition of three types of information:

- Climate model
- Deficit distribution method
- Use a soil and runoff model for all fields

8.4.1   Climate model

The Climate model accepts a number of commonly available climate inputs and converts them into precipitation and the input required by the reference ET model, according to the selected Climate model.

Two Climate model types are currently available:

- **Rainfall only**. This is the simplest climate model, only requiring a Rainfall time series and a reference ET time series as input.
- **FAO 56**. The FAO 56 model requires the specification of Climate time series as input for the calculation of the FAO 56 reference evapotranspiration.

The required inputs are:

- **Rainfall time series**. A rainfall data time series is required for both the Rainfall only and the FAO 56 models.

- **Climate time series**. Climatic data are required input when the FAO 56 model is selected. The climate time series must include Relative humidity, Air temperature (min and max), Wind speed and Sunshine hours.

- **Coordinates**. When the FAO 56 Climate model type is selected, the Latitude and Altitude of the location of the irrigation water user are required. These coordinates will be set automatically if a coordinate system is defined for the features (cf. 5.1 Coordinate System *(p. 49)*)

- **Disaggregate rainfall**. If rainfall is provided on a low temporal resolution (e.g. monthly data), functionality to disaggregate the data into a higher temporal resolution is provided. Checking the 'Disaggregate rainfall' checkbox enables the field where the new aggregation period can be specified. Setting the time span to 10 days, for example, results in a rain event every 10 days, of a magnitude that corresponds to:

  - Rain depth = (Aggregation period) / (Input time step) * (Rainfall depth for input time step)
  - Disaggregation of rainfall is possible for both the Rainfall only and the FAO 56 climate model.

- **Reference ET time series**. When the climate model type is Rainfall only, the time series of reference evapotranspiration must be entered. If the FAO 56 model is selected, the reference ET is calculated according to the FAO 56 method (see Appendix A.3 for additional details).

8.4.2   Deficit distribution method

Deficit distribution methods are used when the irrigation demand exceeds the available water at the sources. In such cases, the deficit distribution methods describe how the available water should be distributed among the fields represented by the node. When the deficit distribution method is selected, the fields that are a part of the Water user need to be defined in the section Water users → Irrigated field (cf. Section 8.5).

Three options are available:

- **Equal shortage**. The fields get the same percentage of the demand covered, and hence suffer the same relative shortage.

- **By priority**. The water is distributed according to the priority that has been specified for the field. The fields with the highest priority (lowest priority number) will receive the full demand first. If several fields have the same priority, water is distributed according to the equal shortage method. The priorities are specified in the Irrigated field section.

- **By Ky**. This method is also called yield stress deficit distribution, since the water is distributed according to how sensitive the crops are to soil water stress at the time of water shortage. This implies that the crop with the highest yield response factor (*Ky*) will be given the highest priority. In case of several crops with the same yield response factor, water is distributed among these fields according to the equal shortage method. The yield response factor of the crops is specified in the Crops module.

8.4.3 Use a soil and runoff model for all fields

If this option is selected, it is possible to specify in the dedicated box a Soil and Runoff model that will be applied to all fields in the irrigation scheme (unless a specific model has been selected for a field, see Section 8.5).

8.5 Irrigated Field

An irrigation Water user has one or more irrigated fields. Each irrigated field consists of a sequence of crop shifts (a crop sequence). A crop shift is characterised by a starting date (sowing date), a crop, and a reference to the irrigation method that will be used to irrigate the crop. A crop shift lasts until the end of the last growth stage of the crop in the field.

In the Irrigated field section, the crop sequence for each field has to be specified.

Each row added to the Irrigated field tab represents one of the crops growing on the field in a particular period and on a specific area of the field. An irrigated field for a specific crop is defined by various properties, such as the name of the crop, the area of the field, the properties of the soil in the field and its ability to generate runoff.

The buttons at the bottom of the Irrigated field table can be used to add, or delete the input for the different irrigated fields of the water user node. A field is added to the Irrigated field node by clicking the 'Append' button '+', which results in a line being added to the Irrigated fields table. To delete a field, select the field's line and click Delete '-': . The properties of the field can be edited directly in the table.

Some attributes of the field are visible on the row of the specific field in the field's tab and they must be specified. These are:

- **ID**. The field identification code, ID, is automatically given every time a new field is added to the model.
- **Name**. This corresponds to the name of the selected crop grown in the field is entered.

- **Priority**. If 'by priority' has been selected as deficit distribution method in the Irrigation scheme section, priority is an integer defining the priority with which the different fields will receive water in condition of scarcity. The fields with the highest priority (lowest priority number) will receive the full demand first.

- **Area**. Size of field growing with specified crop.

- **Min. cycle time**. The so-called minimum cycle time, expressed in days, is an important parameter, if the farmer cannot irrigate the whole field within a single time step, but in multiple time steps. In this case, sections of the field will be created accordingly. Hence, soil moisture, and consequently yield, may be different on the different field sections and internally these are treated as different fields. In each output time step, the average field conditions are written to the result file. To disable the minimum cycle time, set it to a value less than the simulation time step. Example: If, for instance, it takes a farmer two days to irrigate the entire field and the time step of the simulation is one day, then the field is divided into two field sections and on the first day he irrigates one half and the next day the other half.

- **Recycle crop sequence**. If the same sequence of crops is cultivated in the field, this box should be ticked. The frequency with which the crop sequence is recycled depends on the total length of the crop sequence. If the sowing years of the different crops differ, the sequence is repeated as soon as the last crop is harvested and a new year starts.

- **Soil and Runoff model**. Here, the appropriate Soil and Runoff model for the crop of interest can be selected. By clicking on the box, the list of the available models appears. The available models are those entered in the section Irrigation data → Soil and runoff.

When clicking on the arrow next to the field's ID, the tab expands and more attributes of the field appear, which also have to be specified. These are:

- **Crop model**. The Crop model can be chosen from those entered in the section Irrigation data → Crops (to jump to this section, right click on the specific crop's line in the tab and select 'Go to Crops'). To select the model, click on the Crop model field and the available option will appear.

- **Sowing year**. The first sowing year for the crop has to be specified.

- **Irrigation method model**. The Irrigation method can be selected from those entered in the section Irrigation data → Irrigation method (to jump to this section, right click on the specific crop's line in the tab and select 'Go to Irrigation method'). The list of the available models appears when clicking on the Irrigation method model box.

- **First sowing date**. This field displays the date when the crop is sowed for the first time, given in dd-mm-yyyy format. This field cannot be edited.

- **First harvest date**. This field displays the date when the crop is harvested for the first time, given in dd-mm-yyyy format. If a crop is harvested before the next crop is planted, the model assumes that there are no crops at this field in the time between crops, and hence that the irrigation demand is zero. During the period with no crops in the field the water dynamics in the soil continues and effects of rainfall, evaporation and drainage are simulated. When the next crop is sowed the water content in the root zone will be determined by these dynamics. This field cannot be edited.

8.6   Irrigation Data

Irrigation data comprise a variety of definitions related to Water users defined as Irrigation type of users.

Regular water users and Irrigation water users functions very similar. The main difference being that the Regular water user works on prescribed demands and return flows, and the Irrigation user calculates the two variables dynamically. An Irrigation water user represents one irrigation area comprising one or more irrigated fields, which are all drawing water from the same sources. Each Irrigation user therefore represents the total irrigation demand for all fields defined for this specific user.

The calculation of water demand and return flow for irrigation users are based on detailed information about the soil and runoff properties of the fields as well as the type of crop grown within these fields. Generally, several fields in a basin model have the same soil properties and eventually identical type of crops and the Irrigation Data entry enables the definition of specific crop-types, Irrigation method definition as well as overall Soil and runoff methods that can then easily be applied when defining the specific details for Irrigation Users within the Water Users page.

Irrigation data includes sub-pages for definition of:

8.6.1   Irrigation method

The Irrigation method section is used to specify how and when a given field is irrigated.

A **Name** should be specified for each irrigation method. A new method is created by clicking on the 'Append' button, designated with the symbol ➕ . A new line is then added to the tab displayed at the bottom of the page and the input for the new method can be filled in.

Alternatively, predefined irrigation methods can be loaded from a template file by clicking on the 'Load template' button, ✖ , and browsing to the specific template file. An existing template used in another MIKE HYDRO model can

also be used, by clicking on 'Import existing', ■ , and browsing to the previously created .mhydro file.

By clicking on the 'Delete' button, ━ , the currently displayed Irrigation method is deleted.

To switch between the different Irrigation methods, click on the lines relative to different methods in the tab in the lower part of the *Tabular view*.

Type of irrigation methods

The following three Irrigation methods are available:

- None (No irrigation method)
- FAO 56 irrigation method
- Rice Crop irrigation method

The *Irrigation method* dialogue also contains fields to set the Spray loss time series, the Wetting fraction time series, the Trigger option and the Application option. The Trigger option determines when the irrigation will start and the Application option determines how the depth is calculated when the irrigation has started.

The available options for Trigger and Application options depend on the selected Irrigation method, while the Spray loss and the Wetting fraction time series do not.

All input are specified as time series where the time is relative to the day of sowing.

- **Spray loss time series.** The Spray loss time series correspond to the fraction of the irrigation water that is evaporated before the water reaches the soil surface. For sprinkler irrigation, this fraction may be relatively high, whereas it is relatively low for drip irrigation, for example. The value has to be strictly less than 1.

- **Wetting fraction time series**. The irrigation model requires a wetting fraction that determines the fraction of the field surface that is being wetted during irrigation. For example, sprinkler irrigation this fraction will be close to 1, whereas for drip irrigation it may be as low as 0.1. The wetting fraction is also an important factor for determining how much irrigation is required before the surface soil storage is filled and hence when the root zone starts to fill.
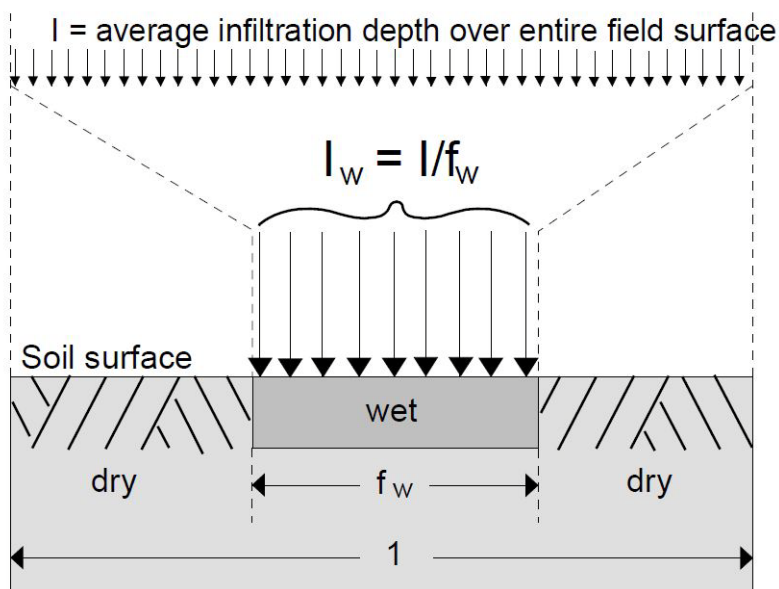
$$I = \text{average infiltration depth over entire field surface}$$

$$I_w = I/f_w$$

Soil surface

dry | wet | dry

f_w

1

Figure 8.3   Illustration of the wetting fraction (after /1/) where I and Iw are the irrigation depth for the field and the irrigation depth for the part of the wetted surface, respectively, and fw is the fraction of the surface wetted by irrigation

None (No irrigation method)

When this option is chosen, no irrigation is applied to the fields.

FAO 56 irrigation method

For the FAO 56 irrigation model the following parameters have to be specified for the Trigger and Application options.

Trigger option

- **Option**. The following four alternatives are available in the Trigger option of the FAO 56 irrigation method:

  - **Fraction of TAW** (Total Available Water). Irrigation starts when the soil moisture content reaches the specified fraction of TAW. TAW is defined as the volume of water contained in the root zone when at field capacity.
  - **Fraction of RAW** (Readily Available Water). Irrigation starts when the soil moisture content reaches the specified fraction of RAW. RAW is defined as the volume of water that can be transpired by the crop without exposing the crop to soil water stress. It is defined as:

    $$RAW = p \cdot TAW$$

    where p is related to the crop model (see more details in the Depletion fraction section, presented in the Crops section).

- **Specified depletion depth**. Irrigation will start when the soil moisture content reaches the specified depletion depth.
- **Prescribed**. No Time series in the Trigger option box are required as input when this option is selected. Irrigation is only described by the irrigation Application option time series given as Fixed irrigation rate option.

- **Time series**. The Trigger option values are given as time series when any other Trigger option than the Prescribed Depletion is selected.

  Depending on the selected Trigger option, the required time series file must contain:

  - **Time in the first column**, which indicates the number of days from start of crop season, and either
  - **Fraction [-]**, which is used for the first two options above, or
  - **Depletion [mm]**, which is used for the specified depletion option.

Application option

- **Option**. The following three alternatives are available in the Application option of the FAO 56 irrigation method:

  - **Fraction of TAW**. Irrigation stops when the soil moisture content reaches the specified fraction of TAW.
  - **Fraction of RAW**. Irrigation stops when the soil moisture content reaches the specified fraction of RAW.
  - **Fixed Irrigation Rate**. The specified irrigation rate is applied to the field.

- **Time series**. The Application option values are given as time series. Depending on the selected application option, the required time series file must contain:

  - **Time in the first column**, which indicates the number of days from start of crop season, and either
  - **Fraction [-]**, which is used for the first two options above, or
  - **Irrigation rate [mm/h]**, which is used for the specified depletion option.

Rice Crop irrigation method

For the Rice Crop Irrigation model the following parameters have to be specified for the Trigger and Application options.

Trigger option

- **Option**. The following two alternatives are available in the Trigger option of the *Rice Crop irrigation* method:

  - **Prescribed**. No Time series in the Trigger option box is required as input when this option is selected. Irrigation is only described by the irrigation Application option time series given as Fixed irrigation rate option.

— **Water Depth**. Irrigation starts when the water depth on the ground surface reaches a certain depth.

Application option

- **Option**. When the irrigation is triggered, the application depth is calculated according to the Application option. The following options are available:

  — **Fixed irrigation rate**. Specified irrigation rate is applied to the field.
  — **Fraction of Ponded Water Depth**: Irrigation stops when the water depth reaches the specified fraction of Maximum Water Depth (MWD). Note: This option is not compatible with Prescribed Trigger option.

The FAO 56 Irrigation method is also compatible with the Rice Crop model. The Rice Crop irrigation method, however, is only compatible with the Rice Crop model.

8.6.2 Crops

Given the soil moisture content and reference evapotranspiration, the crop module takes the input to calculate the corresponding crop evapotranspiration and soil evaporation.

To create a new model click on the 'Append' button, designated with the symbol + . A new line appears in the window displayed at the bottom of the screen. A **Name** should be specified by for the crop type. The inputs related to the crop of interest can be entered in the boxes of the Crops section.

Alternatively, predefined crop models can be loaded from a template file by on the 'Load template' button, ⨉ , and browsing to the specific template file. An existing template used in another MIKE HYDRO model can also be used, by clicking on 'Import existing', ▪ , and browsing to the previously created .mhydro file.

To delete the currently displayed crop type information click on 'Delete': − .

To switch between the different Crops models, click on the lines relative to different crop types in the tab in the lower part of the *Tabular view*.

Type of Crop models

Two different Crop model types are currently available:

- FAO 56 DualCropCoefficient model
- Rice Crop model

FAO 56 DualCropCoefficient model

The FAO 56 method is based on the dual crop coefficient method described in FAO 56. The FAO 56 DualCropCoefficient model calculates the transpiration and soil evaporation separately and thus allows for a more accurate quantification of the consequences of using different irrigation technologies.

Following the FAO 56 terms, the Crop stages are divided into the following:

- **Initial**
- **Development**
- **Mid season**
- **Late season**

The parameters to be assigned to each Crop stage are:

- **Length**. The length, in days, of the specific period.

- **Kcb.** For each stage, a so-called basal crop coefficient (*Kcb*) is assigned. The basal crop coefficient is defined as the ratio of the crop evapotranspiration over the reference evapotranspiration (*ETc/ET0*) when the soil surface is dry but transpiration is occurring at potential rate. *Kcb* is assumed constant in the initial and middle stages, and assumed to follow a linear variation between the stages.

- **Root depth**. The Root depth *R* determines the maximum depth from which the crop can extract water. The minimum and maximum depth has to be specified. It is assumed that the maximum depth is obtained at the beginning of the middle stage, and that the variation between the initial depth and the maximum depth is determined by the following relationship:

$$R = \frac{(K_{cb} - K_{cb,ini})}{(K_{cb,mid} - K_{cb,ini})} \cdot (R_{max} - R_{min}) + R_{min}$$

where:

$K_{cb,ini}$: Initial crop basal coefficient [-]

$K_{cb,mid}$: Basal crop coefficient in middle stage [-]

R_{max} : Maximum root depth [m]

R_{min} : Minimum root depth [m]

- **Max height**. The influence of the vegetation surface roughness on the evapotranspiration is taken into account through a climatic correction factor applied to the basal crop coefficient. The vegetation height is assumed to scale with the basal crop coefficients and is calculated as:

$$H = \frac{(K_{cb} - K_{cb,ini})}{(K_{cb,mid} - K_{cb,ini})} \cdot H_{max}$$

for the initial and development stage, after which the height is assumed to have reached its maximum (H_{max}).

- **Length (Yield)**. When the FAO 33 yield model is applied, each stage is assigned a length that may, but does not have to, be the same as the growth stages in the Crops model to which it is related.

- **Ky**. Yield response factor that must be specified when the FAO 33 yield model is applied. A crop is usually sensitive with different intensities to soil moisture stress at different stages of its development.

Rice Crop model

For the Rice Crop model, one additional stage has been included: the **Nursery** phase. This stage is the period in which the rice plants are developing before transplantation.

The supplementary inputs for the Nursery stage are:

- **Length**. In days.

- **Nursery area**. It is the ratio of the total field area in which Nursery takes place. This is usually a small part of the future rice field.

- **Land preparation**. The Land preparation stage is the period, expressed in [days], before transplanting (at the beginning of the Initial stage). During this period the land is irrigated and the soil is puddled.

Note: As the Nursery stage also covers the Land preparation, the Nursery period should be longer than the Land Preparation period.

Growing period

In this section two inputs should be specified:

- **Sowing day/month**. The starting date (sowing date) of the shift of the specific crop.

- **Last irrigation day**. The irrigation can be stopped after a specified day. This is a useful option when irrigation demand is calculated by the model.

Yield

- **Depletion fraction**. The depletion fraction p expresses the sensitivity of the crop to soil moisture stress, or more specifically, the fraction of the totally available water (TAW) at which soil moisture stress will start to reduce crop transpiration. The amount of water that may be depleted without stressing the plant is called the readily available water (RAW).

    The relationship between RAW and TAW is:

    $$RAW = p \cdot TAW$$

    For soil moisture contents below RAW, transpiration is assumed to decrease linearly with soil moisture content and reach zero when the wilting point is reached (see Figure 8.4).
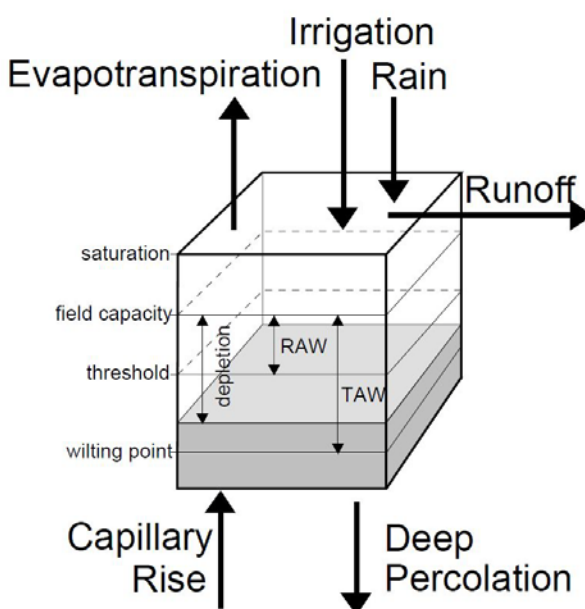
Figure 8.4   Relationship between RAW and TAW (after /1/)

Attaching a Yield model to a crop model allows the conversion of soil water stress into the corresponding yield loss, and hence, to quantify the costs of a soil moisture deficit.

One single yield model is currently available:

- **FAO 33 Yield model**

The following three options are available for the FAO 33 Yield model:

- **Calculate yield**. By ticking this box a yield model is attached to the crop model. When choosing this option, the Potential yield can be entered and so the K_y parameters for the different crop stages.

- **Potential yield**. This parameter, expressed in [kg/ha], represents the crop production per unit area of cultivated field and it must be entered when the yield model is included.

- **Use yield stage length**. Ticking this box enables the user to specify the length of the periods with different K_y in the Length (Yield) section. These may differ from the crop stage given in the Length section.

Note: The total length of the Length (Yield) periods has to be equal to the total length of the Length periods (without including Nursery).

8.6.3   Soil and runoff

In the Soil and Runoff section, the parameters for different combinations of soil type and runoff models are specified.

To create a new model click on the 'Append' button, designated with the symbol **+** . A new line appears in the window displayed at the bottom of the screen. You can specify a **Name** for the new soil type, i.e. the name of the dominant soil type in the soil profile. The Soil model and Runoff model inputs can be entered in the relative sections.

Alternatively, predefined soil and runoff models can be loaded from a template file by clicking on the 'Load template' button, ⚒ , and browsing to the specific template file. An existing template used in another MIKE HYDRO model can also be used, by clicking on 'Import existing', ▪ , and browsing to the previously created .mhydro file.

To delete the currently displayed crop type information click on 'Delete': **−** .

To switch between the different soil and runoff models, click on the lines relative to different crop types in the tab in the lower part of the *Tabular view*.

Soil model

The Soil model, also referred to as soil water model, keeps track of the water flow between different layers in the soil profile and the time-varying water content in each layer (see Figure 8.5**)**. The main task of the soil water model is to keep track of the amount of soil water available for soil evaporation and crop transpiration at any time during the simulation. The soil water content may also be used by the irrigation module to determine the irrigation demand.
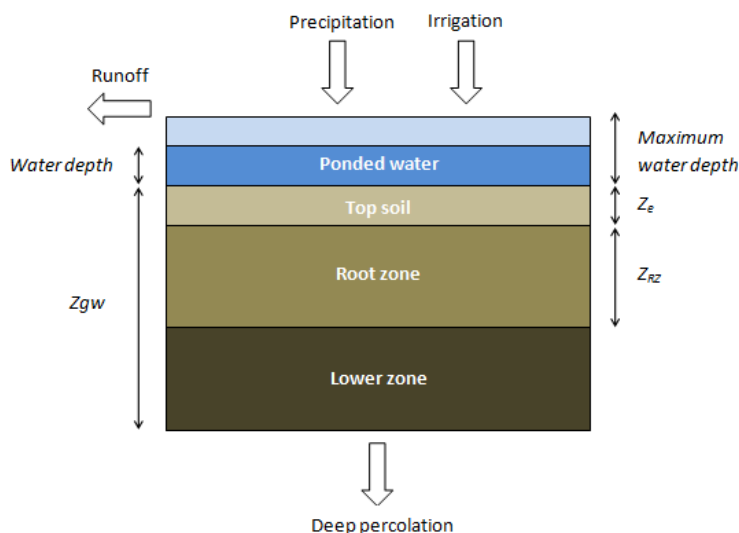
Figure 8.5   Schematisation of the Soil Water models. Each model uses a different number of layers

You can chose between three soil water model types:

- **FAO 56 soil water model**. The FAO 56 soil water model is a simple water balance based model that follows the recommendations provided in FAO 56 /1/ for use with the dual crop coefficient method. This water model describes the dynamics of soil moisture content only in Top soil and Root zone. The description of the Lower zone and Ponded water are not included.

- **ZIMsched soil water model**. The ZIMsched soil water model is one step more complex than the FAO 56 soil model. It simulates the water balance in the unsaturated soil column in three compartments (Top soil, Root zone, Lower zone), instead of the two of FAO 56.

- **ZIMsched for Rice field soil water model**. The ZIMsched for Rice field, a soil water model for rice crops, is an extension of the ZIMsched model with an additional storage on the ground surface (Ponded water). This allows water to be stored on the ground surface during growth of rice.

See 'Irrigation' (Section A.3) in Appendix A for more detailed information on these models.

FAO 56 soil water model

In the *FAO 56 soil water model* dialogue, the required soil properties are defined, and, here, the following parameters need to be specified (see also Table 8.1 for suggested representative values):

Soil moisture contents

- **Initial**. This is the starting soil moisture content at the beginning of the simulation [0-1]. The Initial soil moisture content is set for the Top soil and the Root zone. For the Lower zone the initial soil moisture is by default, at Field capacity.

- **Field capacity**. This is the maximum water content held by the soil against gravity [0-1]. Water cannot be retained in the top soil and the root zone layer when the water is above field capacity, as it will drain away under gravity. Soil moisture below field capacity is available for evapo-transpiration until the soil moisture reaches wilting point.

- **Wilting point**. This is the lowest soil moisture content at which plants can draw water from the top soil and the root zone layers [0-1].

Other parameters are:

- **Porosity**.This is the maximum soil moisture content that the soil can contain [0-1].

- **Depth of evaporable layer**. This is the depth of the top soil layer from which evaporation occurs [m].

Example

The amount of water in a soil column depends on the soil moisture content and the length of the soil column. If the field capacity is 0.15, the wilting point is 0.05, and the length of the root zone is 0.5 meter, then the available amount of water for transpiration is: 50 mm.

Table 8.1   The representative soil property values and maximum depletion by evaporation for an evaporation layer of 0.1 m

Soil type	Field capacity	Wilting point	Difference	Max. depletion by evaporation [mm]
Sand	0.07-0.17	0.02-0.07	0.05-0.11	6-12
Loamy sand	0.11-0.19	0.03-0.10	0.06-0.12	9-14
Sandy loam	0.18-0.28	0.06-0.16	0.11-0.15	15-20
Loam	0.20-0.30	0.07-0.17	0.13-0.18	16-22
Silt loam	0.22-0.36	0.09-0.21	0.13-0.19	18-25
Silt	0.28-0.36	0.12-0.22	0.16-0.20	22-26
Silt clay loam	0.30-0.37	0.17-0.24	0.13-0.18	22-27
Silt clay	0.30-0.42	0.17-0.29	0.13-0.19	22-28
Clay	0.32-0.40	0.20-0.24	0.12-0.20	22-29

ZIMsched soil water model

The parameters for the ZIMsched soil water model are the same as those of the FAO 56 model. Additionally, two extra parameters need to be specified:

- **Sat. hydraulic conductivity**. The saturated hydraulic conductivity is also called the saturated drainage coefficient [mm/h]. Since it is known that the drainage of water from an unsaturated zone diminishes with diminishing soil moisture, water may not drain instantaneously from the time when the water is above field capacity. In such cases water will be available for transpiration for a short period depending on the magnitude of the saturated drainage coefficient.

- **Depth to groundwater**. The unsaturated soil zone extends from the ground surface to the groundwater table [m]. Based on the depth of the top soil and the root zone layers, the lower soil layer can be calculated. From this zone deep percolation to the groundwater takes place using the same drainage formula as for the root zone layer.

ZIMsched for Rice field soil water model

The ZIMsched for Rice field soil water model requires an extra parameter describing the maximum water depth that can be stored on the ground (often called detention storage):

- **Max water depth**. The maximum water depth [m] specifies the maximum water depth that can be held back on the ground surface before surface runoff takes place. From this storage water can evaporate and infiltrate into the top soil storage.

Note: It is possible to plant non-rice crop on ZIMsched for Rice soil. However, max water depth is always set to zero for non-rice crop and for periods when there is no crop.

Runoff model

When it rains on the ground surface, a certain amount of the water may run off, either as surface flow or as subsurface flow or both. The percolation to groundwater is calculated whether a surface runoff model is included or not. The task of the Runoff model is to calculate the surface flow part of the total runoff. Surface runoff may occur either when it rains, when the crop is over-irrigated, or when a combination of both happens (more details about the Runoff Model can be found in Appendix A, Section A.3.5).

When the runoff model option is selected, an 'f' direct surface flow runoff is calculated and subtracted from the rainfall. The remaining rainfall (called the net rainfall, PN) is input to the soil column water balance. If PN together with the applied irrigation cannot be stored in the soil (or on the ground surface, in case of rice fields), additional surface flow runoff may occur. If the irrigation node is connected to the river network, the calculated surface runoff will return through the return flow channel to the river. If the irrigation node is not

connected to a river with a return flow channel, the water will be lost from the model.

When the runoff model option is not selected, the total rainfall enters the soil column water balance. If the rainfall and/or the irrigation input cannot be stored in the soil surface (or on the ground surface, in case of rice fields), the surplus water is considered surface runoff and removed from the field. However, the surface runoff is not returned to the river, but lost from the model, even if the irrigation node is connected to a river with a return flow channel.

There are four Runoff model options available:

- **None**
- **Linear runoff model**, available for all crops (i.e. for all types of Soil model)
- **Modified SCS runoff model**, available for all crops except rice (i.e. for all types of Soil models except the ZIMsched for Rice field)
- **Modified SCS for Rice field runoff model**, available only in combination with the ZIMshed for Rice field Soil model

None (No runoff model)

If the 'None' runoff model is selected, the rainfall enters the soil column water balance. if rainfall and/or irrigation cannot be stored in the soil surface, the surplus water is removed from the field and the model.

Linear runoff model

The Linear runoff model assumes a linear relationship between the rainfall [mm] and the amount of surface runoff. The runoff will be generated according to two specified linear formulas, valid when the accumulated rainfall is above or below a given threshold.

The parameters for the linear runoff model are:

- **When Rainfall is lower than.** It is the threshold of the accumulated rainfall [mm] on the field below which runoff occurs. In this case the runoff follows the formula specified in the line below:

  Runoff = linear coefficient [ ] (fraction to be inserted in the formula) x Rainfall + constant (to be inserted in the formula) in [mm]

- **When Rainfall is greater than**. It displays the threshold of the accumulated rainfall [mm] on the field above which runoff occurs. If the rainfall is above the threshold, the runoff follows the formula specified in the line below:

  Runoff = linear coefficient [ ] (fraction to be inserted in the formula) x Rainfall + constant (to be inserted in the formula) in [mm]

Regardless of the specified parameters, the runoff will never be negative.

Note: When including the Linear runoff model, for any crop including rice, setting both coefficients in the linear equation to zero will have the same effect as not including a surface runoff model. In that case, any excess surface water will to return to the river.

Modified SCS runoff model

When the Modified SCS runoff model is selected, the surface runoff is estimated using the Soil Conservation Service (SCS) storm flow equation (USDA, 1985) /2/ as modified by Schulze (1995) and used in the ACRU agro-hydrological simulation model in South Africa (cf. Modified SCS under Section A.3.5.2 in Appendix A for further details).

The parameters to be specified for the Modified SCS runoff model are:

- **Soil moisture integration depth**. The soil moisture integration depth [m] which has an influence on the infiltration losses in the modified SCS model.

- **Coefficient of initial abstraction**. This coefficient determines also the infiltration losses.

Modified SCS for Rice field runoff model

The input parameters for the Modified SCS for Rice field runoff model are the same as those for the Modified SCS.

The difference between the Modified SCS for Rice field model and the Modified SCS model is that the space (storage) between the maximum ponded water depth (see the ZIMsched for Rice field model section) and the actual water depth is also taken into account in the calculation of the water volume of the potential maximum water retention of the soil, S.

In addition, excess water above the maximum ponded water depth at the end of a time step is added to the already calculated surface runoff (this can only occur when a specified irrigation application is used).

8.6.4 References

/1/ Allen, R.G., Pereira, L.S., Raes, D. and Smith, M., 1998. Guidelines for computing crop water requirements. FAO Irrig and Drainage Paper No. 56, FAO, Rome, Italy.

/2/ USDA, 1985. National Engineering Handbook, Section 4, Hydrology. United States Department of Agriculture (USDA), Soil Conservation Service, Washington DC, USA

9   Reservoirs

The Basin module accommodates multiple multi-purpose Reservoir systems.

Individual Reservoirs can simulate the performance of specified operating policies using associated operating rule curves. These define the desired storage volumes, water levels and releases at any time as a function of current water level, the time of the year, demand for water, and losses and gains.

For Reservoirs, which have a significant surface area compared to the catchment area, in which they are located, it may be important to extract the reservoir surface area from the catchment area. This is done by enabling '*Subtract area of irrigation users and reservoirs from catchment area to calculate runoff'* in the *Simulation specifications|modules* page*.

Note that in order to use this option the Reservoir must be located on the catchment node.

Inserting a reservoir

To insert a Reservoir in the model select the '*Add*' button under the *Reservoirs* ribbon in *Map view* and click on the desired location of the Reservoir. The Reservoir must be located on a river Branch.
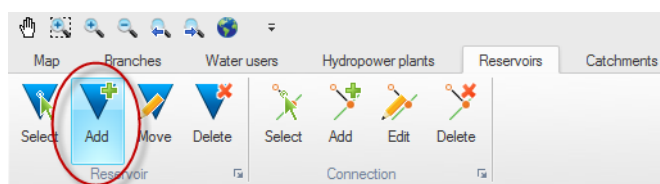
Figure 9.1   Inserting a Reservoir

The Reservoir may be connected to other Reservoirs and/or Remote flow control points by using the '*Add Connection*' button. The digitisation of a Connection must always be done in the direction of the flow.

Note that the mouse cursor changes to a cross when the cursor is properly placed.

Specifying reservoir properties

The *Reservoir properties* dialogue contains the following tabs:

- General
- Operations
- Users

- Remote Flow Control
- Storage Demand
- Reservoir Outlets

9.1 General

Under the *General* tab five reservoir properties are specified:

- Reservoir type
- Level-area-volume (LAV) table
- Characteristic levels time series
- Losses and gains time series (optional)
- Sediment distribution type

9.1.1 Reservoir type

Three types of storage/reservoirs can be modelled and the input requirements will depend on the reservoir type selected:

- **Rule curve reservoirs** regard the reservoir as a single physical storage and all users are drawing water from the same storage. Operating rules for each user apply to that same storage and the users compete with each other to fulfil their water demand.

- **Allocation pool reservoirs** also have physical storage, but the individual users have been allocated certain storage rights within a zone of water levels. An accounting procedure keeps track of the actual water storage in individual pools allocated for water supply users and in a single pool for downstream minimum flow releases (water quality pool). Thus, a particular water level is not uniquely related to a set of volumes in all pools (one can 'shift' some volume from one pool to another without any effect on water level).

- **Lakes** are specific reservoirs for which no operation rules apply. The outflow from a lake can be restricted by a spillway relationship. If no such relationship is given and the water level is at dead zone level (top of dead storage), all inflow that is not allocated to water users will flow out from the lake immediately. Lakes may be used to represent wetlands.

Note: When simulating systems with Reservoirs, it is recommended to use small time steps (days). During large time steps, water levels might move through several zones in the operating rule curves, making results inaccurate.

9.1.2 Level-area-volume (LAV) table

Computing water levels in a Reservoir requires the relationship between level (elevation/height), area and volume (LAV) to be known. This information must be specified as a table in a file:

Click on the [...] button and click *Browse...* to select an existing LAV table or click *Create a new file…* to open the '*Create a new file'* dialogue.

The LAV table is checked for physical plausibility. Particularly, it should hold that:

$$V(H(i+1)) \geq V(H(i)) + A(H(i)) \cdot (H(i+1) - H(i)) \qquad (9.1)$$

Where i is the i'th value in the table of increasing water level elevations H. In other words, for every step in elevation, the increase in volume should at least be the base area (at the previous level) times the increase in height. A smaller increase corresponds to a 'narrowing' Reservoir (as water level increases). A warning is issued at the end of the simulation to a 'narrowing' Reservoir is detected.

During the simulation, linear interpolations between the user-specified neighboring data triplets in the table is performed to arrive at a piece-wise linear LAV function.

Note: If the calculated Reservoir water level is outside the range of tabular values specified in a LAV file during a simulation, an error message will appear on the screen. No extrapolation is carried out beyond the range of the specified elevation values in the LAV file. If a preliminary simulation is nonetheless desired, a work-around is to specify the Reservoir level in the LAV table to start at zero and end at a very high level. This will avoid the necessity to extrapolate and thus avoids the error message.

9.1.3 Characteristic levels time series

The following characteristic levels in a Reservoir must always be specified:

- **Bottom level:** the bottom of the Reservoir
- **Dead zone level:** the minimum level from which water can be utilised (Top of dead storage). If the water level is below this zone water can only be lost due to evaporation or bottom infiltration.
- **Dam crest level:** the highest water level in the Reservoir before spill occurs. note that Dam crest level is not used for Lakes.

9.1.4 Losses and gains time series (optional)

The following three losses / gains time series can be included:

- Precipitation

- Potential evaporation

- Infiltration

To include losses/gains, enable '*Include losses and gains*', click on the button and click *Browse...* to select an existing time series or click *Create a new file…* to open the '*Create a new file*' dialogue.

To create a file with non-uniform values, the file must first be created with a uniform value and then edited in time series editor.

This is done by **clicking** [...] and selecting '*Edit in time series editor*'.

The losses/gains are included in the water balance calculations and the actual loss/gain in each time step (in terms of volume per time unit) will depend on the actual water level (and thereby surface area) in the reservoir at that time step.

If groundwater is defined in a downstream Catchment, infiltration from the Reservoir will be added to this groundwater. However, with the current implementation the Reservoir has to be a Catchment node as well. Therefore to observe the infiltration from a Reservoir to groundwater, the Reservoir must have an immediate upstream and downstream Catchment (and the latter must have a groundwater model specified).

9.1.5 Sediment distribution type

(Only relevant if Reservoir sedimentation is included in the Basin modules.)

If reservoir sedimentation is included in Basin modules, the Reservoir must be assigned a 'sediment distribution' type. The type depends on the shape of the Reservoir as well as its operation. There are four types of reservoir shapes and four types of reservoir operation (cf. Table 9.1).

Table 9.1   Reservoir shapes and Reservoir operations

Reservoir shapes	Reservoir operations
Type I (Lake)	Type I (Sediment submerged – continuous high pool level)
Type II (Floodplain-Foothill)	Type II (Moderate drawdown)
Type III (Hill-Gorge)	Type III (Considerable drawdown)
Type IV (Gorge)	Type IV (Normally empty)

The sediment distribution type is a weighted average of the reservoir shape type and the reservoir operation type as shown in Table 9.2.

Table 9.2   Guide to selection of sediment distribution types

Operations type	Shape type	Sediment distribution type
I	I	I
	II	I or II
	III	II
II	I	I or II
	II	II
	III	II or III
III	I	II
	II	II or III
	III	III
IV	All	IV

Where a choice of two sediment distribution types is given, or in borderline situations, the type should be selected according to whether reservoir shape or operation is expected to be most influential. This selection may also be guided by the sediment grain size, as shown in Table 9.3. However, in most cases sediment grain size has been found to be the least important factor influencing sediment distribution.

Table 9.3   Links between sediment grain size and sediment distribution type

Predominant grain size	Typical sediment distribution type
Sand or coarser	I
Silt	II
Clay	III

The sediment distribution type determines the formulas used to distribute the sediment load within a Reservoir. Based on this, the level area volume (LAV) curve is updated. The sediment distribution types reflect the tendency for sediment in lake-type reservoirs to accumulate in shallower water and in gorge-type reservoirs to accumulate in deeper water.

For more information, please refer to Morris & Fan (1997): Reservoir Sedimentation Handbook, McGraw-Hill, 1997, p. 10.32-10.39.

9.2   Operations

Under the *Operations tab* six different operating rules can be specified:

1. **Minimum release requirement** (optional): release required to support environmental flows in the river downstream of the reservoir. This release takes place as long as the water level is above the Dead zone level (Top of dead storage).

2. **Maximum release requirement** (optional): a restriction on the flood control release when the water level in the Reservoir is above Flood control level (or Spillway bottom level, if defined).

3. **Minimum operation level requirement** (optional): the minimum level at which water is supplied to water users. Below the minimum operation level, only minimum flow requirements are released. If minimum operation level is not specified, the model uses the Dead zone level (Top of dead storage) as the minimum operation level.

4. **Flood control level time series** (mandatory): the level above which water is released for flood control purposes.

5. **Guide curve level** (optional only for allocation pool reservoirs): the level below which the Reservoir is 'divided' into pools.

6. **Pool ownership for downstream river time series** (mandatory only for allocation pool reservoirs), cf. 9.3.3

All operation rules are specified as time series. This is done by enabling the relevant operating rule (for optional rules) and specify the time series

by **clicking on the** [...] button and selecting *Browse* to include an existing time series or selecting *Create a new file* to open the '*Create a new file*' dialogue.

Note that minimum and maximum release requirements may be influenced by remote flow requirements (cf. Remote Flow Control).

Based on the rule curve levels and reservoir characteristics, Reservoirs can be divided into different zones depending on the type of reservoir.

9.2.1   Lakes

All Lakes have an Inactive zone (Dead storage). If the Level-area-volume (LAV) table includes levels above Dead zone level and a Spill capacity table defined, Lakes also have an Inundation zone. In the Inundation zone excess water may be stored. The highest level defined in the LAV table thus represents the 'Dam crest level' for Lakes. If no Spill capacity table is given and the water level is at Dead zone level (top of dead storage), all inflow that is not allocated to Water users will flow out from the Lake immediately.
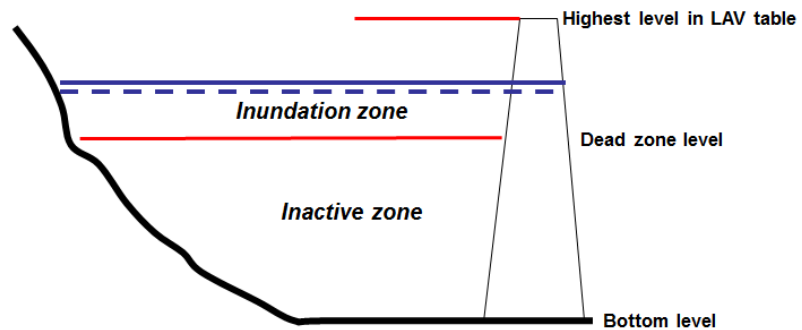
Figure 9.2   Zones in a Lake

9.2.2   Rule curve reservoirs

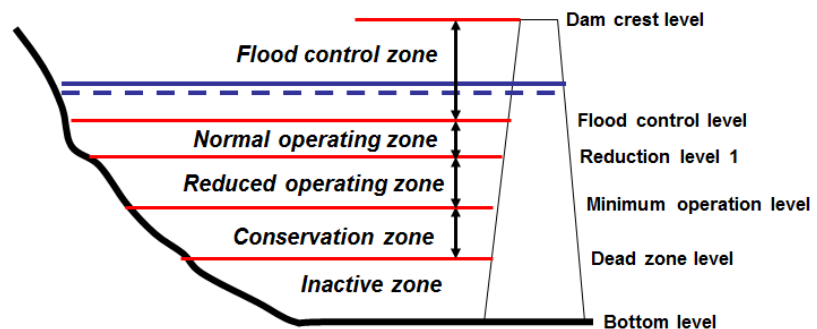Rule curve reservoirs are divided into five zones as illustrated in Figure 9.3,

Figure 9.3   Operating zones in a Rule curve reservoir

Flood control zone

This zone serves as storage buffer to diminish the impacts of high floods. Under normal circumstances the water level in the reservoir is kept at Flood control level to maintain optimal protection and reserve water for supply. If the water level is within the Flood control zone, water is released at a rate up to maximum downstream release. The release can be limited by spillway conditions.

If a Spillway bottom level time series is specified (see Reservoir Outlets, section 9.6) the lower level of Flood control zone is defined by the Spillway bottom level, no matter whether this is above or below the Flood control level.

Normal operating zone

This zone is between the Flood control level (lower level of the flood control zone) and the first Reduction level for a given water user. In the Normal operating zone all demands are fulfilled. The extend of this zone can vary for individual water users.

Reduced operating zone

If the water level is in this zone, a demand is only partially fulfilled. A variable number of reduction level curves and corresponding reduction factors can be specified for each connected water user as described in Section 9.3 '*Users*'. The lower limit of the Reduced operating zone is the Minimum operation level. If this level is not specified then the lower limit is the Dead zone level (Top of Dead Storage).

Conservation zone

If the water level reaches this zone, only downstream release (minimum release requirement for environmental flows is maintained). No water for usage is being released. The Conservation zone only exists if a minimum operation level has been specified.

Inactive zone (Dead storage)

If the water level reaches the Inactive (Dead storage) zone, water can only be lost due to evaporation or bottom infiltration.

9.2.3 Allocation pool reservoirs

In *Allocation pool reservoirs* the main storage is divided into four zones as illustrated in Figure 9.4:
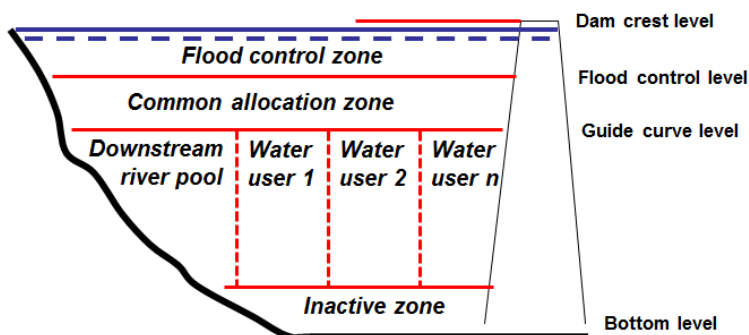
Figure 9.4   Operating zones in an allocation pool reservoir

The **Flood control zone** and the **Inactive zone** are identical to those of a Rule curve reservoir.

The **Common allocation zone** only exists if a guide curve level has been set below a flood control level thus allowing storage of water between these two levels. The guide curve corresponds to the top of the pool allocation zone.

The **Pool allocation zone** is divided into a pool for releases to the downstream river (sometimes called a 'water quality pool') and a number of water supply pools defined by the user. All these pools are purely conceptual accounting storages used internally in the program, and should not be regarded as physical storages.

9.3   Users

Under the *Users tab* five types of attribute data are specified for each water user connected to the reservoir:

- Priority
- Number of reduction levels
- Pool ownership time series (only available for Allocation pool reservoirs)
- Flow loss time series
- Flow capacity time series

9.3.1   Priority

You can connect multiple Water users, Hydropower plants and Reservoirs to a Reservoir. By default, connections are given a priority according to the sequence in which they are digitised. The priority can be changed in the priority column. Number 1 has first priority; number 2 has second priority, and so forth.

Note that the minimum release from the Reservoir always has a higher priority than all other users. In order for a user to effectively have first priority, the specified minimum release for the Reservoir must be set to zero and there must be no minimum remote flow requirements.

9.3.2 Number of reduction levels

You can specify operating rules for supplying water to the individual users. This is done by specifying the No. of Reduction levels. For each Reduction level a time series containing Reduction water levels and corresponding Reduction factors must be specified.

Note that Allocation pool reservoirs have Reduction thresholds (fraction of pool volume) instead of Reduction levels (reservoir water levels). Reduction thresholds are specified in the same way as Reduction levels.

To specify Reduction level time series for a Water user, click on the *small arrow* to the left of the Water user name. A grid cell then becomes available for each Reduction level.

Click on the grid cell and a [...] button appears in the right-hand side of the grid cell.

Click on the [...] button and click *Browse...* to include an existing time series or click *Create a new file…* to open the '*Create a new file*' dialogue. Two items must be specified: Reduction level and Reduction fraction.

When the reservoir water level falls below Reduction level 1 for a specific user, the actual extraction is calculated as the water demand times the specified Reduction factor 1. If the reservoir water level falls below Reduction level 2, a more drastic Reduction factor 2 is applied, and so on. Each user has its own set of reduction levels and corresponding reduction fractions and can have as many sets as required.

Figure 9.5 illustrates how different Reduction levels and factors can apply to different Water users. In the figure, a low priority user (e.g. industrial production) is getting its demand reduced earlier and more drastically, than a high priority water user (e.g. public water supply).
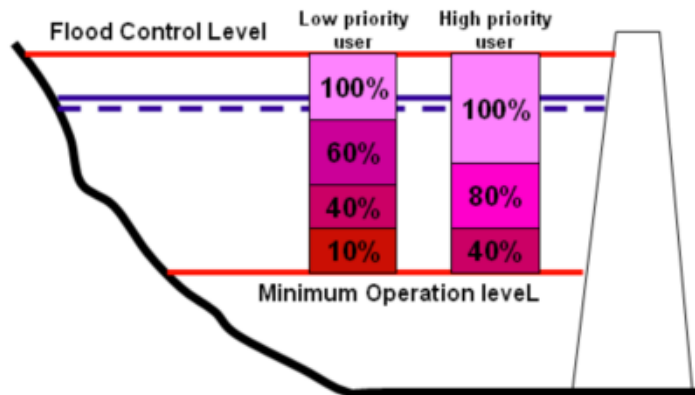
Figure 9.5   Illustration of reduction levels and reduction fractions for two users

Example:

Assuming two Water users extracting water from a Reservoir. The following rules apply for the two water user nodes:

The demand for both Water user nodes is 13.89 m^3/sec.Operating policy:

- *User node 1 (W4): Reduction level at:541 m, Reduction factor: 0.8*
- *User node 2 (W6): Reduction level at 540 m, Reduction factor: 0.6*

The initial water level in the Reservoir is 544 m, and the inflow is zero until the 20th January, where the inflow is 35 m^3/s. Figure 9.6 illustrates how the water level is falling as water is extracted from the Reservoir. When the water level reaches level 541 m, the withdrawal by user node 1 is reduced to 80%, and the water level falls with a lower rate. When the water level falls below level 540 m, the withdrawal to user node 2 is also reduced. Once the inflow to the Reservoir starts the water level is rising with a reduced rate as the withdrawal increases.
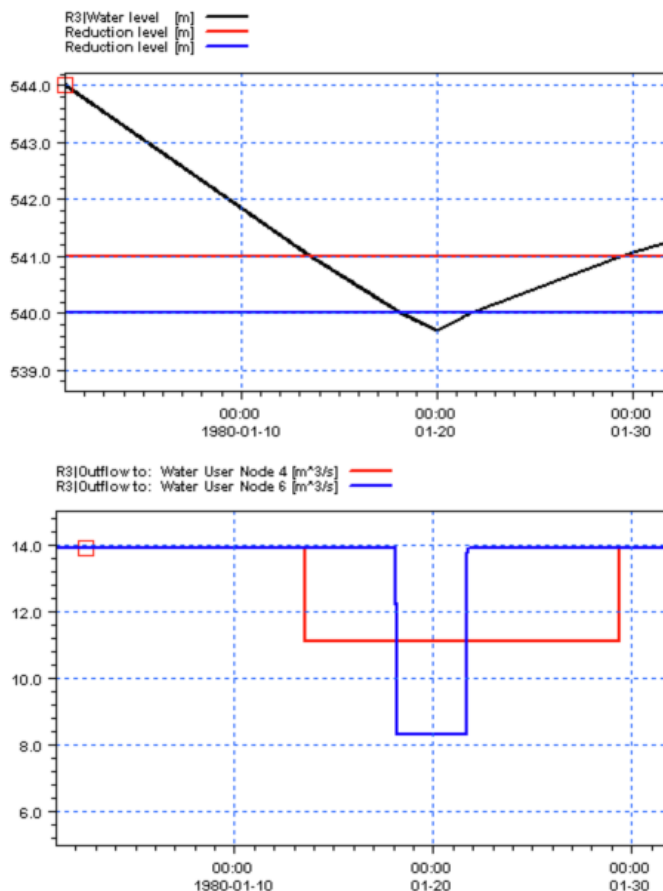
Figure 9.6   Water supply as a function of water level and reduction fractions

9.3.3   Pool ownership time series

(Only available for Allocation pool reservoirs).

Allocation pool owners can be Water users, Hydropower plants or other Reservoirs that extract water from the Reservoir. Additionally, the downstream river 'owns' an allocation pool (sometimes called 'water quality pool') that releases water to the downstream river. Pool ownership time series for the downstream river is specified in the *Operations* properties dialogue (cf. Section 9.2).

To add pool ownership time series to a user click on the '*pool ownership grid cell'*,

and a [...] button appears in the right-hand side of the grid cell.

Click on the [...] button and select *Browse* to include an existing time series or select *Create a new file*... to open the '*Create a new file*' dialogue.

Pool ownership time series must contain two items:

- Fraction of total storage owned by that water user;
- Fraction of inflow owned by that user.

Note that for both items the sum of the fractions owned by all users including the downstream release should add up to 1.0.

9.3.4 Flow loss time series

Supply connections may lose water due to seepage and /or evaporation. If these processes are considered to be of importance they may be included in the model as a time series that specifies the losses. Both seepage and evaporation can be specified as a fraction of the actual flow (dimensionless), or as flux (volume per time).

To add a flow loss time series, enable '*has flow loss*', click on the [...] button and click *Browse...* to include an existing flow loss time series or click *Create a new file…* to open the '*Create a new file*' dialogue.

The user-specified water demand D of the connected user is automatically adjusted to take the loss into account. The adjusted water demand D^* is calculated as follows:

$$D^* = \frac{D}{(1 - \text{loss factor})} \tag{9.2}$$

This means that the Water user demands will still be fulfilled if sufficient water is available in the Reservoir. This also applies if the flow loss is given as a flux.

If groundwater is defined in the Catchment where the Reservoir is located the seepage loss from the connection is added to this groundwater.

Note that if the Reservoir is located on a Catchment node, seepage loss is added to the next downstream Catchment if a groundwater model is included for that Catchment.

Note that for Water users, flow loss time series in supply connections may also be specified in the *dialogue*dialog.

9.3.5 Flow capacity time series

Supply connections may be assigned a flow capacity [Volume per time] that can never be exceeded. To add a flow capacity time series, enable

'has flow capacity', click on the [...] button and click *Browse...* to include an existing flow capacity time series or click *Create a new file…* to open the '*Create a new file'* dialog.

The flow capacity overrules all other rules that may try to force more water through the connection.

Example
A Water user may call for 5 m3/s. If the flow capacity of the supply connection has been set to e.g. 4 m3/s, the user will only receive max. 4 m3/s and hence suffer a deficit, even though there may be plenty of water at the source.

If the flow capacity of a supply connection is reached, the model will attempt to force the water in alternative direction. If this is not possible, the simulation will terminate with a message.

Note that for Water users, flow capacity time series in supply connections may also be specified in the Water user dialog.

9.4 Remote Flow Control

Remote flow controls are logical relationships between a Reservoir and one or more flow control points downstream from the Reservoir, possibly with many intermediate inflows and offtakes.

To connect a Reservoir to a remote flow control point, click the '*Add connection*' button under the *Reservoirs ribbon* in *Map* view. Connect the Reservoir to the desired downstream point (anywhere on downstream Branches, but not Water users). A regular River node is automatically inserted on the desired flow control point and adds the Remote flow control to the Reservoir properties. A Reservoir can have multiple Remote flow connections.

Under the *Remote flow control* tab minimum or maximum remote flow requirements can be specified.

To add a minimum or maximum flow requirement to a remote flow connection, enable '*min. flow requirement or max. flow requirement',* click on

the [...] button and click *Browse...* to include an existing flow requirement time series or click *Create a new file…* to open the '*Create a new file'* dialog.

During simulation, MIKE HYDRO will try to adjust the reservoir releases to meet the remote flow requirements after considering all inflows, extractions and flow delays due to routing. If the adjusted minimum reservoir release (to

meet the remote flow requirements) is higher than a specified minimum release requirement for the reservoir, the adjusted reservoir release determines the actual minimum reservoir release. Similarly, if the adjusted maximum reservoir release (to meet the remote flow requirements) is lower than a specified maximum release requirement for the reservoir, the adjusted reservoir release determines the actual maximum reservoir release.

9.5   Storage Demand

Transfers between two Reservoirs are specified under the 'Storage demand' tab of the receiving reservoir.
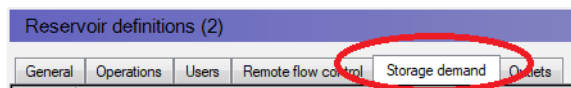
The connection between the reservoirs must first be digitized in the 'Map view' using the 'Add Connection'-button under the 'Reservoir'-ribbon. The digitization of a connection is in the direction of flow, i.e. from 'upstream' (supplying) reservoir to 'downstream' (receiving) reservoir. To model bi-directional transfers between two reservoirs, two connections must be digitized - one in each direction.

Once two reservoirs are connected, the connection will appear as user in the upstream reservoir's 'Users'-tab and in the downstream reservoir's 'Storage demand'-tab. Here, the critical water level or water volume that has to be maintained in the receiving reservoir must be set as 'Storage demand time series'. To do so, click on '…' and select either 'Browse...', to use an existing time series, or 'Create a new file…', to create a new one.

Note, that bi-directional transfers can only be operational, if the supply connections' priority is higher than that of any other user connected to the reservoir(s). Minimum flows still have higher implicit priority. Thus, any minimum releases are performed before transfers to users or other reservoirs.

9.5.1 Use downstream river check box (release 2019 update 1)
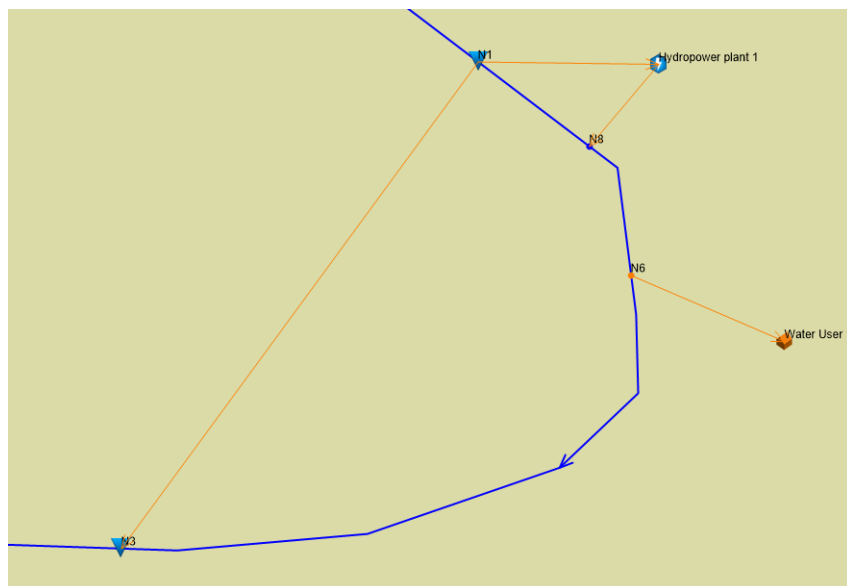
A new specification for the storage demand rule is included in release 2019, update 1. This new feature allows to route the storage demand water for a downstream reservoir through the river branch instead of through the direct connection between the two reservoirs. This option is only possible available, if the two reservoirs are located or connected by the same river branch. If they are not, the feature is meaningless and will be ignored by the numerical engine.

The feature can be found in the 'Storage demand'-tab, where a new checkbox 'Use downstream river channel' is included:
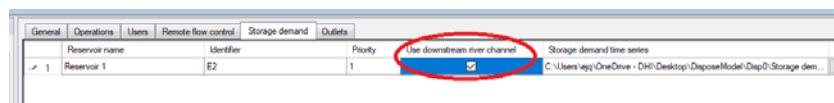
Figure 9.7   Check box in Storage demand tab

If the checkbox is ticked and the requirements as mentioned above are fulfilled, the release will be routed through the channel. Note, that for numerical reasons, there must be at least one river node specified in the branch between the two reservoirs, to route the water through the channel.

This feature allows storage demand water to be released through a connected hydro power plant, which is an operational requirement in many reservoir systems.

Note, that this option will not prevent intermediate downstream users to "steal" water, as the local priority algorithm applies a "first-in-line line-first-serve" rule. To avoid this side effect of the feature, the checkbox should be kept unticked.

9.6 Reservoir Outlets

From Release 2019 update 1, MIKE HYDRO Basin will include outlet specifications for all types of reservoirs (Lakes, Rule Curve and Pool Allocation reservoirs). The approach extends MIKE HYDRO Basin's dam reservoir operation capability while maintaining the simplicity of the modelling process.

Reservoir releases during flood control operations can now be controlled by one or more outlets for each connected node. To avoid unnecessary complexity, the number of outlets connected to water users or reservoirs is limited to one.

Figure 9.8 illustrates an example, where a reservoir is connected to a water user, a hydro power plant, and a reservoir:
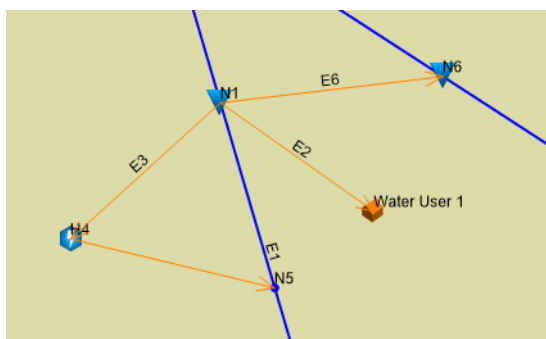
Figure 9.8   Example of connections to reservoir

Outlets connected to different downstream elements in the network seldom have identical capacities and conditions for releasing water. It must therefore be possible to specify individual outlet parameters for each outlet. This will also provide a more realistic water level simulation in response to extreme inflows.

The in previously MIKE HYDRO Basin versions existing spillway tab has been replaced by the 'Outlets'-tab, as illustrated in Figure 9.9 below:

Figure 9.9   Outlet tab in user interface

Figure 9.9 illustrates an example where three outlets have been specified: 1) to the downstream channel, 2) to water user 1, and 3) to the reservoir N6. Notice, that no outlet has been specified for the hydropower plant node, since the hydropower plants has its own specifications to control the outflow through its outlet for power generation.

Example: Specifications of outlet_1 in the outlets table:

1. **ID**: Outlet_1; is the name of the outlet and used as item name in the results file.

2. **Location** states the name of the connected node or 'Downstream' for a connection down the river.

3. **Type**: Two types of outlets are available:

    a. Controlled, as an outlet for which the outflow can be decided by the reservoir operation by closing the outlet partwise or completely.

    b. Uncontrolled, where the outflow is determined by the hydrodynamic conditions of the reservoir and the outlet (reservoir water level and outlet bottom level). In other words, the outflow of an uncontrolled outlet cannot be determined by the reservoir operations.

4. **Use Maximum Flow**: This parameter specifies a possible capacity limitation of the outlet. The maximum flow can be specified in three ways:

    a. None: there is no maximum outflow limitation. It is the default value.

    b. Table: the maximum flow is determined by the hydrodynamic conditions of the reservoir and the outlet bottom level, described by a Q-H table. Q as the maximum outflow is derived by the water depth relative to the outlet bottom level (see point 5.), H.

    c. Timeseries: the maximum flow of the outlet is determined by a discharge time series.

5. **Use outlet bottom**: if the checkbox is ticked, the user must provide a time series for the bottom level of the outlet. If unchecked, the considered outlet bottom level is the top of dead storage or, if one exists, the minimum operational level of the reservoir, depending on the type of connected node.

6. **Outlet Bottom TS**: if the "Use of outlet bottom" check box is ticked, the user must create, or select a time series for the outlet bottom levels. The specified item in the time series must be of water level type.

Note: If an outlet is of uncontrolled type, only a table can be used to determine the maximum outflow. As it is uncontrolled, that value is used for the simulation as routed water through the outlet. In a controlled outlet, the Q value from the table gives the outlet's maximum possible outflow. The actual outflow can be much lower or even zero.

Additional outlets connected to the downstream channel can be added by clicking the '+'-button. In this case, the reservoir operations will release water starting from the outlets with no specified bottom outlet followed by the ones from lowest to highest outlet bottom level. Furthermore, outlets can, if more than one is present, be deleted by selecting them and clicking the '-'-button.

The reservoir will only release water up to the capacity of the outlets (plus release to hydro power plants). An exception is, when the water level of the reservoir raises above the crest level of the reservoir. In this case it is understood that the spill doesn't go through any of the existing outlets, but over the crest level as an uncontrolled spill. In this case, it is possible to model a spill over the crest level as a controlled and unlimited outlet with a bottom level equal to the crest level.

Default controlled rules for controlled outlets
A number of default assumptions apply:

1. When no bottom level is specified for an outlet, the model assumes that the downstream channel has a bottom outlet at the dead storage level;

2. When no bottom level is specified, and the outlet is connected to a water user, the bottom level is assumed to be at the minimum operational level of the reservoir. If no minimum operational level is present, the dead storage level is used;

3. For a controlled outlet connected to a water user, the outlet will release water until the water user demand is completely satisfied, or the reservoir runs empty;

4. For a controlled outlet connected to another reservoir, the outlet will be controlled by the storage demand requirements. When these are satisfied, the outlet gate will be closed to stop any further release;

5. When an outlet is created by pressing the '+'-button in the outlets table, it will by default be created as controlled outlet with no limitation in its outflow capacity. Furthermore, no outlet bottom level will be specified. By

doing so, it is ensured that all MIKE HYDRO Basin models created with MIKE versions before Release 2019 update 1 can run with current and future versions of the software and obtain equivalent numerical results;

6. There is no specification of outlets for hydro power plants, since these have a complete specification of the outflows for power generation and its hydrodynamics limitations with respect to the reservoir water levels;

7. It is assumed that the non-specified outlets to hydropower plants are by default-controlled outlets, controlled by the hydropower plant parameters, defined in the hydropower plant table.

It is relevant to mention that the downstream releases from the reservoir are affected by the existence of connected hydropower plants, particularly if these have excess capacity. In this case the hydropower plant will use the downstream release and return it back to the river.

Order of use when several outlets in the downstream direction

If a reservoir has two or more outlets discharging directly to the downstream river channel(i.e., not discharging through a hydropower plant or a water user with return flow), release will start from the outlets with lowest bottom level. When the capacity of this outlet has reached its maximum capacity, the next lowest outlet will be used, etc. Finally, when the the outlet with highest bottom level has reached its maximum outlet capacity, the reservoir will stop releasing water for the given time step. If two outlets have the same bottom level, the order is irrelevant for the simulation and the model will just release in the order given by the table.

Backward Compatibility of the legacy Spillways tab.

In models created before Release 2019 update 1, there was an optional specification of spillways, which is a simplified specification of an outlet management system. In a model created in a MIKE HYDRO Basin 2019 (not update 1!) or previous, the spillway tab looked like the following:

Figure 9.10   Values in old Spillway tab in User Interface

In the table a spillway capacity table, a spillway bottom outlet level time series, and a bottom outlet capacity time series have been specified. Opening the example above with a version including the outlets module, it is automatically translated to the following:

Figure 9.11   Values in new Outlets tab in User Interface

Two outlets to the river are created, 'Outlet_1', which is a controlled outlet with a maximum outflow specified by the 'Spill capacity table' and its outlet bottom given by the 'Spillway bottom levels' time series.

The second outlet, 'Outlet_2' is a controlled outlet as well. It is specified by the time series 'Bottom outlet capacity' as the maximum outflow. Besides, the outlet bottom TS is given explicitly as the top of dead storage, using the reservoir characteristics time series.

Note that, when creating a new reservoir, the default parameters for outlets are enough to run any model. Thus, the existence of the outlet parameters does not necessarily make the basin modelling more complicated than what is has been in the previous version of the software.

Outlet Simulation results

The simulation will from now on include extra items, specifying the outflow at each outlet. These appear as sub-items for each reservoir node in the results tree as seen in Figure 9.12.
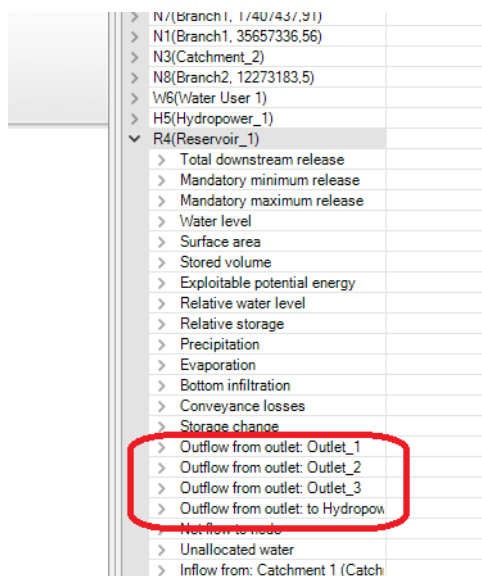
Figure 9.12   Sub-items for reservoir nodes in output

As one can see, there is an additional item for the outflow to the hydropower plant, even though there is no explicit specification of its outlet.

The item called "Downstream Spill" has been removed from Release 2019 update 1 on.

9.7   Guide Curve Level (for Global Ranking)

When Global ranking has been selected as water allocation rule (see 12 Global Ranking *(p. 153)*), at least one Guide curve level must be defined for each Reservoir. Otherwise, if no Guide curve level is defined for a Reservoir, then the reservoir will not release any water at all (except for the basic reservoir operations).

A Guide curve level defines the level to which water may be released to meet the water demand of a Water user (For Global ranking, Water users are defined by the unique Ranking number of their Supply connections). The Guide curve level must be defined above Dead zone level (or Minimum operation level if defined).

Only Water users with corresponding Ranking numbers (priorities) for which Guide curve levels are defined will receive water.

The reservoir will determine whether to release water for a given global ranking supply connection by verifying two conditions:

1. The water level of the reservoir is above the water level stated in one of the guide curve level for the current date (data given in a time series), and

2. The global priority of the supply connection to the water user is higher (lower numerical value) than the priority level related to the same guide curve level of the reservoir.

10 Hydropower

MIKE HYDRO Basin can perform advanced hydropower simulations for either existing systems or for evaluation of the feasibility of new developments. A Hydropower plant extracts water from one or more reservoirs, produces power according to effective head difference and power efficiencies, and returns water to one or more downstream locations.

It is hence required to initially define at least one Reservoir in the model to be able to connect and include a hydro power plant within the model setup (illustrated in figure below).

Figure 10.1   Hydropower plant defined with a single reservoir connection and return flow definition to river location downstream of reservoir

Calculation of hydropower can include special options such as conveyance and head losses as well as tail water level and backwater effects from cascading reservoirs.

A hydropower plant is equivalent to a single hydropower turbine. If the user would like to simulate two or more turbines at a single reservoir, then one hydropower plant should be defined for each turbine.

Inserting a hydropower plant

Hydropower generation is simulated by inserting a hydropower plant and connecting it to a reservoir. Return flow back to the river is simulated by connecting the hydropower station to one or more downstream locations.

To insert a Hydropower plant, select the '*Add*' button under the *Hydropower plants* ribbon in *Map view* and click on the desired location of the Hydropower plant in the map.
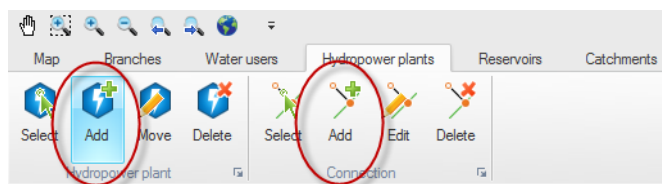
Figure 10.2   Insert a Hydropower plant

Note that the location is for visual purposes only and has no impact on the model setup.

The Hydropower plant is connected to the reservoir and the return flow point by using the '*Add Connection*' button under the *Hydropower plants* ribbon in *Map view.*

Note: The formula used to calculate the hydroelectric power produced is described in Appendix A (Equation (A.2.1) under A.2 Hydropower - Formula).

Specifying hydropower properties

In the *Hydropower plants properties* dialogue the following properties can be specified:

- Power Demand Time Series
- Head Approximation
- Use Minimum Release from Reservoir Option
- Power Efficiency Table
- Tailwater
- Turbine Flow Constraint Table
- Head Loss Table

10.1 Power Demand Time Series

The *Power Demand* time series file contains the five following items, which are all selected while selecting the time series file, from the table in the 'Select item' tab:

1. **Target power**. Target power demand, either as Target Power [MW or equivalent] or as Water Flow [m^3/s or equivalent]. The definition of Target power demand is mandatory.

2. **Installed capacity**, Specified as Power [MW or equivalent] or as Discharge [m3/s or equivalent]. The use of Installed capacity data is mandatory..

3. **Surplus capacity fraction**. Describes the Surplus capacity usage with a dimensionless factor. Fraction between 0 and 1. If this value is greater

than 0, then a fraction of excess turbine capacity can be used for releases triggered by flood control, global ranking, and storage demand rules. The use of Surplus capacity fraction is optional and is included in the simulation only when the corresponding option is ticked (the item must have been selected beforehand, while selecting the time series).

4. **Minimum head**. Minimum head for turbine operation, specified as Head Difference [m or equivalent]. If head (difference between reservoir level and tailwater level or **reservoir bottom level if no tailwater is specified**) drops below this threshold, no water is routed through the turbines, regardless of power demand. The use of Minimum head is optional and is included in the simulation only when the corresponding option is ticked (the item must have been selected beforehand, while selecting the time series).

5. **Minimum flow.** Minimum flow for turbine operation, specified as Discharge [m³/s or equivalent]. The use of Minimum flow is optional and is included in the simulation only when the corresponding option is ticked (the item must have been selected beforehand, while selecting the time series).

The option to produce more power than demanded (up to installed engine capacity) at times where there is a surplus of water in the Reservoir is applicable when the reservoir water level is above Flood control level. Water that would otherwise be spilled can be routed through a turbine. Any release that - due to limited turbine capacity - cannot be exploited for hydropower generation is spilled from the Reservoir.

In addition, releases triggered as a result of global ranking and storage demand rules are also routed through hydropower, provided that capacity is available and the amount to be released exceeds the minimum flow required for turbine operation.

10.2 Head Approximation

Calculation of hydropower demands and actual generation is subject to numerical inaccuracy when tailwater and/or power efficiency tables are used. There are two approximations available

- **explicit** method based on head in the supplying reservoir at the start of the time step, or
- **time step average** method based on the average head at start and end of the time step.

10.3 Use Minimum Release from Reservoir Option

If this option is enabled the minimum release (as specified under the *Operation tab* in the *Reservoir properties* dialogue) is assumed to be routed through

the hydropower station/plant and to be contributing to the total flow necessary for generating the required hydropower.

Note that this option must only be used when releases from the Hydropower plant and releases from the Reservoir are routed to the same river.

Example
If the hydropower demand requires a discharge of 100 m³/s through the tur-bines and the downstream release has been specified as 10 m³/s. Enabling this option will give a total release of 100 m³/s. If it is disabled the total release will be 110 m³/s provided there are no restrictions in water delivery for hydropower.
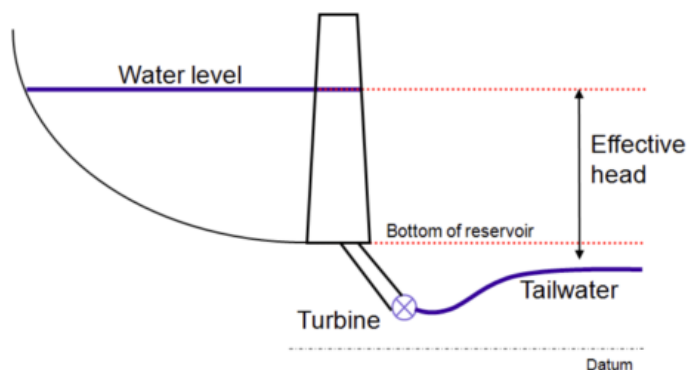
10.4 Tailwater

Figure 10.3   Effective head as difference between reservoir water level and tailwater level

Tailwater table

The *Tailwater* table specifies tailwater level as a function of release from the reservoir.

To include a Tailwater table, enable '*Use Tailwater table*' and click [...] to browse to an existing table or to create a new table.

Note that a discharge-dependent tailwater level necessitates iterations in the solution of the water allocation problem (demand becomes a function of tail-water level, which depends on reservoir release, which finally depends on demand). Thus, for precise results, often a daily simulation time step should be chosen in connection with hydropower simulations.

Use downstream release from reservoir

Besides the flow through the turbines other releases through spillways and gates may contribute to the tailwater level. If this is the case, enable '*Use downstream release from reservoir'*, and this flow together with the flow through the turbines will be used to estimate the tailwater level in the Tailwater Table.

Backwater effects

In cascades of reservoirs, tailwater elevation for a hydropower station can be determined by water level in the next reservoir downstream rather than by discharge from the supplying (upstream) reservoir. If this is the case for a particular model, enable the '*Has backwater from reservoir below*' option.

10.5 Turbine Flow Constraint Table

The Turbine flow constraint table gives the maximum turbine flow rate as a function of head difference.

To include a Turbine flow constraint table, enable '*Use Head constraint for maximum turbine flow*' and click ... to browse to an existing table or to create a new table.

10.6 Power Efficiency Table

The *Power efficiency* table specifies power (machine) efficiency as a function of head difference, discharge, or both. When a Power efficiency table is used, the Power Demand time series will be adjusted accordingly.

Three options are available for the Power efficiently table:

1. Efficiency as a function of head
2. Efficiency as a function of flow
3. Efficiency as a function of head and flow

To include a Power efficiency table, enable '*Use Power efficiency table*', select the table option, and click ... to browse to an existing table or to create a new table.

Note that the efficiency calculation necessitates iterations in the solution of the water allocation problem. Thus, for precise results, a daily or shorter simulation time step should be chosen when the Power efficiency table is used.

10.7 Head Loss Table

Significant head loss in hydropower systems can be caused by friction in the pipes connecting reservoir and turbines (conveyance loss). This can be included in the simulation by enabling '*Use Head loss table*' and specifying the head loss table (head loss Δh as a function of discharge Q) - by clicking on the [...] button to select an existing table or to create a new table.

Specifications of a head loss table is optional, and if no table is provided, head loss will not be taken into account.

11　Calibration Plots

It's possible to create calibration plots at any point in a river network. This can be useful when checking model results against measurements made downstream of water management locations such as reservoirs or diversions to water use locations.

The calibration plot definition functionality is located in the setup tree under 'Calibration plots'.
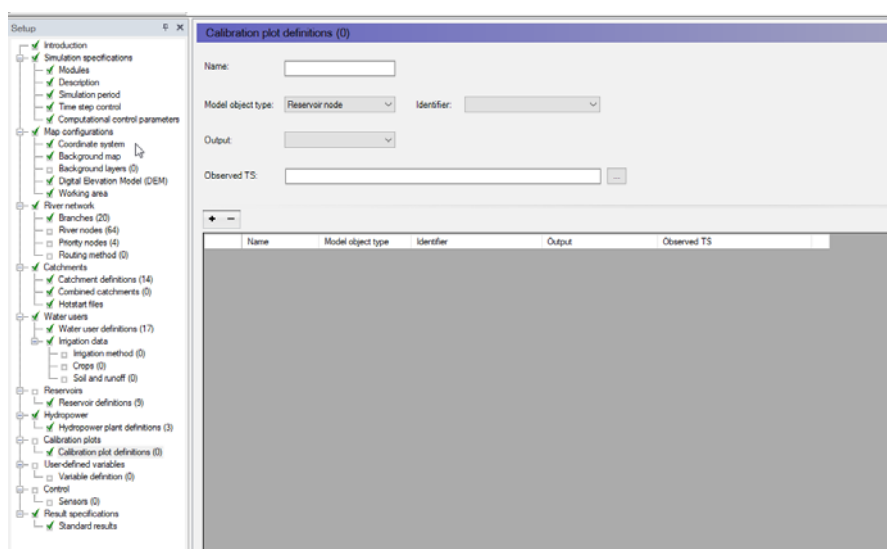

Figure 11.1　Calibration plot definition dialog in User Interface

To create a calibration plot definition, click on the '+' button. Calibration plot definitions can also be removed by clicking on the '-' button.

The following parameters need to be specified for each calibration plot created:

1. A descriptive **Name**.

2. The **Model object type**. The following types are available:
   – River node
   – Reservoir node
   – Water user node
   – Hydropower node
   – River reach

3. The model object Identifier. The list of available identifiers is filtered based on the model object type.

4. The Output type. The output type is also filtered based on the model object type.

5. An Observed TS to compare to the selected model output in the calibration plot.

Calibration plots are written to the simulation results directory after a simulation run.

12 Global Ranking

Global ranking can be used to allocate water in a river basin using a priority ranking scheme.

When global ranking is selected, all other allocation rules and priorities are ignored. This includes local priorities, reservoir storage demand rules, and reservoir demand reduction rules.

Note: If Allocation pool reservoirs are included in the present version, they will behave as Rule curve reservoirs when using global ranking.

Note: Routing and bi-directional connections cannot be used with the current implementation of global ranking.

12.1 Global Ranking Parameters

When global ranking is activated, all supply connections in the model setup automatically appear in the global ranking dialogue, and each supply connection must be assigned a unique ranking number. The following overall conditions apply to the global ranking feature:

- Ranking numbers are float numbers

- Ranking priority decreases with increasing ranking numbers.
  That is, the supply connection with ranking number 1 has first priority, the supply connection with ranking number 2 has second priority, and so on.

- A ranking number is unique. Ranking numbers cannot be shared and each supply connection must have a unique ranking number.

- For Water users with more than 1 supply connection, consecutive ranking numbers should be assigned to ensure consistent priority.

- All supply connections must be assigned a ranking number.

- Only Reservoirs with defined Guide curve levels (pg. 142) will release water to fulfil demands.

Ranking numbers may be displayed in *Map view* by selecting 'GlobalPriority' in the label field for supply connections (in the Symbology tree).

The Global Ranking algorithm
The global ranking allocation algorithm allocates water during each time step as follows:

- First, the algorithm estimates uncontrolled available water at all locations in the model. This includes runoff from catchments, as well as minimum releases from reservoirs and bifurcated water flows.

- Then, unallocated water is allocated by priority order given physical flow constraints until no more unallocated water is left.

This procedure is executed for every time step of the MIKE HYDRO simulation.

When using the global ranking algorithm, users should be aware of how the algorithm handles return flows, bifurcations, and reservoirs.

Return flows

During the allocation process, return flows are estimated and added to the unallocated supply available to downstream users. However, return flows are estimated sequentially as the algorithm moves through the ranking list. Therefore, if a user is located upstream of a downstream user with higher priority, the upstream user's return flow is not included in the available supply to the downstream user. In some cases, this means that a downstream user can cause an unnecessary shortage upstream because the supply available to the downstream user is underestimated.

Consider the example in Figure 12.1. In the example, the total supply available is 2 m^3/s and the net demand is 1.75 m^3/s (1 m^3/s + 0.5*1 m^3/s + 0.5*0.5 m^3/s). Therefore, supplies should be sufficient to meet demands.
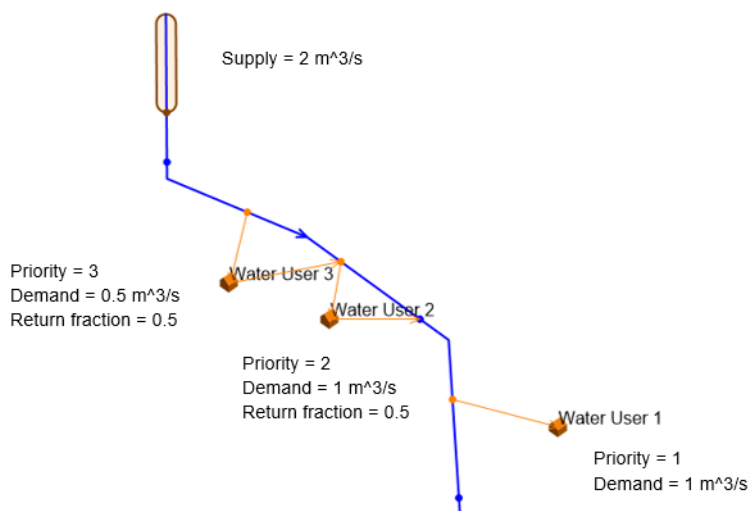

Figure 12.1   Return Flow example

However, the global ranking algorithm will not allocate any water to Water User 3 in this example. This is because Water User 3 has the lowest priority of the three and is also the most upstream location. The water allocation sequence therefore does not consider a return flow from Water User 2 as part of the available supply to Water User 1, nor a return flow from Water User 3 as part of the available supply to water users 1 and 2. If the priority sequence is reversed, however, no shortages are simulated because return flows from the upstream users are included in estimates of available supply to the downstream users.

Reservoirs and bifurcations

If reservoirs or bifurcations are present in the model, the algorithm can make additional reservoir releases or alter the distribution of flows through bifurcations to meet priority demands. Additional reservoir releases are only possible if a guide curve has been defined at the supplying reservoir with a lower priority than the downstream demand (see Guide Curve Level (for Global Ranking)). If more than one reservoir or bifurcation is available to meet a priority demand, then the supplying reservoir or bifurcation is determined based on 'model distance' rather than priority. The behavior of reservoirs and bifurcations is demonstrated with the following two examples, the first with two parallel reservoirs and the second with a bifurcation and a reservoir.

Reservoir example

In the example in Figure 12.2, water demands are the same as in the previous example. The system now includes two upstream reservoirs. Reservoir 1 and Reservoir 2 have guide curve levels defined and both guide curve levels have priorities that are lower than all three of the downstream demands. Therefore, both reservoirs can make releases to meet downstream demands. However, the distribution of downstream demands between the two reservoirs is determined by the principle of 'model distance', rather than the relative priorities of the two guide curve levels. The model distance is defined as the number of nodes between two model objects.

In the example, the global ranking algorithm will release 1 m^3/s from Reservoir 2, along with 1.5 m^3/s from Reservoir 1. Because the model distance from Water User 1 to Reservoir 2 is smaller than the distance to Reservoir 1, Water User 1's demand is met by releases from Reservoir 2. On the other hand, the demands at other two water use locations can only be met by Reservoir 1 due to water balance constraints.
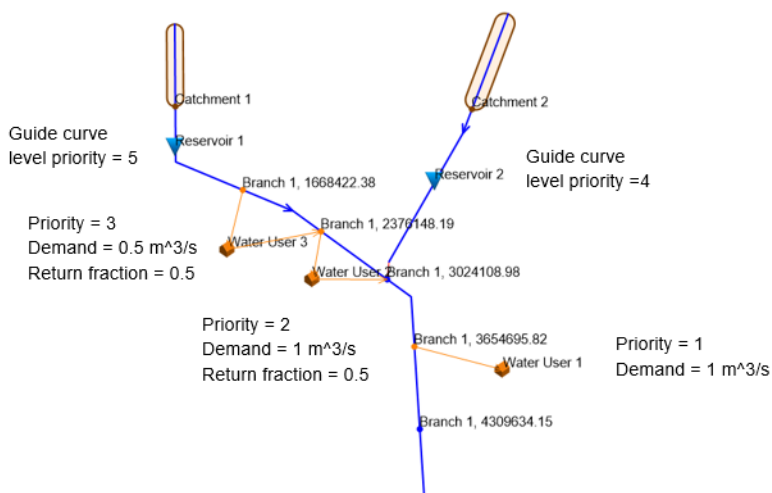
Figure 12.2   Reservoir example

Because of the way that return flows are handled by the global ranking algo-rithm (see previous section), the total reservoir release from the two reser-voirs (2.5 m^3/s), is higher than the net demand (1.75 m^3/s).

Bifurcation example

The global priority algorithm will also re-direct water away from bifurcations in order to meet priority demands. Let us consider bifurcation example in Figure 12.3. In the example, global ranking has not been applied. Therefore, the bifurcation routes flows away from the most downstream water user (Water User 1), resulting in a shortage. In the example, no releases are made from the upstream reservoir.
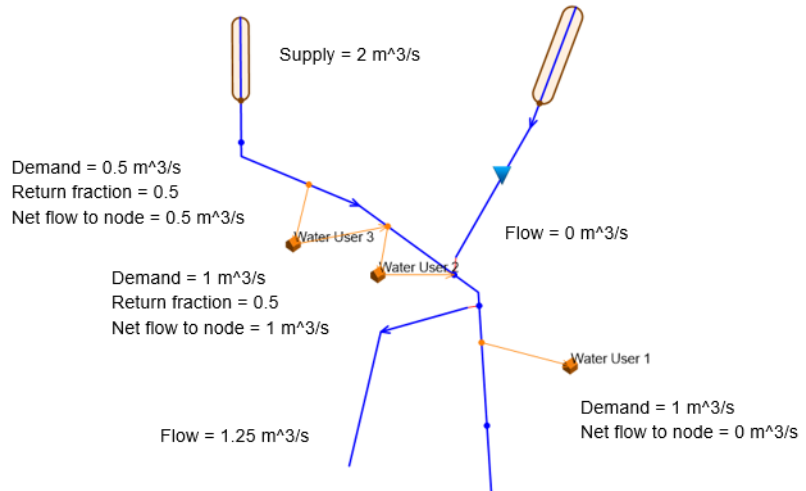
Figure 12.3   Bifurcation example before application of global ranking

After the application of global ranking, the ranking assigned to Water User 1 routes water away from the bifurcation in order to meet Water User 1's demand (Figure 12.4). Because the model distance to the bifurcation node is shorter than the model distance to the reservoir, Water User 1's demand is met by re-routing from the bifurcation rather than releasing from the reservoir.
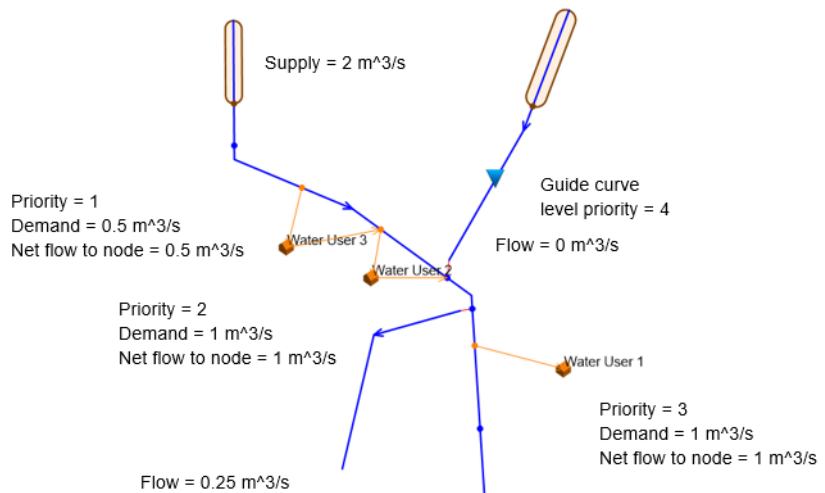

Figure 12.4   Bifurcation example after application of global ranking

13   Sediment Transport

A simple sediment transport option is available in MIKE HYDRO Basin for use in connection with Reservoir sedimentation simulations.

The Sediment transport page and the Reservoir sedimentation simulations are enabled through activating the *'Include reservoir sedimentation'* option in the *'Simulation specifications/modules'* page.

The overall approach for the sediment transport and reservoir sedimentation feature is, that sediment is transported un-disturbed through the river network and potentially deposited in Reservoirs.

The input required for the sediment transport calculations are sediment properties for the sediment fractions included. These definitions are made in the *Sediment properties* dialogue.

Additionally, sediment loads must be defined. These are defined as time series input through a combination of a Total sediment load time series and sediment fraction percentage time series. Sediment load are defined as input to Catchments and consequently, added as sediment load to the river at the location of the specific Catchment node.

Sediment transport and Reservoir sedimentation approach

The reservoir sedimentation module is not a sediment transport model. Sediment is supplied by catchments and is routed through the river network under the assumption that no erosion or deposition takes place in the rivers and deposition of sediments will solely occur in the Reservoirs.

If a reservoir is filled with sediments or other conditions prevents inflowing sediments to deposit in the specific Reservoir, sediment surplus will be transported further downstream in the system. The sediment is routed to the river downstream of the reservoir and to users supplied by the Reservoir. Sediment is distributed between connected rivers and users according to the discharge distribution.

Reservoir sedimentation change the properties and storage capacity of the reservoirs. The Level-Area-Volume (LAV) curve of each reservoir is continuously updated during a simulation based on the amount of sediment deposited in the reservoirs.

The frequency at which LAV is updated during simulation can be controlled by the user. The update frequency is specified in the *Time step control* dialogue (cf. 4.4 Time Step Control *(p. 46)*).

If the bottom level of the Reservoir due to deposition reaches the dead zone level, a warning will be issued, and no more sediment will be deposited in this reservoir.

13.1   Sediment Properties

If reservoir sedimentation is included in Basin Modules the following Sediment properties must be defined:

- Density
- Porosity
- Settling velocities

Density is assumed equal for all sediment types. By default the model uses a density of 2650 kg/m³, which is the density for quartz.

Porosity (the volume of pore-space in sediment) is assumed equal for all sediment types. By default the model uses a porosity of 0.4 which is the typical porosity of naturally deposited, non-cohesive sediments.

The sediment can be defined by one or more grain size classes in the table. Each grain size class must be assigned a settling velocity. To add a sediment grain size class click the '+' in the tool bar above the sediment class table and specify the associated settling velocity.

Table 13.1 shows some average settling velocities for various grain sizes

.

Table 13.1   Sieved diameter d_s, Spherical diameter dv, fall (or sedimentation) diameter d_f and associated settling velocities w in water with a temperature of 10ºC and 20ºC.Source: Engelund & Hansen, 1972: 'A monograph on sediment transport in alluvial streams'

d_s (mm)	d_v (mm)	d_f (mm)	w (10ºC) (m/sec)	w (20ºC) (m/sec)
0.089	0.10	0.10	0.005	0.007
0.126	0.14	0.14	0.010	0.013
0.147	0.17	0.16	0.013	0.016
0.208	0.22	0.22	0.023	0.028
0.25	0.25	0.25	0.028	0.033
0.29	0.30	0.29	0.033	0.039
0.42	0.46	0.40	0.050	0.058
0.59	0.64	0.55	0.077	0.084
0.76	0.80	0.70	0.10	0.11
1.25	1.4	1.0	0.15	0.16
1.8	1.9	1.2	0.17	0.17

Note that the sediment properties are global values used throughout the entire model. It is not possible to make spatial variation of definitions for sediment fraction properties.

14 Water Quality

MIKE HYDRO includes options for Water Quality (WQ) simulations. WQ-simulations in MIKE HYDRO utilises the MIKE ECO Lab framework and hence, offer a versatile and extremely flexible options for WQ-modelling. Water Quality simulations can be performed using either standard, predefined MIKE ECO Lab templates or user-defined templates with tailored Water Quality models.

14.1 Notes on MIKE ECO Lab

MIKE ECO Lab is a numerical lab for Ecological Modelling. It is an open and generic tool for customising Aquatic Ecosystem models to describe water quality, eutrophication, heavy metals and ecology. The module is mostly used for modelling water quality as part of an Environmental Impact Assessment (EIA) of different human activities, but the tool is also applied in aquaculture for e.g optimizing the production of fish, seagrasses and mussels. Another use is in online forecasts of water quality.

The need for tailor-made ecosystem descriptions is big because ecosystems vary. The strength of this tool is the easy modification and implementation of mathematical descriptions of ecosystems into the hydrodynamic MIKE engines.

MIKE ECO Lab utilises so-called Template files with the filename '.ecoLab' (in the following named MIKE ECO Lab File). The MIKE ECO Lab template files are customized collections of equations and parameters required for a specific type of Water Quality simulations. The user may use one of the predefined MIKE ECO Lab Templates, which is installed together with the MIKE Zero installation - or may choose to develop own model concepts. The module can describe dissolved substances, particulate matter of dead or living material, living biological organisms and other components (all referred to as state variables in this context).

The module was developed to describe chemical, biological, ecological processes and interactions between state variables, but it can also describe the physical sedimentation process of components. State variables included in MIKE ECO Lab can either be transported by advection-dispersion processes based on hydrodynamics, or have a more fixed nature (e.g. rooted vegetation).

More details on MIKE ECO Lab can be found in the specific MIKE ECO Lab documentation, which can be accessed from the MIKE Zero Documentation Index.

14.2 Water quality definitions

Water Quality must be enabled by the user if it should be included in the simulation and in the MIKE HYDRO tree view. To activate Water Quality it is required to enable the '*Water quality*' checkboxes under *Simulation specifications\Modules*.

The Water quality definitions page includes the following tab-pages:

- General
- Locations and Parameters

General

Default Water Quality model
The current version support only running one water quality model (MIKE ECO Lab template) at a time. The selected and active template is therefore always the default water quality model - and this setting cannot be changed presently.

Solution method
Water quality calculation is based on predefined MIKE ECO Lab templates which contains a model defined by a number of coupled differential equations, which is solved through numerical integration and interactions between each equation.

Solution method is a selection of the Integration Method for solving the coupled ordinary differential equations defined in the MIKE ECO Lab file. Three different built-in integration routines (solution methods) are available (please consult the MIKE ECO Lab Reference Manual for details on the methods for solving the coupled linear differential equations in the MIKE ECO Lab framework):

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

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

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

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

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

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

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

Update frequency

The update frequency is a parameter that allows to define how often the water quality processes will be calculated during the simulation. The update frequency is defined as a multiplum of the simulation time step but currently it is decided to keep the frequency equal to one in MIKE HYDRO Basin in order to keep the water quality processes simulated on each Basin simulation time step.

MIKE ECO Lab template file

The WQ modelling is based on predefined MIKE ECO Lab templates (Water Quality Models). Two predefined MIKE ECO Lab templates are included in the MIKE HYDRO installation and they can be located in the folder: '*C:\Program Files (x86)\DHI\MIKE Zero\2024\Templates\ECOLab*'.

Templates supplied with the MIKE HYDRO installation are:

- 'MikeHydroWQ_DO.ecolab'

- 'MikeHydroWQ_noDO.ecolab'

Predefined MIKE ECO Lab templates includes BOD, Dissolved oxygen (DO), NH_4, Nitrate (NO_3), Phosphorus (P_{total}), bacteria E.Coli and a User-defined degradation component.

The difference between the two templates is that the former includes dissolved oxygen processes while the latter does not include dissolved oxygen processes.

The State variable overview of a MIKE HYDRO MIKE ECO Lab template is presented in Figure 14.1.

Figure 14.1   MIKE ECO Lab editor with overview of State Variables for the MIKE ECO Lab template: 'MikeHydroWQ_DO.ecolab'

To add an ECO Lab template (Water Quality Model) click on the '*Add*' button at the right hand side of the **Water Quality Models** table and a new row will appear in the table. Click on the row, then click on the '...' button in the right-hand corner of the row (or click on the '...' button next to the MIKE ECO Lab template file cell) and open the appropriate MIKE ECO Lab template from the pop-up window. The predefined default templates can be found in the installation path under 'MIKE Zero\2024\Templates\ECOLab\'.

Once successfully loaded, all parameters (i.e constants, state variables, derived variables and forcing) of the MIKE ECO Lab template and their properties are automatically shown in the **Variables** overview table. The information in this table cannot be edited.

Note: The current version allows only one MIKE ECO Lab template to be loaded.

Locations and Parameters

In the *Locations and Parameters* dialogue, the location and values of water quality parameters may be specified. This includes both initial values and pollution loads from (point) sources.

Locations

Once a MIKE ECO Lab template has been loaded (see *MIKE ECO Lab template file* in General *(p. 164)*) a 'Global Parameter' location is automatically added to the list of locations. The parameters specified for the 'Global Parameter' location (see *Location sub type* below) are used as default values for the WQ modelling.

Local WQ Parameter locations are included by clicking '*Add from MIKE HYDRO*'. This opens a new dialogue window with all possible WQ parameter location in the setup. This includes Branches, Water users (return flows), Reservoirs and Catchments. Click '*Select'* to add a location in the WQ model.

Location sub type

Locations are defined by their type (i.e. Global, Branch, Water user, Catchment or Reservoir) and their 'Location Sub Type'. The available 'Location Sub Types' depends on the 'Location Type' but includes the following:

- Initial

- Source

- Point Source

- Groundwater (various types) - only available for catchments with groundwater

A Local WQ Parameter Location may be assigned several 'Location Sub Types'. This is done by adding the location from MIKE HYDRO as many times as required.

Conservative transport

By default Conservative Transport is enabled, and the WQ parameters are simply transported through the model setup (i.e. no processes alter the WQ parameter concentrations). If Conservative Transport is disabled, various decay processes will affect the WQ parameters. This is specified below as Residence Time Calculation Type.

Residence time calculation type

Residence time intuitively determines the degree of solute decay. Mathematically, it is the upper integration limit of the first-order differential equations shown in the '*MIKE HYDRO Basin Water Quality Scientific Documentation'*. There are five different Residence Time Calculation types:

- *User defined Residence Time and Water Depth*: Time series files must be specified for Residence time (and Water depth if DO is included as a parameter). Click on Residence/Water Depth file cell and click the browse '...' button to select the right time series file. This option is convenient if you do not have any other data that allow residence time to be calculated. Water volume is calculated as the outflow rate times the residence time. If you model DO, you also need to specify water depth and reach length.

- *User defined Water Depth:* Time series files must be specified for Water depth. Click on Water Depth file cell and click the browse '...' button to select the right time series file. With this option Reach length 'l' must be specified in the Locations table (*Length* column) and Width 'w' must be specified under Routing (see). Water volume V is calculated as follows $V = w \cdot l \cdot h$ where h is water depth. Residence time T is calculated as $T = V/Q$ where Q is the reach outflowrate (by convention, it could also be argued that Q should be the average of inflow and outflow rates, if those differ due to routing).

- *Calculated Water Depth*: To use this option, Water level calculation must be specified under Routing (see). Calculation of volume and residence time is the same as above (User defined Water Depth).

- *None No Decay*: Conservative transport.

- *From Routing*: To use this option, a Routing method must be selected under Routing (see). The calculated reach storage S allows residence time T to be found as $T = S/Q$. Reach length is not required.

Parameters

To add parameters to a location, select the relevant location from the list and click the '*Add*' button in the upper right-hand corner of the dialogue window. This opens a new dialogue with all available parameters. Select the required parameters (our press Crtl-A - to select all). The selected parameters are then associated with the location (and Location Sub Type) and the following properties may be edited.

Note that depending on the chosen Location Sub Type, the values of the state variables represent either 'Initial' values (used only during the initial time-step) or 'Load' values applied at each time-step.

Parameter type

By clicking on the cell, the Parameter Type may be set to either '*Constant*' or '*Time Varying*'.

Fixed value

If a '*Constant*' Parameter Type has been selected, the value may be edited here.

Data file

If a '*Time Varying*' Parameter Type has been selected, a time series must be specified by activating the Data File cell, clicking on the '...' button in the right hand side of the cell and select an existing Time series.

Factor

A factor may be multiplied to the value. This may be used to evaluate impacts of reduced pollution loads or to represent related substances.

15 User Defined Variables

The purpose of the user defined variables is to be able to identify and measure any relevant simulation quantity and to store it as a specific item into the result files. The user defined variables are defined by a mathematical expression, which can contain double numbers, analytic functions or arithmetic operations.

The 'User defined variables' table is in the User-defined-variables tree page, under 'Variable definition' page as shown in Figure 15.1.
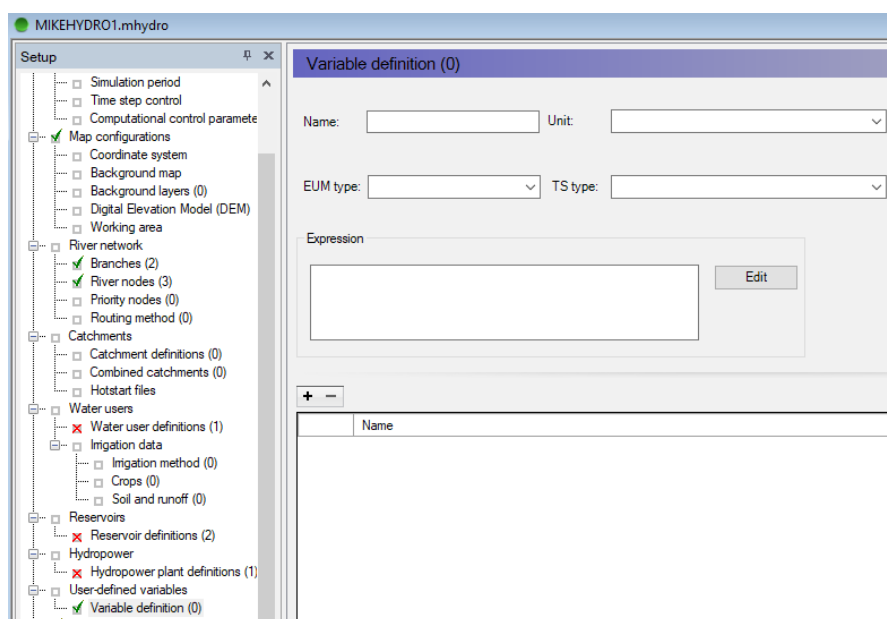

Figure 15.1　User defined variables dialog in User Interface

To add a variable definition, click on the '+'-button, to remove a variable, click on the '-'-button. The user must specify the variable's definition, given by the following parameters:

1. **Name** is used for identification and as name in the result file's output item.

2. **EUM type**: to specify the quantity type of the result item.

3. **Unit**, in which the user defined variable will be stored in the result item.

4. **TS Type**: the time series step type with the options:

   a. Instantaneous

   b. Accumulated

   c. Step accumulated

      d.   Mean Step accumulated

      e.   Reverse mean step accumulated.

5. **Expression**: the mathematical expression defines the value of the output in the result file. It can be formed by:

      a.   External time series: any dfs0 file with allowed EUM type and unit.

      b.   Sensors from the list of sensors of the basin model.

      c.   A large list of mathematical functions.

      d.   Any arithmetical operation for which no formulation error exists.

Connectors as logical operators for which no logical error is present.

16 Control

16.1 Control Rules

The Control Rules module enable users to modify model inputs based on the model state or external time series values.
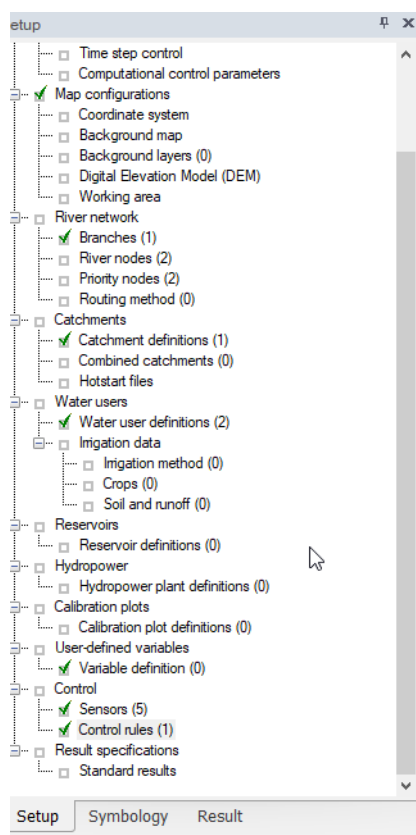
Figure 16.1 The Control Rules editor in setup tree
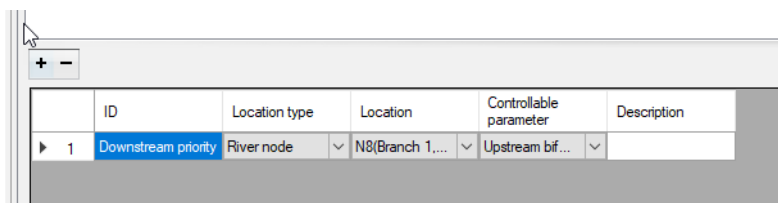
The Control Rules editor is available under 'Control' in the setup tree.

The Control Rules editor includes two tabs: 'General tab' and 'Conditions and actions tab'

16.1.1 General tab

The 'General' tab is used to create a control rule and define general properties.

To create a new rule, click on the plus sign in the bottom section of the tab.

Figure 16.2   Click '+' to create new control rule

A new rule requires specification of the following properties:

- **ID**: User-specified identifier for the rule.

- **Location type**: Control rules can be specified at one of three types of locations: reservoirs, river nodes, and water users. The control rule specification is limited to these types because these are typical locations for water management activities.

- **Location**: A specific location must be specified for applying the rule. The list is filtered based on the location type selections.

- **Controllable parameter**: This is the parameter that will be modified by application of the control rule. Any input timeseries associated with the location selected in the previous step can be used as a controllable parameter.

- **Description**: This is an optional field available for descriptive text.

Figure 16.3   Control rules editor, General tab

16.1.2   Conditions and actions tab

Control rules are implemented through evaluation of conditions and actions. Conditions are evaluated in order and, if a condition evaluates as true, then the associated action is executed, and lower-priority conditions are not evaluated. The functionality is analogous to an IF loop. Conditions are evaluated and actions taken, if appropriate, during each model time step.

New conditions are added by clicking on the plus sign in the conditions list. The order in which conditions are added is the order in which conditions are evaluated during a model run.
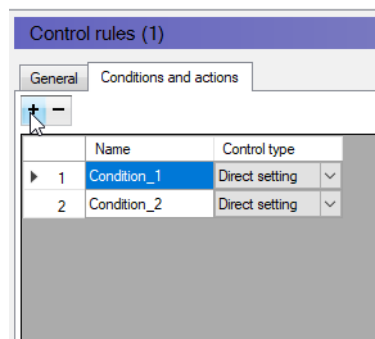

Figure 16.4   Click '+' to create new control condition

Two 'Control type' options are available for each condition: 'Direct setting' and 'Unchanged'. The 'Unchanged' option is available for cases in which a condition evaluating to true will not trigger an action.

Conditions and associated actions are displayed to the right of the conditions list. The displays are linked to the selected condition in the left-hand list.

Figure 16.5   Control rules editor, Conditions and Actions tab

Conditions and actions are specified using editors.

Conditions editor

The conditions editor provides a field for entry of the condition statement. The condition field can be edited directly or by using insert buttons.

Conditions are specified as a function of Sensors and/or User Defined Variables.

In the screen capture below, five sensors have been defined and are available for insertion using the 'Sensors' insert button. The condition statement is a function of three of these sensors.
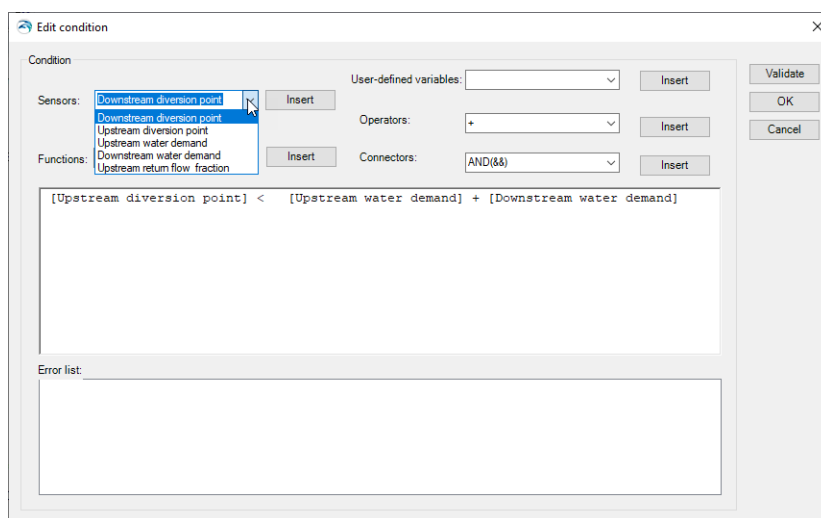

Figure 16.6   Conditions editor

User-defined variables, if defined, can be inserted using the same logic.

A list of mathematical functions is available for insertion into the condition statement. If appropriate, sensors or user-defined variables can be used as function arguments.
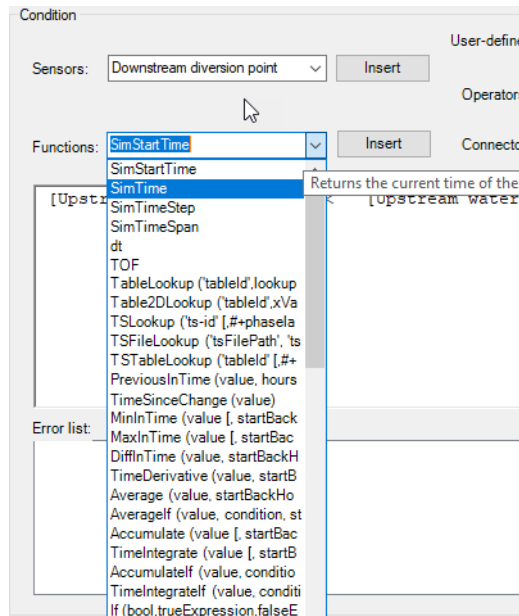
Figure 16.7 Functions in condition editor

Insert buttons are also available for inserting operators and connectors into the condition statement.

A validate button is available for checking condition statements. Results of the validation are printed in the 'Error list' at the bottom of the editor.

Actions editor

The Actions editor is like the Conditions editor. Actions can be entered directly or with the assistance of insert buttons. Actions should be written as expressions that represent the value that the controllable parameter (specified in the control rule definition) should take if the associated condition evaluates to true. An example is shown below.
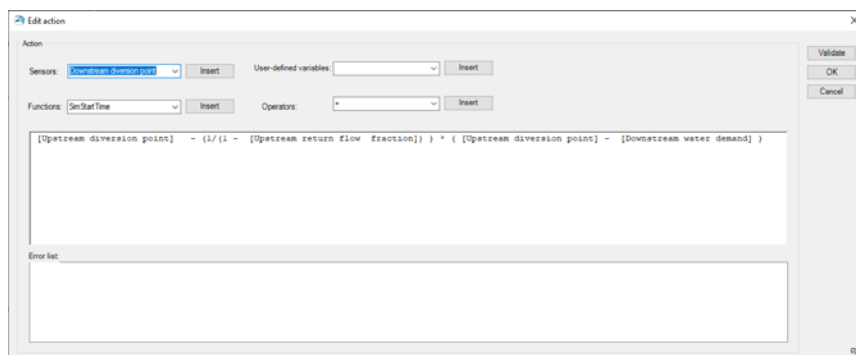
Figure 16.8   Actions editor

16.2   Sensors

Sensors exist to support the Control Rules module. Conditions and actions defined in the Control Rules module must be functions of either model state variables or external timeseries values. Before being used in a condition or action, state variables and external timeseries must first be defined as 'sensors'.

16.2.1 The Sensors editor

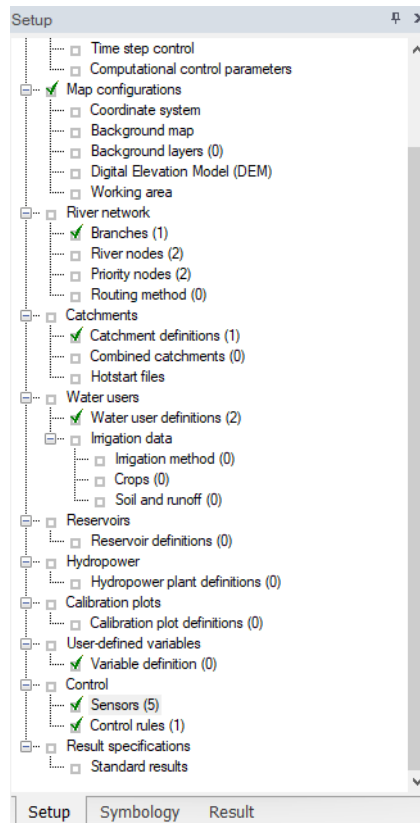Sensors are available under 'Control' in the setup tree.

Figure 16.9   The Sensors editor in setup tree

To create a new sensor, click on the plus sign in the bottom section of the Sensors tab.

A new sensor requires specification of the following properties:

- **ID**: User-specified identifier for the sensor. This identifier is then used when creating conditions and actions.

- **Location type**: In this field one specifies whether the sensor is a model state variable (Model object) or an external timeseries (External).

The next definitions depend on whether the 'Location type' is 'Model object' or 'External'. If 'Model object' is selected, then the following properties need to be specified:

- **Model object type**: One of the MIKE HYDRO Basin model object types is selected here (Branches, River nodes, Reservoirs, etc.).

- **Location**: The location of the state variable is selected. The list is filtered based on the selection in the previous step.

- **Branch location (only applies to Branches)**: If the model object type is 'Branches', then the 'Branch location' field appears, which allows the user to select a location on the selected branch.

- **Sensor**: This is the model state variable associated with the sensor. The list of available state variables is filtered based on the previous selections and is equal to the list of model outputs available at the selection location.

Figure 16.10   The Sensors editor: Location type 'Model object', Model object type 'Branches'. The 'Branch location' selection is available because the model object type is set to "Branches".

If 'External' is selected, then fields are provided for file and item selection.

Figure 16.11   The Sensors editor for Location type 'External'

16.3   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 HYDRO, such as:

- Variables
- Conditions for control rules
- Actions for control rules.

For variables, the return type is a number. For control rules, the return type for a condition is a Boolean, and for actions the return type is a number.

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 dialogs opened from the 'Edit' buttons (Figure 21.1).
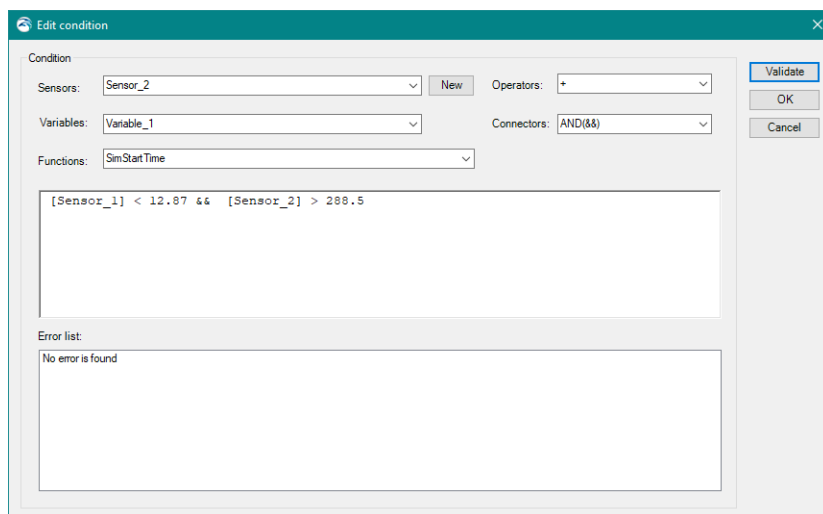
Figure 16.12 An Expression Editor in MIKE HYDRO

The exact content of the dialog depends on the context in which the expression is going to be used, therefore the Expression editor may differ depending on which 'Edit' button was used to open it.

Expressions are built in the central part of the dialog using combinations of Sensors, Variables, Functions, Connectors and Operators.

The "Error list" reports any syntactic error in the expression and provides advice on how to complete the expression, after pressing the 'Validate' button.
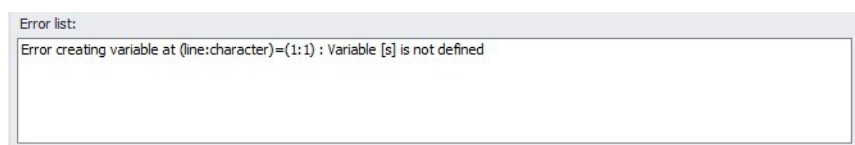
Figure 16.13 The Error list section reports identified errors

16.3.1 Expressions

Expressions are built in the Expression field using combinations of Sensors, Variables, Functions, Connectors and Operators.

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, etc. 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 ("").

"Variables" is a list including all user-specified items in the 'Variables' page. "Sensors" is a list including all sensors defined in the 'Sensors' page.

"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.

16.3.2 Operators

The dropdown shows the operators that may be used to create expressions.
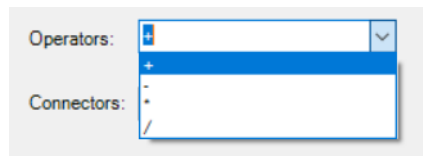
Figure 16.14   Operators in the Expression Editor

It contains Arithmetic operators that work on numbers and always return 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 16.1   List of arithmetic operators

Item	Description
+	Adds two numbers
-	Subtracts two numbers
*	Multiplies two numbers
/	Divides the first number with the second number
%	Remainder after integer division of the first number with second number

16.3.3 Connectors

The dropdown shows the connectors that may be used to create expressions.

Connectors work on Booleans and always return a Boolean value..

Table 16.2   List of connectors

Item	Description
&&	Logical AND operator. If both the operands are True, then operator returns true.
||	Logical OR Operator. If any of the two operands is non-zero, operator returns true.
!	Logical negation operator. It returns true if its operand is false and vice versa.

16.3.4 Comparison operators

Comparison operators are not available in a list but can be typed manually in the expression field.

The result of a comparison is always a Boolean value. The two operands must be of the same type to be compared.

Table 16.3   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 16.3   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

16.3.5 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.

16.3.6 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]))

16.3.7 Expressions involving numbers

A fixed value can be specified directly in the expression editor, using a dot as the decimal separator.

Table 16.4   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 16.5   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
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

Table 16.5   List of math functions

Function	Description
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.

DoubleFromString(arg, sep)

Get double value from string representation arg, where sep is the decimal separator, either "." or ",".

16.3.8 Expressions involving DateTime and TimeSpan

A DateTime value can be specified directly in the expression editor in several ways.

Table 16.6   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
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 16.7   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 16.8   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 16.9   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
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 16.10   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

16.3.9 Expressions involving strings

Strings are enclosed in either double or single quotes.

Table 16.11   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 16.12   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 16.13   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

17 Result Specifications

MIKE HYDRO simulations creates results from the selected type of model executed. Results are saved in result files of a specific type depending on the type of simulation.

The *Results specifications* dialogue enables the definition of location and names of the result files created during the simulations.

MIKE HYDRO suggests a default location and file names for simulation results, and it is therefore not strictly required to edit the result specifications by the user prior to running a simulation.

However, the *Results specifications* dialogue also enables the user to define a user-specified location as well as file names for results produced from the MIKE HYDRO simulation.

17.1 Standard Results

Simulation Results are stored in result files and the location as well as file names may be defined either using default definitions from MIKE HYDRO or using user defined folders and file names.

Overall, four settings can be made:

- Definition of Result folder path
- Definition of Result file names
- Selection of runoff hotstart files
- Selection of statistics type saved for NAM rainfall-runoff simulations

Details on result folder path specification

The 'Use default result folder' checkbox will determine whether you will use the default defined result folder path or wish to define a result folder path manually.

The default suggested result folder path will appear only after the MIKE HYDRO file has been saved initially. Default folder path is then created as a sub-folder to the folder where the .mhydro file is located. The name of the sub-folder will be a mix of the mhydro file folder name and path as indicated in the following example.

Example: A MIKE HYDRO setup file with the name 'WB-sc1.mhydro' has been saved to the location; 'C:\projects\WB-CC\Models'. In this case, the proposed default folder will be:
'C:\projects\WB-CC\Models\WB-sc1.mhydro - Result Files'

To make a user-defined file path definition it is required to deactivate the checkbox and then use the ⌷⌷⌷ button to browse for required folder. Press

'OK' when the correct folder has been located and result files will then be saved to this specific folder during simulation.

Details on result file name specifications

The 'Use default names' checkbox defines whether the default proposed file names or user-defined file names for results are used.

The default proposed result file name is created from the user-specified string in the 'Title' text-field in the 'Simulation specifications -> Description' page of the tree view, and the default result file name can hence be adjusted by editing the 'Title' definition of the project file.

Additionally, the specific type of simulation is included in the default file name to distinguish between multiple result files created in one simulation (see example below)

Example: The user has defined the following 'Title' in the mhydro file: 'CC scenario 1-A'. This will result in the following default result file names:

– Basin results: 'RiverBasin_CC scenario 1-A.dfs0'
– Rainfall-Runoff results: 'RainfallRunoff_CC sceario 1-A.dfs0'

Result file-type is always '.dfs0' type.

Note that simulation results with a title already used will overwrite the existing results unless a new result folder/file has been specified.

To make a user-defined result file name definition, simply deactivate the checkbox specify the requested result file name in the file name edit field.

Details on runoff hotstart files

When runoff hotstart files are selected, the result files from the Rainfall-runoff simulation will also be saved in '.res1d' file format (with the same name as the main 'Rainfall-runoff result file'). These files can later be used as initial conditions for a future NAM simulation, from the catchments' 'Hotstart files' page.

Details on water balance summary for NAM catchments

During Rainfall-runoff simulations, a text file named with the suffix '-RRStat.csv' is created and contains some statistics for each catchment using the NAM rainfall-runoff model. If the option 'Create monthly water balance summary for NAM catchments' is left unticked, this file will only contain yearly statistics. If the option is ticked, the file will instead contain monthly statistics.

18 Ribbon tools

There are a number of effective tools to assist in model creation which are accessible from the various tabs in the ribbon. Tools which are specific to a given type of feature are available in the corresponding tab in the ribbon (for example, all cross section tools are available in the 'Cross sections' tab). Generic tools are available in the 'Tools' tab. Tools described in this section include:

- Import and Export of MIKE HYDRO model data
- Load calculator

18.1 Import and Export of MIKE HYDRO data

MIKE HYDRO includes import and export features for model data components. The import and export options are available from the Tools tab in the ribbon.

18.1.1 Import from Shapefile

MIKE HYDRO includes a comprehensive shapefile conversion feature, enabling import of most georeferenced model objects stored in Shapefiles into a MIKE HYDRO model.

It is possible to import and define River branches, Water users, Reservoirs, Hydropower features as well as Catchment and Rainfall Runoff method from Shapefiles for the Basin module.

Figure 18.1 presents the dialogue for import of Basin model data from Shapefiles.

Figure 18.1   Import of Basin model features from Shapefile

To import model features from shapefile, do the following:

1. Activate the checkmark in front of the model features that should be imported (multiple features can be imported in one operation).

2. Browse and select the specific shape file using the file selection button '…'.

3. Define feature specific attributes from the shape file using the dropdown menus. Each dropdown menu will present the shape file attributes that are possible to use for the specific MIKE HYDRO feature parameter. **Note**: It is not required to specify all attributes in drop down menus. If some are left blank, the import utility will just assign default values for these features.

Some attributes in the shape files must be assigned specific numerical values to be recognized and imported correctly by MIKE HYDRO, as some of the features in MIKE HYDRO are defined by a numerical value rather than a string. The table below presents the attribute values for specific feature type definitions used during import from shapefile to MIKE HYDRO.

Table 18.1   Attribute values for specific feature type definitions

Feature attributes	Options	Values
Water User Type	Regular User Irrigation User	0 1
Reservoir Type	Rule Curve Reservoir Allocatio nPool Reservoir Lake	0 1 2

Table 18.1   Attribute values for specific feature type definitions

Feature attributes	Options	Values
Use Polyl. Area	True False	1 0
RR method (Rainfall Runoff Model Type)	None NAM UHM Time Area Kinematic Wave	0 1 2 3 4

When selections and filename specifications are complete, press the 'Run' button to execute the import of model data from shapefiles. Press 'Close' to stop the conversion or to close the dialogue.

Note: Reservoirs are attached to river branches in MIKE HYDRO, but imported points (for reservoirs) must not necessarily be strictly superimposed to the imported branches. These objects will be inserted on the closest location on the river branch they belong to. Additionally, their chainages will be automatically computed based on this location.

18.1.2 Export to Shapefile

The majority of georeferenced model features defined in MIKE HYDRO Basin can be exported to Shapefiles.

The export option is activated from the through *'Export to Shape files'* button, which opens the dialogue as presented in Figure 18.2.
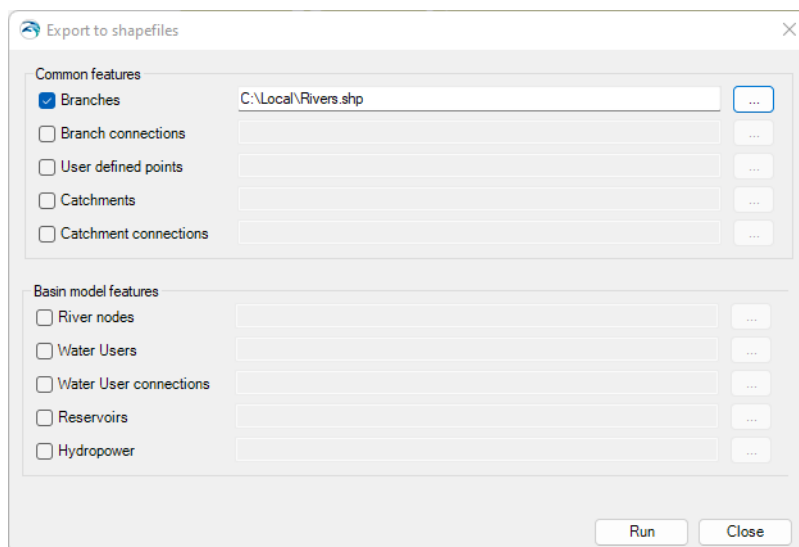
Figure 18.2   Export MIKE HYDRO Data to Shapefile dialogue

The dialogue contains two groups of data; 'Common features' which are features that are also used by other modules than Basin, and 'Basin Module Features' which are module specific features used only in the specific MIKE HYDRO module.

Export the requested model data by activating the checkbox in front features that should be processed and specify a shapefile names for the individual features. Use the file selection button '…' for specifying the output shapefile names.

After completing selections and filename specifications, press the 'Run' button to execute the export of model data to shapefiles. Press 'Close' to stop the conversion or to close the dialogue.

Shapefile content from Export of MIKE HYDRO features will include the following information:

Table 18.2   Export from MIKE HYDRO; Shapefile content

MIKE HYDRO features	Shape type	Key variables to export as shapefile attributes	Attribute name	Attribute type
Branches	River shapes (Polylines)	Branch name Identifier Topo-ID Start Chainage End Chainage Flow direction Branch Type	BR_BrName BR_ID BR_TopoID BR_StartCh BR_EndCh BR_FlowDir BR_Type	Text Text Text Double Double Short integer Short integer
Branch connections	Connecting lines (Polyline)	Branch name Upstr. Type Upstr. Branch name Upstr. Chainage Upstr. Storage ID Downstr. Type Downstr Branch name Downstr. Chainage Downstr. Storage ID	BC_BrName BC_UpsType BC_UpsName BC_UpsCh BC_UpsStID BC_DwsType BC_DwsName BC_DwsCh BC_DwsStID	Text Short integer Text Double Text Short integer Text Double Text
User defined chainages	Points	Branch name Chainage	UDC_BrName UDC_Ch	Text Double
Catchments	Catchment shapes (polygons)	Name Use Polygon Area Catchment Area Rainfall-Runoff model Branch Chainage	Cat_Name Cat_AreSel Cat_Area Cat_RRMod Cat_BrName Cat_Ch	Text Short integer Double Short integer Text Double
Catchment connections	Connection (Polyline)	Catchment name Branch name Chainage Upstream Ch Downstream ch	CC_CatName CC_BrName CC_Ch CC_UpsCh CC_DwsCh	Text Text Double Double Double
River Nodes	Node locations (Points)	Branch name Chainage Identifier Type	RN_BrName RN_Ch RN_ID RN_Type	Text Double Text Short integer
Water Users	User Locations (Points)	Name Type	WU_Name WU_Type	Text Short integer
Water User Connections	Polyline	Water user name Identifier Supply Branch name Supply Chainage Supply priority Return Branch name Return Chainage	WUC_Name WUC_ID WC_SuppName WC_SuppCh WC_Priorit WC_RetName WC_RetCh	Text Text Text Double Long integer Text Double

Table 18.2   Export from MIKE HYDRO; Shapefile content

MIKE HYDRO features	Shape type	Key variables to export as shapefile attributes	Attribute name	Attribute type
Reservoirs	Reservoir locations (Points)	Name Identifier Branch name Chainage Type	RES_ResName RES_ID RES_BrName RES_Ch RES_Type	Text Text Text Double Short integer
Hydro power	Hydro power locations (Points)	Name Identifier Reservoir Branch name Chainage	HP_HPName HP_ID HP_Reserv HP_BrName HP_Ch	Text Text Text Text Double

18.2   Generic Tools

18.2.1   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 HYDRO to trace e.g. rivers or catchments, in which case the DEM data must be supplied in a single file.

Input tiles

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

Supported formats are *.dfs2, *.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 HYDRO project. For the other formats, it is obtained from the files' data.
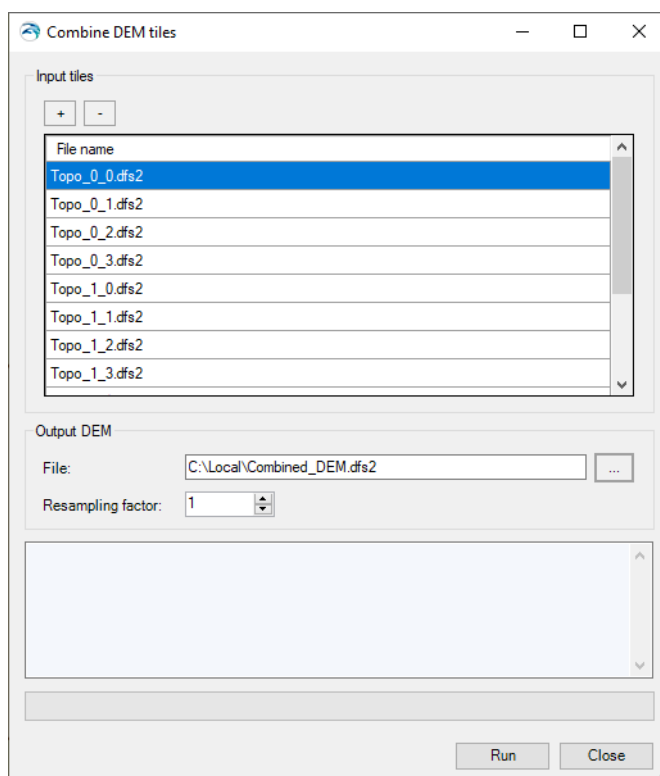
Figure 18.3   The Combine DEM tiles tool

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.

File

The path to the output DEM file. The output 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.

Resampling factor

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 the 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.

Buttons

- Run: Creates the output DEM. When it is created, the tool offers the option to use the created file as input DEM for data processing in MIKE HYDRO.

- Close: Closes the tool.

18.3 Catchment tools

18.3.1 Load Calculator

The Load calculator is a tool to determine the amount of pollutants that are absorbed by the water in its path through catchments and groundwater areas towards the river network. The pollutants are computed considering different pollutant sources related to farming, agriculture, or industrial activities, including animal or human waste, sewage sources, or use of fertiliser, etc. The input consists of the pollutant sources and amounts, and the output consists on the actual flux of pollutants that makes it into the river network.

The tool can be applied as a stand-alone tool for calculating average mass fluxes of pollutants for individual sub-catchments (e.g. kg/catchment/year) or on a raster grid basis (e.g. kg/grid/year). Alternatively, it can be used to provide the pollution loading for a MIKE HYDRO Water Quality model. Pollution loads may include both point and non-point sources.

All loads are initially calculated as constant mass fluxes for each sub-catchment, e.g. kg/year, however when applying the Load Calculator together with e.g. the MIKE HYDRO Water Quality model there are several ways to translate the constant mass fluxes into mass flux time series depending on e.g. runoff time series or any other known temporal variations.

Distance specific decay or retention of pollutants can be included taking into account the distance between the location of the pollution sources and the presumed outlet in the river network.

The *Load Calculator* dialogue consists of four main tabs that will hold the required information for the engine to construct the loads:

- Sources - for specifying pollution sources,

- Catchments - for specifying the transport,

- Decay - for specifying the retention of pollutants, and

- Output - for specifying how the output is to be stored.

Sources

The sources tab is used to define the various sources that are to be modelled in the setup. The individual sources are inserted by clicking the '*Add*' button. The user will then have to enter the name of the source e.g. 'manure'.

This will generate a new sub tab within the main source tab. The dialogue may hold multiple sources, each added or removed using the buttons at the lower right corner of the main tab. Each source must be defined using the associated dialogue.

Definition layer

Shape file

Click the browse '...' button to browse for and select a shape file that includes information about the source, e.g. data on fertiliser application, population numbers, etc. The shape file may either be a point or a polygon file. The text field cannot be edited but will be populated with the path to the shape file. The shape file is used to define the source and the following two attributes must be defined using the drop-down menues:

- **Name Field**: Select a field in the attribute table of the layer that includes a unique ID or Name of the administrative or statistical unit.

- **Value field**: Select a field in the attribute table of the layer that includes data of the fertiliser application, head count, release amount, etc.

Category

This field allows an implicit aggregation of point sources. It defines sub sets of the total loads, in such a way that the contribution of each category can be disaggregated from the total, in order to have an overview of the relative contribution of each category in relationship with the total load.

Components

Select the components that are to be transported in the system. The combobox is populated through querying the MIKE ECO Lab template in the setup of the constituents available (see section 14 Water Quality *(p. 163)*).

Note that only 'State variables' where 'Transport = ADVECTION_DISPERSION' will be shown and handled in the Load Calculator.

Time distribution

The user may select to modulate the load with the runoff. This is done by enabling 'Distribute according to runoff'. The runoff time series is applied as a temporal multiplication factor to distribute the mass fluxes of the pollution source in time for the current load source. Time distribution of components can be defined individually using an alpha series stored in a dfs0 file.

Modification factors

The table is populated by the user inserting a line. The districts are selected from a combobox based on the IDs in the shape file. The columns are added

based on the components selected above. Modifications factors not supplied are equal to unity (n=1).

Catchments

The catchment tab holds information specific to the catchments. It consists of two main group boxes. The top one holding information concerning runoff hydrographs the lower focusing on the concentrations of the baseflow in the catchments.

Click '*Load all catchments*' to load the existing catchments in the setup. For updating existing catchments click '*Update all catchments*', this will add missing catchments to your tables.

Catchment runoff

This box is holding information concerning runoff hydrographs. For non-point sources, pollutant transport to the river (as a flux) is typically positively correlated with the Rainfall runoff. Non-point sources include all sources specified as Livestock or Fertiliser source types. To account for this runoff dependent flux of non-point pollutants, it is assumed that pollutant concentrations in the runoff are constant.

The Runoff Start and End times are applied to specify a period for which annual non-point loads (e.g. kg/year) are translated into an average pollutant concentration (e.g. mg/l). The total annual load (i.e. total mass) is divided by the total accumulated runoff (i.e. total volume) for the specified period to calculate the mean runoff concentration (e.g. in mg/l) of each pollutant originating from non-point sources. To provide a variable load flux input time series, this concentration is then multiplied by the runoff time series specified for each Catchment in the setup.

Typically, the period specified will be of one year duration and represent the calibration period for a MIKE HYDRO Water Quality model.

If the assumption of a constant concentration of a pollutant in the runoff originating from non-point sources is not satisfactory, it is possible to apply an alpha time series to distribute the non-point sources in time.

Catchment baseflow concentrations

The table holds the baseflow concentration values of the constituents for each catchment. The columns are added based on the selected components (see section Sources *(p. 202)*).

This option is only relevant if a groundwater runoff component has been explicitly included in the setup. This must be included as separate time series files, either as a user specified time series or as a NAM model simulation result. When a groundwater component is included, only the fraction of the total load corresponding to:

(Total Runoff - Groundwater runoff)/Total Runoff

will be added to the catchment node in the river network. The rest of the loads are ignored. Instead a user specified baseflow concentration must be specified for each pollutant component representing the expected pollutant concentration in the groundwater discharging into the river section. Baseflow concentrations can be identified from water quality measurements as concentrations found during low flow situations in parts of the river where domestic and point sources are absent. Thus, the baseflow concentration is often a calibration parameter.

This approach has been introduced based on the assumption that most non-point loads are derived from the overland or drainage flow components of the hydrological cycle. This is often seen in rivers dominated by non-point sources resulting in a high concentration of pollutants during high flow and low concentrations during low flow. The fluctuation of concentrations in this case is typically determined by the relation between groundwater and surface/drainage water discharge to the river. Baseflow concentrations are typically considerably lower (e.g. <10%) compared to surface/drainage flow.

In some cases, though, transport through groundwater may be significant. In those situations it may be recommended not to include groundwater separately as described above. Instead use the Distance decay function to describe the overall retention of pollutants in the total runoff not distinguishing between different types of runoff components.

Decay

The decay tab is only active if '*Apply Distance Decay*' has been enabled. This is then applied globally to all catchments.

A river network shape file is used to calculate the distance to the nearest river network point. Please note that the shape file may use a finer resolution than the one used for the modelling of water movement in the rivers.

A fixed temperature is used throughout the model.

Output

The Output tab holds the information on the output type along with the mechanism for launching the load calculation.

A switch '*Basin*' controlling whether the output should be saved as Basin data or not.

Statistics in the form of a shape file with attributes describing the statistics will be produced if the switch '*Maps and Statistics*' is checked.

Calculation process is started with '*Generate loads*' button and a progress bar is updated based on the number of catchments handled.

APPENDICES

APPENDIX A

A.1 The Linear Reservoir Model

In a linear reservoir model, groundwater discharge, i.e., flux through the outlet(s) is proportional to the water level, and because catchment area is constant, it is also proportional to the storage. Specifically, the coupled differential equations solved are:

$$\frac{\partial h_1}{\partial t} = (-k_1 - k_1)(h_1 - L_1) + q_{recharge} + q_{stream\_seepage} \qquad (A.1.1)$$

$$\frac{\partial h_2}{\partial t} = k_i(h_1 - L_1) - k_2(h_2 - L_2) - q_{pumping} \qquad (A.1.2)$$

where the variables related to the geometry (h and L) and time constants k are defined in Fig A.1.1. The dimensions of L and h are [Length].

Fig A.1.1   The geometry of the linear reservoir model

The mathematical solution of the linear reservoir equations in MIKE HYDRO is also valid for situations where the groundwater storage is emptied (when outflows permanently exceed inflows), or when overflow occurs (when inflows permanently exceed outflows). This is also valid when the deep groundwater level reaches the shallow outlet, causing flow back into the shallow reservoir.

A.2   Hydropower - Formula

MIKE HYDRO calculates the hydroelectric power produced from the following formula:

$$P = \Delta h(Q) \cdot Q \cdot \varepsilon(\Delta h) \cdot g \cdot \rho_{water} \qquad \text{(A.2.1)}$$

Where P is the power generated, Δh is the effective head (difference) [L], Q is the discharge/release through turbine(s) [L³/T], ε is the machine (power) efficiency [-], g is the gravitational constant [L/T²] and ρ_{water} is the density of water [M/L³]. Machine (power) efficiency ε may also be specified as a function of Q, in which case $\varepsilon(\Delta h)$ is replaced by $\varepsilon(Q)$ in equation (A.2.1).

The effective head difference is:

$$\Delta h(Q) = h_{reservoir} - h_{tailwater}(Q) - \Delta h_{conveyance}(Q) \qquad \text{(A.2.2)}$$

The hydropower formula is non-linear because of the dependencies of head difference on discharge and machine efficiency. Tailwater levels are generally a function of discharge, and so are additional conveyance head losses in the channel (both increase with discharge). In addition the tailwater can also become governed by backwater from the reservoir downstream rather than discharge of the supplying reservoir. In the simulations, the applicable tailwater level for use in equation (A.2.2) is found from

$$h_{tailwater} = max(h_{tailwater} \cdot (Q), h_{downstream\_reservoir}) \qquad \text{(A.2.3)}$$

In MIKE HYDRO, the following inter-dependencies between variables can be assumed constant or insignificant by leaving out the respective detailed specifications:

$h_{tailwater}(Q) = h_{reservoir\_bottom\_level}$ (leaving out the tail water table)

$\Delta h_{conveyance} = 0$ (leaving out the conveyance head loss table)

$\varepsilon(\Delta h) = 0.86$ (leaving out the power efficiency table)

Water demand for power generation is calculated by solving the power formula, equation (A.2.1), for Q (the solution must be found iteratively). When the effective head difference is small, turbines are however shut off, both because they are inefficient and because the required discharge would grows very large. Accordingly, a minimum head for operation can also be specified. If the head is less than the minimum head, Q is set to zero, i.e., no water is routed through the turbines, regardless of demand.

A.3 Irrigation

A.3.1 Climate Models

The climate module accepts a number of commonly available climate inputs and converts them into precipitation and the input required by the reference ET model, according to the selected Climate model. Two model types are currently available:

- The **Rainfall only** model. This is the simplest climate model, only requiring a Rainfall time series as input.
- The **FAO 56** climate model. This model is only relevant if the evapotranspiration model is of the type FAO 56. The required inputs for this model are climatic data, rainfall data and geographical information about the location of the irrigated area

A.3.2 Reference ET Time Series Calculation

The reference ET model is responsible for providing the Crops module with reference evapotranspiration in each time step of the simulation. The evapotranspiration rates may either be calculated based on the input from the Climate model, or provided directly as a time series.

Two types of the reference ET models are currently available:

- Reference ET time series
- FAO 56 reference ET

A.3.2.1 Reference ET time series

As an alternative to the FAO 56 calculation of reference ET, this model type accepts time series of reference evapotranspiration. This method works with fewer data and must be used with the Rainfall only climate model type.

A.3.2.2 FAO 56 reference ET

The FAO 56 reference ET model is applied by default when selecting the FAO 56 climate model. The FAO 56 reference ET model uses the standardised Penman-Monteith equation for calculating reference evapotranspiration and it requires no additional input.

The reference ET is calculated as:

$$ET_0 = \frac{0.408\Delta(R_n - G) + \gamma\dfrac{900}{T + 273}u_2(e_s - e_a)}{\Delta + \gamma(1 + 0.34u_2)}$$
(A.3.1)

Where:

ET_0: Reference evapotranspiration [mm/day],

R_n: Net radiation at the crop surface [MJ/m^2/day],

G: Soil heat flux density [MJ/m^2/day],

T: Air temperature at 2 m height [°C],

u_2: Wind speed at 2 m height [m/s],

e_s: Saturation vapour pressure [kPa],

e_a: Actual vapour pressure [kPa],

e_s-e_a: Saturation vapour pressure deficit [kPa],

Δ Slope vapour pressure curve [kPa/°C],

γ Psychrometric constant [kPa/°C].

The FAO Penman-Monteith equation provides the evapotranspiration from a hypothetical grass reference surface and provides a standard to which evapotranspiration in different periods of the year or in other regions can be compared and to which the evapotranspiration from other crops can be related.

A detailed description of the calculation procedure for the Penman-Monteith formulation is described in FAO 56, and contains the following calculation steps:

- Derivation of all required climatic parameters from the daily maximum (T_{max}) and minimum (T_{min}) air temperature, altitude (z), mean wind speed (u_2) and geographical location.
- Calculation of the vapour pressure deficit (e_s - e_a). The saturation vapour pressure (es) is derived from the mean temperature, which is assumed to be the average of T_{max} and T_{min}. The actual vapour pressure (e_a) is be derived from the minimum temperature, which is assumed to equal the dew-point temperature.

- Determination of the net radiation (R_n) as the difference between the net short-wave radiation (R_{ns}) and the net long-wave radiation (R_{nl}). These variables are derived from geographical location, sunshine hours and vapour pressure. The effect of soil heat flux (G) is ignored for time steps smaller than 10 days as the magnitude of the flux in this case is relatively small and ET_0 is obtained by combining the results of the previous steps.

A.3.3 Crops

FAO 33 Yield Model

Attaching a yield model to a crop model allows the conversion of soil water stress into the corresponding yield loss, and hence, to quantify the costs of a soil moisture deficit. The yield model currently available is the FAO 33 Yield Model.

This model is based on a so-called potential yield (Y_p), which is the crop yield under optimal conditions (no soil moisture stress). The sensitivity of a crop to soil moisture stress depends on the growth stage. A crop will usually be more sensitive to soil moisture stress in early growth stages than in late stages. This is taken into account with a yield response factor (K_y). A yield response factor has to be specified for each of four growth stages. Each stage is assigned a length that may, but does not have to, be the same as the growth stages in the Crops model to which it is related.

The crop yield is calculated as:

$$\frac{Y_a}{Y_p} = \prod_{i=1}^{i=G} \left[1 - K_{yi}\left(1 - \frac{Et_a}{Et_p} \right) \right] \qquad (A.3.2)$$

Where:

Y_a: Actual yield [kg/ha]

Y_p: Potential yield [kg/ha]

Et_a: Actual transpiration [mm/day]

Et_p: Potential transpiration [mm/day]

Index i is the i'th growth stage in a growing season with a total of G growth periods.

A.3.4 Soil Model

The Soil model, also referred to as soil water model, keeps track of the water flow between different layers in the soil profile and the time-varying water content in each layer (see Fig A.3.2). The main task of the soil water model is

to keep track of the amount of soil water available for soil evaporation and crop transpiration at any time during the simulation. The soil water content may also be used by the irrigation module to determine the irrigation demand.
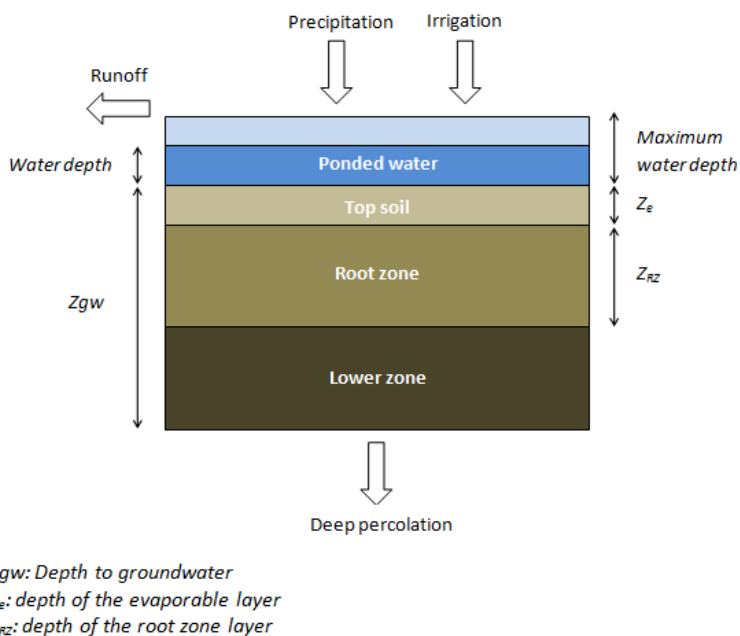
Fig A.3.2   *Schematisation of the Soil Water Models. Each model uses a different number of layers*

You can chose between three Soil model types:

- FAO 56
- ZIMsched
- ZIMsched for Rice field

A.3.4.1 FAO 56

The FAO 56 Soil water model is a simple water balance based model that follows the recommendations provided in FAO 56 /3/ for use with the dual crop coefficient method. More details are given in the following sections:

- FAO 56 Soil model properties
- FAO 56 Soil model computations

FAO 56 Soil model properties

The following parameters need to be specified for the FAO 56 Soil water model (see also Table A.3.1 for suggested representative values):

In the **Soil moisture content** dialogue:

- **Initial**. This is the starting soil moisture content at the beginning of the crop season [0-1]. The Initial soil moisture content is set for the Top soil and the Root zone. For the Lower zone the initial soil moisture is by default, at Field capacity.

- **Field capacity**. This is the maximum water content held by the soil against gravity [0-1]. Water cannot be retained in the top soil and the root zone layer when the water is above field capacity, as it will drain away under gravity. Soil moisture below field capacity is available for evapo-transpiration until the soil moisture reaches wilting point.

- **Wilting point**. This is the lowest soil moisture content at which plants can draw water from the top soil and the root zone layers [0-1].

- **Porosity**. This is the maximum soil moisture content that the soil can contain [0-1].

- **Depth of evaporable layer**. This is the depth of the top soil layer Ze from which evaporation occurs [m].

Example: *The amount of water in a soil column depends on the soil moisture content and the length of the soil column. If the field capacity is 0.15, the wilting point is 0.05, and the length of the root zone is 0.5 meter, then the available amount of water for transpiration is: 500 mm.*

Table A.3.1   The representative soil property values and maximum depletion by evaporation for an evaporation layer of 0.1 m

Soil type	Field capacity	Wilting point	Difference	Max. depletion by evaporation [mm]
Sand	0.07-0.17	0.02-0.07	0.05-0.11	6-12
Loamy sand	0.11-0.19	0.03-0.10	0.06-0.12	9-14
Sandy loam	0.18-0.28	0.06-0.16	0.11-0.15	15-20
Loam	0.20-0.30	0.07-0.17	0.13-0.18	16-22
Silt loam	0.22-0.36	0.09-0.21	0.13-0.19	18-25
Silt	0.28-0.36	0.12-0.22	0.16-0.20	22-26

Table A.3.1   The representative soil property values and maximum depletion by evaporation for an evaporation layer of 0.1 m

Soil type	Field capacity	Wilting point	Difference	Max. depletion by evaporation [mm]
Silt clay loam	0.30-0.37	0.17-0.24	0.13-0.18	22-27
Silt clay	0.30-0.42	0.17-0.29	0.13-0.19	22-28
Clay	0.32-0.40	0.20-0.24	0.12-0.20	22-29

FAO 56 Soil model computations

The model keeps track of the soil moisture content in two soil water compartments:

- The top soil layer from which only soil evaporation is drawn. The length of this layer is specified as the "Depth of evaporable layer" and is denoted *Ze*.

- The root zone layer. The length of this layer corresponds to the actual root depth (*Zroot(t)*), which can vary in time depending on the crop development. Transpiration takes place from this layer.

The wetting fraction (equals 1.0 for rain and specified by the user for irrigation) is taken into account when the exchange between the evaporable layer and the root zone is calculated. For rain (wetting fraction 1.0) the sub model assumes that the evaporable layer drains to the root zone as soon as the water content has reached field capacity (above which free drainage can occur). For wetting fractions less than 1, water will start draining at average water contents lower than field capacity. (see Wetting fraction time series)

- **Water balance for the top soil layer**. The water balance, per unit time, is calculated in the following three steps:

  – **Infiltration of rain and irrigation water** (calculated in two steps). First rainfall is added, and if storage is still available, irrigation is added:
  $$Infiltration = min((\theta sat - \theta old) \cdot Ze \cdot 1000, P)$$

  $$Infiltration = min((\theta sat - \theta) \cdot Ze \cdot 1000 \cdot Wf, Irr)$$

  Where:

  θsat : Soil moisture content at saturation [-]

  θold : Soil moisture content to the old time [-]

  θ : Soil moisture in the top soil layer after rain is applied [-]

P: Net rainfall after runoff [mm]

Irr: Irrigation demand [mm]

Ze: Depth of the top soil layer [m]

Wf: Wetting fraction [-]

– **Percolation to the root zone**: After water from rain and irrigation has been applied, the amount of water above field capacity percolates to the root zone layer assuming that water is instantaneously drained, by gravity, out of the top soil layer.

$$Perc = (\theta - \theta fc) \cdot Ze \cdot 1000$$

Where:

θ : Updated water content after irrigation has been applied [-]

θfc : Soil moisture content at field capacity [-]

– Updating the water content: The water content is now updated taking into account the evaporation from the top soil layer. Notice that the water content in the top soil during periods with high infiltration will be equal to field capacity minus the evaporation.

$$\theta new \cdot Ze \cdot 1000 = \theta old \cdot Ze \cdot 1000 - Ev + Infiltration - Perc$$

Where:

θnew : Final soil moisture content in the top soil at the new time [-]

Ev: Evaporation [mm]

- **Water balance for the root zone layer**.The water balance, per unit time, is calculated in the following two steps:

– Root zone percolation. The percolation out of the root zone layer is assumed to take place, by free drainage, instantaneously as soon as the soil moisture content is above field capacity. It is calculated from the following equation:

$$PercR = (\theta - \theta fc) \cdot Zroot \cdot 1000$$

Where *Zroot* is the thickness of the root zone. *Zroot* is time varying depending on the crop stage. θ is the intermediate soil moisture content in the root zone layer after it has been updated with the percolation from the top soil layer.

– **Updating the water content**. The new water content after the time step is then calculated by:

$$\theta new \cdot Zroot \cdot 1000 = \theta old \cdot Zroot \cdot 1000 - Et - PercR + Perc$$

Where *Et* is the transpiration [mm].

A.3.4.2 ZIMsched

ZIMshed properties

The parameters for the ZIMsched soil water model are the same as those of the FAO 56 model. Additionally, two extra parameters need to be specified:

- **Sat. hydraulic conductivity**. The saturated hydraulic conductivity is also called the saturated drainage coefficient [mm/h]. Since it is known that the drainage of water from an unsaturated zone diminishes with diminishing soil moisture, water may not drain instantaneously from the time when the water is above field capacity. In such cases water will be available for transpiration for a short period depending on the magnitude of the saturated drainage coefficient.

- **Depth to groundwater**. The unsaturated soil zone extends from the ground surface to the groundwater table [m]. Based on the depth of the top soil and the root zone layers, the lower soil layer can be calculated. From this zone deep percolation to the groundwater takes place using the same drainage formula as for the root zone layer.

ZIMsched Soil model computations

The ZIMsched soil water model is one step more complex than the FAO 56 soil model and simulates the water balance in the unsaturated soil column in three compartments. The first two are identical to the ones presented in the FAO 56 soil model. The third layer is denoted as:

- The lower zone layer. It covers the soil column from the root zone to the groundwater table. If the depth to the groundwater table below ground surface is called *Zgwt*, the thickness of this layer is time varying equal to: *Zlower(t) = Zgwt - Zroot(t) - Ze*.

Fig A.3.2 shows the schematic presentation on the soil column and the flows involved.

In the ZIMsched model, the root zone percolation also only occurs when the water content is above field capacity, and but the percolation is not instanta-

neous and is soil-dependent, described by the saturated drainage coefficient of the soil:

$$RootZonePercolation = \frac{(\theta - \theta fc)}{(\theta por - \theta fc)} \cdot Ks \qquad \text{(A.3.3)}$$

Where *Ks* is the saturated drainage coefficient [m/s].

The updating of the soil moisture content in the root zone is calculated by:

$$\theta new \cdot Zroot \cdot 1000 = \theta old \cdot Zroot \cdot 1000 - Et - PercR + Perc \qquad \text{(A.3.4)}$$

The deep percolation *PercD* from the lower soil layer is calculated by a similar equation as for the root zone percolation, and the new soil moisture content in the lower zone is calculated by:

$$\theta new \cdot Zlower \cdot 1000 = \theta old \cdot Zlower \cdot 1000 + PercR - PercD \qquad \text{(A.3.5)}$$

ZIMsched for Rice field

The ZIMsched for Rice, a soil water model for rice crops, is an extension of the ZIMsched model with an additional storage on the ground surface. This allows water to be stored on the ground surface during growth of rice. The model requires an extra parameter describing the maximum water depth that can be stored on the ground (often called detention storage):

- **Max water depth**. The maximum water depth [m] specifies the maximum water depth that can be held back on the ground surface before surface runoff takes place. From this storage water can evaporate and infiltrate into the top soil storage.

ZIMsched for Rice field computations

In the computations of the soil model for rice crops the actual water depth is added to rain (not to the irrigation because the wetting factor would be applied) and at the end of the calculation any excess of water on the top soil is considered as ponded water (up to the maximum water depth). The exchange of flow between different soil layers is identical to the ZIMsched model.

As in ZIMsched model, first Top soil layer is filled, then the Root zone layer and finally the lower zone. If there is an excess of water in the lower compartment (above field capacity), deep percolation occurs. At the end of the calculations, the excess of ponded water may generate runoff.

At the beginning of nursery stage (until start of land preparation) only a fraction of the field is used. The fraction, called Nursery area is specified in the Crops module.

A.3.5 Runoff Model

When it rains on the ground surface, a certain amount of the water may run off, either as surface flow or as subsurface flow or both. The irrigation module of MIKE HYDRO is considering a vertical water balance model for the soil column from the ground surface to the groundwater table. The horizontal extent of this soil column represents the field in which a certain crop sequence is grown.

The total runoff from an irrigation field can be divided into two components, a surface water runoff and a groundwater runoff. The latter is the amount of water, which escapes the root zone and percolates to the groundwater storage. The percolation to groundwater is calculated whether a surface runoff model is included or not, see further details in the Soil model section (wherever the link between irrigation node and catchment groundwater is described).

The task of the runoff model is to calculate the surface flow part of the total runoff. Surface runoff may occur either when it rains, when the crop is over-irrigated, or when a combination of both happens.

A.3.5.1 Using the Runoff model or not

The surface runoff models are based on the assumption that when the soil is wet, the infiltration capacity is small and a larger portion of rain will immediately run off as surface flow. This direct surface flow runoff can either be calculated as a function of the storage in the soil (and rice ponds) or as a function of the actual rainfall.

When the runoff model option is selected, an "a priori" direct surface flow runoff is calculated and subtracted from the rainfall. The remaining rainfall (called the net rainfall, PN) is input to the soil column water balance.

If PN together with the applied irrigation cannot be stored in the soil (or on the ground surface, in case of rice fields), additional surface flow runoff may occur.

If the irrigation node is connected to the river network, the calculated surface runoff will return through the return flow channel to the river. If the irrigation node is not connected to a river with a return flow channel, the water will be lost from the model.

Note: The return flow fraction should normally be set to 1 for irrigation nodes, as it does not reflect the fraction of water supplied to the node as for normal water user nodes, but the fraction of calculated surface runoff.

However, if it is set to less than 1, it indicates that some surface runoff may be lost on its way to the river. This effect could be described more correctly by

specifying a percolation loss in the return flow channel, because it would enter the groundwater reservoir.

When the runoff model option is not selected, the total rainfall enters the soil column water balance. If the rainfall and/or the irrigation input cannot be stored in the soil surface (or on the ground surface, in case of rice fields), the surplus water is considered surface runoff and removed from the field. However, the surface runoff is not returned to the river, but lost from the model, even if the irrigation node is connected to a river with a return flow channel.

A.3.5.2 Available Runoff models

Four surface runoff model options are available:

- None
- **Linear**, available for all crops (i.e. for all types of Soil model)
- **Modified SCS**, available for all crops except rice (i.e. for all types of Soil models except the ZIMsched for Rice field)
- **Modified SCS for Rice field** available only in combination with the ZIMshed for Rice field Soil model

None

No runoff model option applied.

Linear

The linear runoff model assumes a linear relationship between the rainfall [mm] and the amount of surface runoff. The runoff will be generated according to two specified linear formulas, valid when the accumulated rainfall is above or below a given threshold. The parameters for the linear model are:

- **When Rainfall is lower than**. It is the threshold of the accumulated rainfall [mm] on the field below which runoff occurs. In this case the runoff follows the formula specified in the line below:

  Runoff = **linear** coefficient [ ] (fraction to be inserted in the formula) x Rainfall + **constant** (to be inserted in the formula) in [mm]

- **When Rainfall is greater than**. It displays the threshold of the accumulated rainfall [mm] on the field above which runoff occurs. If the rainfall is above the threshold, the runoff follows the formula specified in the line below:

  Runoff = **linear** coefficient [ ] (fraction to be inserted in the formula) x Rainfall + **constant** (to be inserted in the formula) in [mm]

Regardless of the specified parameters, the runoff will never be negative.

Note: When including the Linear runoff model, for any crop including rice, setting both coefficients in the linear equation to zero will have the same effect as not including a surface runoff model. In that case, any excess surface water will to return to the river.

Modified SCS

The parameters to be specified for the Modified SCS runoff model are:

- **Soil moisture integration depth**. The soil moisture integration depth [m] which has an influence on the infiltration losses in the modified SCS model.

- **Coefficient of initial abstraction**. This coefficient determines also the infiltration losses.

When this runoff model is selected, the surface runoff is estimated using the Soil Conservation Service (SCS) storm flow equation (USDA, 1985) /4/ as modified by Schulze (1995) and used in the ACRU agrohydrological simulation model in South Africa.

The modified equation is given by:

$$Q = \frac{(P - c \cdot S)^2}{P + S \cdot (1 - c)} \qquad \text{(A.3.6)}$$

Where:

Q: Surface runoff [mm]

P: Rainfall [mm]

c: Coefficient of initial abstraction (default value 0.25) [-]

S: Potential maximum water retention of the soil, taken as the soil water deficit below porosity over the depth $Zinte$ [mm]

$Zinte$: Integration depth (default $Zinte$ = 0.25 m) [m]

S is calculated from the following formulas:

For $Zinte > Ze$:
$$S = (\theta sat - \theta e) \cdot Ze \cdot 1000 + (\theta sat - \theta root) \cdot (Zinte - Ze) \cdot 1000$$

For $Zinte \leq Ze$:
$$S = (\theta sat - \theta e) \cdot Zinte \cdot 1000$$

Where:

θsat : Water content at saturation [-]

θe : Water content in the Top Soil [-]

$\theta root$: Water content in the Root Zone [-]

Ze: Depth of the Top Soil layer [m]

Example:

Input P = 100 mm, Soil moisture integration depth = 0.2 m.
Porosity = 0.45, and actual soil moisture content is equal to 0.20.

The runoff is calculated from:

$$Q = \frac{(P - c \cdot S)^2}{P + S \cdot (1 - c)}$$

For C= 0 and S = 50 mm Q = 67 mm
For C=0.5 and S = 50 mm, Q = 45 mm

Modified SCS for Rice field

The difference between the Modified SCS for Rice field model and the Modified SCS model is that the space (storage) between the maximum ponded water depth (see the ZIMsched for Rice field model section) and the actual water depth is also taken into account in the calculation of the water volume of the potential maximum water retention of the soil, S.

In addition, excess water above the maximum ponded water depth at the end of a time step will be added to the already calculated surface runoff (can only occur when a specified irrigation application is used).

The input parameters for the Modified SCS for Rice field model are the same as those for the Modified SCS.

A.4 References

/3/ Allen, R.G., Pereira, L.S., Raes, D. and Smith, M., 1998. Guidelines for computing crop water requirements. FAO Irrig and Drainage Paper No. 56, FAO, Rome, Italy.

/4/ USDA, 1985. National Engineering Handbook, Section 4, Hydrology. United States Department of Agriculture (USDA), Soil Conservation Service, Washington DC, USA

APPENDIX B

B.1 Reservoir sedimentation in MIKE HYDRO Basin

B.1.1 Prediction of reservoir trap efficiency

In MIKE HYDRO Basin a reservoir is characterised by the following:

- A (surface) area elevation curve/table
- Top of dead storage level
- Bottom level
- Dam crest level (optional)
- Spillway Discharge/water level (Q/H) relation
- Spillway spill level
- Spillway bottom outlet

MIKE HYDRO Basin determines at each time step the reservoir inflow and outflow (distributed on outlets at level of dead storage and spillway), as well as the reservoir water level, volume and surface area.

The reservoir sediment trap efficiency is determined in a number of steps. First the reservoir retention time T can be calculated as reservoir volume (V) divided by inflow (Qi), i.e.

$$T = \frac{V}{Q_1}$$

The retention time indicates how long water on average will stay within the reservoir.

The trap efficiency (TE) for each size fraction i, with a settling velocity w_i, can then be calculated as (see Chen 1975):

$$TE_i = 1 - e^{\frac{w_i T}{D}}$$

In which D is a characteristic reservoir water depth (calculated from storage elevation curve). D may be calculated as

$$D = \frac{V_r}{A_{r,surf}}$$

Where V_r is the instantaneous volume of water in the reservoir and $A_{r,surf}$ is the instantaneous surface area of the reservoir. Thus the characteristic depth is equivalent to the average depth in the reservoir.

The trap efficiency indicates the fraction of sediment that is deposited in the reservoir. The total sediment deposited is calculated as

$$Deposition = \sum_{i=1}^{N} TE_i Q_{i,load}$$

The sediment transport of each size fractions released downstream (and thus used in the routing calculation) is given by

$$Q_{i, load, out} = (1 - TE_i) \, Q_{i, load, in}$$

B.1.2 Update of reservoir storage surface curve

A storage volume elevation curve is constructed from the storage area elevation curve. The assumption is that the storage area varies linearly between the given elevations.

Thus for a given storage elevation area tabulated curve, the reservoir volume column will be calculated by MIKE HYDRO Basin.

Table B.1.1   Calculated reservoir volume

Elevation	Reservoir Surface Area	Reservoir Volume
h_N	A_N	V_N
h_{N-1}	A_{N-1}	V_{N-1}
...	...	...
h_{i+1}	A_{i+1}	h_{i+1}
h_i	A_i	h_i
h_{i-1}	A_{i-1}	h_{i-1}
...	...	...
h_1	A_1	h_1
h_0	A_0	h_0

The surface area varies between levels h_i and h_{i+1} as

$$A = A_i + (h - h_i) \frac{(A_{i+1} - A_i)}{(h_{i+1} - h_i)}$$

The volume varies between the levels h_i and h_{i+1} as

$$V = V_i + A_i \, (h - h_i) + \frac{1}{2} \, (h - h_i)^2 \, \frac{(A_{i+1} - A_i)}{(h_{i+1} - h_i)}$$

Which for $h = h_{i+1}$ reads

$$V_{i+1} = V_i + \frac{1}{2} \, (A_{i+1} + A_i) \, (h_{i+1} - h_i) = \frac{1}{2} \sum_{j=0}^{i} (A_{j+1} - A_j)(h_{i+1} - h_i)$$

To update the storage area curve requires the calculation of a dimensionless quantity F which signifies the amount of deposited sediment volume above the reservoir volume relative to the total volume of a cylinder formed reservoir with surface area given by the current level

$$F = \frac{S - V}{HA}$$

Where S is the total volume deposited sediment within the considered time-span. H is the maximum depth for a full reservoir i.e.

$$H = h_N - h_0$$

The relative depth p is calculated as

$$p = \frac{h - h_0}{H}$$

The user selects the curve (I, II, III, IV) (see Morris & Fan (1997) p. 10.32-10.39) of corresponding values of (p,F). The intersection with the curve made up of the calculated set of values (p,F) is determined. The intersection is denoted (p_{in}, F_{in}) and the value of p_{in} indicates the so-called zero capacity elevation. The corresponding values of reservoir area and volume are calculated through the formula given earlier. The zero-capacity level indicates the new bottom level of the reservoir.

The relative sediment area is calculated from the equation below for each of the elevations (including the zero capacity level)

$$a = c_0 \, p^{c_1} (1 - p)^{c_2}$$

Where the coefficients (c_0, c_1, c_2) depend on the reservoir type selected (I, II, III, IV) (Morris and Fan (1997)). The relative sediment area at the zero capacity level is denoted by a_{in}. The area occupied by the sediment at the various levels is determined from

$$A_{sediment, i} = A_{in} \frac{a_i}{a_{in}}$$

Where A_{in} is the surface area at the zero capacity level and the area occupied by the sediment is only calculated above the zero capacity level (below this level the reservoir is assumed to be filled). The sediment volume may be evaluated as

$$V_{sediment, i+1} = V_{sediment, i} + \frac{1}{2} (A_{sediment, i} + A_{sediment, i+1})(h_{i+1} - h_i)$$

The revised storage area is determined as

$$A_{revised, i} = A_i - A_{sediment, i}$$

The revised storage volume is given by

$$V_{revised, i} = V_i - V_{sediment, i}$$

Below the zero capacity level the both the storage area and the storage volume are set to zero

The storage elevation curve is updated at regular intervals (typically different from the time step). The interest lies in the time as to when the bottom of the

reservoir reaches the top of the dead storage level. At this point the reservoir will quickly fill and after this point the inflow will be equal to the outflow.

INDEX

Z