MIKE

Powered by **DHI**

# MIKE 11

A modelling system for Rivers and Channels

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. For details please refer to your 'DHI Software Licence Agreement'.

**LIMITED LIABILITY**

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

# CONTENTS

# SIMULATION EDITOR

# 1 Simulation Editor

The simulation editor serves three purposes:

1. It contains the simulation and computation control parameters.
2. It is used to start the simulation.
3. It provides a link between the network editor and the other MIKE 11 editors as illustrated in Figure 1.1 (only a subset of editors are shown).
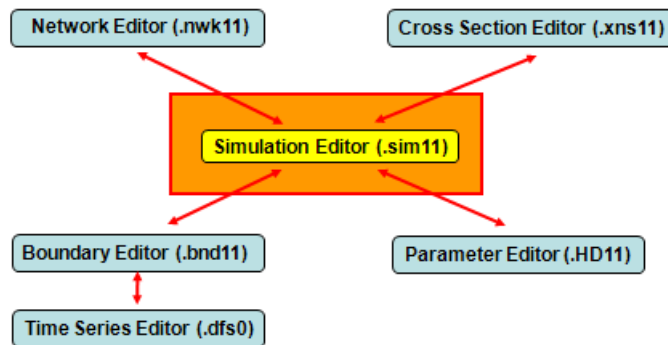


Figure 1.1     The MIKE 11 Simulation Editor

The editing of cross sections is a typical example of this link, where the graphical view of the network editor e.g. is used to present and select cross sections from the cross section editor.

The linkage requires a filename to be specified for each of the required editors. The filenames are input on the Input Property Page of the simulation editor. An alternative is to select a file from the File Menu which will activate the appropriate editor.

# 1.1　Models
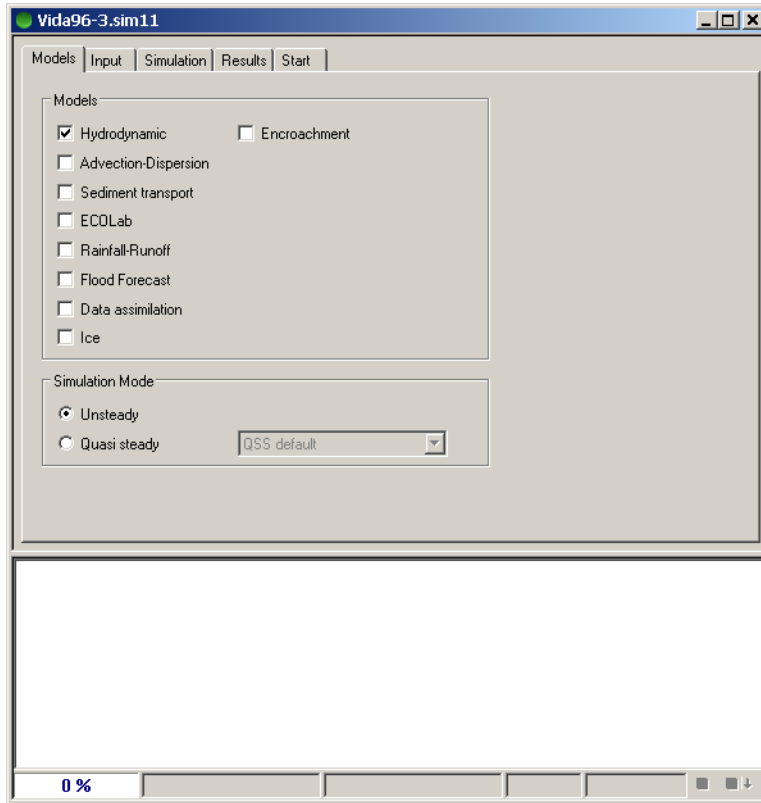
Figure 1.2　The Models tab. Note that the Simulation Mode Box may differ if a quasi two dimensional steady state solver with vegetation is not installed

This page is used to define the simulation models to execute and the simulation mode (unsteady or quasi steady).

## 1.1.1　Models

The following abbreviations of model names are used:

| | |
|---|---|
| HD | Hydrodynamic |
| AD | Advection-Dispersion |
| ST | Sediment Transport |
| MIKE ECO Lab | (Including Water Quality modelling) |
| RR: | Rainfall-Runoff |
| FF | Flood Forecast |
| DA | Data Assimilation |
| Ice | River Ice modelling |

Some of the models, which can be selected, are dependent on other modules in a simulation and it is therefore required to have the related models selected. This rule for model-dependency is implemented such that once a model is selected there will be an automatic selection of possible dependent models (e.g. selection of the FF model also selects the HD model and selection of MIKE ECO Lab also selects the AD model, etc.).

The RR model can be selected to run together with the HD model or on its own, in which case the runoff result file can be specified as input into a subsequent run of the HD model.

When selecting the hydrodynamic model an additional tick box entitled Encroachment becomes active. When selecting the latter all other tick boxes become inactive since the encroachment model is only designed to function in conjunction with the hydrodynamic module.

Further, when carrying out an encroachment simulation please ensure that the simulation mode is set to Quasi steady. If the latter is not the case the program will issue a warning and terminate.

Water Quality modelling takes place through the MIKE ECO Lab model entry where a variety of Water Quality models can be selected from so-called MIKE ECO Lab templates.

The River Ice model as listed above is the result of a development project and the model is not yet completed and publicly available as a standard model Please consult DHI for the current state if you have interest in modelling river ice formation, etc.

## 1.1.2   Simulation Mode

### Unsteady
The HD calculations are based on hydrodynamic flow conditions.

### Quasi steady
At every time step the calculations are based on steady flow conditions.

If a quasi two dimensional steady state solver with vegetation is not installed the Simulation Mode Box will differ from Figure 1.2. (quasi dimensional steady state solver with vegetation is a separate module which must be present in the license file). Otherwise a total of four possible settings are available:

1. QSS default: The classic MIKE 11 steady state solver is used.

2. QSS with vegetation: The quasi two dimensional steady state solver with vegetation is used for the simulation.

3. QSS with energy equation: A submodule of the quasi two dimensional steady state solver with vegetation. The energy equation is used for obtaining the water level in the network.

4. QSS with Ida's method: A submodule of the quasi two dimensional steady state solver with vegetation. An approximate solution of the governing equation (Ida's method) is used for obtaining the water level in the network.

## 1.2    Input
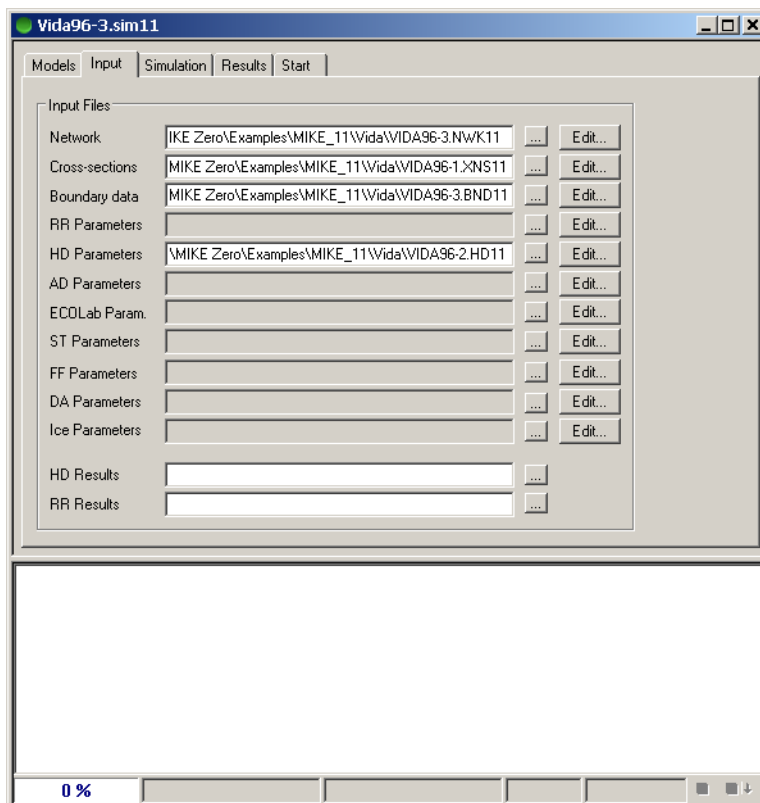


Figure 1.3    The Input tab

Based on the model selection from the Models Property Page a number of filename fields becomes active, and the user is required to specify a range of input filenames.

This button opens a file selection box.
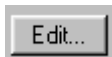
Edit...     This button opens the relevant editor if a valid filename of the given file-type has been specified in the filename field.

**Note** that files required for the simulation are indicated by the active fields on the page. Two exceptions exists though; the fields for the result files. It is important to notice, that files specified in the Models page are all input files for a simulation, so the result-filename fields are Not to be interpreted as being result filenames for the simulation. Instead, these result-files indicate an input to the simulation as described below.

A hydrodynamic result file (HD Results) is required if:

- a stand alone Advection-Dispersion/Sediment transport simulation is to be carried out where hydrodynamic conditions/results are read from results of a previous HD-simulation,

or

- if lateral sources from a previous MIKE SHE/MIKE 11 coupled model run are to be included in a hydrodynamic simulation (lateral inflow from a MIKE SHE simulation requires an additional activation of the parameter; 'LATERAL_INFLOW_FROM_RESULT_FILE' in the MIKE11.ini file).

A Rainfall Runoff result file (RR Results) is only required if the hydrodynamic and rainfall models are executed independently of each other.

## 1.3     Simulation

The simulation property page contains details of simulation time, time stepping specifications and initial conditions for each of the chosen types of models.
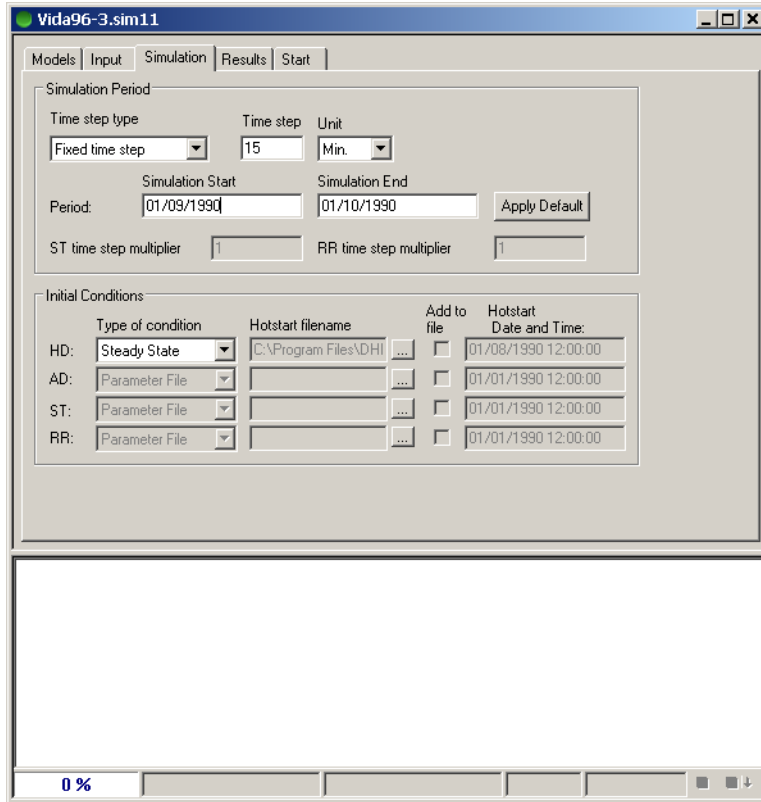
Figure 1.4    The Simulation tab

## 1.3.1    Simulation Period

### Time step type

Time stepping type is specified as either: Fixed time step, Tabulated time step or Adaptive time step. In case of fixed time step, the time step is specified in the time step edit field as a number and the unit is given in the unit selection list.

In case the time stepping type is specified as 'Tabulated', the time steps are specified by activating the settings button and selecting a time series from a time series file (dsf0) with Item-type defined as 'Time step' in the dialog presented in Figure 1.5.
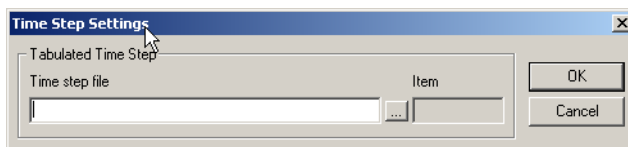


Figure 1.5    Dialog for specification of the tabulated time step time series file

The settings for time step adaptation is similarly specified in a menu activated by the Settings button. The menu is presented as Figure 1.6.



Figure 1.6     Time step settings for adaptive time stepping

### Minimum/maximum/unit
Defines the limits for the adaptation of the time step.

### Change ratio
The time step is successively lowered with change ratio until the criteria specified in this menu are met. The starting value for the adaptation within the time step is change ratio times the previous time step.

### Criteria
The time step adaptation model offers seven criteria which may each be enabled/disabled and given threshold values.

- |resid(BC)/BC| is a measure for the largest acceptable error introduced at the boundaries. MIKE 11 interpolates the boundary values between t and t+$\Delta$t using linear interpolation. In case the boundary values has a resolution finer than $\Delta$t this may introduce unfavourable behaviour where details are neglected. The term resid(BC) describes the residual between the actual value in the time series boundary condition (BC) file and the value found using linear interpolation between t and t+$\Delta$t. The term BC refers to the actual value in the time series boundary condition file.

- |delQ| is a measure for the largest acceptable discharge change anywhere in the grid within a time step. The criterion helps to lower the time step when sudden changes appears in the discharge. The changes may be either physical changes due to a sudden increase in inflow for instance or mathematical changes due to numerical instability. In either case a decrease in time step may well be desirable.

- |delQ/Q| is a measure for the largest acceptable relative discharge change anywhere in the grid within a time step.The criterion helps to lower the time step when sudden changes appear in the discharge. The changes may be either physical changes due to a sudden increase in inflow for instance or mathematical changes due to numerical instability. In either case a decrease in time step may well be desirable. The use of this criterion is well suited for dam break studies where it can be used for refining the time step in the period after the break.

- |delh| is measure for the largest acceptable water level change anywhere in the grid. As for the |delQ| and the |delQ/Q| criteria they help lowering the time step when large changes appear in the water level due to either physical changes or mathematical instabilities.

- |delh/h| is measure for the largest relative acceptable water level change anywhere in the grid. As for the |delQ|, |delQ/Q| and the |delh| criteria they help lowering the time step when large changes appear in the water level due to either physical changes or mathematical instabilities.

- |Courant| (HD) specifies the maximal allowed Courant number within the grid and time step.

$$Cr = \frac{(v + \sqrt{gD})\Delta t}{\Delta x}$$

The Courant number, defined above, expresses the length in terms of grid cells that information travels within a time step. The HD Courant number refers to the momentum equation and $\Delta x$ is hence in this context the distance between two h-points.
Mike 11 applies a 6-point Abbott scheme for solving the equation which does not have the typical Courant number below unity demand. Good results are obtained up to Courant numbers as high as 10-20.

- |Courant| (AD) specifies the maximal allowed Courant number for the advection dispersion calculation.

$$Cr = \frac{v\Delta t}{\Delta x}$$

The AD Courant number, defined above, is a measure for the length in terms of grid cells that the species are convected within a time step. The applied computational scheme is stable for AD Courant numbers less than unity. The AD solver includes both h- and Q- points as species grid points, hence the $\Delta x$ is half the distance between two h-points when calculating the AD Courant number.
The criterion is well suited for ensuring stability of AD calculations by lowering the time step when the flow velocity increases and increasing the time step when the flow velocity decreases.

### Period
The date and time for the start and end of the simulation period. The standard windows date time format is used.

Apply Default   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 the time series files active for the actual simulation takes place and the earliest possible start-date and latest possible end-date is automatically transferred to the Period date-fields. If no dates are proposed from activating the button the most likely reason will be that there are no common time period in the setup.

### ST Time Step Multiplier
The ST module will typically not operate using the same time step as the HD model. The ST Time Step Multiplier hence specifies the time step for the Sediment Transport simulations as a multiple of the HD time step.

### RR Time Step Multiplier
The RR module generally requires a time step different from the HD model when running a coupled simulation of HD and RR. The RR Time Step multiplier therefore defines the time step for the RR-model as a multiplum of the HD time step.

## 1.3.2   Initial Conditions

For each of the modules HD, AD, ST and RR the following can be specified:

## Type of condition

- **Steady State**: (HD only).
  The initial conditions will be calculated automatically assuming a steady state condition with discharges and water levels at the boundaries corresponding to the start time of the simulation.

- **Parameter File**:
  The initial conditions will be taken from the parameter file relevant to the module in question.

- **Hotstart**:
  The initial conditions will be loaded from an existing result file.

- **Steady+Parameter**: (HD only).
  The initial conditions will be established using both the steady state and parameter file method. In those grid points where data are specified in the Initial *(p. 312)* Property Page of the Hydrodynamic Parameters Editor *(p. 311)* the initial conditions will be taken from the parameter file, other grid points will be calculated using the steady state option.

## Hotstart Filename

The name of an existing result file from which initial conditions for the simulation should be loaded. For hotstart to be valid using control structures an additional hydrodynamic result file (*HDAdd.res11 or *ADDOUT.res1d) file must be present in the same location as the standard hydrodynamic result file. Further this additional result file should hold structure information as selected in the additional result tab in the hydrodynamic editor.

## Add to File

The results of the current simulation will be added to the end of the hotstart file. Any information (in the hotstart file) after the simulation start date will be lost. This part of the file will be replaced by the new simulation results.

## Hotstart Date and Time

The date and time at which the initial conditions are loaded from the hotstart file. If the "Add to File" has been selected the hotstart date and time will be taken as the simulation start.

## 1.4     Results
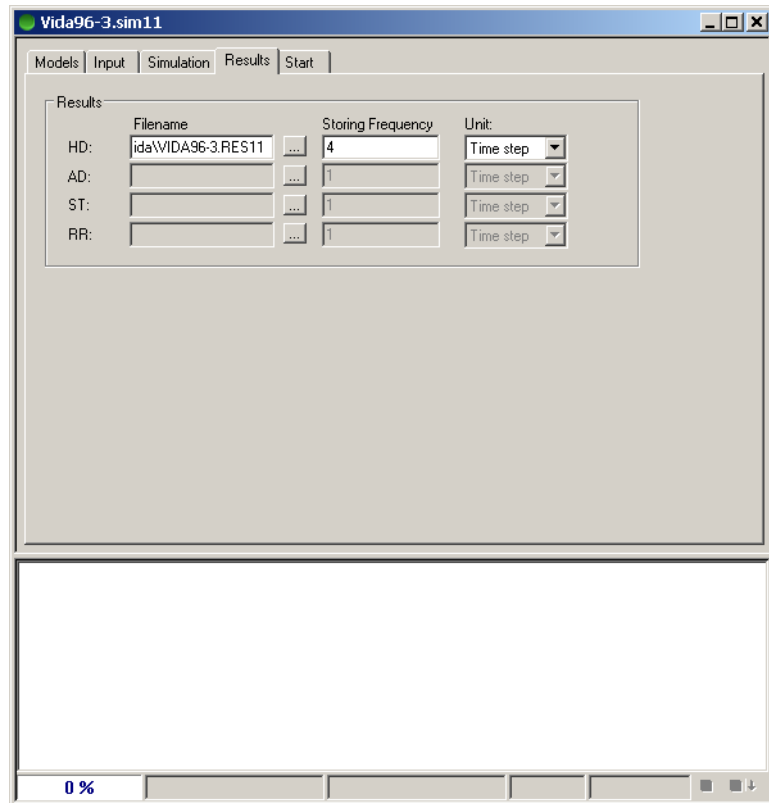
Figure 1.7     The Results tab

For each of the modules selected on the Models Property Page the user should specify a filename for saving of the simulation results.

The filename can not be edited if the flag "Add to File" has been selected on the Simulation Property Page. In this case the selected hotstart file will become the result file as well.

### Storing Frequency and Unit

To limit the size of the result files the user can specify an interval for saving of results to the result files. The storing frequency may be specified either as the number of time step intervals between each saving of the results or as a specific time. The latter, however, requires that the specified storing time frequency is a divisible by the time step.

# 1.5 Start
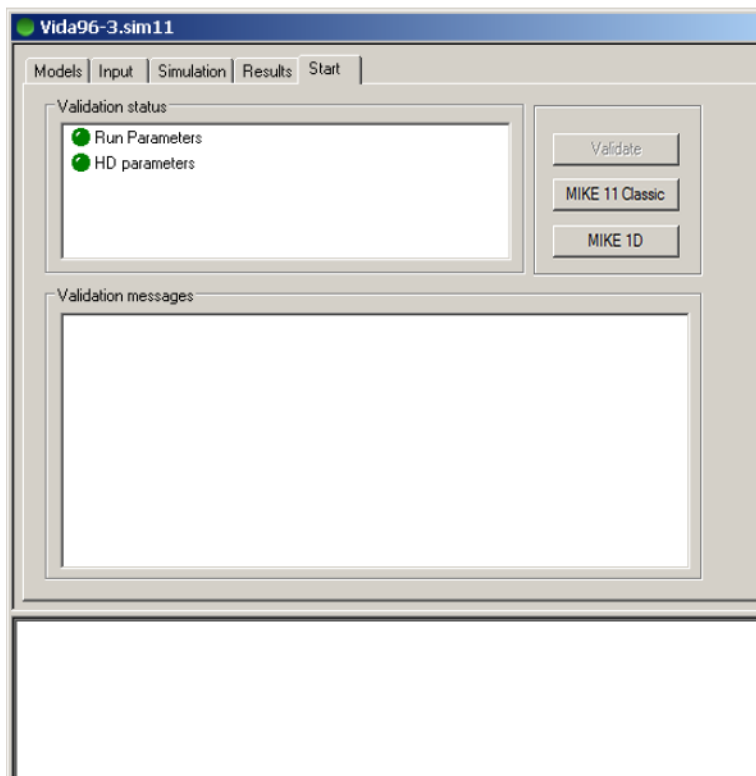


Figure 1.8    The Start tab

If all specified input files exist, the simulation can be started. It is possible to choose either the classical MIKE 11 simulation engine (using the 'MIKE 11 Classic' button) or the MIKE 1D engine (using the 'MIKE 1D' button). The progress of the simulation will be reported in the info bar of the editor. Please refer to separate documentation on the difference between these two engines and the possibilities provided with them.

Any error or warning message from the simulation will be presented in the log-part of the editor and additionally, saved in a file with the same name as the simulation file and a .log extension.

Upon completion the simulation results may be viewed using MIKE View.

# 1.6 Volume Balance Summary Output from Simulation

MIKE 11 produces an overall volume balance for a simulation. The balance is presented in a file named m11sim11VolumeBalance.HTML where the 'm11sim11' is the MIKE 11 simulation filename without extension. The file is

overwritten for every simulation. An example of the volume balance file is shown below:

## Volume balance Summary

| | | | | | | |
|---|---|---|---|---|---|---|
| A: Initial volume in model area | | | | | 239719.94 | m^3 |
| B: Final volume in model area | | | | | 466794.30 | m^3 |
| MIKE 21 lateral inflow | 83719.09 | m^3 | | | | |
| MIKE 21 boundary inflow | 0.00 | m^3 | | | | |
| Lateral sources inflow | 0.00 | m^3 | | | | |
| Lateral correction | 0.00 | m^3 | | | | |
| Open boundaries inflow | 895655.63 | m^3 | | | | |
| C: Total inflow | | | | | 979374.72 | m^3 |
| MIKE 21 lateral outflow | | | 343896.10 | m^3 | | |
| MIKE 21 boundary outflow | | | 0.00 | m^3 | | |
| Lateral sinks outflow | | | 0.00 | m^3 | | |
| Open boundaries outflow | | | 409156.18 | m^3 | | |
| D: Total outflow | | | | | 753052.28 | m^3 |
| E: Continuity Balance = B-A-C+D = | | | | | 751.92 | m^3 |
| Relative deficit E/max(A,B,C,D) = | | | | | 0.08 | % |

Figure 1.9    Volume balance summary from MIKE 11 simulation

The volume balance is broken down into the following items:

- Inflow through open boundaries
- Lateral inflow
- The runoff contribution from rainfall runoff models
- Evaporation from rainfall runoff model

Further depending on the coupling applied the MIKE 11 will also report

- Total exchange with MIKE SHE with separate items for drain flow, base flow and overland flow

- Total exchange with MIKE 21 (through MIKE FLOOD) consisting of lateral exchange over river banks and through coupled branch ends.

- Total exchange with MIKE URBAN CS either as lateral point exchanges or at branch ends.

The file respects user settings with respect to units and thus reports in cubic feet if set up to do so.

Deviation in volume balance may be caused by a number of factors:

- A large time step may give rise to water level changes within a time step which undermines the assumption of a gradual change of the water level dependent parameters such as the flooded area, cross sectional area and hydraulic/resistance radius.

- Lateral extraction of water from a dry section of the river network through SHE, MIKE 21 or evaporation (net rainfall in a RR model)

- Non coinciding bed levels of rivers meeting at junctions. Since MIKE 11 uses water level compatibility at nodes the lower level of the bed level must coincide to ensure volume conservation.

- Introduction of slots of a distance of delh (default 0.1 m) above the river bed. These slots modify the flooded area expression giving rise to a modification of the governing equations for shallow water. They are introduced to ensure a stable and smooth solution for shallow water. They still satisfy the volume conservation in the model though the numerical volume is different from the physical volume when the slot is in use.

To evaluate the effect of the large time step one may select to activate the mass error switch under additional output in the MIKE 11 hd11 file. To get a correct accumulated value all time steps must be saved. The correction of lateral discharge from dry sections of the river is reported in the volume balance as an extraction item. Non-coinciding junction levels are reported in the log file. To report the numerical volume in the volume balance the user may choose to activate the slot volume correction switch in the MIKE11.ini file

SLOT_VOLUME_CORRECTION = ON

This will calculate the overall volume based on the full depth of the slot.

# RIVER NETWORK EDITOR

# 2 River Network Editor

The River Network Editor is a key editor of MIKE 11 as it supplies the user with an overview of the current model setup components defined in the network file it self but also by presenting data which is stored in the other MIKE 11 editors.

The network editor has two main functions:

1. River network input and editing.
2. Overview of all model information in the current simulation.

The former includes:

- Digitising river networks and branch connections.
- Definition of hydraulic structures (weirs, culverts etc.).
- Definition of catchment inflow points (for rainfall run-off model).

The Network editor is composed of two different editors (the Graphical view and the Tabular View) of which the graphical editor provides an overview display in a graphical window as presented in Figure 2.1. Settings for the appearance of the graphical view and selection of which data to be presented graphically are found in the Settings menu of the Main menu bar.
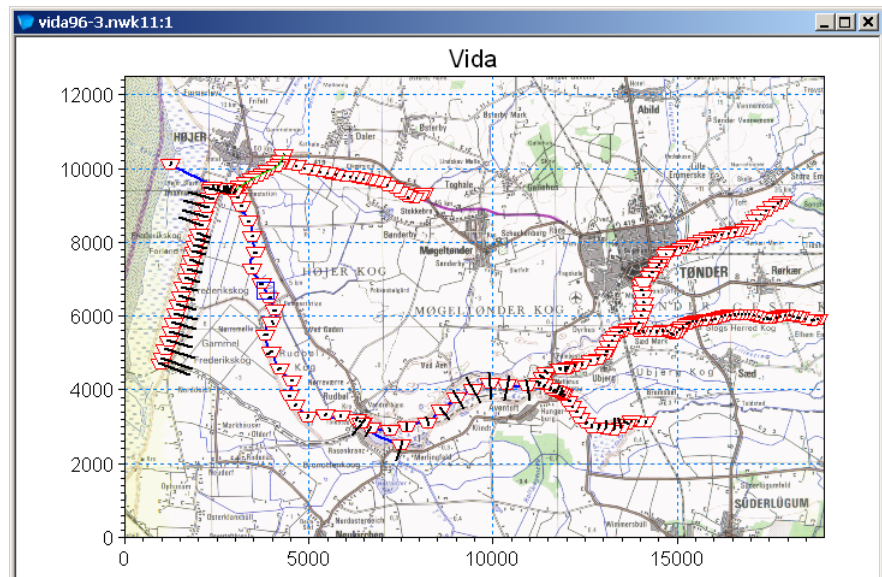


Figure 2.1    The graphical overview display of the MIKE 11 Network Editor (= the Graphical View)

The Tabular View of the network editor contains a large group of sub-pages in which the numerical values behind most of the objects being presented in the Graphical view are stored for editing. The tabular view contains individual pages with edit fields and tables for digitization points, river branches definitions and connections, hydraulic structures details and other data.

The Tabular view can be opened from the Graphical view through the 'View'->'Tabular view' option in the main menu bar or simply by using the hot-key combination; '<CTRL>+T'. The Tabular view, branch page is presented below in Figure 2.2.
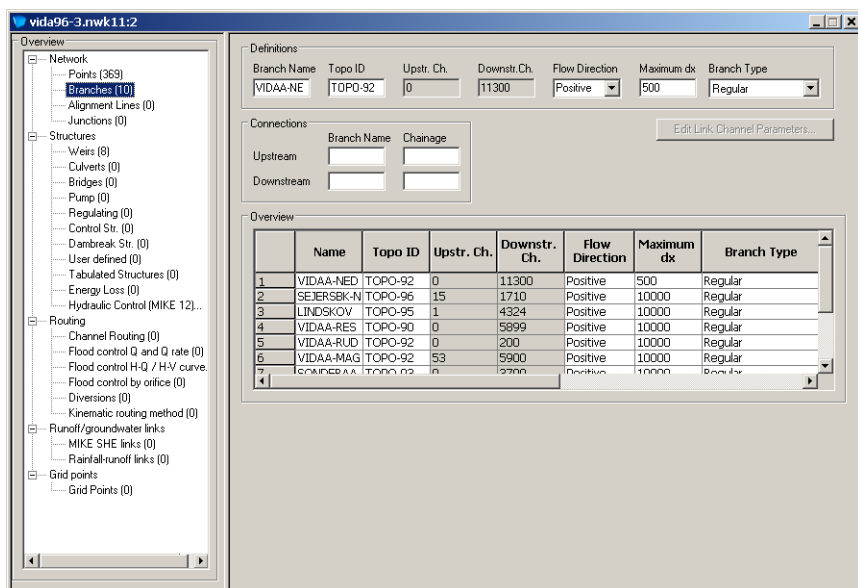


Figure 2.2    The Network Editor, Tabular View

Details on features and usage of the Network Editor including both the Graphical View and the Tabular View is described in the following.

Some of the features available in the Network Editor have been developed in cooperation with CTI Engineering, CO., Ltd., Japan. Amongst these are; Tabulated structures, Honma's weir formula, bridges (D'Aubuisson and submerged bridge), Routing along channels, Outflow from Dams/retarding basins and the Steady flow with vegetation.

## 2.1    Graphical View

The graphical view is the default view and will be activated automatically when a river network file is opened or created. Creation of a new network file (.Nwk11) requires an initial definition of the working area coordinates and map projection for the river model to be created. The opening window when creating a new network file is presented in Figure 2.3.
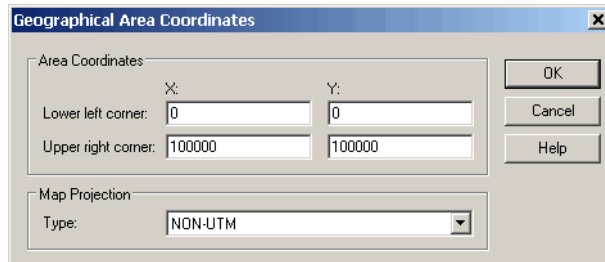
Figure 2.3    Geographical Area coordinates and map projection definition dialog appearing when creating a new network file (.nwk11)

Pressing OK opens a blank Graphical view - ready for the generation of the river network to be used in the project.

Note that additional graphical view sessions can be opened using the New Window item under the Window Menu.

Editing of the river network (i.e. the points and branches) is undertaken using the Graphical Editing Toolbar. Editing tools are also found using the Pop-Up Menu (right mouse button) where features like insert, edit and delete functions are included. Typically the Pop-Up Menu is used for editing of cross section geometry, parameters, hydraulic structures and data stored in other MIKE 11 editors.

Note that to access information from another editor other than the Network Editor, an editor filename must be specified using the Simulation File Editor.

Example of insertion of a Weir the Pop-Up menu is shown in Figure 2.4
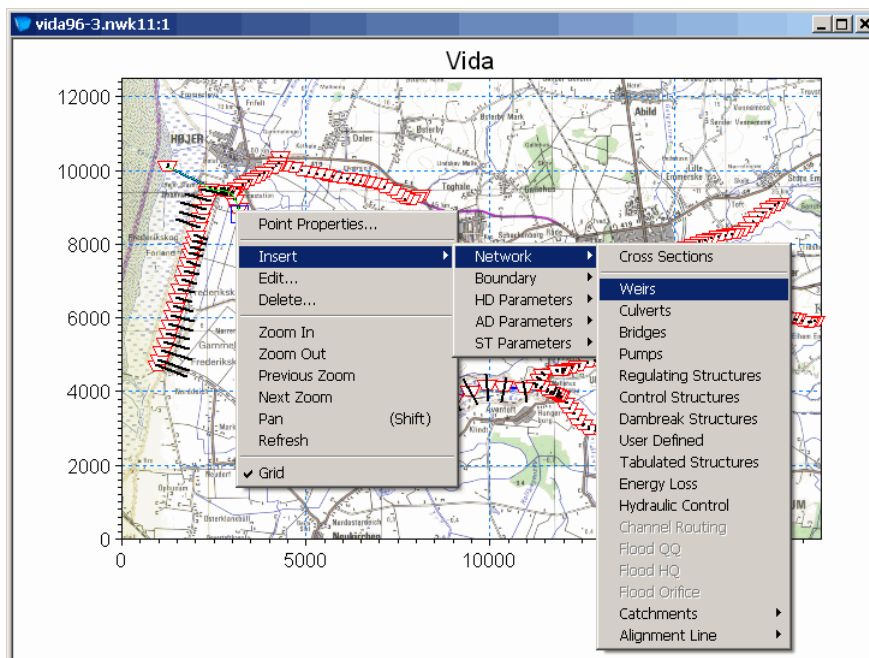
Figure 2.4    Illustration of right mouse pop up menu from where all data editors can be accessed

## 2.1.1    Network editing tool bars

The graphical view is facilitated with two tool bars. One for graphical editing of the river network, and one for graphical editing of alignment lines (see 2.2.3 Alignment Lines *(p. 67)* for more details about alignment lines)

### Tool Bar for River Network

The tool bar for graphical editing of the river network is shown in Figure 2.5. In the following the functionality of each of the icons in the tool bar is explained.



Figure 2.5    Tool bar for editing river network

**Select object.** This icon activates the selection mode which is the default mode. Points, layers and other objects can be selected by pointing and clicking with the left mouse button. Multiple

objects can be selected by moving the mouse to a corner of the area of interest, clicking and dragging with the left mouse button. Objects located within the marked area will be selected. Selected objects are identified by a red frame indicator.

**Add new points.** New points can be added by a point and click operation using the left mouse button. Multiple points can be added by pressing the left mouse button and holding it down while moving the mouse along the desired path. New points will be created with a spacing determined by the "minimum digitize distance" specified in the Mouse Settings property page of Network Settings. Points added with this tool will always be added as free points, i.e. not connected to a river branch. Alternatively you can add new points and define a river branch in one operation using the following tool.

**Add points and define branch.** This tool creates points and branches in a single operation. Point and click at successive locations along a desired path. Points can also be added by pressing the left mouse button and holding it down while moving. Double click on the last point to end the branch.

**Delete points.** This tool deletes both free points and points connected by a branch. Move the cursor over the point (the cursor will change style to indicate that a point has been detected) and press the left mouse button to delete. Multiple points can be deleted by holding the left mouse button down while moving the cursor over the points.

**Move points.** This tool moves both free points and points connected by branching. Select the point using the left mouse button and then drag to the desired location.

**Define branch.** This tool creates one or more branches by drawing a line through two or more free points. Select the first point to be included in the branch and drag the cursor through the free points to be included in the branch. Alternatively new points can be added in one operation by using the Add points and define branch

**Auto route branch.** This tool automatically determines a river branch route from a set of free points. To use this tool you select the first point and drag to the last point on the branch. The editor automatically determines a path through intermediate free points by always searching for the closest point.

**Delete branch.** This tool deletes a branch without removing the river points. Point at the branch to delete and click once with the left mouse button.

**Cut branch.** This tool divides a single branch into two separate branches. Move the cursor to the required segment where the

break is required. When the cursor changes style press the left mouse button once to cut the branch.

**Merge branch.** This tool merges two separate branches into one. Move the cursor to the beginning or the end of a branch, click at this point with the left mouse button and drag to the connection point on another branch.

**Insert point.** This tool will insert free points into an existing branch. Move the cursor to a point on an existing branch, click with the left mouse button and drag the cursor to the free point for inclusion into the branch path.

**Exclude points.** This tool will exclude points connected along a branch. Move the cursor over the point to be excluded and click the left mouse button once. The point is excluded from the branch path but is not deleted.

**Connect branch.** This tool is used to connect two river branches at a junction point.Click and drag with the left mouse button from a river branch end point (upstream or downstream) to the junction point on a neighbouring branch.

Please be careful when connecting four or more branches. In such cases all branch connections should be made to a single junction point as shown in Figure 2.6.
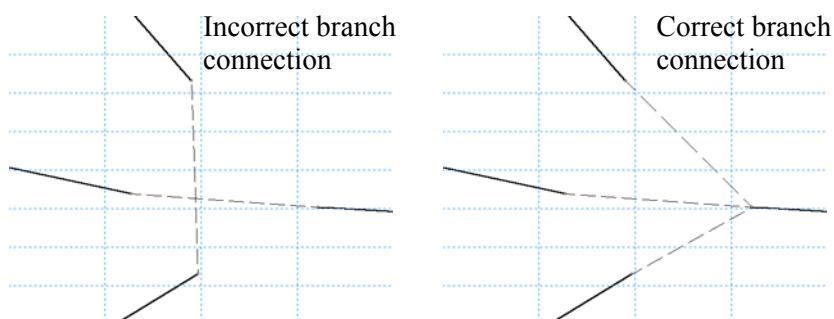


Figure 2.6     Connection of four or more branches.

**Disconnect branch.** This tool deletes a branch connection. Select the point at the end of the branch to be disconnected and click with the left mouse button once.

**Repeat insert.** The repeat insert tool will add a copy of the latest object (weir, cross section, boundary condition, initial condition etc.) created using the Insert facility in the Pop-Up Menu. The repeat insert button is a fast and convenient way of inserting multiple objects to the river network. The current object type is shown in

the status bar when the repeat insert button is activated. After activating the repeat insert tool you should point and click once at the desired location of the new object.

**Select & edit.** This tool is similar to the Edit facility found in the Pop-Up Menu. The tool is a fast convenient way of accessing the various editors required for objects on the river network. To control the number of editor windows activated use the Select and Edit Settings Property Page of the Network Settings property sheet.

## Tool Bar for Alignment Lines

The tool bar for graphical editing of the river network is shown in. In the following the functionality of each of the icons in the tool bar is explained.



Figure 2.7    Tool bar for editing alignment lines.

**New alignment line.** Add a new alignment line by pointing and clicking at successive locations along a desired path. Points can also be added by pressing the left mouse button and holding it down while moving. Double click on the last point to end the line. Once added the line should be given the correct type and be connected to a branch.

**Move alignment line points.** This tool moves points on an alignment line. Select the point using the left mouse button and then drag to the desired location.

**Delete alignment line points.** This tool deletes points on an alignment line. Move the cursor over the point (the cursor will change style to indicate that a point has been detected) and press the left mouse button to delete. Multiple points can be deleted by holding the left mouse button down while moving the cursor over the points

**Insert points to alignment line.** This tool will insert free points in an existing alignment line. Move the cursor to a point on an existing branch, click with the left mouse button and drag the cursor to the free point for inclusion into the branch path. The free point to be inserted must be added using the tool "Add new point" in the available in the toolbar for river network editing.

**Add points to alignment line.** Using this tool you can add points to an existing alignment line. Point are added at the upstream or downstream end. Click once at the point to which you

want to add new points. Then point and click at successive locations along the desired path.

**Spline alignment line.** Splines an alignment line by automatically adding new points in between the existing points. Once you have clicked at the icon in the tool bar, you should click once at the first point in the branch to be splined, then click at the last point. Points will be added only between the first and last point clicked at. The coordinates of the existing points will not change a result of the splining. Figure 2.8 shows an alignment line before and after splining. Five points have been added between all existing points

Before splining    After splining
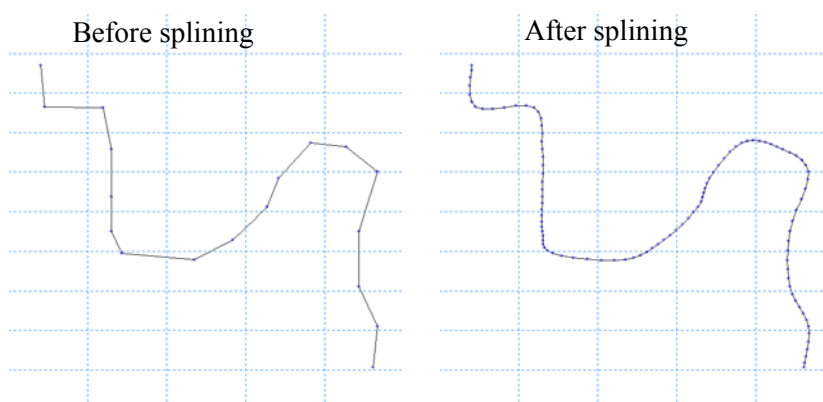
Figure 2.8    Alignment line before and after splining

**Merge alignment lines.** Merges two existing alignment lines into one such that the properties for the merged line equals the properties of the first line. Click once at the downstream end of the first line, then click once at the upstream end of the second line, and the lines are merged.

**Connect alignment line.** Connects a new alignment line to a branch. Click once at the alignment line to be connected, then click once at the branch the alignment line should belong to.

**Dead water line in vegetation.** Adds a dead water zone behind a vegetation zone (see Figure 2.39). The vegetation zone must be connected to a branch before this tool is applied. Using this tool the should select (by clicking once) the two points along the vegetation zone at which the two dead water lines should start. Once the user has selected the two points the tool automatically finds the direction of the flow by finding the point on the branch which is closest. This defines the guide lines, and once the angle between the guide line and the dead water line is specified by the user, the dead water lines are created.

**Dead water line along bank.** Adds a dead water zone adjacent to an expansion (see Figure 2.38). This tool is only available for left/right levee bank alignment lines that has been connected to a branch. Using this tool the should select (by clicking once) the two points along the bank line at which the two dead water lines should start. Once the user has selected the two points the tool automatically finds the direction of the flow by finding the point on the branch which is closest. This defines the guide lines, and once the angle between the guide line and the dead water line is specified by the user, the dead water lines are created.

## 2.1.2 Main menu bar - File

The File menu under the main menu bar (Figure 2.9) contains the standard Windows file operation features (New/Open/Close/Save) as well as features for opening the most recent files or projects edited in previous editing instances.

Additional features comprise the printing and the key issue in relation to the Network Editor is the possibility of importing network data from external text-file containing network data for either points, branches or both.
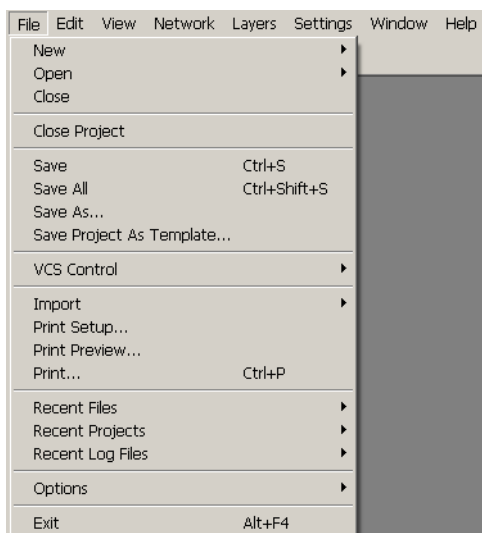


Figure 2.9     The File menu under the Main menu bar

The import options available are described in the following:

## Import

### Point and Branch Data from Cross-Section ASCII File

If a cross section file has been exported to a text file, this text file can be imported to the network editor. In this way point and branch information is passed from the cross section file to the network file.
**Please note** that this option only is relevant when the cross section file holds information about the coordinates.

### Point and Branch Data from Point-Branch ASCII File

Point and branch information can be read into the network file using a text file with one line for each point to be imported.

The text-file must have the following format:

x-coordinate y-coordinate Branch_Name Chainage

Example:

312333.00 6829000.00RIVER_1 0.000
312830.00 6829050.00RIVER_1 499.00
311385.00 6829260.00RIVER_11959.00
311421.00 6829871.00RIVER_12571.00

### Alignment Points and Lines from PFS Files

This feature is only appropriate if the Quasi Two Dimensional steady state with vegetation module is used. It provides a way of importing alignment line data into a setup.

The data in the file must be of the form shown in Figure 2.10. The file should contain a section of the type "[AlignmentLine]... EndSect // AlignmentLine" for each alignment line.

```
[AlignmentLines]
    [AlignmentLine]
        Point = X(1), Y(1)
        Point = X(2), Y(2)
                |
                .
                |
        Point = X(N), Y(N)
    EndSect  // AlignmentLine
 EndSect  // AlignmentLines
```

Figure 2.10    The format used for importing alignment line data.

## 2.1.3    Main menu bar - Edit

The Edit menu (Figure 2.11) contains the standard Windows options for copy, paste, select and delete as well as some specific features for the network editor, Tabular view.
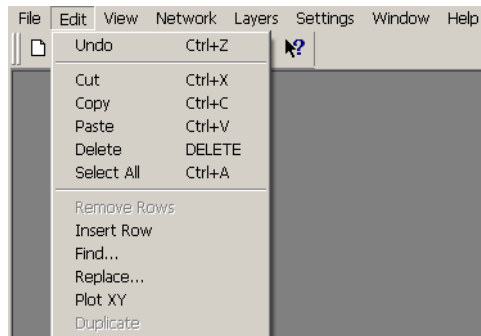


Figure 2.11    The Edit menu of the Network editor Main menu bar

The following features are applicable only when located in one of the tables in the Network Editor, Tabular View property pages.

### Remove rows

Delete a selected row in a table.

### Insert rows

Insert additional row at highlighted location in table.

### Find...

Search for a specific string in an active table

### Replace...

Search and replace string values in tables.

### Plot XY

Plot of two selected column in a XY-like plot.

### Duplicate

Duplicate the currently selected item in an overview grid.

## 2.1.4    Main menu bar - View

The View menu of the Main menu bar (Figure 2.12) contains a number of features for viewing or presenting type of data related to the Network editor, Graphical View.
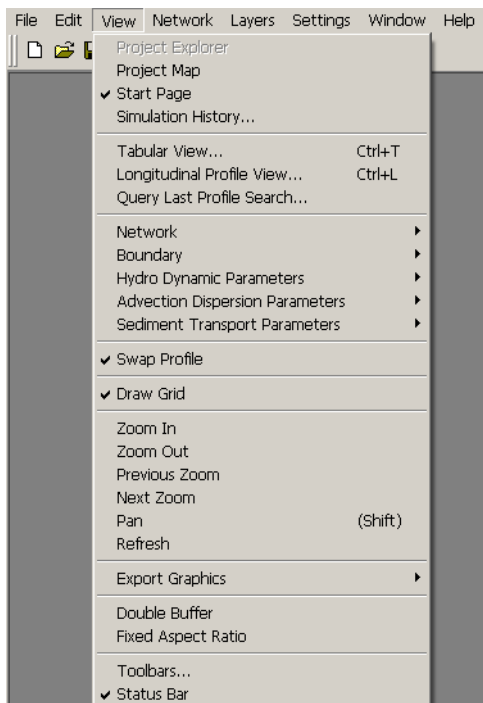


Figure 2.12    The View menu of the Network Editor Main menu bar

### Project Explorer

General MIKE Zero switch for showing the project explorer as a visible component in the MIKE Zero interface (Primarily useful if the MIKE 11 editor files are part of a MIKE Zero project)

### Project Map

General feature for the MIKE Zero project map feature. Switch for making the project map visible within the MIKE Zero GUI.

### Start Page

Switch on or off the MIKE Zero Start Page.

### Tabular view

Used to open the tabular view of the network file.

> **Note** that a hot-key combination; **'CTRL+T'** is hard-coded for an easy entry to opening the Tabular View directly from the Graphical view.

## Longitudinal Profile View

Used to select a longitudinal profile for viewing. Select the profile by clicking the mouse at the first and at the last branch to be included in the profile.

## Query Last Profile Search

Selecting a longitudinal profile in a looped network by clicking at the first and the last branch in the profile sometimes results in more than one profile. All the possible profiles can be examined by using the 'Query Last Profile Search' option.
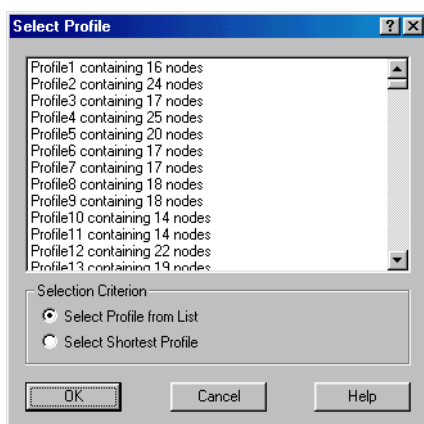


Figure 2.13    Dialog for selecting the required longitudinal profile amongst all possible options available.

## Network

The Network entry offers an easy way of turning on and off a number of the most common used graphical network objects in the graphical view. More detailed layout of the graphical view see Graphics *(p. 52)*.

## Boundary

Here the presentations of the different boundary types can quickly be turned on or off. For a more detailed layout of the graphical view see Graphics *(p. 52)*.

## Hydrodynamic Parameters

Here the presentations of the different hydrodynamic parameters can quickly be turned on or off. For a more detailed layout of the graphical view see Graphics *(p. 52)*.

## Advection Dispersion Parameters

Here the presentations of the different advection dispersion parameters can quickly be turned on or off. For a more detailed layout of the graphical view see Graphics *(p. 52)*.

## Sediment Transport Parameters

Here the presentations of the different sediment transport parameters can quickly be turned on or off. For a more detailed layout of the graphical view see Graphics *(p. 52)*.

## Swap Profile

The Swap Profile option relates to the plotting of longitudinal profiles, where the profile - once plotted - can be swapped in orientation (plotted either Upstream towards Downstream locations or swapped; Downstream towards Upstream locations)

## Draw Grid

A switch for drawing the grid or not in the graphical view.

## Zoom features

Standard zoom features for zooming in and out, previous zoom, panning or refreshing the view if required.

Note on panning: It is also possible to pan the graphical view by simply holding down the <SHIFT> button and move the pane with the cursor of the mouse (click and drag the view with the mouse).

## Export Graphics

The graphical view can be exported in the following ways:

- Copy to Clipboard.
- Save to metafile.
- Save to bitmap.
- Export layer graphics to file.

## Fixed Aspect ratio

Fixed aspect ratio are per default active in the graphical view such that the branches etc. are not distorted when changing the width and height of the graphical editor window.

Switching off the Fixed Aspect Ration allows for distorting the content in the graphical view such that the view always fills the entire window but the width-height relation is not locked when adjusting the window size.

### Toolbars...

Selection tool for MIKE Zero toolbars.

### Status bar

Switch for showing or hiding the status bar in the bottom of the view.

## 2.1.5 Main menu bar - Network

The Network menu as presented in Figure 2.14 offers a number of features for assisting the user in generating the river network.
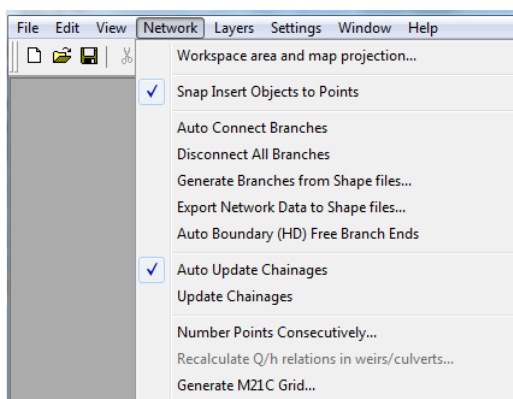


Figure 2.14    The Network menu in the Main Menu bar

A description of the menu items follows here under.

### Workspace Area and Map Projection...
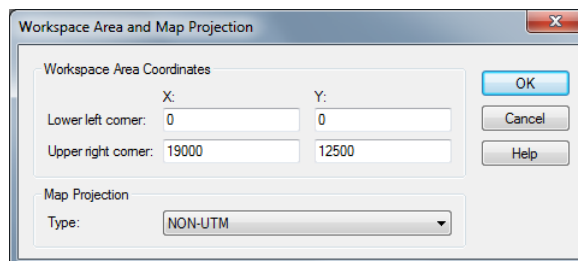


Figure 2.15    Menu for resizing and changing map projection settings for the area of the graphical view.

The graphical view can be resized by entering the minimum and maximum coordinates for both the x-axis and the y-axis.

Also, if required, it is possible to change the defined map projection for the network file through this property page.

## Snap Insert Objects to Points

Here the 'Snap Insert Objects to Points' option can be switched on and off.

## Auto Connect Branches

When selecting this option all the branches are automatically connected by use of the method selected in the 'Settings'->Network'->'Network Data' dialog (Network data *(p. 54)*).

## Disconnect All Branches

Choosing this options will remove all branch connections.

## Generate Branches from Shape files...

Selecting this item will open a dialog that allows for utilizing information in Shape files for automatic generation of points and/or branches.

**Note:** Prior to using the network data in a Shape file for generating (parts of) the river network it is required to load the same shape file as a background layer. Otherwise, it is not possible to select the appropriate shape-file from the file selection drop-down boxes as shown in below.
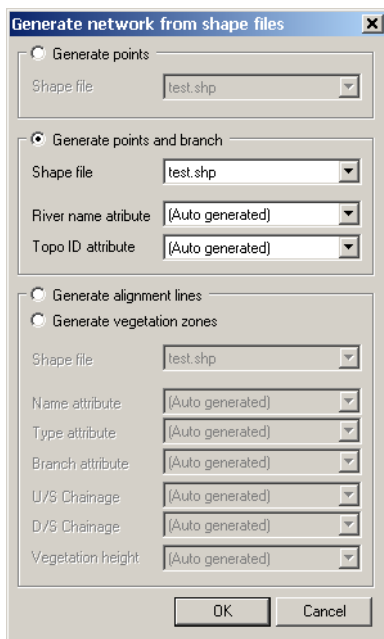
Figure 2.16    Generate river network from Shape file dialog

Shape files with point information can be used for generating points, and Shape files with polylines can be used to generate points and branches or only points. When generating points and branches together, the attributes defined in the Shape files can be used to define the river name and eventually the Topo ID (Topographical Identifier) for each branch defined from the Shape file.

The additional features of generating Alignment lines and vegetation zones are used only in connection with the Quasi two dimensional steady state with vegetation module (requires a special module in the license to activate this feature).

## Export Network Data to Shape files

The Export Network Data to Shape files dialog offers the possibility for generating shape-files from different Network data information.

Press the file-selection button ⬚ to define the name of the Shape-file that will contain the river network data.
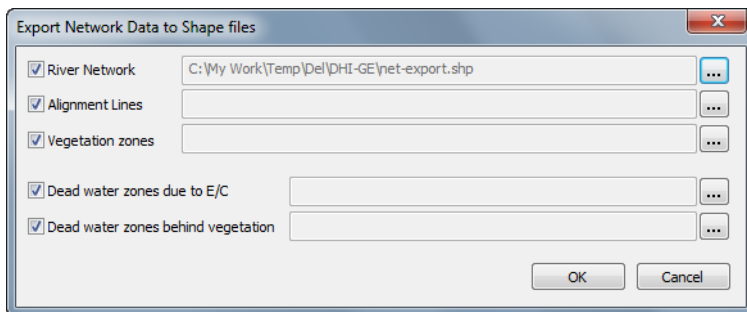
Figure 2.17     Export Network Data to Shape file dialog

**Note** that the last four options (Alignment lines, Vegetation zones, Dead water zones due to E/C and Dead water zones behind vegetation) are connected to the special Japanese version of MIKE 11 which requires special modules in the license to activate.

## Auto Boundary (..) Free Branch Ends

An easy way to create boundary definitions for all free ends in river branches is through this feature, which can create boundaries for the HD module, the AD module or the ST module depending on the selections in Network data *(p. 54)*.

**Note** that in order to use this feature, it is required to have both the network file and a boundary file 'connected', which is done by having a simulation file open in which both the network and the boundary files are selected in the 'Input' page of the simulation editor

## Auto Update Chainages

When this option is selected the chainages of the points will be updated automatically during additional editing processed where river branches are adjusted, points moved etc. which can influence the chainage of points in the network.

## Update Chainages

This option is only meaningful if the Auto Update Chainages option is not selected. The Update Chainages option could be used after having moved one or several points.

### Number Points Consecutively
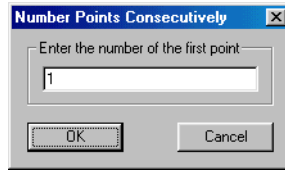


Figure 2.18    The menu in which the number of the first point can be entered.

When joining two network files (see B.1.1 Merging .pfs files *(p. 499)*) it is necessary that the number of the points in the two files do not overlap. To avoid this it is possible to renumber the points internally in one of the network files.

**Note** that the renumbering will not re-arrange the location of points in the Tabular view, Points page but only change the internal numbering of points in the network file which amongst other things are used to determine which points is used to define each river branch in the network.

### 2.1.6    Main menu bar - Layers

The Layers menu (Figure 2.19) includes features for managing layers in the Graphical view.
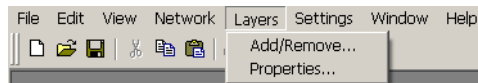


Figure 2.19    The Layers menu of the Network editor Main menu bar

### Add/remove

The Add/remove dialog as presented in Figure 2.20 offers the features for managing layers in a network file.

The dialog contains two tab-pages:

1.  'Add/Remove Layers':
    The Add/Remove property page includes options for adding or removing layer files in the network file. Use the ▭ button to add an additional line to the Layers table and select the layer type to import in the 'File Type' drop-down selection box.
    Browse for a layer file using the file browsing button; ▭ to search for the layer file to import.

Use the delete-button ❌ to remove selected layers from the layers dialog.
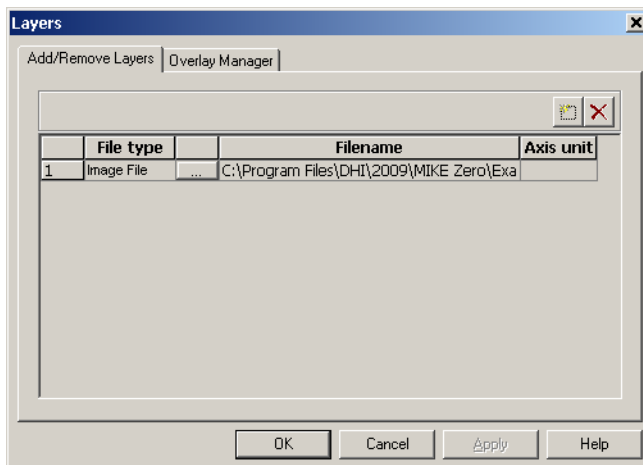


Figure 2.20    Add/Remove Layers page

It is possible to import the following type of Layer files:

- Image files (.bmp, .jpg and .gif)

- Shape files. (.shp). Please note that Shape files including m-and z-data are not possible to load in the present version

- Grid files (.dfs2, dt2). 2-dimensional grid-files stored in the DHI dfs2 file format.

**Note:** When loading image files, the geo-reference is automatically set to (0,0) and (10000, 10000) for lower left and upper right image corners respectively. Image coordinates can thereafter be adjusted through the "Properties..." page in the Layers menu.

2.  'Overlay Manager'
    If multiple layers have been imported it is possible to arrange the order of the different layers in the 'Overlay Manager' dialog (Figure 2.21) by selecting a specific layer and using the arrows; ⬆⬇ to either move the specific layer up or down in the list of layers.
    Layers are displayed in the Graphical View following the order defined in this Overlay manager page such that layer in line 1 is plotted first, then layer 2 on top of layer 1, then Layer 3 on top of Layer 1 and 2 etc.
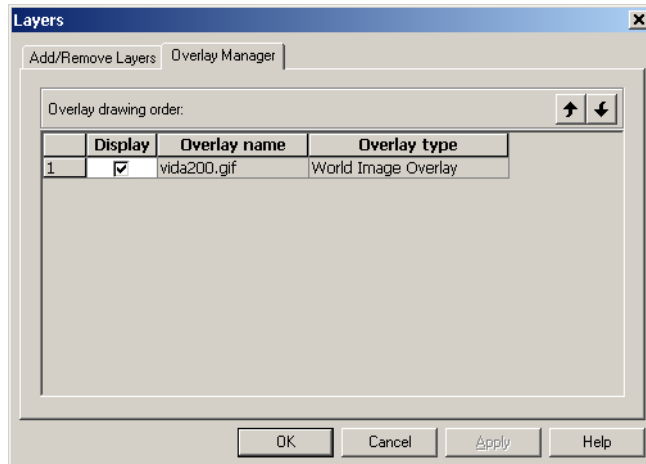
Figure 2.21    Overlay Management page for Layers management

## Properties...

The properties dialog (Figure 2.22) defines the appearance and characteristics for the individual layers imported to the Network editor file.
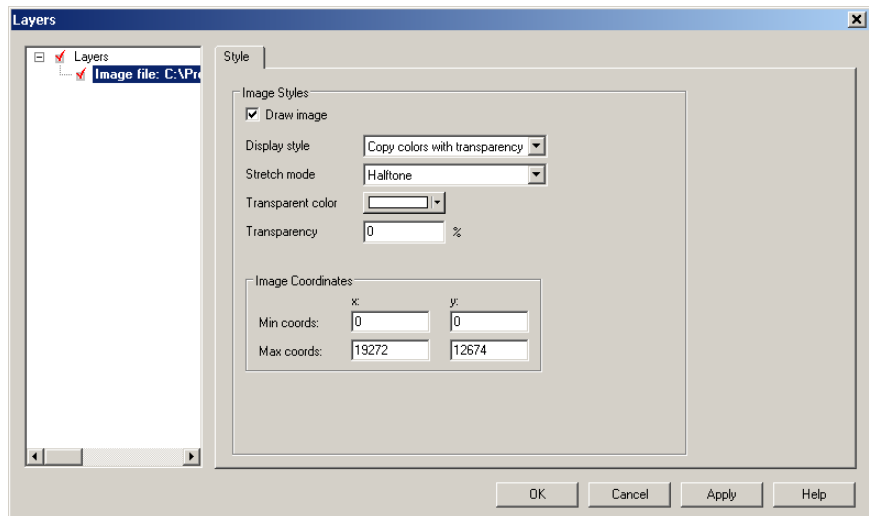


Figure 2.22    Network editor, Layer Properties dialog

Property definitions depends on the specific layer type:

- Image files properties:
  Display style and the image coordinates for the image file in order to make a proper geo-referenced location corresponding to the location and geographical projection of the River Network

- Shape file properties:
  Shape file imported contains either points, polylines or text-strings and therefore, properties include definitions of colours, style and size/thickness.

- Grid file properties:
  Grid file layers can be presented with or without Element mesh, Grid lines, colour legend and Iso-lines as well as different contour-types can be applied.

Layer settings are saved in the Nwk11-file such that these are maintained when loading the file at a later stage.

## 2.1.7 Main menu bar - Settings

The Settings menu of the main menu bar offers features for customizing the appearance and functionality of the Graphical View. Two entries are present as presented in Figure 2.23.
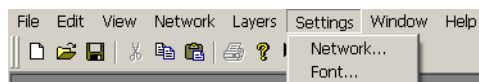


Figure 2.23    The Settings menu of the Network Editor Main menu bar

### Network...

The 'Network...' option opens the network settings dialog containing the following four property pages:

- Graphics
- Mouse
- Network data
- Select and edit

Details on the property pages content follows below.

#### Graphics
The Graphics property page (Figure 2.24) controls the appearance and partly the functionality of the graphical view.
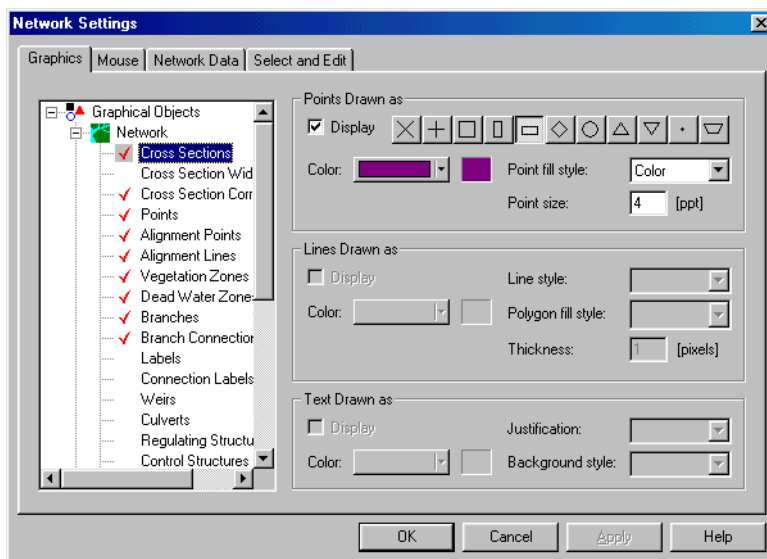
Figure 2.24    The Graphics property page

The left hand side of the page presents the graphical object possible to present on the graphical view - organized in a tree structure. Each object in the tree-view has an entry for defining individual settings for points, lines, labels, etc. Open settings for editing a specific object by opening the tree-view and highlight the object you wish to edit. Selecting an object allows for changes in the right hand side of the dialog.

The 'Display' tick-box determines whether the specific object will be presented in the Graphical view (in case the object is present in the network file presently edited).

Additionally, it is possible to switch on or off the display of all items in an entire branch of the tree-view (Network, Boundary, Hydro Dynamic Parameters, etc) by using the right mouse button on a branch.

### Mouse

This property page sets the properties for the cursor (mouse) when digitizing and editing points on the graphical view.
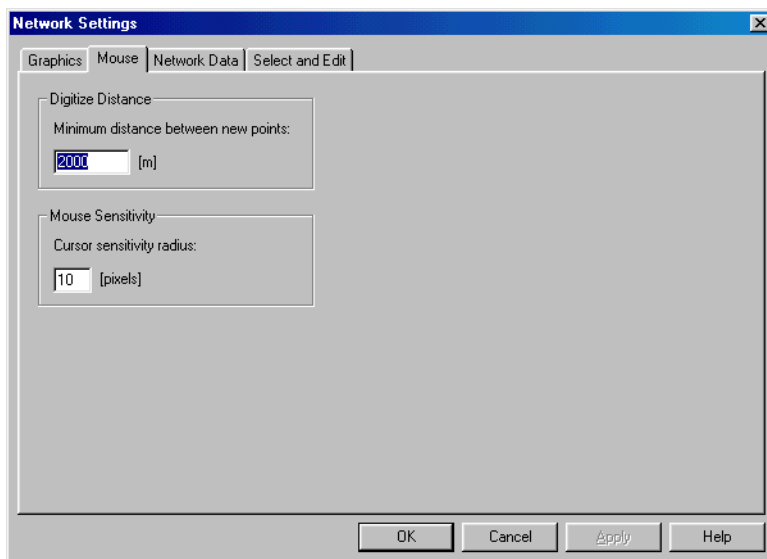
Figure 2.25    The Mouse property page

The Digitize distance; 'Minimum distance between new points' sets this mini-mum distance at which a new point can be defined following a previous digi-tized point. Consequently, a smaller value allows for a fine detail digitization (many points) whereas a large value reduces the number of points during a digitization process.

The Mouse Sensitivity; 'Cursor sensitivity radius' determines the radius in pix-els for which the mouse detects points in the graphical view.

## Network data

The Network data property page (Figure 2.26) contains global, default set-tings for misc. parameters related to specific graphical view features.
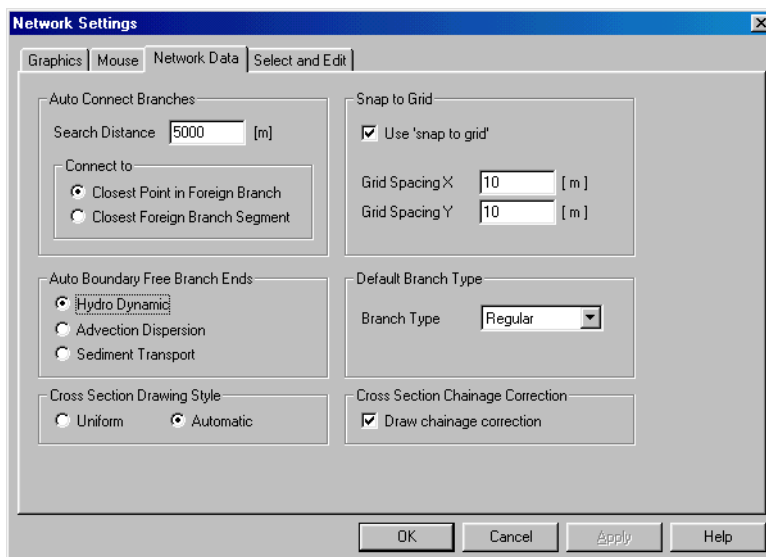
Figure 2.26    The Network Data property page

The options available in the Network page are:

*Auto Connect Branches*

> **Search Distance**:
> The maximum search radius applied when using the Auto Connect Branches facility under the Network Menu can be specified here.

> **Connect to**:
> The automatic connection can either be made to the nearest point or to the nearest branch segment.

*Auto Boundary Free Branch Ends*

> The selection here determines which type of boundary conditions will be defined in the boundary file when using the 'Auto Boundary (xx) Free Branch Ends' tool in the View menu of the Main Menu bar. Options are; HD, AD or ST models.

*Cross section drawing style*

> The cross section drawing style may be set to uniform or automatic

*Snap to grid*

> This facility may be used for snapping points to a user defined grid. The spacing of the grid may be defined here as well.

> **Note** the grid spacing used for snapping is not shown.

*Default branch type*

The Branch Defaults determines which type of branch MIKE 11 will create during branch digitizing in the graphical view. Options are; Regular, Link Channel and Routing. Following the creation of branches in the Graphical view by digitization, it is possible to change the branch type if required in the Tabular view, Branches page.

The 'Rotate Branch Graphical Symbols' check-box enables rotating of graphical symbols such as triangles, rectangles etc. on the plan plot. Unselecting the checkbox results in symbols always being oriented 'north-south'. If enabled, symbols will be oriented towards the direction of the river branches as illustrated in Figure 2.27.
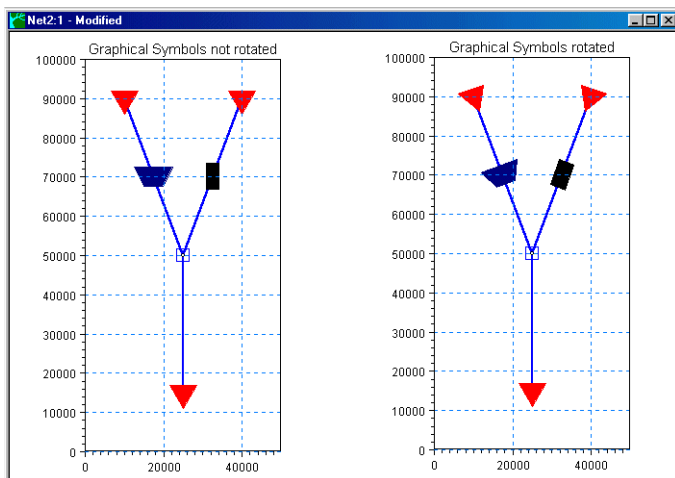


Figure 2.27    Illustration of 'Rotate Symbols' feature in MIKE 11 Network Editor

*Drawing Title*

Defines a title, or header string, which will be presented on top of the Graphical View.

## Select and edit

The 'Select and edit' page (Figure 2.28) includes a number of tick-boxes which simply determines which editors shall be possible to access using the 'Select & Edit tool' from the Graphical View toolbar.
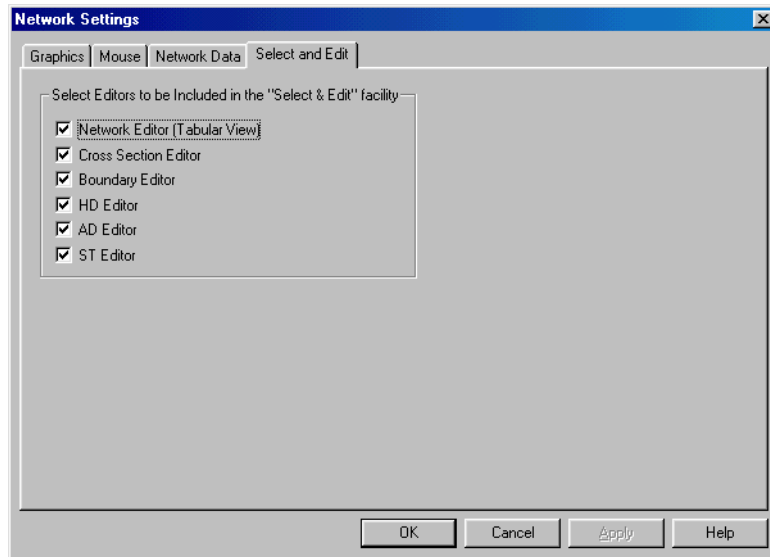
Figure 2.28    The Select and Edit property page

## Font...

Feature for defining Font settings of the Graphical View (Figure 2.29).
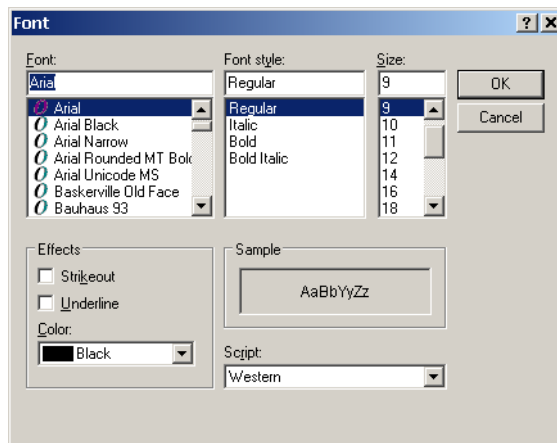


Figure 2.29    Font settings dialog

## 2.1.8    Main menu bar - Window

The Window menu option (Figure 2.30) offers the standard Windows features for arranging open windows (tile, cascade, arrange) as well as a possibility for opening an additional instance of a Graphical View window.
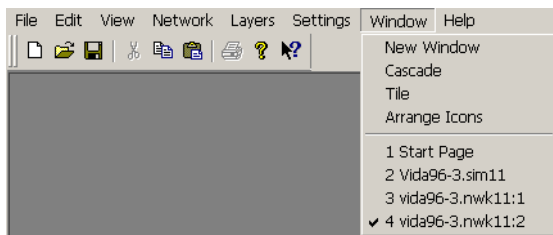
Figure 2.30     The Window menu of the Network editor, Main menu bar

Additionally, all open windows are listed in the lower part of the menu allowing for an easy location of and access to a specific editor file.

### 2.1.9    Main menu bar - Help

The Help menu (Figure 2.31) is a general feature located in all MIKE Zero editors.
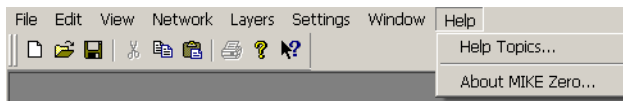


Figure 2.31     The Help menu of the Network editor Main Menu bar

The Help menu follows the standards of Windows programs and offers two options:

#### Help Topics...

The Help Topics option opens the MIKE Zero on-line help framework through which any help item of a specific MIKE Zero product can be accessed.

#### About MIKE Zero...

The About dialog supplies details on the present MIKE Zero installation such as; Current Version, License owner and License Type.

## 2.2    Tabular view: Network

The tabular view contains a number of property pages for editing specific data types such as digitization points, river branches, hydraulic structures, rainfall catchments, etc.

Use hot-key; '<CTRL>+T' to open the tabular view from the graphical view of the network editor.

## 2.2.1　Points

The geographical coordinates (x,y) of the digitization points in the river network may edited here.

**Note** that the Points in this dialog are only digitizing points used to define the appearance of the river network in the graphical view. The digitization points are not used in the calculation at all, so they must not be confused with calculation points.

The points dialog is shown in Figure 2.32.
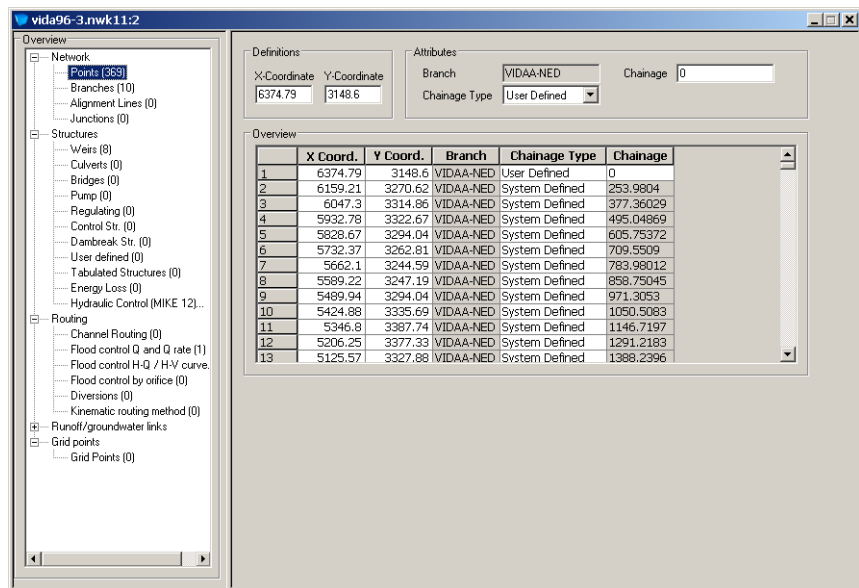


Figure 2.32　The points property page, Network editor, Tabular view

Digitization points are typically created through editing sessions in the graphical view where individual points or entire branch-segments are created from digitization.

However, it is also possible to generate a bulk of points in one single operation by importing points from a text-file through the 'File->Import' feature of the Main menu bar.

### Definitions

The X- and Y-coordinate of the selected point may be edited here.

### Attributes

Different attributes are available for editing.

### Branch

This displays the river branch to which the present point belongs and is only for verification purposes. If no branch is indicated, the point is free and hence not part of the model.

### Chainage type

The chainage may either be chosen as user defined or system defined, where system defined chainage type means that the chainage of a point will be calculated as a function of the location in the plan coordinate system and the distance to up- and downstream points.

At user-defined points, the user determines the exact chainage value. (e.g. at a bridge where the surveyor knows the exact chainage of the bridge measured from an upstream reference point). The points at the upstream and downstream ends of branches are best set to 'User-defined', and unless greater accuracy is essential the chainage is best rounded to the nearest metre.

### Chainage

If the chainage type is set to user defined the chainage may be edited using this box.

### Type - not used

## Overview table

An overview of the points is given in the overview table and all points data can also be edited in this overview table instead of using the above mentioned group-boxes for points editing.

## 2.2.2 Branches

The Branches property page (Figure 2.33) contains parameters defining the river branches and their connections in the river model.

**Please note** that it is not possible to create new branches within this page. If new river branches are to be included in the model then it is required to shift to the Graphical view and digitize a new river branch in the graphical View. The creation of a river branch in the Graphical View automatically inserts a new line in the present Branches page.

**Also note** that it is not possible to adjust the chainages for the upstream and downstream point of the river branch in this page! These chainages are being ported to the Branches page from the Points page where chainages for digitization points belonging to specific river branches are defined. Therefore, it is strongly recommended in order for the user to control the exact length of a river branch to define - in the Points property page - at least the first and the last points in a river branch as 'User-Defined' points such that an exact chainage can be defined at the upstream and downstream points of a river branch

respectively. With this, you will be certain that MIKE 11 uses exactly a river branch length as you defined it should be!
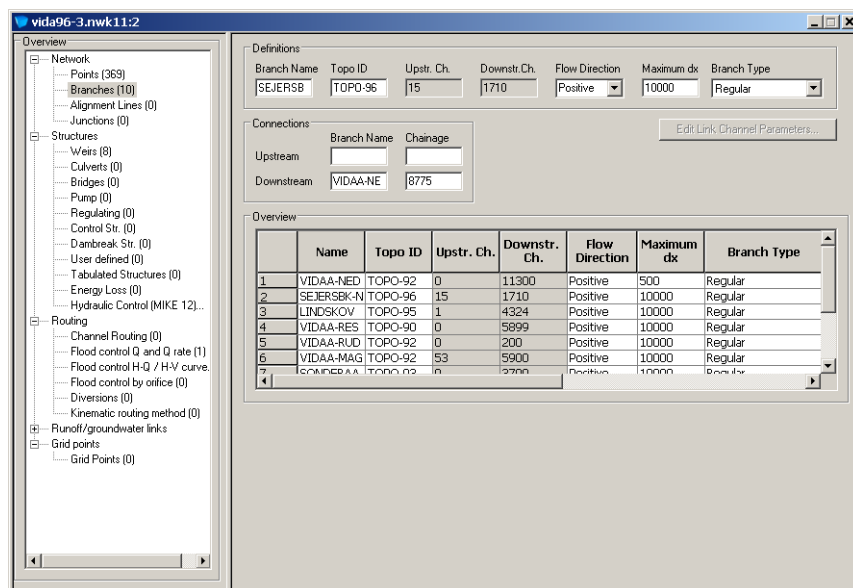


Figure 2.33    The branch property page

Property page parameters are described below:

## Definitions

### Branch name
Name of the actual river branch.

### Topo ID
Topo ID is an abbreviation of 'Topographical Identifier', which refers to the data defined in the Cross section file (xns11). The cross section file offers the possibility to store several measurement campaigns (e.g. from different years or before and after the construction of a bridge structure). The main purpose of this Topo ID is to ease the implementation of an alternative scenario simulation once the river model has been properly defined. As it is normally a sound assumption to say, that the river digitization and length, etc. does not change then it is possible to change the topographical data used in a simulation simply by changing name of the Topo ID for one or more river branches (and eventually save the nwk11 with a different name than the original file).

**Note** that the String defined for the Topo ID must be identical to a Topo ID present in the cross section file for the actual river branch. If not, the calculation can not be performed.

### Upstr. Ch.
The chainage of the first point in the branch.
In case the chainage value of this point should be adjusted it must be changed in the 'Points' property page.

### Downstr. Ch.
The chainage of the last point in the branch.
Again, in case the chainage value of this point should be adjusted it must be changed in the 'Points' property page.

### Flow Direction
Indicator for how MIKE 11 should interpret the chainage definition with respect to the natural flow direction. The default assumption of MIKE 11 is to define flow direction positive with increasing chainages looking downstream of the river.

If flow direction is selected as positive, the simulated discharges will be positive when the flow direction is from upstream chainage to downstream chainage. Vice versa if the flow direction is defined as negative.

Hence, a river survey with river chainages starting with 0 at the river mouth and chainages increasing upstream should be defined as a 'Negative Flow' branch as illustrated in.
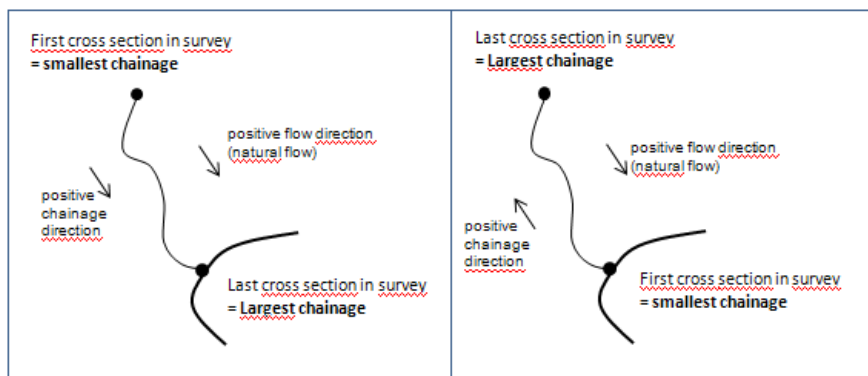


Figure 2.34    Illustration of flow direction definition. Left figure illustrates situation where increasing chainages follow positive flow direction and branch should therefore be defined as 'Positive Flowdir' whereas the right part of the figure illustrates the situation where a survey has increasing chainages from the river outlet and upstream. This situation requires the river branch to be defined as 'Negative Flowdir'

## Maximum dx

Maximum dx is the maximum grid-spacing (or distance) allowed by the calculation engine between two adjacent h-points (water level calculation points). At locations where Cross sections are present in the cross section file, the calculation engine will always create h-points at these location and in case the Maximum dx is defined with a value smaller than the distance between cross sections, the calculation engine will automatically insert a number of additional calculation points by interpolation in between existing cross sections such, that the maximum distance between h-points in the calculation will less than or equal to the defined value of Maximum dx.

Hence, the Maximum dx parameter is a possibility for increasing the spatial resolution of the river model by forcing an additional number of calculation points in between physically defined cross sections in the cross section file.

## Branch Type

- **Regular:**
  A 'normal' branch composed by a number of calculation points defined from cross sections. A minimum of one cross section is required

- **Link Channel**:
  No cross sections are required. Instead the parameters given in the Link channel dialog must be specified using the The 'Edit Link Channel Parameters...' button button.
  Note that LINK channels are ONLY to be used when conducting Hydrodynamic simulations.

- **Routing:**
  A special routing feature is available in the network branch definition in case a special module is present in the MIKE 11 license. This feature has been developed as part of a dedicated Japanese version of MIKE 11 and hence only applicable if a special M11-Japanese module is purchased and added to the MIKE 11 license.
  The Routing option in the branch type column of the Branches dialog as well as the 'Routing' option in the tree view as described in section 2.4, is only visible to users holding the required special module in their license.

  No cross sections are required. Only the flow is calculated, no water levels. See section 2.4 Tabular view: Routing *(p. 148)*.

- **Kinematic Routing**:
  Kinematic 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. The Kinematic Routing method does not facilitate the use of structures at Kinematic Routing branches. Moreover, the method does not account for backwater effects.

  At Kinematic Routing branches, it is possible to run the model without information on cross-sections. In turn, this indicates that Kinematic Routing branches can not be used to model a looped part of a river network.

Employment of Kinematic Routing branches requires that all branches located upstream of a Kinematic Routing branch are defined in the same way.

- **Stratified - no longer available:**

## Connections

The connection point of one branch to another can be specified here. However, it is recommended that branch connections be defined using the Connect Branch tool in the Graphical Editing Toolbar.

## The 'Edit Link Channel Parameters...' button

This button is only enabled when a branch has been defined with the branch type of 'link channel'.

### Link Channels - what are they?

The link channel is a short branch used to connect a flood plain to the main river branch and therefore typically represents the embankment geometry between parallel rivers (e.g. main river branch and flood plain branch). Link channels do not require cross sections to be specified and are consequently simpler to use than regular channels. The link is modelled as a single structure branch of only three computation calculation points (h-Q-h). The Q-point in the link channel branch is defined through an 'open culvert' type of structure. That is, an overflow weir with a length to include the friction loss component.

**Note:** Following the internal formulation of the link channel where no true cross sections are defined in a link channel branch, the **usage is restricted to Hydro-Dynamic simulations ONLY**. Thus, for Advection-Dispersion, Water-Quality and Sediment-Transport calculations the set-up should be void of link channels and instead connecting branches between main rivers should be defined out of 'regular' branches with cross sections defined and eventual a weir included to describe embankment levels if this is what should be represented with the connecting branch.

The link channel dialog (see Figure 2.35) is used for specifying all parameters appropriate for the link channel e.g. geometry, head loss coefficients etc.

Figure 2.35     Link channel property page

Typical Link channels representation are illustrated in Figure 2.36.
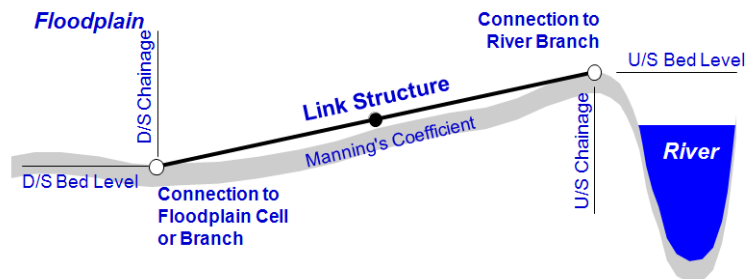


Figure 2.36     Longitudinal parameters and representation of a Link Channel

### Geometry

The link-channel geometry comprises typically the definition of a longitudinal geometry of the embankment along the river. The geometry is defined from the following parameters:

- **Bed Level US**:
  Upstream bed level of the link channel.

- **Bed Level DS:**
  Downstream bed level of the link channel.

- **Additional Storage**:
    Link channels do not contain cross sections and do not contribute to the storage capacity at nodal points where the link connects to a main branch. The Additional Storage parameter can be used to avoid zero storage at nodal points to which only link channels and no regular channels are connected.

    The Additional Storage combo-box defines whether additional storage is to be added at the upstream, downstream or both ends of the link channel. The actual storage is specified in the additional flooded area column of the processed data in a cross section defined at the same location as the link channel.

### Bed resistance

The bed resistance along the length of a link channel can be described using Manning's *M* or Manning's *n*.

### Head Loss Coefficients

All four factors are dimension less and must be within the range 0.00 - 1.00.

### Cross Section Geometry

A depth-width table defines the cross section geometry of the flow area in a link channel. Both the depth and the width must be increasing.

### Q/h - relations

Free outflow Q/h-relation must be calculated prior to calculation. To calculate the *Q/h* relationship, specify the number of relationships required and press the Calculate button. The result of the calculation will appear in the table. If any of the parameters defining the link channel geometry, or loss coefficients are changed the *Q/h* relations must be re-calculated.

Table values in the Q/H relations can be changed if required but general recommendation is to leave the table as is from the automatic calculation when pressing the Calculated button. Only the columns 'US Type' and 'DS Type' can not be changed. These are only indicators of flow type derived from the calculated flow parameters in the link channel and the values of these columns are not included in the calculations.

Table parameters are:

> y: water depth in link channel structure
>
> A: Flow Area in link channel structure
>
> R: Hydraulic Radius in link channel structure
>
> C: Conveyance indicator in link channel structure
>
> Qc:Critical flow in link channel structure

Hus: Water level in h-point upstream of link channel structure

Hds:Water level in h-point downstream of link channel structure

## 2.2.3    Alignment Lines

The alignment lines features are part of the quasi two dimensional steady state with vegetation module.

### Purpose

The purpose of using alignment lines is to save geo-referenced information in the network editor, and to utilize this information to update information in the cross section editor. Alignment lines information in the network file will influence the simulation results only when transferred to the cross section editor, and such transfer is requested in the cross section editor. The information in the cross section editor which is subject to be updated as the result of transferring alignment line information is:

- Positions of markers indicating left and right bank/levee (marker 1 and 3), left and right low flow bank (marker 4 and 5), and lowest point (marker 2).
- Zone classification.
- Vegetation height.
- Angle between cross section and direction of flow/branch.

### Definition.

An alignment line is similar to a branch in the sense that it is a line going through an ordered list of points with x- and y- coordinates. The following list of types of alignment lines are available:

- Left levee bank.
- Right levee bank.
- Left low flow bank.
- Right low flow bank.
- Thalweg.
- Vegetation zone.

An alignment line must belong to a branch in order to be taken into account. Only one alignment line of each type can belong to a branch. However, with the exception that any number of vegetation zones can be belong to a branch.

# User Interface

Figure 2.37 shows the property page for alignment lines. Each alignment line is shown as a row in the overview in the bottom of the dialog, and the x- and y-coordinates of the points along the actual line (the line in the row being high lighted in the overview) is shown in the details in the top of the dialog.
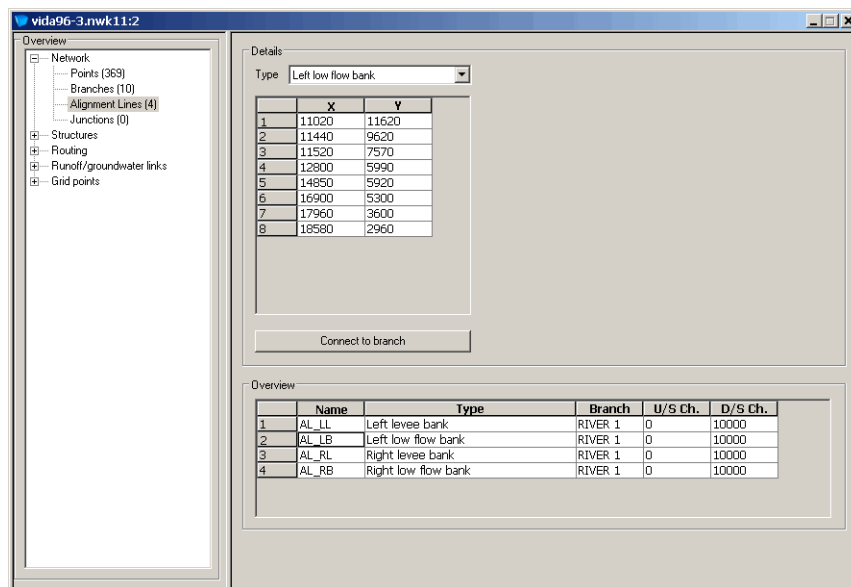


Figure 2.37    The alignment lines property page

Depending on the type of alignment line there may, in addition to the x- and y-coordinates, be other data shown in the details part of the dialog. These additional data are:

- **Left and right bank**:
  Each pair of expansion and contraction lines creates a dead water zone along the bank. The dead water zone is defined by the bank line between the expansion and the contraction point, and by two straight lines starting at the expansion and the contraction point. Each of these lines are defined by two angles. One being the angle (relative to the x-axis of the coordinate system) of an artificial guide line parallel to the main flow direction, and one being the angle between the guide line and the dead water line.

- **Vegetation zone**.
  A vegetation height is assigned to each vegetation zone. Similar to the dead water zone adjacent to an expansion there is a dead water zone downstream of a vegetation zone. There are two straight lines pointing in the downstream direction which defines the dead water zone. These lines are each defined by two angles. One being the angle (relative to the x-axis of the coordinate system) of an artificial guide line parallel to the main flow direction, and one being the angle between the guide line and the dead water line.
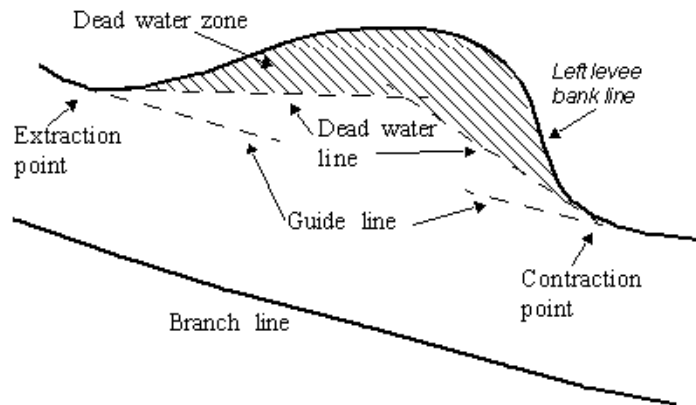
Figure 2.38    Definition of dead water zone along bank
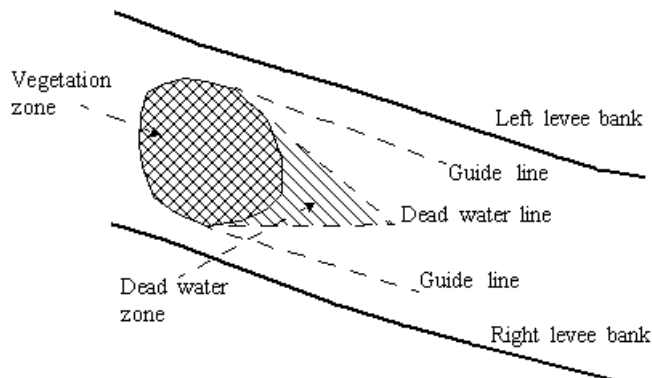
Figure 2.39    Definition of dead water zone behind vegetation zone

The x- and y-coordinates for the points along the alignment lines can be edited in three ways: 1) Using the tools available in alignment lines tool bar in the graphical view (see  Tool Bar for Alignment Lines *(p. 37)*). 2) Editing the numbers in the tabular view. 3) Using the File menu to import the coordinates from a text file.

Figure 2.40 shows a river network including alignment lines as visualized in the graphical view of the network editor.

Once the alignment data are added, the information is ready to be transferred to the cross section editor.
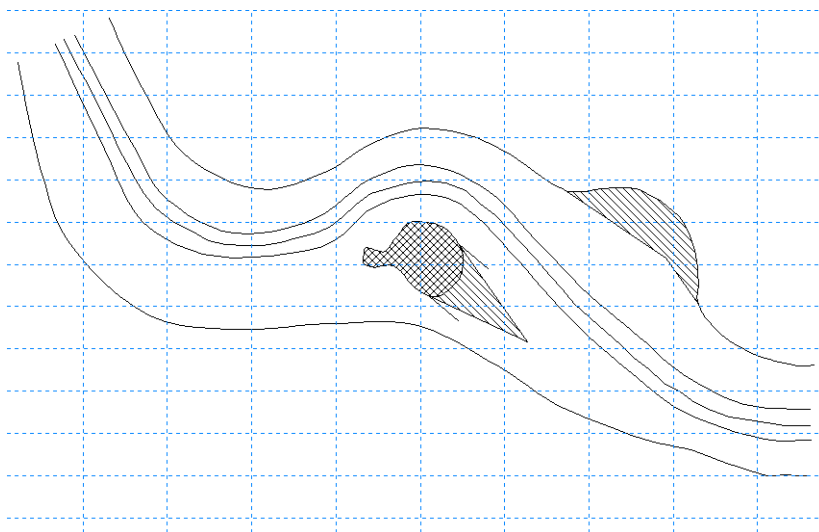


Figure 2.40    Example of a river network with alignment

## 2.2.4    Junctions

The junctions feature is part of the quasi two dimensional steady state with vegetation module.
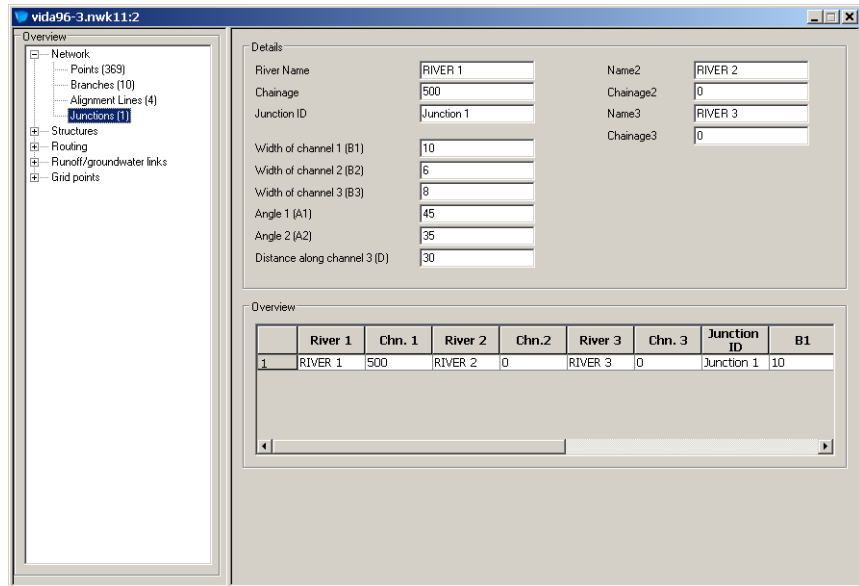
Figure 2.41    The Junctions dialog

### Details

**Name**, **Name2** and **Name3**: The river name of the three rivers meeting at the junction.

**Chainage**, **Chainage2** and **Chainage3**: The chainage of the three rivers meeting at the junction.

**Width of Channel1**, **Channel2** and **Channel3**: User defined width of the respective channels.

**Angle 1** and **Angle 2**: The direction angle of channel 1 and 2 with respect to channel 3.

**Distance along channel 3 (D)**: The distance along channel 3 at which the local water depth should be used for the determination of the water level in the downstream points of channel 1 and 2.

## 2.3    Tabular view: Structures

The Structures part of the Tabular View contains property pages for specifying details on all available structures in MIKE 11. Select the appropriate structure type in the Structures section of the tree-view to access the page for the structure type required to include in the model.

### 2.3.1 Introduction on structures modelling

A Number of Structures such as weirs, culverts, bridges etc. may be included in the river network set-up. The flow through most of the structures is calculated using the energy equation allowing inclusion of local head losses, but more simple type of structure equations such as weir formulas are also available. The effect of the bed friction is generally not taken into account thus it is recommended that the h-points up- and downstream are situated close to the structure.

Structures are always located at Q-points in which the momentum equation is normally solved, and the reason for including a structure in a model is to replace the momentum equation with something more suitable for the structure in question.

### 2.3.2 Composite structures definition

Composite structures are easily defined using an approach of combining a number of individual structure types. A composite structure such as a bridge with several openings and road over topping can be defined as a combination of multiple culverts and an overflow weir. The only requirement for a composite structure is that all individual structures in the composite structure are defined at the exact same location (same River Name and Chainage). An example is illustrated in Figure 2.42.
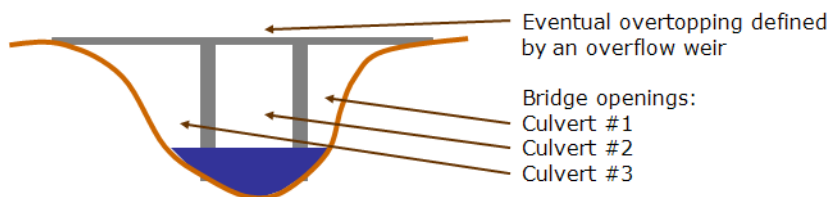


Figure 2.42   Composite structure illustration. Can be defined in MIKE 11 with a number of individual structures defined at exact same location

One of the benefits of defining a composite structure as illustrated above is that it is possible to obtain and analyse flow results through/over each of the individual structures forming the composite structure. To obtain the results for each structure use the 'Save Results in Structures' options in the additional output feature of the HD Parameter editor (see Add. Output *(p. 347)*).

### 2.3.3 Structure Types definition

Most hydraulic structures available within MIKE 11 can be defined as one of three different types; 'Regular', 'Side Structure' or 'Side structure with Reservoir'. Structures without this option works as regular structure.

### Regular

Regular structures are internal structures that specify a flow in the river branch typically based on up- and down- stream water level conditions.

### Side Structure

Side structures are special type of a regular structure. Side structures extract water from the river network at the location where the side structure is defined.

Internally in the calculation engine, side structures are handled through an automatic generation of an 'artificial' side branch with the side structure included as a regular structure placed midway in the side branch as illustrated in Figure 2.44.

Specific characteristics of a Side Structure include:

- Head losses are excluded from the flow calculation for side structures by setting internally the flow areas of the cross sections in the side structure branch as extremely large ($1e^{20}$ m$^2$) and hence, the velocity component in these sections (V=Q/A) is a negligible number.

- For culverts and weirs, an assumption of free out- and over-flow respectively is implemented through an internally defined water level boundary condition for the outflow point of the artificial side structure branch. The boundary condition is defined as a water level boundary with a low water level.

- The naming convention used for the artificial side branch is: "SS_"<original branch name>"_"<original chainage>

- In case of multiple side structures defined at the same location, these will all be included in the same side branch as a 'composite side structure'.

- A side structure branch is a h-Q-h branch with two cross section h-points in each end of the branch and a structure in the centre Q-point. Up- and downstream cross sections in the Side Structure Branch are automatically assigned with a shape identical to the main river cross section located immediately downstream of the side structure location. Upstream cross section of the Side Structure branch is identical to the section in the main river, but the downstream section has been shifted 10 m down in order to obtain a free outflow condition in the Side Structure. See Figure 2.43.
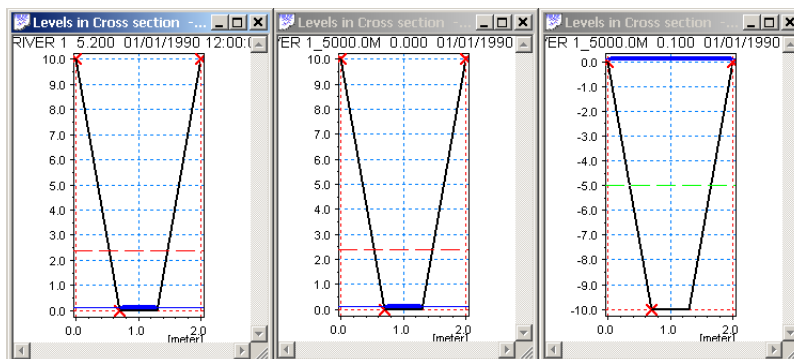
Figure 2.43    Cross sections as defined in the artificial Side Structure branch. Left part illustrates the cross section in the main river downstream of the chainage defining the side structure. Middle part illustrates the upstream cross section at chainage 0 of the side structure branch (=copy of main branch section) and right part illustrates the downstream cross section of the Side Structure Branch, which is identical in shape as the other sections but shifted by -10 m

An example of a Side Structure branch is illustrated in Figure 2.44:
Side Structure included in "RIVER 1" at chainage 5000 meter will effectively be placed in "SS_RIVER 1_5000.0M" in chainage 50 meter. The side branch, "SS_RIVER 1_5000.0M", will have a length of 100 meter.
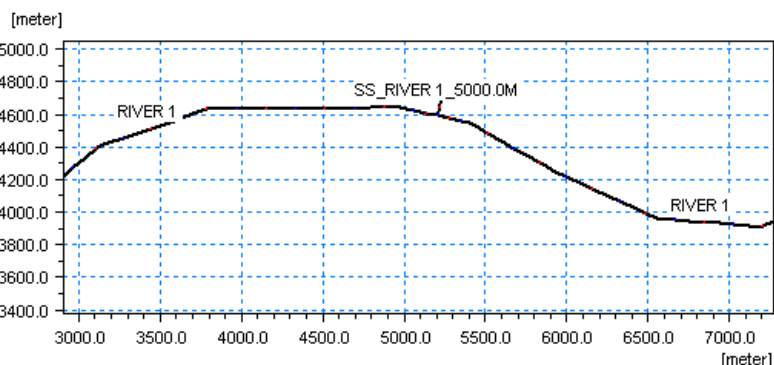


Figure 2.44    Illustration of automatically generated Side structure branch

## Side Structure with Reservoir

Inclusion of Side Structures with Reservoir in a branch results in an automatic generation of a side branch with the structure placed midway as described for Side Structures without reservoirs.

The downstream cross section of the side structure branch has a user-defined, additional storage area assigned, which assigns the reservoir data specified as a level dependent additional storage function (ref. Section 3.2.2 Processed data, Tabular View) defined in the downstream (outflow) h-point of

the side structure branch. The significant difference between a structure defined as 'Side Structure' and 'Side Structure with Reservoir' is therefore, that in the latter case, the water conveyed through the side structure is stored in the system and can be analysed, whereas for the structure type without the Reservoir, the water simply leaves the river model and disappears completely.

The Additional Storage relation is defined through the dialog opened when pressing the [ Edit reservoir... ] button (Figure 2.45):
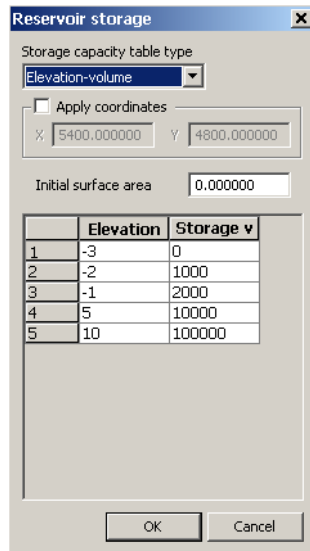


Figure 2.45    Dialog for specifying Additional storage for Side Structures with Reservoir

Two options for defining the Reservoir storage exist:

- *Elevation-area*
  The Elevation-Area relation specified in the table is used directly as the additional storage area as a function of the water level in the reservoir.

- *Elevation-volume*
  Additional storage area is derived from the Elevation-Storage relation following the relation:

$$FIA_{i+1} = \left( \frac{\Delta V_{Table}}{\Delta H_{Table}} - FIA_i \right) \cdot 2 \tag{2.1}$$

where:

$FIA_i$ : Flooded or additional storage Area at level 'i'

$\Delta V_{Table}$ and $\Delta H_{Table}$ are differences in Volumes and Levels respectively between values defined in the Storage-elevation table

Additional specific features of the Side Structure with Reservoir:

- Head losses are excluded from the flow calculation for side structures with Reservoir by setting internally the flow areas of the cross sections in the side structure branch as extremely large ($1e^{20}$ m$^2$) and hence, the velocity component in these sections (V=Q/A) is negligible.

- The boundary introduced through the inclusion of the side weir is internally specified as a no flow boundary. Flow may be included through specification of point source boundaries at the boundary.

- More side structures with reservoir in the same location will result in one side branch for each.

- The side branches will be named with the naming convention: "SSPR"<original branch name>"_"<original chainage>"_"<Structure ID>.

- '**Apply Coordinates**'. The Apply coordinates offers a possibility for defining coordinates for the reservoir - which in the case of the artificially generated side structure branch is identical to the end point of the side structure branch. Hence, the coordinates defines the appearance of the side structure branch and a length of the branch will be calculated according to the specified coordinates.

- The cross sections applied in the artificially generated Side Structure Branch with Reservoir are defined such that the upstream cross section is a copy of the section in the main river downstream of the chainage defining the side structure and the downstream cross section is a defined as a triangular cross section shape as illustrated in Figure 2.46.
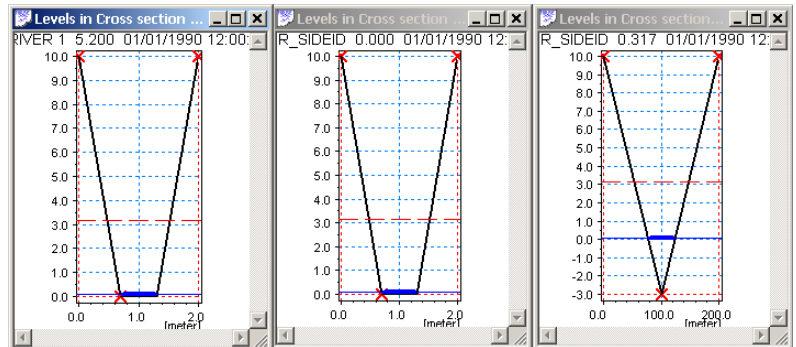
Figure 2.46    Cross sections as defined in the artificial Side Structure with Reservoir branch. Left section illustrates cross section in main river downstream of chainage defined for Side Structure. Middle section illustrates the upstream section of the side structure branch (copy of main branch section) and right section illustrates triangular section applied at downstream h-point of Side Structure Branch

Levels of the cross section is adapted from the defined Level-Area/Volume table, whereas the widths (0 m at bottom and 100 m in top of section are just artificial numbers applied in order to draw a cross section in the results presentation tools. The calculated storage volume of the section is defined from the table-values.

An example of a Side Structure with Reservoir definition: Side Structure included in "Branch1" at chainage 1000 meter with ID SideWeir1 will effectively be placed midway in branch "SSPR_Branch1_1000m_SideWeir1". The length of side branch, "SSPR_Branch1_1000m_SideWeir1", will follow from the position specified if any, otherwise it will be 100 metres.

## 2.3.4    Structure Geometry definition

Many of the structures available in MIKE 11 requires a Structure Geometry definition to complete the structure input.

It is important to notice, that the Structure Geometry to be defined is actually the geometry - or level-width relation - of the sections of the hydraulic structure where the water can potentially flow. That is, the Flow Area definition as illustrated in Figure 2.47 marked with the dashed (blue) section.
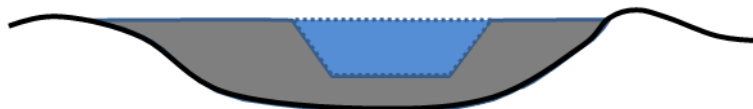
Figure 2.47    Illustration of Structure Geometry definition. The Dashed (blue) area which potentially conveys water over/through the structure is the geometry which must be defined

*Example*: Assuming the following geometrical definition of the dashed (blue) flow area in Figure 2.47: Crest level of structure is at level 3 m (crest level = bottom of blue area in figure), bottom width is 5 m, top-width is 7 m at a level of 5 m. A structure geometry definition in this case is normally defined as presented in Table 2.1 below:

Table 2.1    Weir Geometry definition

| Level [m] | Width [m] |
|-----------|-----------|
| 3.00 | 0.00 |
| 3.05 | 5.00 |
| 5.00 | 7.00 |

## 2.3.5    Structure Plotting

For structures where specification of dimensions and elevations is part of defining the structure, it is possible to make a plot of the structure together with the upstream and downstream cross section. This includes all weirs, all culverts, some bridges and most control structures (all except discharge control structures). Plotting structures and their neighbouring cross sections together helps ensuring that structure dimensions and elevations are specified correctly. A typical source of model instability is a structure which is larger or lower than the neighbouring cross section.

For structures which can be plotted the structure dialog contains a section named "Graphic" with a "Plot..." button. Clicking this will open a separate graphical window showing two plots. The upper plot being the upstream cross section and the structure, and the lower plot being the downstream cross section and the structure. Initially, the centre of the structure is horizontally aligned to marker 2 of the cross section.

Clicking the right-mouse button in the graphical view the user will be given the following options in addition to the regular zoom and pan options:

- Settings...:
  This will open a separate dialog where the graphical properties of cross section, structure etc. can be changed.

- Use same axis scaling U/S and D/S:
  Selecting this the two plots will be shown with the same scaling of vertical and horizontal axis

- Show next:
  The next structure (according to the list in the tabular view) with neighbouring cross sections will be shown instead of the current.

- Show previous:
  The previous structure (according to the list in the tabular view) with neighbouring cross sections will be shown instead of the current.

- Pan structure left:
  The structure will be moved to the left in the graphical view.

- Pan structure right:
  The structure will be moved to the right in the graphical view

- Increase pan step:
  The distance by which the structure moves left/right is increased by a factor 2.

- Decrease pan step:
  The distance by which the structure moves left/right is decreased by a factor 2.

The following figure shows a typical structure plot for a weir.
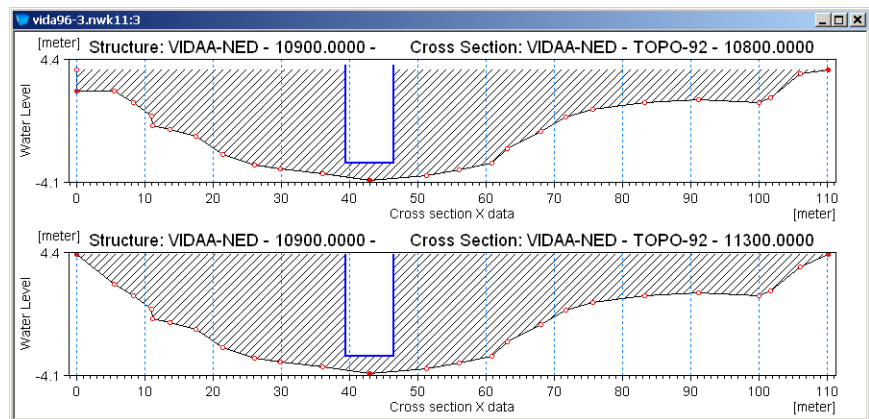


Figure 2.48    Structure Plot

## 2.3.6    Weirs

The Weirs page in MIKE 11 includes different types and formulas for overflow weirs. The weirs property page is presented in Figure 2.49.
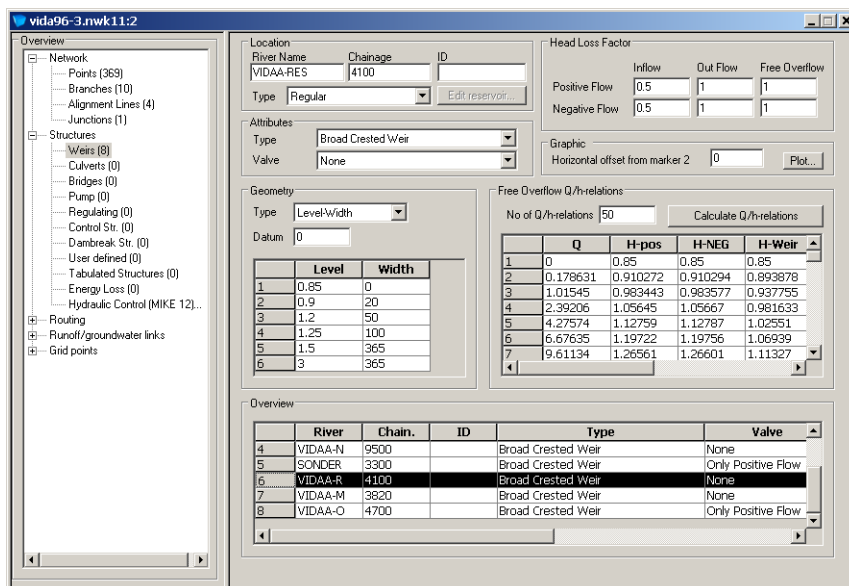
Figure 2.49    The weir property page

Weir definitions comprise the following data.

## Location

- **River Name**:
  Name of the river branch in which the weir is located.

- **Chainage**:
  Chainage at which the weir is located.

- **ID**:
  String identification of the weir. ID will be used in structure results and hence makes it easier to identify a specific structure if a setup contains many structures, or if multiple structures are defined at the same location. It is recommended always to give the weir an ID.

- **Type**:
  The location type may be Regular, Side Structure or Side Structure + Reservoir. See 2.3.3 Structure Types definition *(p. 72)* for details.

## Attributes

- **Type**:
  Type of Overflow Weir formulation:

- **Broad Crested Weir**:
  Broad Crested weir formulation is used for calculating the *Q/h* relation table on the page which contains corresponding values for water level and discharge for critical flow conditions at the crest.

- **Special Weir**:
  Special Weir is chosen in the situation where a Broad Crested weir formulation is not valid. For this weir type, the *Q/h* relationship table for free overflow must be specified manually by the user.

- **Weir Formula 1**:
  A standard weir expression reduced according to the Villemonte formula is applied. See the Reference Manual, under weir formula.

- **Weir Formula 2 (Honma)**:
  The Weir Formula 2 is the Honma weir expression.
  See the Reference Manual, Weir Formula 2 (Honma).

- **Weir Formula 3(Extended Honma)**:
  An extended version of the Honma weir formula is applied in Weir Formula 3. See the Reference Manual, Weir Formula 3(Extended Honma).

- **Valve**:

- **None**:
  No valve regulation applies.

- **Only Positive Flow**:
  Only positive flow is allowed, i.e. whenever the water level downstream is higher than upstream the flow through the structure will be zero.

- **Only Negative Flow**:
  Only negative flow is allowed, i.e. whenever the water level upstream is higher than downstream the flow through the structure will be zero.

## Head Loss Factors
The factors determining the energy loss occurring for flow through hydraulic structures (only active for broad crested weir and special weir).

## Geometry
- Geometry definitions applies only for broad crested weir and special weir types.

- **Type of Geometry**:

- **Level-Width:**
  The weir geometry is specified as a level/width table relative to the datum.

- **Cross Section DB:**
  The weir geometry is specified in the cross section editor. A cross section with a matching branch name, Topo ID and chainage must exist in the applied cross section file. The Topo ID is assumed to be the same as that specified in the Branches Property page, see Topo ID *(p. 61)*.

- **Datum**:
  Offset which is added to the level column in the level/width table.

- **Level/Width table**:
  Weir shape defined as levels and corresponding flow widths. Values in the levels column must be increasing. See also section 2.3.4 Structure Geometry definition *(p. 77)*.

## Weir formula Parameters (only weir formula 1)



Figure 2.50    The weir property page, formula 1

**Width**: Width of the flow.

**Height**: Weir height. See Figure 2.52

**Weir Coeff.**: Multiplication coefficient in the weir formula.

**Weir Exp.**: Exponential coefficient in the weir formula.

**Invert Level**: Bottom datum level. See Figure 2.52

## Weir formula 2 Parameters (only weir formula 2 (Honma))
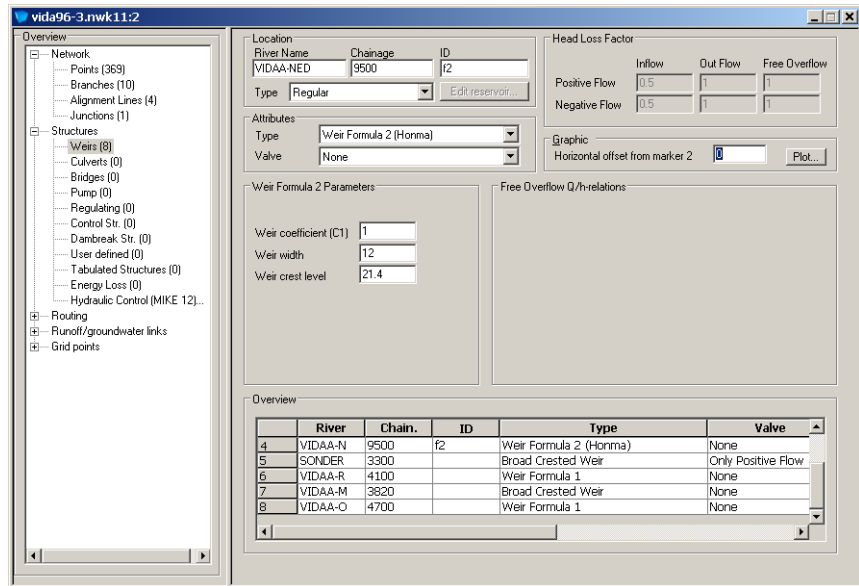


Figure 2.51    The weir property page, formula 2

**Weir coefficient (C1)**: Multiplication coefficient in the Honma weir formula.

**Weir width**: Width of the flow.

**Weir crest level**: Weir level. See Figure 2.52
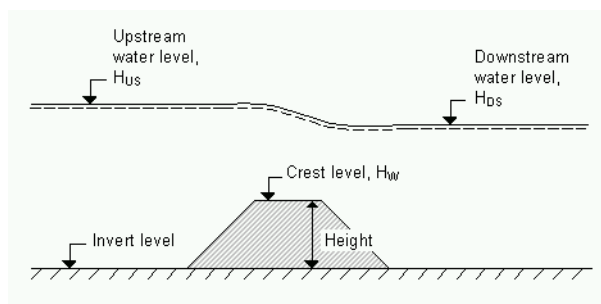


Figure 2.52    Definition sketch for Weir formula

## Weir formula 3 Parameters (only weir formula 2 (Honma))

When choosing weir formula 3 a separate dialog can be opened by clicking the "Details..." button. See the following figure:

Figure 2.53    Parameter definition dialog for Weir formula 3

Using weir formula 3 several parameters are to be specified for calculation of the flow in three regimes: perfect, imperfect and submerged overflow. MIKE 11 does not check if there is a continuous transition from one flow regime to the next. This has to be ensured by the user through proper selection of the parameters.

### Free Overflow Q/h-relations (only broad crested weir and special weir)

- **Broad crested weir:**
  **The** *Q/h* relations are calculated using the Calculate button after all relevant information has been entered. The result of the calculation will appear in the table. In order to compute the *Q/h* relation, the nearest upstream and downstream cross section are used. The cross sections must be within the distance maximum dx (Maximum dx *(p. 63)*) defined for the branch in question. The *Q/h* relation can not be calculated unless the cross sections are defined. It is also necessary that the Simulation File is open in order to load the cross section data from a cross section file.

  Table content are;
  - Q : Structure Discharge
  - H-Pos: Water Level upstream in case of Positive flow
  - H-Neg: Water Level downstream in case of Negative flood
  - H-Weir: Water Level at structure
  - Width: Structure width for actual structure level
  - Area: Structure Area for actual structure level

- **Special weir:**
  Unlike for the broad crested weir, where *Q/h* relations are calculated automatically, the user must manually specify *Q/h* relations corresponding to free overflow conditions for a Special weir. These must be specified for both positive and negative flows.

> **Note** that *Q/h* relations must be re-calculated if any changes are made to the weir or the cross sections up- or downstream have been altered. Further, since a weir in MIKE 11 is defined as a structure causing a contraction loss and subsequently an expansion loss, some constraints are placed on the geometry of a broad crested weir. The geometry of the weir must be such that the cross sectional area at the weir is less than the cross sectional area at both the upstream and the downstream cross section for all water levels!

## 2.3.7    Culverts
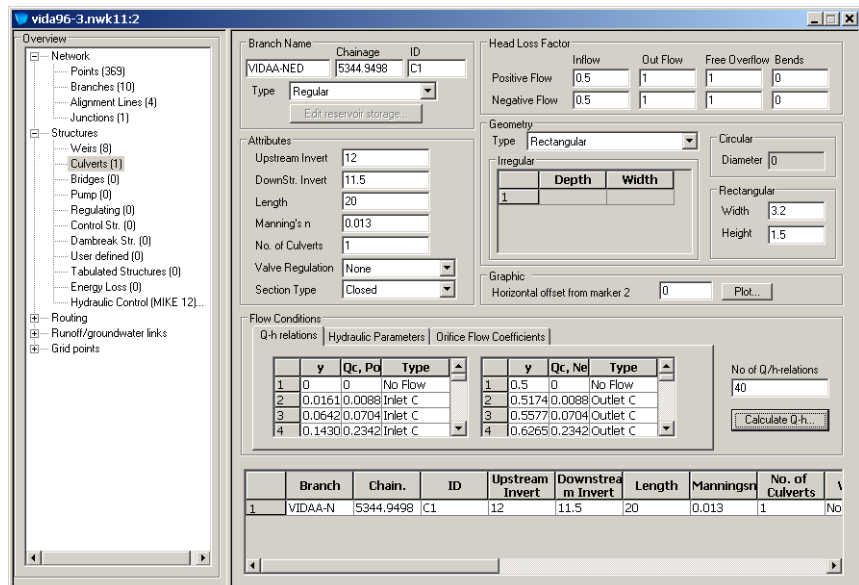


Figure 2.54    Culvert editor page

### Branch name
**River Name**:
Name of the river branch in which the weir is located.

**Chainage**:
Chainage at which the weir is located.

**ID**:
String identification of the culvert. It is used to identify the culvert if there are multiple structures at the same location. It is recommended always to give the culvert an ID.

**Type**:
The location type may be Regular, Side Structure or Side Structure + Reservoir. See 2.3.3 Structure Types definition *(p. 72)* for details.

### Attributes

**Upstream Invert**:
Invert level upstream of the culvert.

**Downstr. Invert**:
Invert level downstream of the culvert.

**Length**:
Length of the culvert.

**Manning's n**:
Manning's bed resistance number along the culvert (for calculation of the Friction loss present in a structure with a certain length).

**No. of Culverts:**
Number of culvert cells.

**Valve Regulation**:

● **None:**
No valve regulation applies.

● **Only Positive Flow:**
Only positive flow is allowed, i.e. whenever the water level downstream is higher than upstream the flow through the structure will be zero.

● **Only Negative Flow:**
Only negative flow is allowed, i.e. whenever the water level upstream is higher than downstream the flow through the structure will be zero.

**Section Type**: Closed or Open. An open culvert functions as a channel.

### Head Loss Factors

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

### Geometry

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

● **Rectangular**:
The width and height specify the geometry.

● **Circular**:
The geometry is specified by the diameter.

● **Irregular Level-Width Table**:
The geometry is specified using a level/width table. Values in the level column must be increasing. Internal levels are converted to depths through substraction of lowest level.

- **Irregular Depth-Width Table**:
  The geometry is specified using a depth/width table. Values in the depth column must be increasing.

- **Section DB**:
  The geometry is specified by a cross section. A cross section with the same branch name, Topo ID and chainage must exist in the cross section file. The Topo ID is assumed to be the same as specified in Topo ID *(p. 61)*.

For further details on structure geometry definition, see section 2.3.4 Structure Geometry definition *(p. 77)*.

## Flow Conditions

Once the above parameters and the desired number of Q/h relations have been filled in the button Calculate Q/h relations can be pressed. The result of the calculation will appear in the table. If any of the parameters defining the culvert is changed the user should remember to re-calculate the Q/h relations. In order to compute the *Q/h* relation, the nearest upstream and downstream cross section are used. The cross sections must be within the distance maximum dx (Maximum dx *(p. 63)*) defined for the branch in question. The *Q/h* relation can not be calculated unless the cross sections are defined. It is also necessary that the Simulation File is open in order to load the cross section data from a cross section file.

The *Q/h* relations are given as *Q/y* relations (where *y* is depth above invert).

The *Q/y* relations table also shows the type of flow occurring for the actual levels. Please note that the flow-types are only indicators derived from MIKE 11. It will have no effect on the calculation results to change the flow types. The possible types are:

- **No Flow**: No flow occurs at the first level ($y = 0$) and when the valve regulation flag prohibits flow in one direction

- **Inlet C**: The flow at the inlet is critical

- **Outlet C**: The flow at the outlet is critical. A backwater curve using a fine resolution is calculated to relate the discharge to the upstream water level in the river

- **Orifice**: The flow at the culvert inlet has an orifice type formation. The discharge is based on the orifice coefficients shown in the menu. These coefficients can be edited, added or deleted, if required. The *Q/h* relations will be re-calculated after editing the coefficients

- **Full Cul**:
  The culvert is fully wet with a free discharge at the outlet.

**Note** that *Q/h* relations must be recalculated if any changes are made to the culverts defining parameters or if the cross sections up- or downstream have been altered. Further note that since a culvert in MIKE 11 is defined as a structure causing a contraction loss, a friction loss (bend loss) and subsequently an expansion loss some constrains are placed on the geometry of a culvert.

The geometry of the culvert must be such that the cross sectional area at the inflow is less than the cross sectional area upstream of the culvert for all water levels. Similarly, the cross sectional area at the outflow end must be less than the cross sectional area immediately downstream of the culvert.

## 2.3.8    Pumps



Figure 2.55    The pump property page

### Location

- **Branch Name**: Name of the river branch in which the pump is located.

- **Chainage**: Chainage at which the pump is located.

- **ID**: String identification of the pump. Used for identification of the pump in case of multiple structures at the same location. Specification of pump ID is recommended.

- **Type**:
  The location type may be Regular, Side Structure or Side Structure + Reservoir. See 2.3.3 Structure Types definition *(p. 72)* for details.

## Control Parameters

- **Start Level:** Water level at the inflow that activates the pump. Note that for pumps with internal outlet the inflow is situated at the previous h-point (previous with regard to chainage) in case of positive discharge and at the next h-point with regard to chainage in case of negative discharge. The sign of the discharge follows from the specifications made under Pump Data.

- **Stop Level:** Water level at which the pump starts closing down.

- **Start-up Period**: Period for changing pump discharge from zero to full. The pump discharge is changed linearly in time.

- **Close Down Period**: Period for changing pump discharge from full to zero. The pump discharge is changed linearly in time.

## Pump Data

- **Specification Type**:

- **Fixed Discharge**:
  Pump rate independent of the local water head expect for the start/stop control.

- **Tabulated Characteristic**:
  Pump rate controlled by specified characteristic (Q-dH-curve) and the water level difference between upstream water level and outlet level/downstream water level.

- **Discharge**:
  Pump rate when applying "Fixed Discharge".

- **Outlet Level**:
  The outlet level is used when defining tabulated (dh-Q) characteristics for the pump. It then represents the outlet level on the downstream side of the pump.

- **Q-dH-curve**:
  Q-dH-characteristic of the pump. The discharge is determined through interpolated look up in the table specified. The dH used for the look-up is given as the difference between Downstream and Upstream water level in case of submerged outlet and as the difference between upstream level and outlet level in case of free outlet. The shift between the two is fully dynamic allowing an outlet to change from being free to submerged and vice versa.
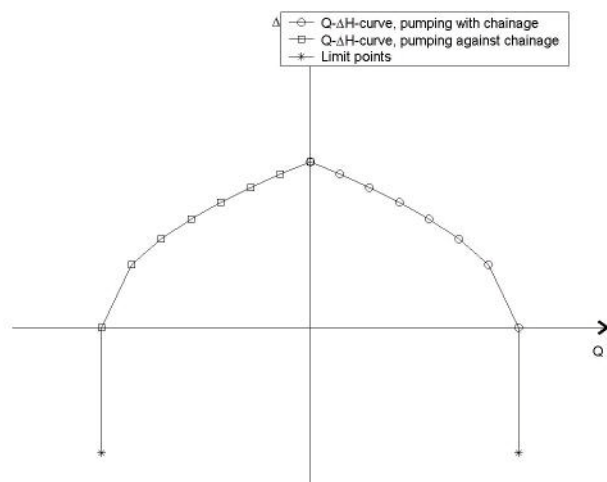
Figure 2.56    Q-dH curves for pumps

**Note** that MIKE 11 does not allow extrapolation. It is therefore recommended to add limit points to the Q-dH -curve. See figure Figure 2.56.

**Note** also that a positive Q always is interpreted as the pump pumping from upstream to downstream (i.e. from upstream lying area to a lower lying area) disregarding any chainage conventions. Typically, this would be from smaller chainage to higher chainage. If a negative flow direction is selected for the branch containing the structure, positive discharge is still interpreted as pumping from upstream to downstream areas. However, in this situation that would mean pumping from higher chainages to lower chainages.

## 2.3.9    Bridges

MIKE 11 offers a number of approaches when modelling flow through bridges. The approach to choose should be based on the assumptions for the different methods and the requirements of the modelling.

The bridge modelling approaches can be divided into pure free flow methods and methods which may be combined with submergence/overflow methods. The pure free flow methods can be further sub-divided into methods for piers and methods for arches.

The methods specially designed for piers are

- Bridge piers (Nagler): An orifice type of flow description with the effect of the piers taken into account through an adjustment factor.

- Bridge Piers (Yarnell): An equation derived from experiments for normal flow conditions in the sub critical flow range. Again the effect of piers is handled through the use of adjustment factors.

The free flow arch methods available are

- Arch bridges (Biery and Delleur): An orifice type of equation is used to describe the discharge through the bridge. The equation is derived under the assumption of a rectangular channel and is based on a single span arch opening. Multiple arch openings are handled by a simple multiplication factor.

- Arch bridges (Hydraulic research method, HR): The HR method is based on laboratory experiments of both single and multi spanned arch bridges in rectangular channels. The method uses tables describing the relation between the blockage ratio, the downstream Froude number and the upstream water level.

The methods that can be combined with both submergence and overflow methods are the following

- Energy Equation: A standard step method where a backwater surface profile determination is used to calculate the discharge through the bridge. The method takes the contraction and expansion loss for bridges of arbitrary shape into account. The method assumes sub-critical flow and may default to critical flow for steep water surface gradients.

- Federal Highway Administration (FHWA) WSPRO method: The FHWA WSPRO method is based on the solution of the energy equation. Contraction loss is taken into account through the calculation of an effective flow length. Expansion losses are determined through the use of numerous experimentally based tables. The method takes the effect of eccentricity, skewness, wingwalls, embankment slope etc. into account through the use of these tables.

- US Bureau of Public Roads (USBPR) method: The USBPR method estimates free-surface flow assuming normal depth conditions. the method is based on experiments and takes the effect of eccentricity, skewness and piers into account.

The submergence methods available are

- Pressure Flow using the Federal Highway Administration method: Two orifice equation descriptions are used. One for situations when the orifice is submerged downstream and a modified equation for situations when only the upstream part of the orifice is submerged.

- MIKE 11 culvert: A standard MIKE 11 culvert description may be chosen for submergence flow. The culvert to be used is specified by the user. The culvert is only active if submergence occurs.

- Energy equation: The flow under the bridge is determined through a standard backwater step method. The flow is assumed to be in the sub-critical range and thus the method may default to critical flow. Both con-traction and expansion loss is taken into account.

Overflow methods available are:

- Energy Equation: The flow over the bridge is determined through a standard backwater step method. The flow is assumed to be in the sub-critical range and thus the method may default to critical flow. Both con-traction and expansion loss is taken into account.

- Road overflow using the Federal Highway Administration method: The overflow is modelled using a weir equation taking tail water submergence into account through the use of a submergence coefficient. The method may be used for both gravel and paved surfaces.

- MIKE 11 weir: A standard MIKE 11 weir description may be chosen for overflow. The weir to be used is specified by the user. The weir is only active if overflow occurs.

Finally there are two additional bridge types which are not pre-processed prior to the simulation. These bridge types form part of a separate module.

- Bridge Piers (D'Aubuisson): The discharge is determined based on a momentum equation assuming no bed slope and that the friction loss is negligible. The method can handle pure rectangular channel descriptions but also gives the option of using cross sections of arbitrary shape.

- Fully submerged bridges: The fully submerged takes the drag of the fully submerged bridge into account. Friction is neglected and the bed is assumed horizontal through the bridge.

Overflow is only available in combination with submerged flow. When the bridge structure bottom level is exceeded the bridge type solution will be ignored and replaced with a submerged solution. When the bridge structure top level is exceeded the submerged solution is combined with overflow.

For all bridges there are some common settings consisting of:

- **Name**: The river name. Used along with the chainage to identify the location of the bridge in the network. River name and branch name are identical,

- **Chainage**: The location of the bridge in the river. The river cannot be located at a location where a cross section exists.

- **Bridge ID**: String identification of the bridge. The string identification for the bridge is used in the output, but also for the preprocessing of the bridges. It is important that every bridge in the set-up has a unique non-empty Bridge ID. The latter ensures that bridges can be uniquely identified by the calculation kernel of MIKE 11.

- **Method**: Pull down menu for selecting the free flow method used in calculating the flow through the bridge. The methods are briefly described above. Please refer to the reference manual for a more thorough description of the underlying assumptions and governing equations.

- **Options**: Some of the bridge methods offer a number of additional options. These options may include e.g. piers, skewness, etc. To include an option simply activate the appropriate box.

- **Geometry and Loss factors**: Through the use of the Edit button the user can access the interface for entering geometric and loss factor parameters. The Detail button gives access to the tables used for determining loss coefficients.

- **Submergence**: (Pressure flow) Available if the Submergence checkbox is marked (See Options). The user must select one of the methods: Energy Equation, FHWA WSPRO or MIKE 11 culvert. For details on FHWA WSPRO see Submergence, FHWA WSPRO. For a description of culverts please refer to the appropriate section (2.3.7). The energy equation method is equivalent to the one used for free surface flow. Note that only some methods allow the submergence option.

- **Culvert no**: When choosing a MIKE 11 culvert details of the culvert structure are in the culvert menu (See section 2.3.7). Culvert no is chosen as the number marked in the overview box in the culvert menu.

- **Bridge level (bottom)**: The bottom level of the bridge. The geometry of the lower part of the bridge consists of the data in bridge cross sections combined with the bridge level bottom.

- **Overflow**: Note that the overflow option is only available if the submergence option is also selected. The choices are FHWA WSPRO, Energy Equation or MIKE 11 weir method. For details on FHWA WSPRO see Overflow, FHWA WSPRO. If the Energy equation method is selected the user needs to supply the bridge level at the top of the bridge. This level indicates the lowest point on the deck. The data supplied for the bridge geometry also constitutes part of the geometry of the overflow part.

- **Weir no**: If a MIKE 11 weir is chosen the appropriate weir from the weir menu must be selected (See section 2.3.6). Weir no is chosen as the number marked in the overview box in the weir menu.
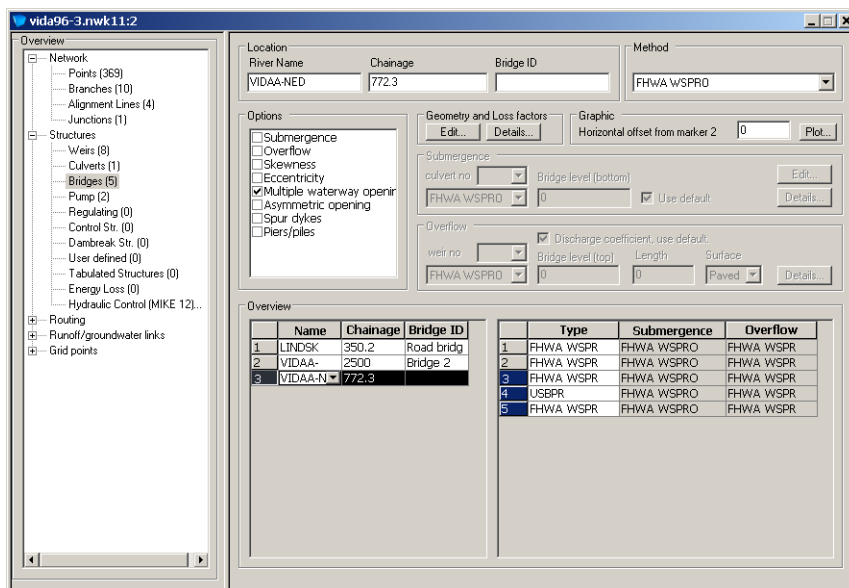
Figure 2.57　Overview, Branch2 has three bridge openings - 2, 3 and 4. Marked in the right part of the overview window

The overview is split in two. The left hand side gives an overview of the physical bridges in the set-up and the right the different openings. If a bridge consists of one opening there will be one entry on the left and one on the right for that particular bridge. If a bridge has multiple openings there will only be one entry on the left for that particular bridge and a number of entries on the right corresponding to the number of openings in the bridge.

- To add a bridge to a set-up simply place the cursor in the field of the last bridge and hit the TAB key until a new line appears. Similarly for additional openings in the same bridge. As an alternative the insert key on the keyboard may be used. To summarize the layout:

- **Overview**: Left part shows River name, Chainage and Bridge ID. Right part show methods for the bridge openings.

- **Multiple waterway openings**: If working with multiple waterway openings all multiple waterway openings are marked when the bridge is activated. (See Figure 2.57). In order to add additional openings, mark a row in the right part of the overview window and press insert on the keyboard.

### Working with Loss factor tables:

Some of the bridge methods require values found in tables. These tables can either be used based on default values taken from the literature or may be user defined/edited. In general it is recommended to use the default values.

The loss / adjustment factor tables are viewed by pressing the Details button. The default loss factor tables are generated by pressing the Edit button.

When having "default" unmarked for a loss factor, changes in the loss factor table will be saved. If "default" is marked changes will not be saved after pressing edit.

In the loss factor tables the user can create more columns and rows. Placing the cursor in the last column (right end) and pressing the arrow button on the keyboard will create a new column. Pressing the tab button on the keyboard when having the cursor in last bottom cell creates an additional line.

## Federal Highway Administration (FHWA) WSPRO & US bureau of Public Roads (USPBR) Bridge

The FHWA WSPRO, Energy Equation and the USPBR methods may be used to describe free surface flow through a bridge opening. The methods use the up and down stream cross sections located in the cross section editor, as well as two additional cross sections to be defined within the network editor.

The location of the cross sections outside the bridge should be so that any potential contraction or expansion loss is taken into account. In other words the optimal location is where the stream lines are parallel prior to a contraction and post a possible expansion. As a rule of thumb the distance between the bridge and the cross sections should be of the order one opening width (see Figure 2.58).
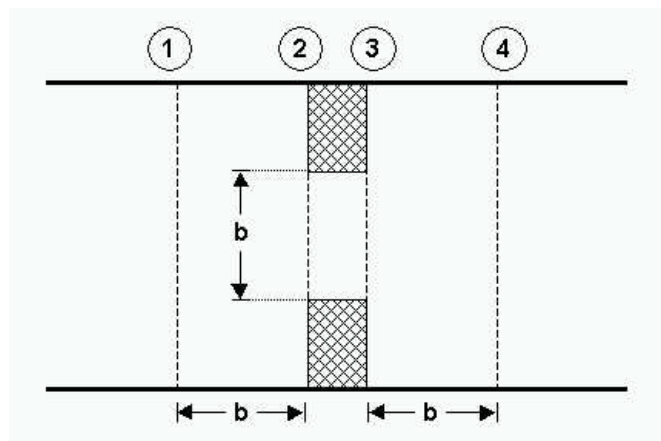


Figure 2.58    Location of up- and downstream cross section. 1: Upstream river cross section. Defined in the cross section editor. 2: Upstream bridge cross section. Defined in the network editor, bridge geometry. 3: Downstream bridge cross section. Defined in the network editor, bridge geometry. 4: Downstream river cross section. Defined in the cross section editor

Available options for FHWA WSPRO Bridge:

● Submergence

● Overflow

- Skewness, Used when the embankments are not perpendicular to the approaching flow.
- Eccentricity, Used when the bridge opening is eccentrically located in the river.
- Multiple waterway opening
- Asymmetric opening, Used for individual definition of left and right abutments.
- Spur dykes
- Piers/piles

Available options for USBPR Bridge:

- Submergence
- Overflow
- Skewness, Used when the embankments are not perpendicular to the approaching flow.
- Eccentricity, Used when the bridge opening is eccentrically located in the river.
- Multiple waterway opening
- Piers/piles

Geometry and loss factors are viewed by pressing the Edit button under Geometry and Loss factors.

## Geometry, Waterway opening:



Figure 2.59    Geometry property page

- **Opening type**: (see definition sketch Figure 2.60 - Figure 2.63). Only used for the FHWA WSPRO method.

- **Embankment slope**: Only for FHWA WSPRO, opening type II, III, and IV. Example, insert 2 for a 1:2 slope.

- Waterway length L

- At level z: Only for FHWA WSPRO, opening type II, III, and IV. Enter the level for witch the Waterway length is measured.
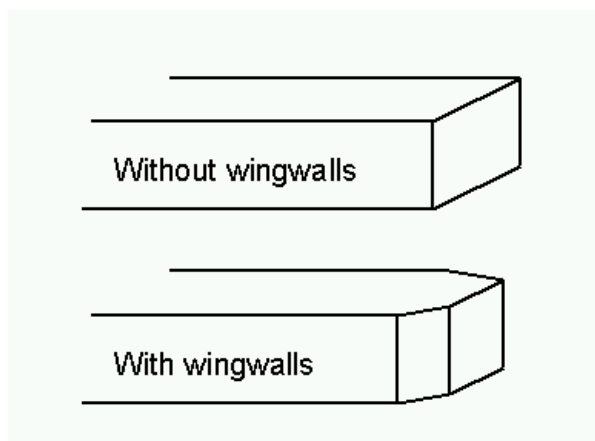
Figure 2.60     Definition sketch of type I opening, vertical embankments and vertical abutments, with or without wingwalls (after Matthai)
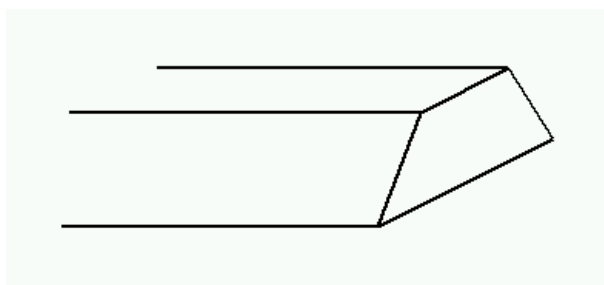


Figure 2.61     Definition sketch of type II opening, sloping embankments without wing-walls (after Matthai)
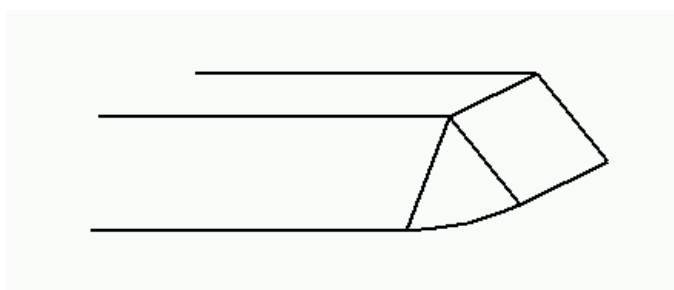


Figure 2.62     Definition sketch of type III opening, sloping embankments and sloping abutments (spillthrough) (after Matthai)
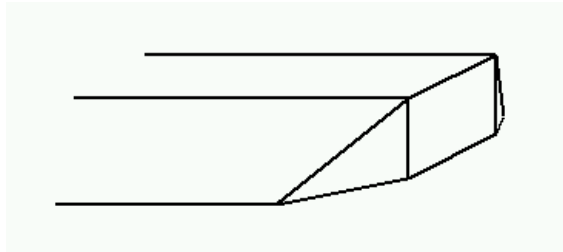
Figure 2.63    Definition sketch of type IV opening, sloping embankments and vertical abutments with wingwalls (after Matthai)

Geometry, Cross-section table:

- **Slope**: If the slope check box is marked then only the upstream bridge cross section must be defined. The downstream cross section is generated by copying the upstream cross section and using the slope defined in the slope edit box. Upstream bridge cross section correspond to section 2 and downstream bridge cross section correspond to section 3. (See Figure 2.58).

- **Datum**: The water level datum is added to the Z values in the Cross section table.

- **X**: Horizontal values for the cross section. Note that the x-values are evaluated with the up and downstream cross section. As a result it is important that the four cross sections (See Figure 2.58) are placed correct in respect to the x-values.

- **Z**: Vertical level of the cross section point.

- **Resistance**: Additional relative resistance in the cross section point. 1 is resistance corresponding to the manning number.

- **Marker**: Define cross section embankments and abutments (See Figure 2.64).

- Cross section definitions: Cross sections 2 and 3 must be defined as illustrated by the blue arrows in Figure 2.64 - i.e. along the floodplain then to the underside (soffit) of the bridge then down the abutment to the channel and similar to the other side. Markers 1 and 3 correspond to floodplain limits and 4 and 5 are bottom of left and right abutments respectively
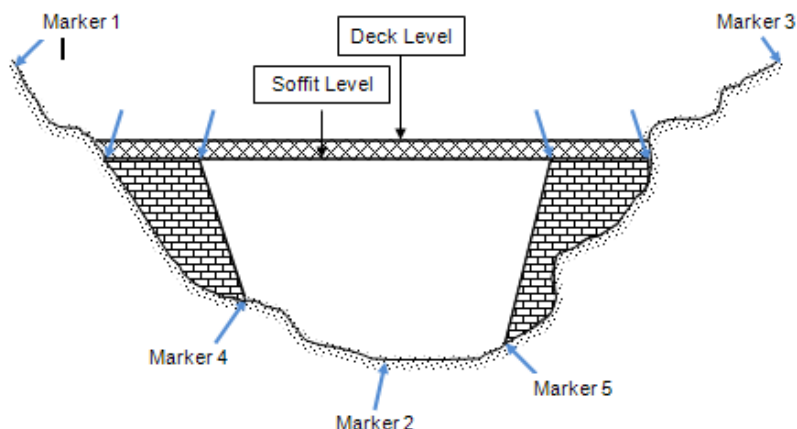
Figure 2.64    Definition of the bridge cross section markers

Geometry, Multiple waterway opening:

Geometry and loss factors are defined for each opening when working with multiple waterway openings (see Figure 2.57). The position of each opening and the corresponding stagnation points are defined from the stagnation point value (if not default) and from the horizontal values in the bridge cross sections. (See Figure 2.65)

- **Use default left stagnation points**: When the checkbox is marked the stagnation point is set by MIKE 11. When the checkbox is unmarked the user must define the left up and down stream stagnation points in the edit boxes.

- **Left stagnation point upstream**: Horizontal value (X value) for the left stagnation point in the upstream river cross section defined in the cross section editor. Section 1 in Figure 2.58. The stagnation point to the right is found from the neighbouring opening. The left stagnation point refers to a lower value of X than the right stagnation point.

- **Left stagnation point downstream**: Horizontal value (X value) for the left stagnation point in the downstream river cross section defined in the cross section editor. Section 4 in Figure 2.58. The stagnation point to the right is found from the neighbouring opening. The left stagnation point refers to a lower value of X than the right stagnation point.
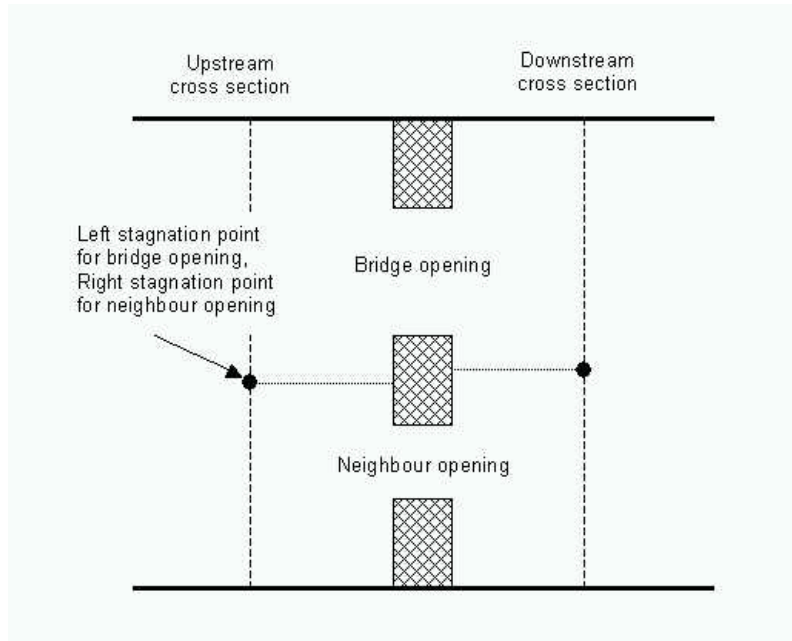
Figure 2.65     Multiple openings and stagnation points

Loss factor:



Figure 2.66     Loss factor property page

- **Entrance rounding**: Loss factor for FHWA WSPRO, opening type I. When 'use default' is ticked a default loss factor table will be generated from the information entered under entrance rounding. Corner type: Enter the radius, r for the corner rounding. Wingwall type: enter the width, W and angle of the wingwall.
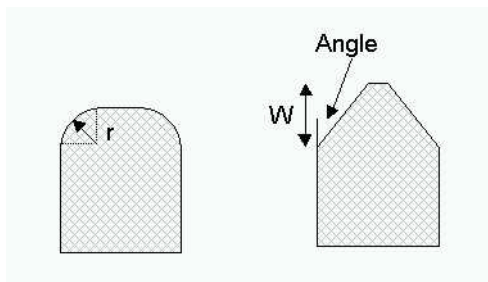


- **Spur dykes**: FHWA WSPRO, Loss factor when spur dykes is marked in options. When use default is selected, a default loss factor table will be generated. For Straight spur dykes the user must enter the length and offset from the bridge opening. For Elliptical spur dykes the user must enter the length and angle.

- **Wingwall**: Loss factor for FHWA WSPRO, opening type IV. When use default is selected, a default loss factor table will be generated from the entered wingwall.angle.

- **Froude number**: Loss factor for FHWA WSPRO, opening type I. When use default is selected, a default loss factor table will be generated.

- **Base Coefficient**: Loss factor for FHWA WSPRO, opening type I, II, III and IV and USBPR. When use default is selected, a default loss factor table will be generated. For the USBPR method an opening type is chosen.

- **Abutment**: Loss factor for FHWA WSPRO, opening type III. When use default is selected, a default loss factor table will be generated.

- **Average Depth**: Loss factor for FHWA WSPRO, opening type II. When use default is selected, a default loss factor table will be generated.

- **Velocity distribution coefficient**: Loss factor for the USBPR method. When use default is selected, a default loss factor table will be generated.

- **Piers / Piles**: Loss factor when 'piers / piles' is marked in options. When use default is selected, a default loss factor table will be generated. Choose Type (piers or piles) and enter the proportion of the waterway blocked by piers/piles. For the USBPR method the user must choose a piers type for generating a default loss factor table.

- **Eccentricity**: Loss factor when eccentricity is marked in options. When use default is selected, a default loss factor table will be generated.

- **Skewness**: When skewness is marked in options an angle for skewness is entered in the edit box, Skewness angle,.

- **Resistance**: Choose Manning's M or n as the type for resistance.

- **Resistance value**: The value for the relative resistance on the bridge structure between markers 1 and 4 and between 5 and 3 (see Figure 2.64). Between markers 4 and 5 the bed resistance given in the HD editor will be used.

- **Opening/Contraction ratio**: Choose channel contraction ration $m$ or bridge opening ratio $M$ as parameter in the loss factor tables.

Table 2.2        Loss factor tables for FHWA WSPRO

| Table | | Opening Type | Function of: | |
|---|---|---|---|---|
| Base coefficient | $C'$ | I | $m$ or $M$ | |
| Base coefficient | $C'$ | II, III and IV | $m$ or $M$ | $L/b$ |
| Froude number | $k_F$ | I | $F_3$ | |
| Entrance | $k_{r/w}$ | I | $m$ or $M$ | |
| Average depth | $k_y$ | II | $m$ or $M$ | $(Y_a + Y_b)/(2b)$ |
| Abutment | $k_x$ | III | $x/b$ | $L/b$ |
| Wingwall | $k_\theta$ | IV | $m$ or $M$ | |
| Eccentricity | $k_e$ | I, II, III and IV | $e$ | |
| Piers | $k_j$ | I, II, III and IV | $m$ or $M$ | |
| Piles #1 | $k_{j=0,1}$ | I | $m$ or $M$ | |
| Piles #1 | $k_{j=0,1}$ | II, III and IV | $m$ or $M$ | $L/b$ |
| Piles #2 | $k_j$ | I, II, III and IV | $k_{j=0,1}$ | |
| Spur dike | $k_d$, $k_{a/b}$ | I, II and IV | $m$ or $M$ | |
| Spur dike #1 | $k_d$ | III | $m$ or $M$ | $L_d/b$ |
| Spur dike #2 | $k_a$ | III, Elliptical | $m$ or $M$ | $L_d/b$ |
| Spur dike #2 | $k_b$ | III, Straight | $m$ or $M$ | |

In the Loss Factor menu the user can choose to use $m$ or $M$ in the tables.

Where:

    *m*    Channel contraction ratio.

    *M*    Bridge opening ratio.

    *L*    Bridge waterway flow length.

    *b*    Bridge opening length.

    $F_3$    Froude number in downstream bridge section.

    $(Y_a + Y_b)/(2b)$  Average water level in bridge section.

    *x*    Unwetted abutment length.

    *e*    Eccentricity.

    *j*    Portion of waterway blocked by piers/piles.

    $L_d$    Spur dike length

Table 2.3    Loss factor tables for USBPR

| Table | | Function of: | |
|---|---|---|---|
| Base coefficient | $k_b^*$ | $M$ or $m$ | |
| Velocity distribution coefficient | $\alpha_2$ | $M$ or $m$ | $\alpha_1$ |
| Eccentricity | $e$ | $M$ or $m$ | $\Delta k_e^*$ |
| Skewness | $\Delta k_\phi^*$ | $M$ or $m$ | |
| Piers | $\Delta k_p^*$ | $M$ or $m$ | |

In the Loss Factor menu the user can choose to use *m* or *M* in the tables.

Where:

    *M*    Bridge opening ratio.

    *m*    Channel contraction ratio.

    *e*    Degree of eccentricity.

    $\alpha_1$    Velocity distribution coefficient in upstream cross-section.

## Fully Submerged Bridge
Press the Edit button under Geometry and Loss factors.

The details of the bridge geometry and location are inserted in the appropriate boxes:

- **Channel width**: The user specified channel width. If a positive value is implemented the water level increment calculation are based on a rectangular channel analysis. If a negative value is implemented a more general momentum equation is solved utilising the cross sections upstream and downstream of the bridge.

- **Section area of submerged bridge**: The cross sectional area of the submerged bridge. Note that the bridge formula assumes that the bridge is fully submerged

- **Drag coefficient**: The drag coefficient of the bridge.

Figure 2.67 shows an example where a submerged bridge is inserted at the chainage 500 m in the river 'RIVER 1'. The channel width is specified as 10 m, the section area of the bridge is set equal to 5 m and the drag coefficient is set to 1.6.
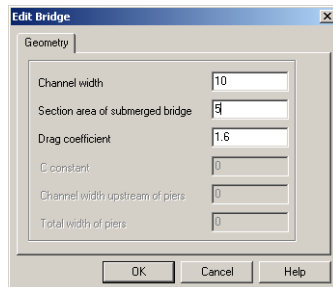


Figure 2.67    Submerged bridge geometry property page

**Note**: If the Froude number downstream of the fully submerged bridge is greater than the criteria (default 0.6) the effect of the bridge is ignored. The criteria value may be changed in the MIKE 11.ini file by setting the variable:

BRIDGE_FROUDE_CRITERIA.

## Arch Bridge (Biery and Delleur) & (Hydraulic Research (HR))
The Arch Bridge methods describe free surface flow through one or more arch bridge openings.

Since the equations used for arch bridges are only valid for free flow, submergence and overflow cannot be selected. Geometry and loss factors are viewed by pressing the Edit button under Geometry and Loss factors.
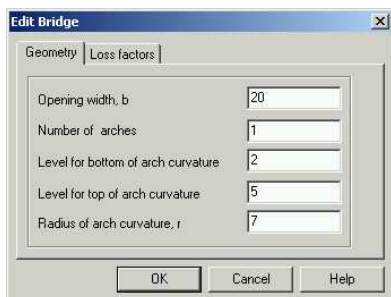
Figure 2.68    Arch Bridge Geometry property page



Figure 2.69    Arch Bridge Loss factor property page

- **Opening width, b**: Opening width at the Arch spring line.

- **Number of arches**: The number of arch openings in the bridge.

- **Level for bottom of arch curvature**: Vertical level for Arch spring line.

- **Level for top of arch curvature**: Vertical level upper most point in the arch.

- **Radius of arch curvature, r**.

- C**oefficient of discharge, Use default**: When use default is selected, a default loss factor table will be generated.

- **Discharge coefficient**: Choose channel contraction ration $m$ or bridge opening ratio $M$ as parameter in the loss factor tables.
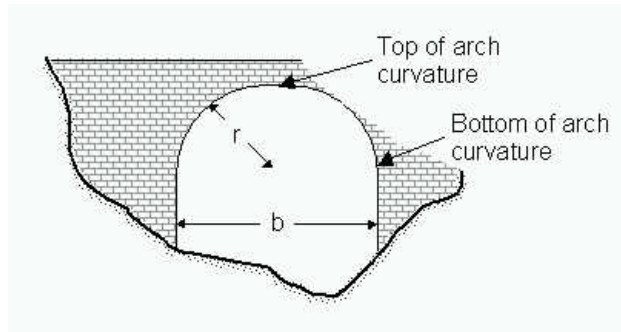
Figure 2.70    Hydraulic variables for Arch Bridge

**Note** that the Invert level is not specified in the geometrical definition of the Arch Bridge. Arch Bridge flow calculation depends on the water levels and depths at the up- and downstream side of the bridge, but this is calculated from the cross sections up- and downstream respectively and hence is not necessary to define in the parameters page.

Loss factor tables for arch bridges:

Biery and Delleur: Coefficient of discharge, $C_D$ is a function of $m$ or $M$ and Froude number $F$.

HR: Backwater ratio $H_1 / Y_4$ is a function of Blockage ratio $j$ and Froude number $F$.

### Bridge piers (D'Aubuisson's formula)
Press the Edit button under Geometry and Loss factors.



The details of the bridge geometry and location are inserted in the appropriate boxes:

- **C constant**: User specified constant (< 1).

- **Channel width upstream of piers**: If the width is positive the water level increment due to the bridge is calculated on the basis of a rectangular channel analysis. If a negative value is given the calculation is based on the cross sections upstream and downstream of the bridge.

- **Total width of piers**.

Figure 2.71 shows an example with bridge piers inserted at the chainage 500 m in the river 'RIVER 1'. The bridge piers have been given the topological identification tag 'Bridge 1'. The geometry dependent non-dimensional con-

stant has been given the value 0.8, the upstream width is specified as 10 m and the total width of the piers is set to 3.5 m.
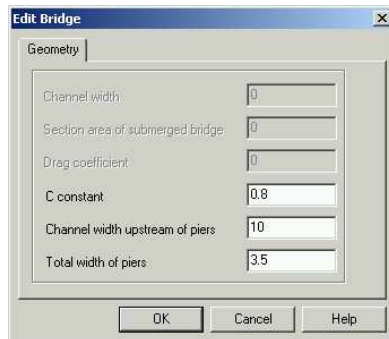


Figure 2.71    D'Aubuisson Bridge piers geometry property page

**Note**: If the Froude number downstream of the piers is greater than the criteria (default 0.6) the effect of bridge piers using D'Aubuisson's formula is ignored. The criteria value may be changed in the MIKE11.ini file by setting the variable:

BRIDGE_FROUDE_CRITERIA.

### Bridge piers (Nagler) & (Yarnell)
The Nagler and Yarnell methods describes free surface flow trough a bridge opening with piers.

Available options for Nagler and Yarnell:

- Submergence
- Overflow

Geometry and loss factors are viewed by pressing the Edit button under Geometry and Loss factors.
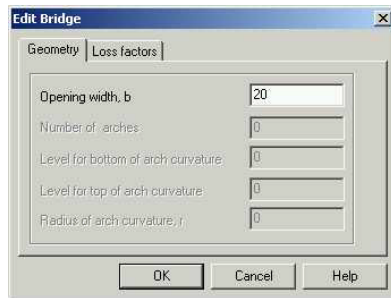
Figure 2.72     Nagler and Yarnell Bridge piers geometry property page
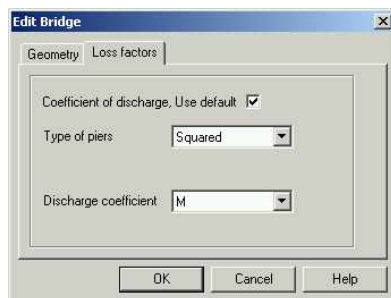


Figure 2.73     Nagler and Yarnell Bridge piers Loss factor property page

- **Opening width, b**: The total opening width between the piers.

- **Coefficient of discharge, Use default**: When, use default is selected, a default loss factor table will be generated.

- **Type of piers**: When, use default, marked choose Type of piers.

- **Opening/Contraction ratio**: Choose channel contraction ration $m$ or bridge opening ratio $M$ as parameter in the loss factor tables.

Loss factor tables for piers bridges:

Nagler: Coefficient of discharge $k_n$ and adjustment factors $\theta$ and $\beta$ are functions of $m$ or $M$.

Yarnell: Pier coefficient K is a function of $m$ or $M$.

## Submergence, FHWA WSPRO
The method describes pressure flow through a submerged bridge and is used in combination with one of the methods describing free-surface flow.

Submergence is available if the Submergence check box is marked (see Options) and FHWA WSPRO is selected in the Submergence box.



Figure 2.74    Submergence, FHWA WSPRO, property page.

- **Bridge level (bottom)**: Vertical level of the bottom of the girders.

- **Use default**: When use default is selected, a default loss factor table will be generated.

- **Details**: Loss factor tables are viewed by pressing the Details button. The loss factor table is only of interest if orifice flow is set to ON in the MIKE11.ini file. Orifice flow is in general not recommended due to the abrupt change of flow regime potentially causing transients in set-ups.

## Overflow, FHWA WSPRO

The method describes weir flow over the bridge and is used in combination with submerged flow. Overflow is available if the Overflow check box is marked (see Options) and FHWA WSPRO is selected in the Overflow box.



Figure 2.75    Overflow, FHWA WSPRO, property page.

- **Bridge level (top)**: Vertical level of the road.

- **Length**: Width of top of embankment in the direction of flow.

- **Discharge, Use default**: When use default is selected, a default loss factor table will be generated.

- **Surface**: When Use default is marked, choose a surface type for generating default loss factor tables.

- **Details**: Loss factor tables are viewed by pressing the Details button.

Loss factor tables for road overflow:

Road overflow #1: Discharge coefficient $C_f$ is a function of the ratio between maximum elevation along the top of the embankment $h_s$ and waterway length $L_R$ for $h_s/L_R > 0,15$.

Road overflow #2: Discharge coefficient $C_f$ is a function of total head available to produce weir flow $H$ for $h_s/L_R \geq 0,15$.

Road overflow #3: Submergence factor $k_t$ is a function of the ratio between minimum elevation along the top of the embankment $h_t$ and total head available to produce weir flow $H$.

## 2.3.10  Regulating
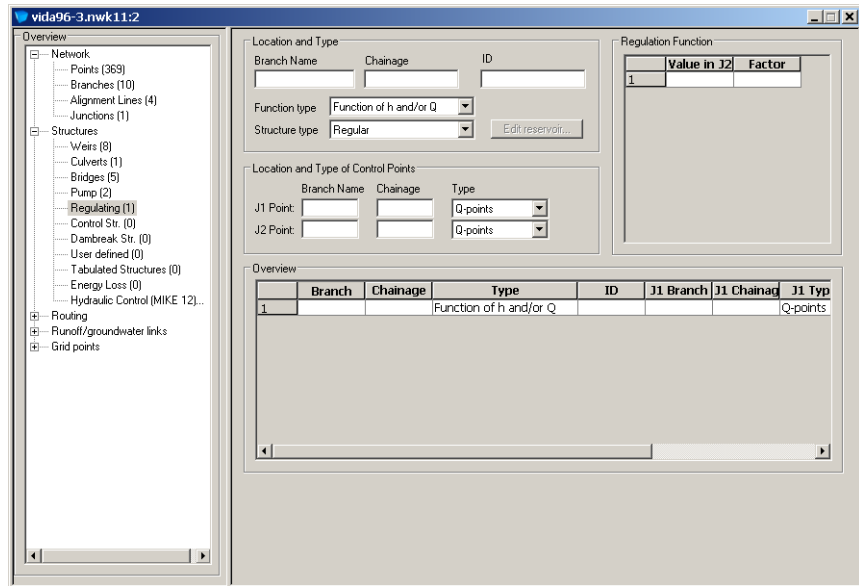


Figure 2.76    Regulating structure property page

This structure type is typically applied where discharge through a dam is to be regulated as a function of the water level, and the inflow into the reservoir.

The Regulating property page is used for defining a regulating structure such as a pump. The property page consists of a number of dialog boxes (see Figure 2.76) which functionality is described below:

### Location and Type

**River Name**: Name of the river branch in which the weir is located.

**Chainage**: Chainage at which the structure is located.

**Type**:
The location type may be Regular, Side Structure or Side Structure + Reservoir. See 2.3.3 Structure Types definition *(p. 72)* for details.

**Function of Time**: The discharge through the structure is specified as a function of time. The actual discharge time series must be specified in the Boundary Editor *(p. 215)*.

**Function of** *h* **and/or** *Q*: The discharge through the structure is defined as a function of *h* or *Q* at two locations (J2 and J1) in the river model: $Q = f(J2) \cdot J1$

**ID**: String identification of the structure. It is used to identify the structure if there are multiple structures at the same location. It is recommended always to give the structure an ID.

### Location and Type of Control Point

This section is only available when the regulation is specified as a *h/Q* function. The locations J1 and the J2 are specified in terms of branch name and chainage. In addition the user must specify J1 and J2 as being an *h*- or a *Q*-point.

### Regulation Function

This section is **only** available when the regulation is specified as a *h/Q* function. The function f(J2) is specified in the Regulation Function table as a series of factors for corresponding values of J2.

**Note** that a regulating structure may be used for implementing an internal *Q/h*-relation. This is done by choosing the J2 point as the *h*-point upstream of the structure and letting the function f(J2) describe the required *Q/h*-relation. Finally a dummy branch must be included in the set-up. This dummy branch should be constructed so that a unit discharge flows through it. The J1 point is then simply chosen as a *Q*-point in the dummy branch.

## 2.3.11  Control Str.

Control structures may be used whenever the flow through a structure is to be regulated by the operation of a movable gate which forms part of the structure.They can also be used to control the flow directly without taking the moveable gate into consideration. In this case it simulates a pump.
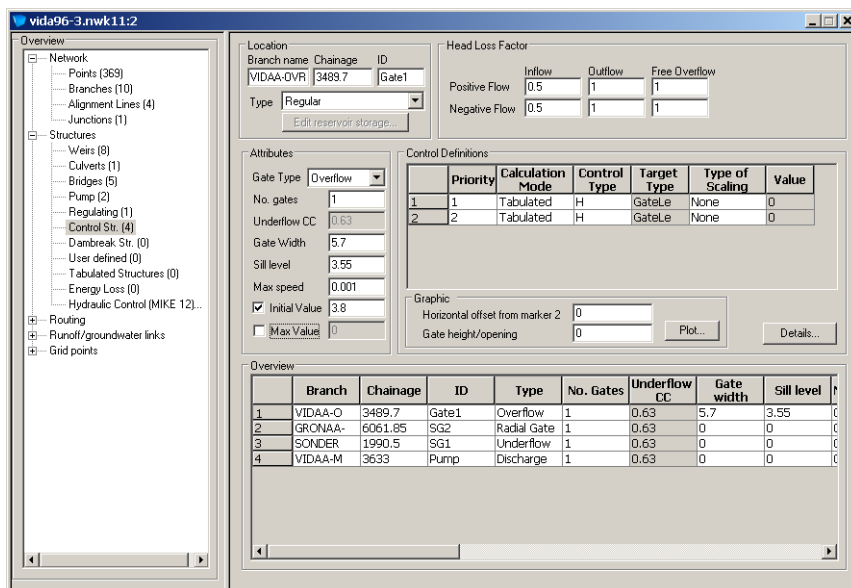
Figure 2.77    The control structure property page

## Location

**Branch name:** Name of the river branch in which the structure is located.

**Chainage:** The chainage in which the structure is located.

**ID:** String identification of the structure. It is used to identify the structure if there are multiple structures at the same location. It is recommended always to give the structure an ID.

**Type**:
The location type may be Regular, Side Structure or Side Structure + Reservoir. See 2.3.3 Structure Types definition *(p. 72)* for details.

## Attributes

**Gate Type**
* **Overflow**: This gate type corresponds to a variable crested weir.

* **Underflow**: This gate type corresponds to a vertical sluice gate.

* **Discharge**: This gate type corresponds to a pump.

* **Radial gate**: This gate type corresponds to a Tainter gate. In contrast to the other gate types a radial gate does not need any information about head loss factors. Instead a number of radial gate parameters must be entered, see Radial Gate Parameters *(p. 114)*.

- **Sluice, Formula:** This gate is physically the same as an underflow gate but instead of using the energy equation a set of flow formulaes are applied, see Parameters for gate type = Sluice, formula *(p. 116)*.

### Number of gates

The number of identical gates is entered here. This variable is used when a series of identical gates are simulated.

### Underflow CC

This is the contraction coefficient used for underflow gates only. Default value is 0.63.

### Gate width

The width of the gate. Not applicable for gates of the Discharge type.

### Sill level

The level of the sill just upstream of the gate. Not applicable for gates of the Discharge type.

### Max. Speed

This variable defines the maximum allowable change in gate level per. time. (If a discharge gate is chosen the variable defines the maximum allowable change in discharge per. time). This variable is introduced because the control strategy defining the variation of the gate level can result in very rapid changes, which can create instabilities in the computation.

### Initial Value

If the Initial Value checkbox is checked the value specified will be used as initial value

### Max Value

If the Max Value checkbox is checked the value specified will interpreted as the highest possible gate level or in case of a discharge structure the highest possible pump discharge.

## Head loss factors

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

## Radial Gate Parameters

The look of the control structure property page when a radial gate is chosen is shown in Figure 2.78.
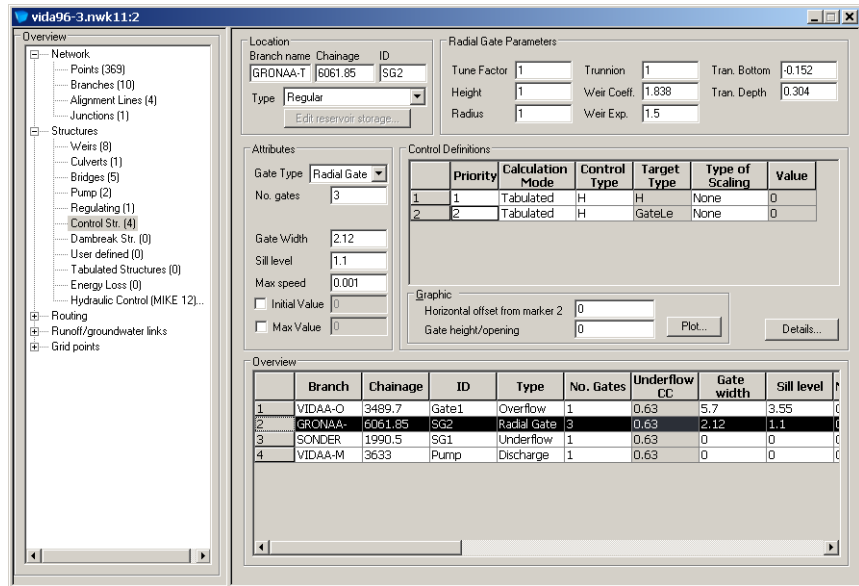
**Figure 2.78** The control structure property page when a radial gate has been selected.

In MIKE 11 radial gates are automatically divided into an underflow part and an overflow part. When specifying gate levels for a radial gate the user should specify the level for the underflow part, i.e. the level of the bottom of the gate. The gate level for the overflow part is then calculated based on geometric considerations.

Flow through a radial gate is calculated as a function of the flow regime of the gate (either; 'Free', 'Submerged' or 'Transition'). Free and Submerged flow regimes are calculated individually as described in 'Hydraulic Aspects - Radial Gates in the reference manual and flow in the transition zone is calculated as an interpolated value between the calculated Free and Submerged flows.

### Tune Factor
Discharge calibration factor. The Tune Factor is applied as a multiplication factor solely on the part of the discharge that flows below the radial gate (not applied to eventual over topping gate flow).

### Height
Height above sill level of the overflow gate crest when the gate is closed, see Figure 2.79.

### Radius
Radius of gate, see Figure 2.79.

### Trunnion
Height above sill level of the centre of gate circle, see Figure 2.79.

### Weir Coeff. and Weir Exp.

The two weir parameters are the Coefficient and Exponent respectively used in the calculation of flow above the top of the radial gate (considered as weir flow above the top of the radial gate).

### Tran. Bottom

The Tran. Bottom parameter is used to define the level where a free flow regime for the gate changes to a transition zone between free flow and submerged flow regime. Corresponds to $y_{Tran,Bottom}$ as defined in Hydraulic Aspects - Radial Gates in the reference manual.

### Tran. Depth

The Tran. Depth parameter is used to define the depth (or height) of the transition zone between free flow and submerged flow regimes and consequently is used to define the level where the flow regime changes to submerged flow. Corresponds to $y_{Tran,Depth}$ as defined in Hydraulic Aspects - Radial Gates in the reference manual.
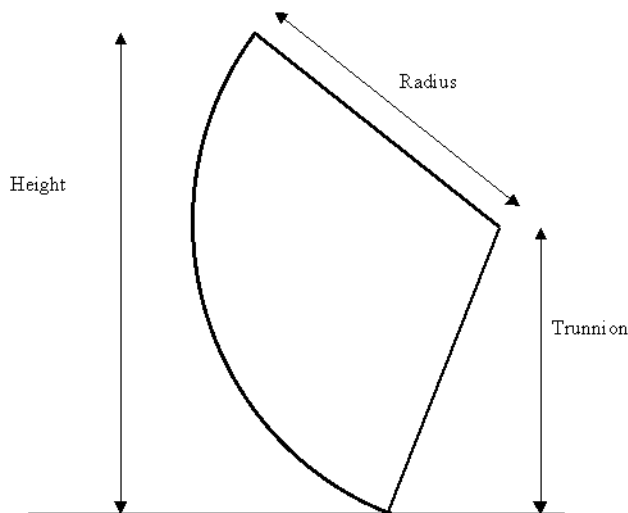


Figure 2.79    Definition of a radial gate

## Parameters for gate type = Sluice, formula

When choosing this type of gate a special set of parameters will be requested in the upper right part of the control structure dialog. See the following figure:
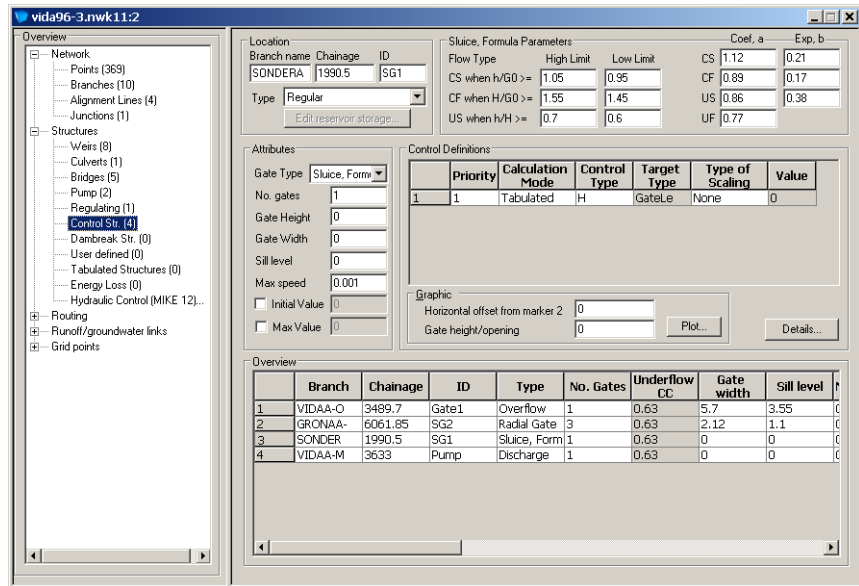
Figure 2.80    Sluice gate parameters

With this gate type the flow under the gate is divided into four flow regimes the choice of which depends on the upstream and downstream water level. The four flow regimes are:

- CS: Controlled Submerged

- CF: Controlled Free

- US: Uncontrolled Submerged

- UF: Uncontrolled Free

Additionally flow over the top of the sluice gate is taken into account when the water level up stream and/or downstream exceeds the gate level plus the gate height.

In short the high limit and low limit parameters are used to smoothen the transitions between flow regimes and coefficient a and exponent b are parameters for the flow equations. Please refer to the reference manual for further details.

## Control Definitions

The way the gate operation is calculated is determined from a control strategy. A control strategy describes how the gate level depends on the value of a Control Point. For a specific gate it is possible to choose between an arbitrary number of control strategies by using a list of 'if' statements. For each of these statements it is possible to define an arbitrary number of conditions that all must be evaluated to TRUE if the 'if'-statement is to be evaluated to

TRUE. It is hereby made possible to use different operating policies depending on the actual flow regime, time etc.

Two things are required to define a control strategy:

1. The conditions that must be fulfilled for the strategy to be executed and

2. the control strategy itself.

The control strategy itself is a relationship between an independent variable (the value of the Control Point) and a dependent variable (the value of the Target Point).

An example: Assume that the position of the gate is determined by the downstream water level. The control point is then the grid point downstream of the gate. The value of the water level in this points thus determines the value of the target point which in this example will be the gate level.

The reason for using the concept 'Target Point' and not just call it gate level is as follows:

In MIKE 11 there are different Calculation Modes and it is hereby made possible both to define control strategies that determines the value of the gate level directly and control strategies that determines the gate level indirectly. Suppose that you want to know how the gate should be operated in order to maintain a certain water level on the upstream side of the gate. The requested upstream water level has a seasonal variation due to a seasonal variation of the flood risk. To do this in MIKE 11 the control point is the time and the target point is the upstream water level. The way to get from the requested water level to a gate level is done by choosing the calculation mode 'Iterative Solution'. In this case MIKE 11 will iterate on the gate level until the upstream water level equals the requested value (or acceptable close to this value).

Five main parameters must be defined: Priority, Calculation Mode, Control Type, Target Type and Type of Scaling.

Further some additional details must be defined. We start with a description of the main parameters.

As seen in Figure 2.77 or Figure 2.78 the control definitions section consists of a table. Each line in this table represents the main parameters of an 'if'-statement.

### Priority

As mentioned under Control Definitions *(p. 117)* it is possible to make MIKE 11 choose between an arbitrary number of control strategies. These control strategies are organised using a list of 'if' statements. The control strategy belonging to the first of these statements that are evaluated to TRUE will be executed. It is thus of importance for the user to define which 'if'-statement that are evaluated first, second, third and so on. This is enabled by the priority

field. In this the user defines the priority of the 'if'-statement by writing an integer number. By default the first line in the table will have priority equal to one, the second line will have priority equal to two and so on. Note that the 'if'-statement with the lowest priority always will be evaluated to TRUE. This is because this statement is connected to the default control strategy that will be executed when all other 'if'-statements are evaluated to FALSE.

## Calculation Mode

- **Tabulated**: This is the default calculation mode, which determines the value of the gate level directly (gate discharge in case of a discharge gate).

- **PID operation**: This calculation mode corresponds to a PID operated gate. With this calculation mode the gate level is determined indirectly using the following equation:

$$u^n = \alpha_1 K \left\{ 1 + \frac{T_s}{T_i} + \frac{T_d}{T_s} \right\} \{ y^n_{ref} - y^n \} \tag{2.2}$$

$$- \alpha_2 K \left\{ 1 + 2\frac{T_d}{T_s} \right\} \{ y^{n-1}_{ref} - y^{n-1} \} - \alpha_3 K \frac{T_d}{T_s} \{ y^{n-2}_{ref} - y^{n-2} \} - u^{n-1}$$

where $u^n$ is the gate level (or discharge in case of a discharge structure) at the *nth* time step, $K$ a factor of proportionality, $T_i$ the integration time, $T_d$ the derivation time, $T_s$ the sampling period, i.e. the simulation time step, $y^n_{ref}$ is the required value of the target point at the *nth* time step, $y^n$ the actual value of the target point at the *nth* time step, $\alpha_1$, $\alpha_2$, $\alpha_3$ are weighing factors. In this way $\{y_{ref} - y\}$ represents a deviation from the desired situation. This deviation is minimized by the PID algorithm in (2.2).

The variables $K$, $T_d$, $T_i$, $\alpha_1$, $\alpha_2$ and $\alpha_3$ are entered by the user, see Iteration / PID *(p. 133)*. The rest is calculated by MIKE 11.

- **Momentum equation**: If 'Momentum equation' is chosen the flow through the structure will be calculated using the momentum equation instead of the energy equation. This corresponds to ignoring the presence of the structure. Because of this no calculation of gate level/discharge will take place. Therefore specification of control point, target point and scale factor has no importance when choosing calculation mode as 'Momentum equation'. An example where this calculation mode could be useful is in a river with an inflatable dam.

- **Iterative solution**: This calculation modes gives an indirect determination of the gate level/discharge. In Control Definitions *(p. 117)* a small example was given explaining how this calculation mode could become useful. When using this calculation mode the user must take great care when choosing the target points. This is because the iteration takes place for a fixed time step. If the target point is placed too far away from the gate the changes in gate level during the iterative procedure will not have any effect on the value of the target point The parameters that the user must enter when 'Iterative solution' is chosen as calculation mode are described in Iteration / PID *(p. 133)*.

- **Fully open**: If this calculation mode is chosen the gate is fully open. For an overflow gate this means that the gate level will equal the Sill Level specified by the user. For underflow gates and radial gates if will correspond to the Max Level defined by the user. For a pump this corresponds to the maximum pump capacity specified as Max Value.

- **Close**: If this calculation mode is chosen the gate is closed. For an overflow gate this means that the gate level will equal the Max Level specified by the user. For underflow gates and radial gates if will correspond to the Sill Level defined by the user. For a pump this corresponds to a pump discharge equal to zero.

- **Unchanged**: If this calculation mode is chosen the gate level or the pump discharge will remain unchanged.

- **Change with**: If this calculation mode is chosen the gate level or the pumps discharge will change with the amount specified in the Value column in the Control Definitions data section.

- **Set equal to**: If this calculation mode is chosen the gate level or the pumps discharge will equal the value specified in the Value column in the Control Definitions data section.

- **Tabulated 2D**: This calculation mode can be chosen for Discharge gates and used to control the discharge in the structure from time of year (dates) and a control point value.

## Control Type
Here the type of control point is chosen.

- **h**: Water level in a point.

- **dh**: Difference between water levels in two points.

- **Q**: Discharge in a point.

- **dQ**: Difference between discharges in two points.

- **abs(Q)**: Absolute value of the discharge in a point.

- **Q_Structure**: The discharge through a structure.

- **Sum_Q**: The sum of flows in points and structures.

- **V**: Velocity in a point.

- **Gate level**: The level of a gate.

- **Acc. Vol.**: Accumulated volume running through a point.

- **Time**: The target point will be given as a time series.

- **Min of hour**: Integer expressing the minutes at the time of calculation.

- **Hour of day**: Integer expressing the hour at the time of calculation.

- **Day of week**: Integer expressing the day of the week at the time of calculation. Monday corresponds to one, tuesday to two and so on.

- **Day of month**: Integer expressing the day of the month at the time of calculation.

- **Month of year**: Integer expressing the month of the year. January corresponds to one, February to two and so on.

- **Year**: The year given as an integer value.

- **Time after start**: This control type is used in control strategies with a gate operation that can not be interrupted. An example could be a gate that closes from fully open to fully closed during half an hour when the water level downstream reaches a certain level. Because it is not known when the closing procedure is initiated it is not possible to describe it using a time series. Instead the gate level is described as a function of time measured relative to the time at which the procedure was initiated, i.e. the first value of the Control Type 'Time after start' MUST always equal zero. If it is decided to operate a gate using this control type no other operating policies can be invoked before the actual gate operation has finished. In the example this means that no other operating policies can be used during the half hour it takes to close the gate.

- **Concentration**: A concentration of any compound.

- **Hups**: Water level just upstream of the structure.

- **Hdws**: Water level just downstream of the structure.

- **Qups**: Discharge just upstream of the structure.

- **Qdws**: Discharge just downstream of the structure.

- **Vol**: The volume of water in a point.

- **Volups**: The volume of water just upstream of the structure.

- **Voldws**: The volume of water just downstream of the structure.

- **BranchVol**: The volume of water in a certain part of a river branch

- **Depth**: The depth in a point.

- **Area**: The area in a point.

- **DepthUps**: The depth just upstream of the structure.

- **DepthDws**: The depth just downstream of the structure.

- **AreaUps**: The area just upstream of the structure.

- **AreaDws**: The area just downstream of the structure.

- **ThisGate dh**: The water level difference across the structure.

- **ThisGate Q_Structure**: The flow through the structure.

- **ThisGate Gate Level**: The gate level of the structure.

- **Day of Year**: Integer expressing the day of the year at the time of calculation.

The use of references to the points up- and downstream of the structure and reference to the ThisGate enables a faster editing of the strategies because no location data must be entered.

## Target Type
Here the type of the target point is chosen.

- **h**: Water level in a point.

- **dh**: Difference between water levels in two points.

- **Q**: Discharge in a point.

- **dQ**: Difference between discharges in two points.

- **abs(Q)**: Absolute value of the discharge in a point.

- **Q_Structure**: The discharge through a structure.

- **Sum_Q**: The sum of flows in points and structures.

- **V**: Velocity in a point.

- **Gate level**: The level of a gate.

- **Concentration**: A concentration of any compound. Note that concentration can not be used as target type if the calculation mode is chosen as 'Iterative solution'.

- **Hups**: Water level just upstream of the structure.

- **Hdws**: Water level just downstream of the structure.

- **Qups**: Discharge just upstream of the structure.

- **Qdws**: Discharge just downstream of the structure.

- **Vol**: The volume of water in a point.

- **Volups**: The volume of water just upstream of the structure.

- **Voldws**: The volume of water just downstream of the structure.

- **BranchVol**: The volume of water in a certain part of a river branch

- **Depth**: The depth in a point.

- **Area**: The area in a point.

- **DepthUps**: The depth just upstream of the structure.

- **DepthDws**: The depth just downstream of the structure.

- **AreaUps**: The area just upstream of the structure.

- **AreaDws**: The area just downstream of the structure.

- **ThisGate dh**: The water level difference across the structure.

- **ThisGate Q_Structure**: The flow through the structure.

- **ThisGate Gate Level**: The gate level of the gate.

The use of references to the points up- and downstream of the structure and reference to the ThisGate enables a faster editing of the strategies because no location data must be entered.

## Type of scaling:

- **None**: This is the default value. When this is chosen no scaling of the value of the target point will take place.

- **Scaling with internal variable**: When this is chosen the value of the target point will be scaled with the value of a specified internal variable. See Control Strategy *(p. 130)* for a list of the internal variables that can be used as scaling factors.

- **Scaling with time series**. When this is chosen the value of the target point is scaled with a factor taken from a time series.

**Value**:
Specific value of a structure change or setting. This column should be edited only if the calculation mode is chosen as either Change With or Set Equal to.

## Details...

When pressing the details button a new dialog pops up. This is used to enter the necessary details in defining the operating rules for control structures in MIKE 11.

There are four property pages: 'Logical Operands', 'Control- and Target point', 'Iteration/PID' and 'Control Strategy'.

### Logical Operands



Figure 2.81    The Logical Operand property page

As stated in Control Definitions *(p. 117)* it is possible to define a number of conditions that all must be evaluated to TRUE if the whole 'if'-statement is to be evaluated as TRUE. These conditions are in MIKE 11 called 'Logical Operands'.The logical operands are entered in the Logical Operand property page, see Figure 2.81. Each row in this table corresponds to a logical operand.

**Note** that it is not necessary to enter any logical operands for the 'if'-statement with the lowest priority. The control strategy belonging to this 'if'-statement is the default strategy and will always be executed when all other 'if'-statements with higher priority are evaluated to FALSE. As an example think of a gate where the gate level is a known function of time. In this case only one control strategy is needed. The control type will be 'Time' and the target type will be 'Gate Level'. Calculation mode is chosen as 'Direct Gate Operation'. It is not necessary to enter any logical operands because when only one control strategy is specified this strategy will have the lowest priority.

**LO Type**: This field holds the type of Logical Operand.

- **h**: Water level in a point.

- **dh**: Difference between water levels in two points.

- **Q**: Discharge in a point.

- **dQ**: Difference between discharges in two points.

- **abs(Q)**: Absolute value of the discharge in a point.

- **Q_Structure**: The discharge through a structure.

- **Sum_Q**: The sum of flows in points and structures. If this is chosen data must be entered in a special dialog. This dialog opens when pressing the 'Sum of Q'-button to the right of the table. How to enter data in this case is described in Sum of Discharges *(p. 128)*.

- **V**: Velocity in a point.

- **Gate level**: The level of a gate.

- **Acc. Vol.**: Accumulated volume running through a point.

- **Min of hour**: Integer expressing the minutes at the time of calculation.

- **Hour of day**: Integer expressing the hour at the time of calculation.

- **Day of week**: Integer expressing the day of the week at the time of calculation. Monday corresponds to one, Tuesday to two and so on.

- **Day of month**: Integer expressing the day of the month at the time of calculation.

- **Month of year**: Integer expressing the month of the year. January corresponds to one, February to two and so on.

- **Year**: The year given as an integer value.

- **Concentration**: A concentration of any compound.

- **TS-Scalar**: The logical operand is here a number given in a time series.

- **Loop number**: This is a special type. To illustrate the use, an example is appropriate: Imagine a situation where a certain water level downstream of the structure is required, but only under the condition that a minimum discharge through the gate is maintained. This requires two iteration loops. In the inner loop (the first one) an iteration will be performed in which the required water level downstream is achieved. In the outer loop (second loop) it is checked if the discharge is larger than or equal to the minimum discharge allowed. This check is performed AFTER the inner loop has converged. If the discharge is too low a new iteration takes place in which it is ensured that the discharge is not smaller than the minimum required. In order to be able to formulate such a problem in MIKE 11 the Logical Operand type 'Loop-Number' has been implemented. The inner loop corresponds to 'Loop Number' equal to one, the next loop corresponds to 'Loop Number' equal to two and so on.

- **TSLGLC**: Making a simulation using a time step of five minutes will result in an update of the gate level for every five minutes. Sometimes this gives too much information. Maybe the user is only interested in updating the gate level every hour. This can be achieved using this TSLGLC (**T**ime **S**ince **L**ast **G**ate **L**evel **C**hange) type of logical operand. This variable counts the time since the gate level last changed and can thus be used to ensure that the gate level is not updated at every time step.

- **TOF**: Time Of Forecast. Used to decide if the simulation is in hindcast or forecast mode.

- **Hups**: Water level just upstream of the structure.

- **Hdws**: Water level just downstream of the structure.

- **Qups**: Discharge just upstream of the structure.

- **Qdws**: Discharge just downstream of the structure.

- **Vol**: The volume of water in a point.

- **Volups**: The volume of water just upstream of the structure.

- **Voldws**: The volume of water just downstream of the structure.

- **BranchVol**: The volume of water in a certain part of a river branch

- **Depth**: The depth in a point.

- **Area**: The area in a point.

- **DepthUps**: The depth just upstream of the structure.

- **DepthDws**: The depth just downstream of the structure.

- **AreaUps**: The area just upstream of the structure.

- **AreaDws**: The area just downstream of the structure.

- **ThisGate dh**: The water level difference across the structure.

- **ThisGate Q_Structure**: The flow through the structure.

- **ThisGate Gate Level**: The gate level of the gate.

**Branch Name LO1**: This field contains the name of the branch with the Logical Operand.

**Chainage LO1**: This field contains the chainage of the Logical Operand.

**Name LO1**: This field is used only when LO Type equals 'Gate Level', 'Q_Structure' or 'TSLGLC'. Then this field holds the structure ID of the relevant structure.

**Comp. No.**: This field is used only when LO Type equals 'Concentration'. The field holds the number of the relevant component.

**Branch Name LO2**: This field is only used if the LO Type equals 'dH' ($H_1$ - $H_2$) or 'dQ' ($Q_1$ - $Q_2$). The field holds the name of the branch in which the $H_2$ or $Q_2$ should be found.

**Chainage LO2**: This field is only used if the LO Type equals 'dH' ($H_1$ - $H_2$) or 'dQ' ($Q_1$ - $Q_2$). The field holds the name of the chainage of the $H_2$ or $Q_2$ point.

**Sign**: Here the operator used in the logical expression is used. The user can choose between {<, <=, >, =>, =, <>}.

**Use TS-value**:

- **No**: If 'No' is selected the value of the Logical operand is compared to the value entered in the 'Value' field.

- **Yes**: If 'Yes' is selected the value of the Logical Operand is compared to the value found in the relevant time series.

**Value**: Here the value that must be compared with the logical operand is entered.

**Time Series File**: This field holds information about the relevant time series file in case that the Use TS-value is chosen as 'Yes' or in the situation where the LO Type is chosen to be 'TS-Scalar'.

**Time Series Item**: This field holds the name of the item chosen from the time series file that was selected in the Time Series File field.

## Sum of Discharges



Figure 2.82    Input page to Sum of Discharges

It is possible to add any number of discharges and use this as a Control Type, Target Type, Scaling Type or a Logical Operand. The discharges can be taken from any grid point and any structure in the setup. Further each discharge can be multiplied with a user defined factor. This factor can be both positive and negative. The sum of Q can be expressed as:

$$\text{sum of } Q = \sum_{i=1}^{i=n} fac_i Q_i \qquad (2.3)$$

$n$ is the number of discharges to sum, $fac_i$ the factor to be multiplied with the $ith$ discharge, $Q_i$.

**Factor**: This corresponds to $fac_i$ in eqn. (2.3).

**Type**: This holds the type of discharge to add.

- **Discharge in grid point**: A discharge in a grid point is selected.

- **Structure discharge**: The discharge in a structure is selected. Note that this is not the same since you can have several structures in the same grid point.

- **Qups**: Discharge just upstream of the structure.

- **Qdws**: Discharge just downstream of the structure.

- **ThisGate Q_Structure**: The flow through the structure.

The use of references to the points up- and downstream of the structure and reference to the ThisGate enables a faster editing of the strategies because no location data must be entered.

**Branch**: The name of the branch with the grid point / structure from which the discharge should be taken.

**Chainage**: The chainage of the grid point / structure.

**Struc. Name**: In case Structure Discharge has been chosen as the type the structure ID must be given here.

## Control and Target Point



Figure 2.83    The Control- and Target point property page

**Control Type**: Here the type of Control Point is chosen. This field is linked to the Control Type field described in Control Type *(p. 120)*.

**Branch, Control Point 1**: This field contains the name of the branch with the control point.

**Chainage, Control Point 1**: This field contains the chainage of the control point.

**Name, Control Point 1**: This field is used only when Control Type equals 'Gate Level' or 'Q_Structure'. The field holds the structure ID of the relevant structure.

**Comp. No., Control Point 1**: This field is used only when Control Type equals 'Concentration'. The field holds the number of the relevant component.

**Branch, Control Point 2**: This field is only used if the Control Type equals 'dH '($H_1$ - $H_2$) or 'dQ' ($Q_1$ - $Q_2$). The field holds the name of the branch in which the $H_2$ or $Q_2$ should be found.

**Chainage, Control Point 2**: This field is only used if the Control Type equals 'dH' ($H_1$ - $H_2$) or 'dQ' ($Q_1$ - $Q_2$). The field holds the name of the chainage of the $H_2$ or $Q_2$ point.

**'Sum of Q for Control Point'-button**: This button is only activated if Control Type is chosen as 'Sum_Q'. How to enter the necessary data in this case is described in Sum of Discharges *(p. 128)*.

**Target Point Type**: Here the type of target point is chosen. This field is linked to the Target Type field described in Target Type *(p. 122)*.

**Branch, Target Point 1**: This field contains the name of the branch with the Target point.

**Chainage, Target Point 1**: This field contains the chainage of the target point.

**Name, Target Point 1**: This field is used only when Target Type equals 'Gate Level' or 'Q_Structure'. Then this field holds the structure ID of the relevant structure.

**Comp. No., Target Point 1**: This field is used only when Target Type equals 'Concentration'. Then this field holds the number of the relevant component.

**Branch, Target Point 2**: This field is only used if the Target Type equals 'dH' ($H_1$ - $H_2$) or 'dQ' ($Q_1$ - $Q_2$). Then this field holds the name of the branch in which the $H_2$ or $Q_2$ should be found.

**Chainage, Target Point 2**: This field is only used if the Target Type equals 'dH' ($H_1$ - $H_2$) or 'dQ' ($Q_1$ - $Q_2$). Then this field holds the name of the chainage of the $H_2$ or $Q_2$ point.

**'Sum of Q for Target Point'-button**: This button is only activated if Target Type is chosen as 'Sum_Q'. How to enter the necessary data in this case is described in Sum of Discharges *(p. 128)*.

**Time Series File**: This field holds information about the relevant time series file in case that the Control Type is chosen as 'Time'. If the button to the right of this field is pressed it is possible to browse for the file. At the same time the relevant item in the time series file can be selected.

**Time Series Item**: This fields hold the name of the item chosen in the time series file that are selected in the Time Series File field.

### Control Strategy

The control strategy page can have two different appearances depending on the selected 'Calculation Mode' on the control structures main page. The default tabular view for definition of regular control strategies (Target point depending on one Control Point variable) is presented in Figure 2.84. If the Calculation mode is selected as 'Tabulated 2D' the Control Strategy page will be presented as shown in Figure 2.85 instead.
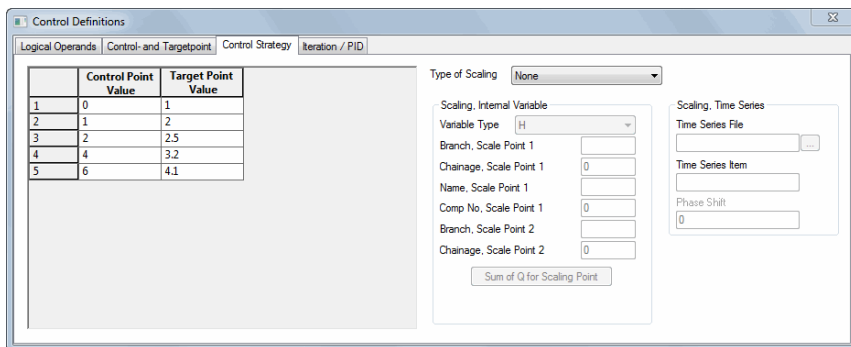
Figure 2.84    The standard view of the Control Strategy property page
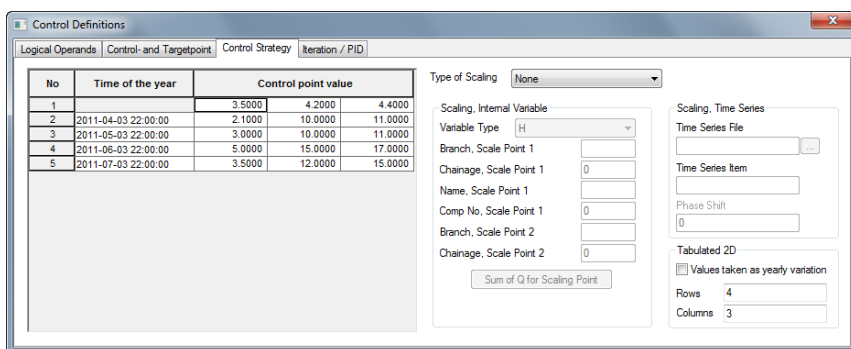


Figure 2.85    Control Strategy page for Tabulated 2D Calculation Mode

Control Strategies are entered in this page either as a relationship between the value of the Control Point and the value of the Target Point for standard tabulated control definitions, or as a relation between dates, a control point variable and the discharge as the target point variable for control definitions using the Tabulated 2D Calculation Mode. These relationships are defined table on the left side of the property page.

Also information about eventual scaling of the target point are entered here. The Type of Scaling field is linked to the Type of Scaling field described in Target Type *(p. 122)*. Below this field there are two sections: A 'Scaling, Internal Variables' section and a 'Scaling, Time series' section. Both of these will be greyed out if 'None' is chosen as scaling type.

If Type of Scaling is chosen as 'Scale with time series' a dfs0 file containing the relevant time series can be allocated by pressing the button to the right of the 'Time Series File'. At the same time the relevant item in the dfs0 file can be selected.

If Type of Scaling is chosen as 'Scale with internal variable' some of the following fields must be filled by the user:

**Variable Type**: The type of internal variables that can be used are:

- **h**: Water level in a point.
- **dh**: Difference between water levels in two points.
- **Q**: Discharge in a point.
- **dQ**: Difference between discharges in two points.
- **abs(Q)**: Absolute value of the discharge in a point.
- **Q_Structure**: The discharge through a structure.
- **Sum_Q**: The sum of flows in points and structures.
- **V**: Velocity in a point.
- **Gate level**: The level of a gate.
- **Concentration**: A concentration of any compound.
- **Hups**: Water level just upstream of the structure.
- **Hdws**: Water level just downstream of the structure.
- **Qups**: Discharge just upstream of the structure.
- **Qdws**: Discharge just downstream of the structure.
- **Vol**: The volume of water in a point.
- **Volups**: The volume of water just upstream of the structure.
- **Voldws**: The volume of water just downstream of the structure.
- **BranchVol**: The volume of water in a certain part of a river branch
- **Depth**: The depth in a point.
- **Area**: The area in a point.
- **DepthUps**: The depth just upstream of the structure.
- **DepthDws**: The depth just downstream of the structure.
- **AreaUps**: The area just upstream of the structure.
- **AreaDws**: The area just downstream of the structure.
- **ThisGate dh**: The water level difference across the structure.
- **ThisGate Q_Structure**: The flow through the structure.
- **ThisGate Gate Level**: The gate level of the gate.

The use of references to the points up- and downstream of the structure and reference to the ThisGate enables a faster editing of the strategies because no location data must be entered

- **Branch, Scale Point 1**: This field contains the name of the branch with the scaling point.

- **Chainage, Scale Point 1**: This field contains the chainage of the scaling point.

- **Name, Scale Point 1**: This field is used only when Variable Type equals Gate Level or Q_Structure. The field holds the structure ID of the relevant structure.

- **Comp No, Scale Point 1**: This field is used only when Variable Type equals Concentration. The field holds the number of the relevant component.

- **Branch, Scale Point 2**: This field is only used if the Variable Type equals dH ($H_1$ - $H_2$) or dQ ($Q_1$ - $Q_2$). The field holds the name of the branch in which the $H_2$ or $Q_2$ should be found.

- **Chainage, Scale Point 2**: This field is only used if the Variable Type equals dH ($H_1$ - $H_2$) or dQ ($Q_1$ - $Q_2$). The field holds the name of the chainage of the $H_2$ or $Q_2$ point.

- **'Sum of Q for Scaling Point'-button**: This button is only activated if Variable Type is chosen as Sum_Q. How to enter the necessary data in this case is described in Sum of Discharges *(p. 128)*.

**Tabulated 2D**: When Calculation Mode is defined as 'Tabulated 2D', an additional group-box is visible on the page. Here, it is required that the user defines the number of lines (dates definitions) and number of columns (control point value definitions) in order to define the size of the 2D table. Dates in the first column must be entered in the format yyyy-mm-dd hh-mm-ss. The top row of all following columns contains the control points variables (e.g. reservoir water levels) and finally the target point values (discharge) as a function of date and a control point variable must be entered in the remaining part of the table.
Please note that all table-fields must be filled with numbers. Blank cells are not allowed.

'Values taken as yearly variation'. This check-box enables the possibility for defining control definitions for one year only and these values will then be repeated using the same variation in consecutive years during a simulation.

### Iteration / PID
**PID-Section**: Here the necessary data is entered if the calculation mode is chosen as PID-operation.

Figure 2.86    The Iteration / PID property page when calculation mode is chosen as PID-Operation.

**Integration Time, Ti**: Corresponds to $T_i$ in eqn. (2.2).

**Derivation Time, Td**: Corresponds to $T_d$ in eqn. (2.2).

**Proportionality Factor, K**: Corresponds to $K$ in eqn. (2.2).

**Weighting factor for time step 1, a1**: Corresponds to $\alpha_1$ in eqn.(2.2).

**Weighting factor for time step 2, a2**: Corresponds to $\alpha_2$ in eqn. (2.2).

**Weighting factor for time step 3, a3**: Corresponds to $\alpha_3$ in eqn. (2.2).

**Iteration-Section**: Here the necessary data is entered if calculation mode is chosen as Iterative solution. When making an iterative solution it is necessary to define some criteria for when the solution is acceptable. MIKE 11 use a criteria that can be expressed like:

$$TP_{Required} - Limit_{Low} \leq TP_{Act} \leq TP_{Required} + Limit_{High} \qquad (2.4)$$

where $TP_{Required}$ is the required value of the target point, $TP_{Act}$ is the actual value of the target point, $Limit_{Low}$ is the amount that the actual value of the target point can be smaller than the required target point and $Limit_{High}$ is the amount that the actual value of the target point can be larger that the required value of the target point.

Figure 2.87    The Iteration / PID property page when calculation mode is chosen as Iterative Solution

**Value -**: This field corresponds to $Limit_{Low}$ in eqn. (2.4).

**Value +**: This field corresponds to $Limit_{High}$ in eqn. (2.4).

**Use absolute or relative value**: Two options exist: 'Absolute' and 'Relative. When choosing 'Absolute' the limits in the convergence interval given in the 'Value -' and the 'Value +' fields are interpreted as absolute values. If 'Relative' is chosen the values are interpreted as fractions of the requested value of the target point. Example: Suppose that the Target Point is the water level downstream of the gate and the requested value of the Target Point is 20. $Limit_{Low}$ and $Limit_{Low}$ are both equal to 0.2. If 'Absolute' is chosen the iteration stops when the actual water level is between 19.8 and 20.2. If 'Relative' is chosen the iteration stops when the actual value is between 16 and 24

**Max. Change of Gate Level**: This field holds the maximum change of the gate level (or discharge in case of a discharge gate) that can take place during one iteration and will be used as the first guess at the change in gate level. Both positive and negative values can be entered. In this way the user can make sure that the first guess at a new level makes the iteration go in the right direction.

## 2.3.12  Dambreak Str.

### General

Most dambreak setups consist of a single or several channels, a reservoir, the dam structure and perhaps auxiliary dam structures such as spillways, bottom outlets etc. Further downstream the river may be crossed by bridges, culverts etc. It is important to describe the river setup accurately in order to obtain reasonable results. There is no limit to the number of dam structures in a MIKE 11 model.

## River channel setup

Setting up the river channel description in the cross-section data base is the same for dambreak models as it is for other types of models. However, due to the highly unsteady nature of dambreak flood propagation, it is advisable that the river topography be described as accurately as possible through the use of as many cross-sections as necessary, particularly where the cross-sections are changing rapidly.

Another consideration is that the cross-sections themselves should extend as far as the highest modelled water level, which will normally be in excess of the highest recorded flood level. If the modelled water level exceeds the highest level in the cross-section data base for a particular location, MIKE 11 will extrapolate the PROCESSED data.

## Reservoir description and appurtenant structures

In order to obtain an accurate description of the reservoir storage characteristics, the reservoir can be modelled as a single h-point in the model. This point also corresponds to the upstream boundary of the model where inflow hydrographs are specified.

The description of the reservoir storage is carried out directly in the processed data. The only columns which contain 'real' data are those containing the water level and the additional flooded area.

In this way the surface storage area of the dam is described as a function of the water level. The lowest water level should be somewhere below the final breach elevation of the dam, and should be associated with some finite flooded area. (This first value, hence, describes a type of 'slot' in the reservoir).

The cross-sectional area is set to a large finite value. It is only used when calculating the inflow headloss into the breach.

It may be practical to locate the dambreak structure on a separate branch containing only three calculation points, as shown in Figure 2.88.
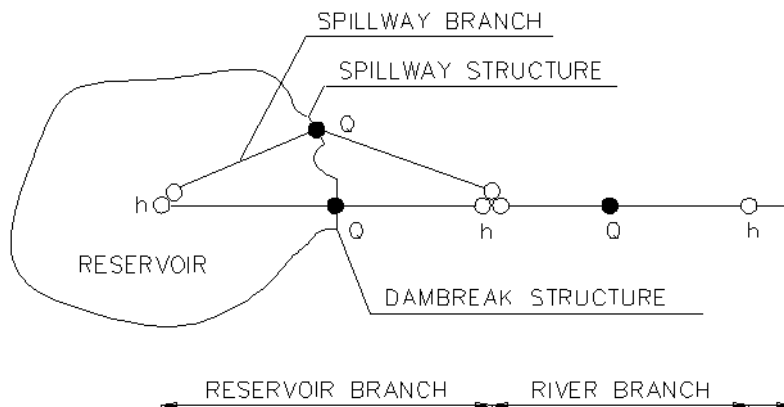
Figure 2.88    Typical setup for dambreak simulation

### The dam

At the *Q*-point where the dambreak structure is located, the momentum equation is replaced by an equation which describes the flow through the structure. This may be either critical or sub-critical. A check on the energy levels at the structure and at the next downstream h-point is first carried out to determine which description is applicable. Refer to the MIKE 11 HD Reference Manual, Dambreak Section.

As the momentum equation is not used at the *Q*-point, the $\Delta X$ step used between the adjoining h-points is of no consequence. The maximum $\Delta X$ step should, however, be greater than the difference between given chainages to prevent the insertion of interpolated cross- sections.

### Spillways and other structures

If a spillway is added to the dam itself, it could be described as a separate branch, see Figure 2.88.

At the node where the two branches meet, the surface flooded area is taken as the sum of the individual flooded areas specified at each h-point.

Hence, if the reservoir storage has already been described in the reservoir h-point, the spillway h-point should contain no additional surface areas. In this case both the width and the additional flooded areas should be set to zero. The cross-sectional area, hydraulic radii, etc. can be given as for the reservoir branch.

It is not a requirement that a separate branch for the spillway structure is defined. The dambreak and the spill way structure can be located in the same grid point, i.e. as a composite structure. The advantage of having two separate branches is that the discharge through the spillway and the dambreak structure is given as two separate time series in the result file.

# Specifying the Dambreak

Figure 2.89    The Dambreak structure property page

This Dambreak structure property page is used for inserting dambreak structures in a given network. The property page (see Figure 2.89) consists of a number of dialog boxes whose functionality is described below.

## Location
**River Name**: Name of the river branch in which the dambreak is located.

**Chainage**: Chainage at which the dambreak is located.

**ID**: String identification of the structure. It is used to identify the structure if there are multiple structures at the same location. It is recommended always to give the structure an ID.

**Type**: The location type may be Regular, Side Structure or Side Structure + Reservoir. See 2.3.3 Structure Types definition *(p. 72)* for details.

## Breach Calc. method
The calculation of the dam break may be carried out using either the energy equation or alternatively the National Weather Service (NWS) dambreak equations. Only the former allows the breach to be erosion based. If the latter is chosen the type of the initial breach must also be selected. The choices are

- Breach failure: The dam break initiates as a breach of the crest.

- Piping failure: The dam break initiates as a piping failure. The shape of the pipe being trapezoidal.

## Dam Geometry

**Crest Level**: The crest level of the dam before failure.

**Crest Length**: The crest length (perpendicular to the flow) of the before failure.

## Limit for Breach Development

Not applicable to the NWS calculation routines.

- **Apply limiting cross section**:
    - **No**: The development of the breach will be unlimited.

    - **Yes**: The development of the breach is limited (e.g. solid rock below the dam). The shape of the limitation should be specified in the Cross Section Editor *(p. 169)*.

- **Topo ID**: Topo ID applied when using a limiting section in the cross section file

- **River Name**: River Name applied when using a limiting section in the cross section file

- **Chainage**: Chainage applied when using a limiting section in the cross section file

- **X-coor at centre breach**: The x-coordinate of the breach centerline specified in the coordinate system applied for the raw data of the limiting section.

## Head Loss Factors

The factors determining the energy loss occurring for flow over/through the hydraulic structure. Only required for the energy loss method.

## Failure Moment and Mode

The moment at which the dam failure commences can be defined in three ways:

1. Hours after Start: The failure is specified to take place a specified number of hours after the start of the simulation.

2. Date and Time: The failure time is specified as a date and time.
3. Reservoir water level: The failure is specified to take place when the water level in the reservoir (assumed to be the grid point immediately upstream of the dam) exceeds a certain level.

The development of the breach can take place in two different ways:

1. Time Dependent: The development of the dam breach is specified by the user in terms of breach level, width and slope as functions of time. Addi-

tionally if a NWS piping failure is selected the top level of the pipe must also be specified. This specification takes place through the Boundary Editor *(p. 215)*.

2. Erosion Based: MIKE 11 calculates the breach development by use of a sediment transport formula for which the parameters are specified in the Dambreak Erosion Dialog.

**Note** that the erosion based method is only available when selecting the energy equation based calculation mode.

### Time Step Control
This feature is only applicable when using a fixed time step. At the specified time after failure the time step is multiplied with the given factor and the remainder of the simulation is carried out with the new increased time step.

- Time after failure when changing the time step: Time for the increase of time step relative to the failure time and specified in hours.

- Factor by which the time step is multiplied: Time step amplification factor. The factor must be larger than one corresponding to an increase in the time step.

It is recommended to use an adaptive time step as an alternative to this feature.

## Making dambreak simulations

### Initial Conditions
In many cases dam failures occur on a dry river bed downstream. However, such initial conditions should be treated with caution in MIKE 11.

Hence, before a dambreak is actually simulated, it is expedient to create a steady-state 'hot start' file which can be used for all subsequent dambreak simulations.

The easiest method of creating such a file is to make a setup identical to that used for the dambreak, with the following exceptions:

1. A small lateral inflow is added at the first h-point in the river downstream of the dam. This will ensure some depth of water in the river from which a steady-state can be reached

2. The inflow into the reservoir can be non-zero, if desired.

3. The dambreak structure should be specified not to fail, i.e. to ensure that the maximum calculated reservoir level is greater than the specified failure reservoir level (i.e. failure will not occur during the generation of the steady- state hot start file). Initial conditions (*h* and *Q*) for this 'hot start'

simulation must be specified in the supplementary data, including the reservoir level.

This setup should be run until a steady-state condition is reached (Q = constant = lateral inflow at the downstream boundary). If this file (*.res11 or *.res1d) is very large, a further simulation can be carried out by using this as a hot start and run it for a few time steps, using the same boundary conditions as previously. This smaller file can then be used for all future hot starts and the larger file can be discarded.

With the hot start file ready, the dambreak simulation can now be carried out. It is suggested that a DELTA value of slightly more than the default of 0.5 be used to damp out short waves which may lead to numerical instabilities. A time step of the order 1-10 minutes is suggested.

## 2.3.13  Dambreak Erosion
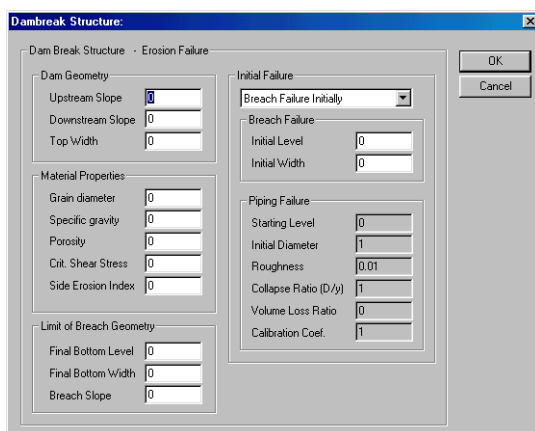


Figure 2.90    The Erosion property dialog

This dialog (Figure 2.90) is accessed from the Dambreak Str. *(p. 135)* property page in the Tabular view: Structures *(p. 71)* by pressing the Erosion Parameters... button (can only be accessed if the Failure Mode is set to Erosion Based). The dialog can only be used to specify erosion based failure modes.

### Purpose
The breach depth relationship is calculated using the Engelund-Hansen sediment transport formula. Breach width is determined from the product of breach depth and the side erosion index specified by user.

### Dambreak Geometry
- Upstream slope: Slope (horizontal: vertical) of the upstream face of the dam structure.

- Downstream slope: Slope (horizontal: vertical) of the downstream face of the of the dam structure.

- Top Width: The top width of the dam crest.

### Material Properties

- Grain Diameter: Representative grain diameter of the dam core material.

- Specific Gravity 2.5 - 2.7: Relative density of the dam core material.

- Porosity 0.3 - 0.5: Porosity of the dam core material.

- Crit. Shear Stress 0.03 - 0.06: Critical shear stress of dam core material used for sediment transport estimation (Shields criteria).

- Side Erosion Index: Multiplication factor used to calculate breach width erosion rates from breach depth predictions.

### Limit of Breach Geometry

The breach will continue developing until it has reached the breach geometry limit, which is defined by

- Final bottom level: The minimum level to which the breach is allowed to develop.

- Final bottom width: The maximum width to which the breach is allowed to develop.

- Breach slope: Slope (horizontal: vertical) on either side of the breach.

### Initial Failure

The failure of the dam can initially take place in two ways:

1.  as a breach starting at the top of the dam

or

2. as a piping failure through the dam.

### Breach Failure

- Initial Level: The level of the breach develops in one time step as an initial breach shape.

- Initial Width: The width of the breach develops in one time step as an initial breach shape.

### Piping Failure

- Starting Level: The level at which piping failure begins to occur.

- Initial Diameter: The diameter of the piping breach which develops in one time step as an initial breach shape.

- Roughness: Pipe roughness used to calculate the Darcy friction factor.

- Collapse Ratio (*D/y*) > 0: When the ratio between the diameter of the pipe (*D*) and the distance from the top of the dam to the top of the pipe is larger than the collapse ratio the pipe collapses.

- Volume Loss Ratio 0 - 1: When the dam collapses some of the material may be carried out without depositing on the bed of the breach. The volume loss ratio is the fraction of the material to be washed out immediately after collapse.

- Calibration Coef. > 0: Calibration multiplication factor used to adjust the calculated change in pipe radius.

## Dambreak geometry output

When the Limiting Section is applied it is possible to activate a feature which creates an ASCII output file after completion of simulation. The ASCII file will contain the final dambreak geometry and it is hence possible to verify which shape has been obtained after simulation.

The ASCII file can be generated as follows: Create an empty text-file in the data-folder where the sim11-file is located. Name of the text file must be 'DBFinalCrossSection.txt'. If this file is present in the data folder, and the Limiting section is active, then the final dambreak section geometry is written to the text-file using the format of a cross section ASCII file. It is therefore possible to import the ASCII file into a cross section file to view the shape of the section at the dambreak location.

## 2.3.14 User Defined Structure
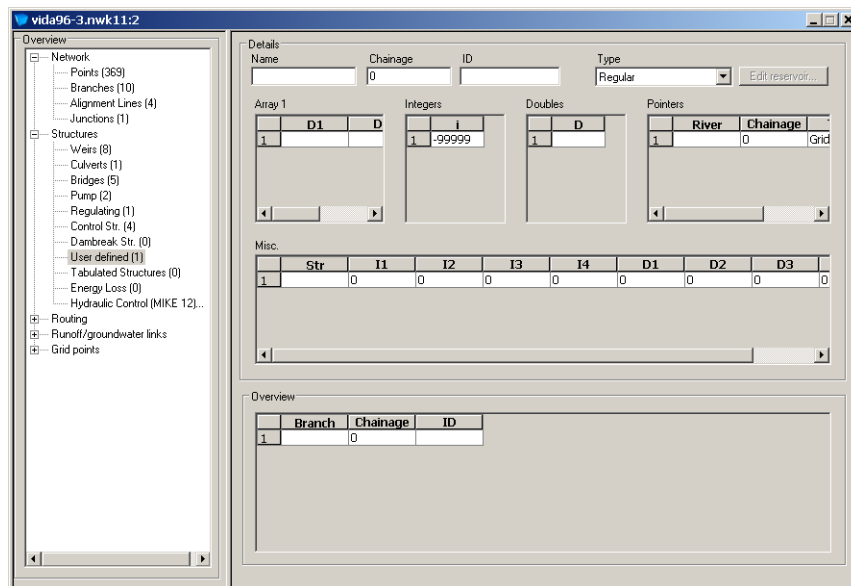


Figure 2.91    The User Defined Structure property dialog

The user defined structure is available to create customised structures in MIKE 11. However, the potential application goes beyond this, allowing for the customisation of almost any specialist application or modification to MIKE 11.

When activated, the user defined structure will access a DLL (Dynamic Link Library) written by the user. The Network Editor interface contains a number of variables that can be used in the DLL. In addition, the DLL can access any variable in MIKE 11 through several records.

MIKE 11 is written in PASCAL using Borland DELPHI. Any code written must be compatible with the compiled unit files (DCU) provided. The easiest way to ensure this is to have Delphi and write your programs in PASCAL.

Please contact your local MIKE support office for use of user defined structures.

For more information, see the Reference Manual.

## 2.3.15  Tabulated Structure
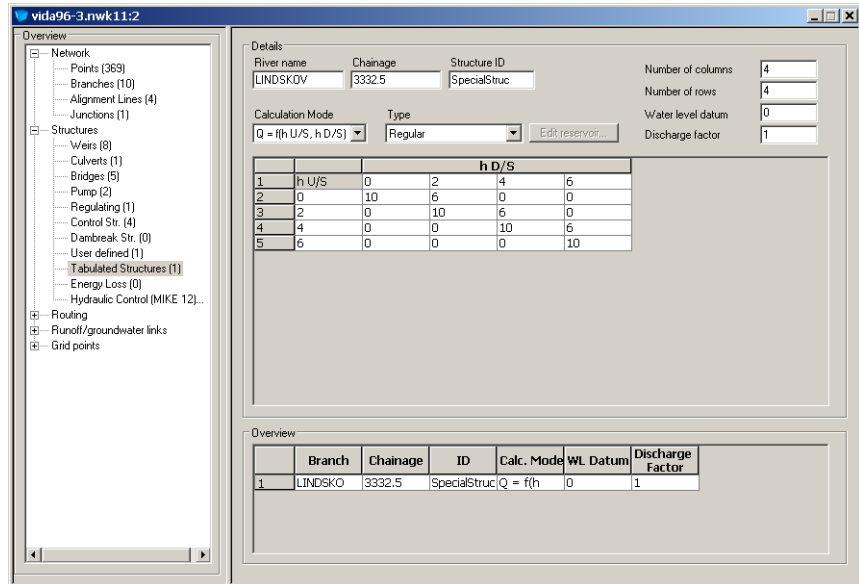


Figure 2.92    The Tabulated Structure property dialog

The Tabulated Structure property page is used for defining a structure regulated by a user defined relation between the discharge through the structure and the up- and downstream water level. The relation is defined in a table. The property page consists of a number of dialog boxes (see Figure 2.92) whose functionality is described below:

### Details

- **River Name**: Name of the river branch in which the structure is located.

- **Chainage**: Chainage at which the structure is located.

- **Structure ID**: String identification of the structure. This has no influence on the simulation. It is only used to identify multiple structures at a single location within a result file.

- **Type**: The location type may be Regular, Side Structure or Side Structure + Reservoir. See section 2.3 Tabular view: Structures for details

- **Calculation Mode**:

– **Q = f(h U/S, h D/S):** The discharge is given as a function of the up- and downstream water level. The upstream water level (h U/S) must be tabulated in the first column and the downstream water level (h D/S) must be tabulated in the first row in the table. Then the corresponding discharges must be tabulated.

The upstream water level must increase in the downward direction and the downstream water level must increase in the right direction. The discharge can not increase in the right direction and it can not decrease in the downward direction.

– **H U/S= f(h D/S, Q):** The upstream water level is given as a function of the discharge and downstream water level. The downstream water level (h D/S) must be tabulated in first column and the discharge must be tabulated in the first row in the table. Then the corresponding upstream water levels must be tabulated.

The discharge must increase in the right direction and the downstream water level must increase in the downward direction. The upstream water level must increase in the right and the downward direction.

– **H D/S= f(h U/S, Q):** The downstream water level is given as a function of the discharge and upstream water level. The upstream water level (h U/S) must be tabulated in the first column and the discharge must be tabulated in the first row in the table. Then the corresponding downstream water levels must be tabulated.

The discharge must increase in right direction and the upstream water level must increase in the downward direction. The downstream water level must decrease in the right direction and increase in the downward direction.

- **Number of Columns:** Set number of columns in the table. The number of columns must be 4 or higher. A large number of columns will increase the accuracy and the stability of the results.

- **Number of Rows:** Set number of rows in the table. The number of rows must be 4 or higher. A large number of rows will increase the accuracy and the stability of the results.

- **Water level datum:** The water level datum is added to the up- and downstream water level in the table.

- **Discharge factor:** The discharge factor is multiplied to the discharge in the table.

## 2.3.16 Energy Loss



Figure 2.93    Energy Loss property page dialog

The Energy Loss property page is used to define energy losses associated with local flow obstructions such as sudden flow contractions or expansions and gradual or abrupt changes in the river alignment. Moreover, a user defined energy loss coefficient can be defined.

At each specified Energy Loss point a discharge grid point is inserted at run-time. At each time level of the computation the discharge at Energy Loss points is computed by use of the energy equation:

$$\Delta H = \zeta \frac{Q^2}{2gA^2} \tag{2.5}$$

in which $\Delta H$ is the energy loss, g is the acceleration of gravity, Q is the discharge and A is the cross-sectional wetted area. The quantity, $\zeta$, denotes the energy loss coefficient as specified in the Energy Loss property page dialog.

Details:
- **River name:** Name of the river in which the Energy Loss point is located.

- **Chainage:** Chainage at which the Energy Loss point is located.

- **ID:** String identification of the Energy Loss point. The specified ID has no influence on the simulation.

- **Apply energy loss:** Determines whether, or not, the associated energy loss type is applied in the simulation.

- **Alignment change:** Denotes the angular change in river alignment at the Energy Loss point in question.

- **Roughness coefficient:** The roughness coefficient is of the order of 0.2 for rough pipes and of the order of 0.1 for smooth pipes.

- **Positive flow:** Denotes the energy loss coefficient in the case of positive flow across the Energy Loss point in question. Applies to user defined loss, contraction loss and expansion loss.

- **Negative flow:** Denotes the energy loss coefficient in the case of negative flow across the Energy Loss point in question. Applies to user defined loss, contraction loss and expansion loss.

- **Overview table:** Contains information on all kinds of energy losses applied at each Energy Loss point within the river network.

## 2.4    Tabular view: Routing

Routing is a simplified hydraulic calculation. Normally, simulation of how a flood wave or a hydrograph propagates along a branch is based on a solution of the St. Venant equations, which requires cross section information. However, if such is not available routing may be an alternative. There are per default no water levels calculated in routing branches. The concept of routing is basically transformation of a hydrograph, i.e. using the inflow hydrograph at the upstream end of a branch (provided either as a boundary condition or coming from the upstream node of the branch) as input routing calculates the outflow hydrograph as function of various user-defined routing elements with specific characteristics and hence able to transform an inflow to a different outflow with e.g. different peak value and adjusted shape of the hydrograph. Typically such a routing element represents a reach of a river or a flood control device such as a reservoir or a hydraulic control structure.

To allow for the insertion of routing components into a branch the branch type must be defined as either "Kinematic Routing" or "Routing". See section 2.2.2 Branches *(p. 60)*.

Any number and combination of routing components are allowed. If no routing components are inserted in a routing branch, the outflow will equal the inflow. The order of the routing components are determined by the chainage of the components. However, additional inflow can also be added as the runoff from a catchment.

A routing component is any of the data types described in the following sections.The order of the routing components are determined by the chainage of the components.

Routing can be combined with normal hydrodynamic simulation such that in some branches routing is applied while in others hydrodynamic simulation is done. The only requirement is that at the upstream end of a routing branch there should either be no other branch connected, or only branches which are routing branches as well.

In order to make a hydrodynamic simulation, MIKE 11 requires a cross section file and a HD parameter file, i.e. if routing is applied in all branches then it will be required to create an empty cross section (.xns11) and a HD parameter file (.HD11) with e.g. default values, and reference made to these in the simulation editor.

If the upstream end of a routing branch has no connection to other branches a discharge boundary condition is required at this location. This also applies if inflow is only to be supplied as catchment runoff. In such case a dummy discharge boundary condition with Q=0 must be specified.

**Note:** The following sections describe a number of different Routing options in MIKE 11. However, it is important to note that only the 'Kinematic Routing option (section 2.4.6) is included in a 'standard' MIKE 11 license.
Other options described (sections 2.4.1 through 2.4.5) are only developed as part of a dedicated MIKE 11 Japanese version and as such only visible and possible to activate for users who have purchased the required modules in their MIKE 11 license. If the required modules are not present in the license, these dialog entries will not appear when opening the Tabular View of the Network Editor.

## 2.4.1 Channel routing

The dialog for specifying the parameters for channel routing is shown in Figure 2.94. (Only visible and applicable is a special M11-Japanese module is included in the actual MIKE 11 license).
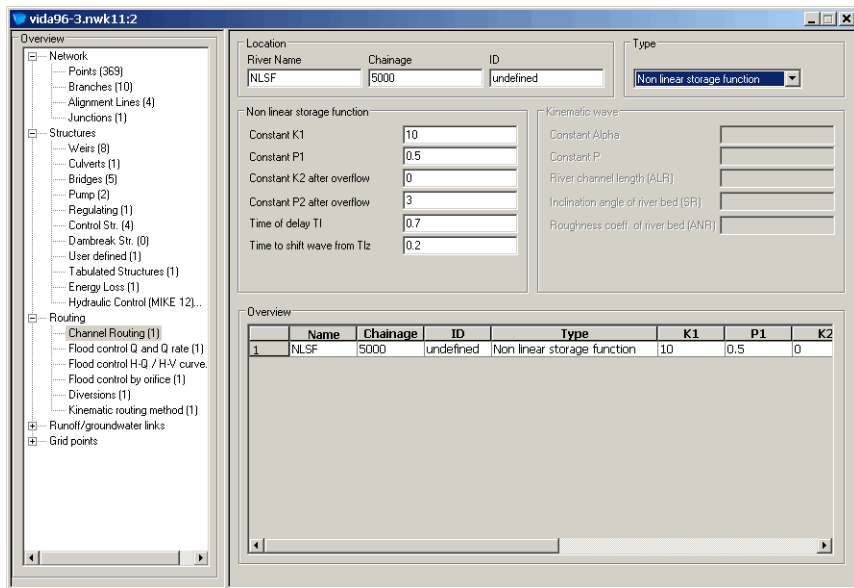
Figure 2.94    Dialog for channel routing

In the dialog the user should specify the following parameters:

**Name:** Name of the branch where the routing component is located.

**Chainage:** Chainage at which the routing component is located.

**ID:** Name of the routing component. Does not influence the simulation.

**Type:** Currently only non-linear storage is implemented.

**K1, P1, Q1, K2, P2, TI, TIz**: Parameters for the calculation. See technical reference for more details.

**Alpha, P, ALR, SR and ANR**: Parameters for kinematic wave. See technical reference for more details.

**NOTE!** The Non-Linear Storage Function method includes a number of default 'Advanced' variables which are editable for the user through the MIKE11.ini file. These variables comprise; 'Error1', 'Error2', 'IR1' and 'IR2'.

## 2.4.2    Flood control Q and Q-rate

The dialog for specifying the parameters for "Flood control Q and Q-rate" is shown in Figure 2.95. (Only visible and applicable is a special M11-Japanese module is included in the actual MIKE 11 license).
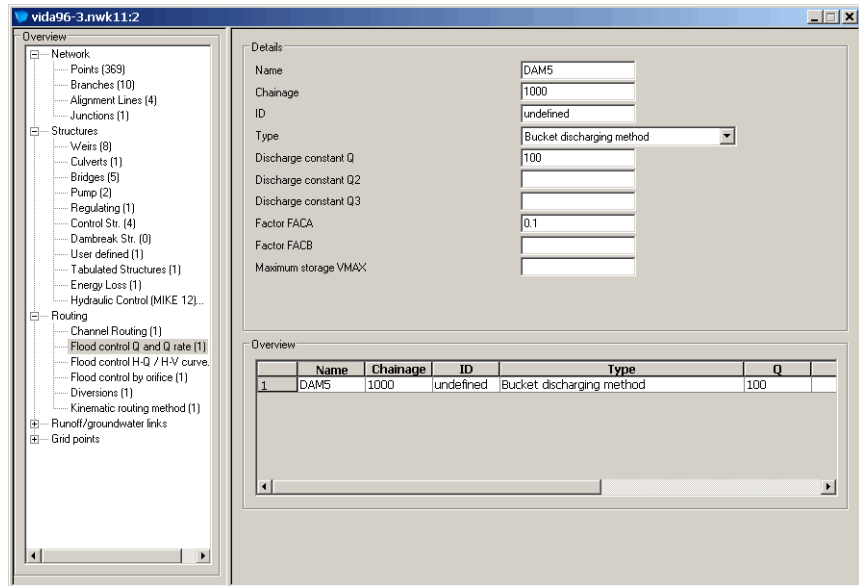
Figure 2.95    Dialog for flood control Q and Q-rate

In the dialog the user should specify the following parameters:

**Name:** Name of the branch where the routing component is located.

**Chainage:** Chainage at which the routing component is located.

**ID:** Name of the routing component. Does not influence the simulation.

**Type:** The user should select the actual type of flood control.

**Q, Q2, Q3, FACA, FACB, VMAX**: Parameters for the calculation. Depending on the selected type of flood control fewer or more or the parameters are required. See technical reference for more details.

## 2.4.3    Flood control H-Q / H-V curve

The dialog for specifying the parameters for "Flood control H-Q/H-V curve" is shown in Figure 2.96.
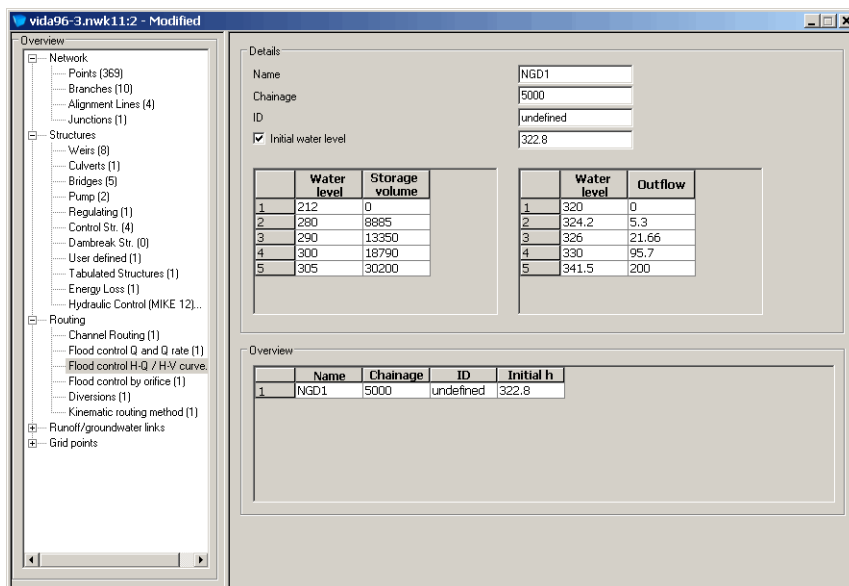
Figure 2.96    Dialog for flood control H-Q / H-V curve

In the dialog the user should specify the following parameters:

**Name:** Name of the branch where the routing component is located.

**Chainage:** Chainage at which the routing component is located.

**ID:** Name of the routing component. Does not influence the simulation.

**Type:** The user should select the actual type of flood control.

**Initial water level:** If checked the water level specified will be applied, otherwise the initial water level will be equal to the water level giving an outflow equal to the initial inflow.

**Water level / Storage volume:** A table of water levels and corresponding storage volumes.

**Water level / Outflow:** A table of water levels and corresponding outflow.

**NOTE!** The Flood Control H-q / H-V method includes a number of default 'Advanced' variables, which are editable for the user through the MIKE11.ini file. These variables comprise 'Error', and 'IBUN'.

## 2.4.4    Flood control by orifice

The dialog for specifying the parameters for "Flood control by orifice" is shown in Figure 2.97. (Only visible and applicable is a special M11-Japanese module is included in the actual MIKE 11 license).
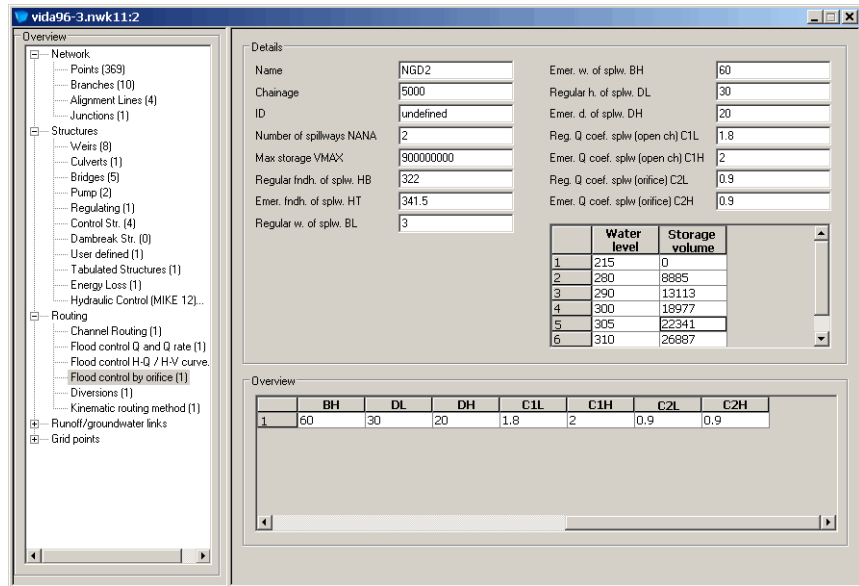
Figure 2.97    Dialog for flood control by orifice

In the dialog the user should specify the following parameters:

**Name:** Name of the branch where the routing component is located.

**Chainage:** Chainage at which the routing component is located.

**ID:** Name of the routing component. Does not influence the simulation.

Additionally a range of parameters should be specified. See the reference manual for more details.

## 2.4.5    Diversions

The dialog for specifying the parameters for a diversion is shown in Figure 2.98. (Only visible and applicable is a special M11-Japanese module is included in the actual MIKE 11 license).
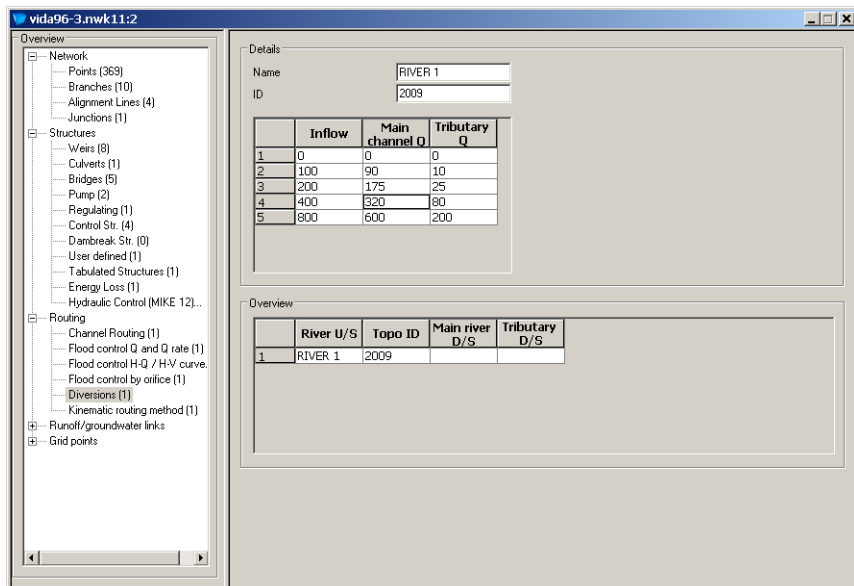
Figure 2.98    Dialog for diversion of flow

Normally when applying the routing facilities the network does not split the flow as a proper calculation of the split requires a water level to be calculated. However, using the diversion facility the user is allowed to specify how a branch splits into two branches. This is done by pre-defining the split of flow, i.e. for a range of inflow discharges the amount continuing along the main branch and along the tributary branch should be specified.

In the dialog the user should specify the following parameters:

**River U/S**: Name of the branch coming from upstream.

**ID:** Name of the routing component. Does not influence the simulation.

**Main River D/S:** Name of the first downstream branch.

**Tributary D/S:** Name of the second downstream branch.

**Inflow, Main channel Q, Tributary Q**: For a range of inflow discharges the amount continuing along the main branch and along the tributary branch should be specified.

Whether the main river downstream carries the majority of the flow does not matter.

This facility does not allow a routing branch to split be into more than two branches. If this is required an artificial routing branch with no routing elements has to be applied. Figure 2.99 shows how this is done when a branch splits into three branches.
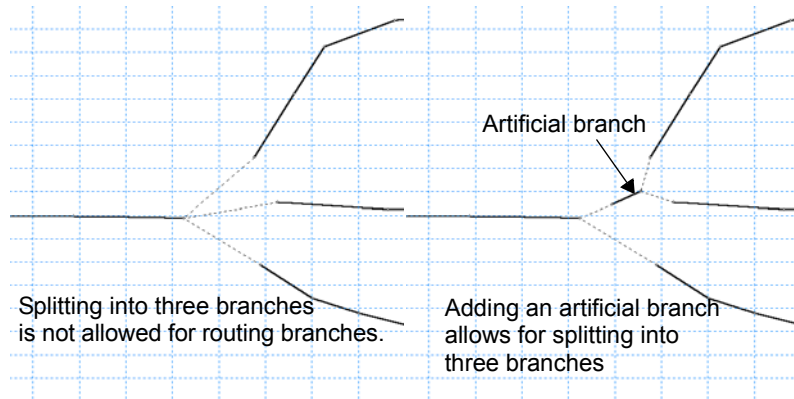
Figure 2.99     Splitting routing branch into three branches

## 2.4.6    Kinematic Routing Method

Kinematic 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. The Kinematic Routing method does not facilitate the use of structures at Kinematic Routing branches. Moreover, the method does not account for backwater effects.

Since the Kinematic Routing method is unconditionally stable, it facilitates the use of large time steps, which is important when running the model in parallel with the hydrological model, MIKE SHE.

At Kinematic Routing branches, it is possible to run the model without information on cross-sections. In turn, this indicates that Kinematic Routing branches can not be used to model a looped part of a river network. Employment of Kinematic Routing branches requires that all branches located upstream of a Kinematic Routing branch are defined in the same way.



Figure 2.100   Definition of Kinematic Routing branches
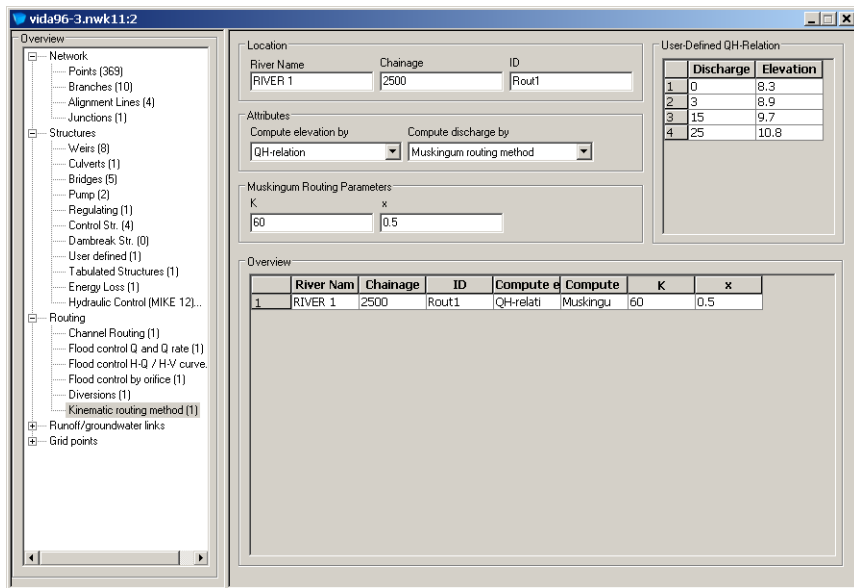
Figure 2.101 Definition of Kinematic Routing elements

The dialog used to define a Kinematic Routing branch is shown in Figure 2.100, while the dialog used to define Kinematic Routing elements is shown in Figure 2.101.

## Details

### Location
**River name:** Name of the river in which the Kinematic Routing point is located.

**Chainage:** Chainage at which the Kinematic Routing point is located.

**ID:** String identification of the Kinematic Routing point. The specified ID has no influence on the simulation.

### Discharge Computation
**Muskingum method**: A routing method that requires the following input parameters:

- **K**: Time scale describing the travel time of the water through the Kinematic Routing element in question.

- **x**: A weighting factor greater than zero and smaller than 0.5.

**Muskingum-Cunge method**: A routing method that does not require any input parameters. At each time level of the computation, the method computes the spatial variation of K and x, cf. above, the intention being to approximate the diffusion of a natural flood wave.

**No transformation:** Employment of this option indicates that the flood wave is not transformed in passing the Kinematic Routing element in question.

### Water Level Computation

**User-defined discharge-elevation method**: Employment of a discharge-elevation relation indicates that the water level is looked up in the specified table using as input to the interpolation scheme the computed discharge. If this method is adopted, cross-sections need not be specified in the cross-section editor.

**Resistance method**: Employment of this option indicates that the Manning resistance method is used to compute the water level. This method requires as input cross-section information, the computed discharge and a bed resistance value.

## 2.5 Tabular view: Runoff / Groundwater Links

This section gives details of how to implement possible links to a rainfall runoff model or linkage to the groundwater model MIKE SHE.

### 2.5.1 MIKE SHE Links
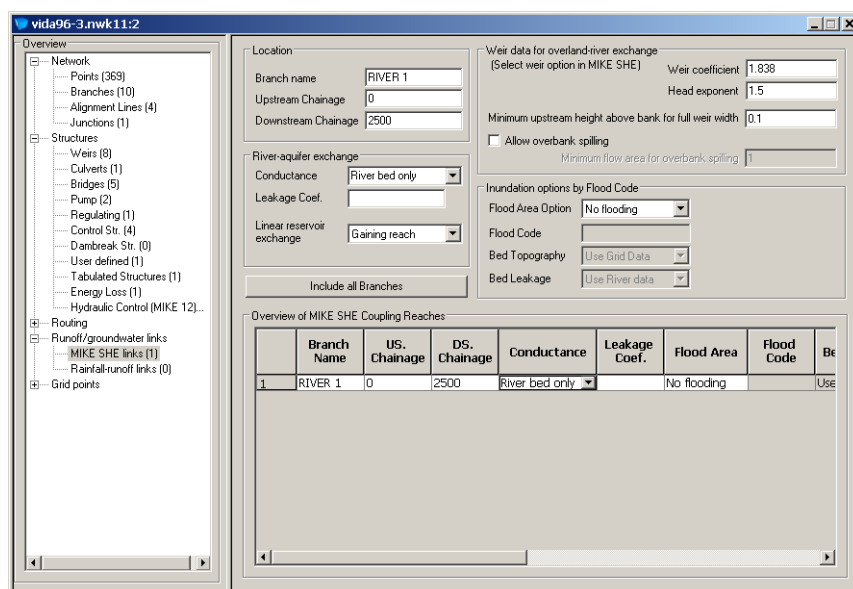


Figure 2.102   MIKE SHE links dialog

### Include all branches

If this button is pressed all branches included in the MIKE 11 set-up are copied to the MIKE SHE coupling page. Branches that should not be in the cou-

pling can subsequently be deleted manually and remaining specifications completed. Thus you may have a large and complex hydraulic model, but only couple (certain reaches of) the main branches to MIKE SHE. All branches will still be in the hydraulic MIKE 11 model but MIKE SHE will only exchange water with branch reaches that are listed in the MIKE SHE coupling definition page.

Observe that the Include all branches feature will overwrite existing specifications.

## Location

### Branch name, US and DS Chainage
The name of the branch and the upstream and downstream chainage for the river reach where the MIKE SHE coupling should be used. One branch can be sub-divided into several reaches. A reason for doing so could be to allow different riverbed leakage coefficients for different reaches of the river.

## Leakage

### Exchange Type
3 different River - Aquifer types of exchange between surface water and aquifer are available (described in the technical documentation of the MIKE SHE User Manual); Aquifer Only, Aquifer + Bed, River Bed only.

When the MIKE 11 coupling is used the exchange type specification in MIKE SHE is ignored.

### Leakage Coefficient (1/s)
Leakage coefficient for the riverbed lining (see exchange documentation). The leakage coefficient is relevant only if the exchange type for the conductance calculation is either 'Aquifer + Bed' or 'River Bed only'.

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

- **Gaining Reach**
  In Gaining Reaches, the river receives baseflow from the baseflow reservoirs in MIKE SHE

- **Losing Reach**
  In Losing Reaches, the river will exchange water to the baseflow reservoirs in MIKE SHE as a function of the Leakage coefficient, Water depth in river, Bank width and length of the coupling reach.

## Weir data for overland-river exchange

When the Weir formula option has been selected in MIKE SHE for Overland-River exchange calculation in MIKE SHE, the overland spilling will be calculated using Weir Formula 1 and weir equation parameters must be defined for each coupling reach.

### Weir Coefficient and Weir exponent

These parameters refer to the Weir coefficient and exponent in the Weir Formula 1 (see MIKE 11 Reference Manual, Weir formula)

### Minimum upstream height above bank for full weir width

In the Weir Formula 1, when upstream water depth above the weir approaches zero, the flow over the weir becomes undefined. To prevent numerical problems, the flow is reduced linearly to zero when the water depth is below the minimum upstream height threshold.

### Allow overbank spilling

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

### Minimum flow area for overbank spilling

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

## Inundation

### Flood Area Option

The Flood Area or Inundation Area option is one of the new facilities in MIKE SHE and allows that a number of model grids are flooded (being part of a river, lake, reservoir etc.). The flood area may be defined as no flooding, auto(matic) or manual. These three may also be used in parallel for different branches or even for specific coupling reaches within the same branch

If the no flooding option is adopted rivers are considered lines located between adjacent model grids. No flooding can occur and over-bank spilling is not possible.

If the auto(matic) or manual option is used a river or a lake (with wide cross-sections) may cause flooding of a number of grids in MIKE SHE. A reference system is established between MIKE 11 *h*-points and individual model grids in MIKE SHE. Subsequently a simple flood-mapping procedure is adopted to calculate water stage on the ground surface (in MIKE SHE). The flood mapping procedure simply compares simulated water levels (in an h-point) with the ground surface elevation in reference grids. If the water level is higher

than the ground surface elevation, flooding occurs. The reference system between *h*-points and model grids may be established automatically by MIKE SHE or it may be established (partly) manually (see below). Each (potentially flooded) MIKE SHE grid point is referenced to the nearest MIKE 11 *h*-point on a coupling reach with the same floodcode value.

- **No Flooding**
  The no flooding option is equivalent to the old formulation in MIKE SHE where rivers are considered a line between two adjacent model grids. If this option is used one of the three river-aquifer exchange formulations will be adopted. River-Overland exchange is always one-way, namely overland to river. Over-bank spilling is not possible when the No flooding formulation is adopted. The river water level may rise above the topographic elevation of the adjacent grids without flooding the grids.

  If the no flooding option is applied the floodcode is not used.

- Automatic Flood Area Option
  The automatic flood-area option is often useful if the geometry of rivers, lakes etc. is not too complex. Thus, for instance, if a large wide river without too much meandering is considered, the automatic flood area option will typically be feasible.

  When the automatic option is chosen, MIKE SHE's set-up program will automatically generate the potentially flooded areas (flood grid code map) depending on the location of the individual rivers and on the width and location of the river cross-sections. The specified coupling reach floodcode is used as grid code, and the flood-mapping procedure described above is applied. Thus it is important to use unique coupling reach floodcode values to ensure correct mapping to the corresponding grid points.

- **Manual Flood Area Option**
  The manual option allows the user to delineate the potentially flooded areas, using a T2 grid code file - the floodcode file specified in MIKE SHE's user interface. If the river system considered is a very complex system with looped networks, meandering generating a complicated geometry, it will typically give the best result to create a floodcode file manually by digitising the floodplain / lake delineation and use this option.

  The flood-mapping procedure above is applied. The potentially flooded area of each coupling reach must be defined with a unique integer grid code value in the floodcode file, and the same integer value specified as coupling reach floodcode.

## Flood Code

Specification needed when the automatic or manual flood area option is chosen.

As described above the coupling reach floodcode is used for mapping MIKE SHE grids to MIKE 11 *h*-points, and for the automatic option also for generating the flood grid codes of the actual coupling reach. It is important to use unique floodcodes to ensure correct flood-mapping.

## Bed Topography

Specification needed when the automatic or manual flood area option is chosen.

The MIKE SHE ground surface elevation can be re-defined in flood area grid points, depending on the bed topography option. It should be emphasised that the flood mapping and dynamic flooding during the simulation requires a good consistency between the MIKE 11 cross-sections and the ground surface elevations of the corresponding MIKE SHE flood grid points.

- **Use Cross-section**
  When this option is specified the ground surface elevations of the actual flood grid points are substituted with values directly interpolated from the MIKE 11 cross-sections of the actual coupling reach. The set-up program performs an inverse-distance-weighted interpolation, using points (elevations) on the MIKE 11 cross-sections as discrete input points. When the distance between individual MIKE 11 cross-sections is higher than ½ Dx (grid size) extra discrete points are generated by linear interpolation between the MIKE 11 cross-sections before the grid interpolation is made. This is done to ensure that an approximate river cross-sectional topography is incorporated in all MIKE SHE grids along the river and not only where a MIKE 11 cross-section is located.

  **Please note** that the interpolated grid values are only used inside the area delineated by the MIKE 11 cross-sections used for interpolation. When the manual flood area option is used, the user defined flood area is not necessarily identical with the flood area covered by the MIKE 11 cross-sections. If the automatic flood area option is used the area covered by the MIKE 11 cross-sections and the flood area will always be consistent, as the flood-area is generated (automatically) based on the MIKE 11 cross-sections.

  In principle the use cross-section option ensures a good consistency between MIKE SHE grid elevations and MIKE 11 cross-sections. There will, however, often be interpolation problems related to river meandering, tributary connections, etc. where wide cross-sections of separate coupling reaches overlap. Thus it is recommended to make the initial MIKE SHE set-up using the Use Cross-section option and then subsequently retrieve and check the resulting ground surface topography (using the MIKE SHE Input Retrieval tool). If needed the retrieved ground

surface topography (T2 file) can be modified (MIKE SHE Graphical Editor) and then used as input for a new set-up, now using the use grid data option described below.

- **Use Grid Data**
  MIKE SHE grid data is used instead of MIKE 11 cross-sections. It is checked whether the optional bed elevation file has been specified in MIKE SHE's user interface:

  – Bed Elevation File specified
    When the bed elevation file has been specified the ground surface elevations of the actual flood grid points are substituted with values from the specified T2 file. The option is useful when the surface elevation data of the flood areas is more detailed than the regional terrain model.

  – Bed Elevation File not specified

The regional MIKE SHE surface topography is also used in flood areas.

As described above the specified T2 file will often be a retrieved and modified surface topography from a previous set-up with use cross-section option.

## Bed Leakage
Specification needed when the automatic or manual flood area option is chosen.

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

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

- **Use grid data**
  The overland-groundwater exchange option and leakage coefficient specified in MIKE SHE's user interface is used. Both can be single value or distributed (T2 file).

- **Use river data**
  The MIKE SHE overland-groundwater exchange option and leakage coefficient in flood grid points are substituted with the corresponding river-aquifer Exchange Type and Leakage Coefficient specified for the actual coupling reach. Please note that the two reduced contact options (exchange types B and C) result in the same overland-groundwater exchange option.

The substitution is made in all flood grid points of the actual coupling reach.

### Overview of MIKE SHE coupling reaches
This box presents an overview of the link with MIKE SHE.

## 2.5.2 Rainfall-runoff links
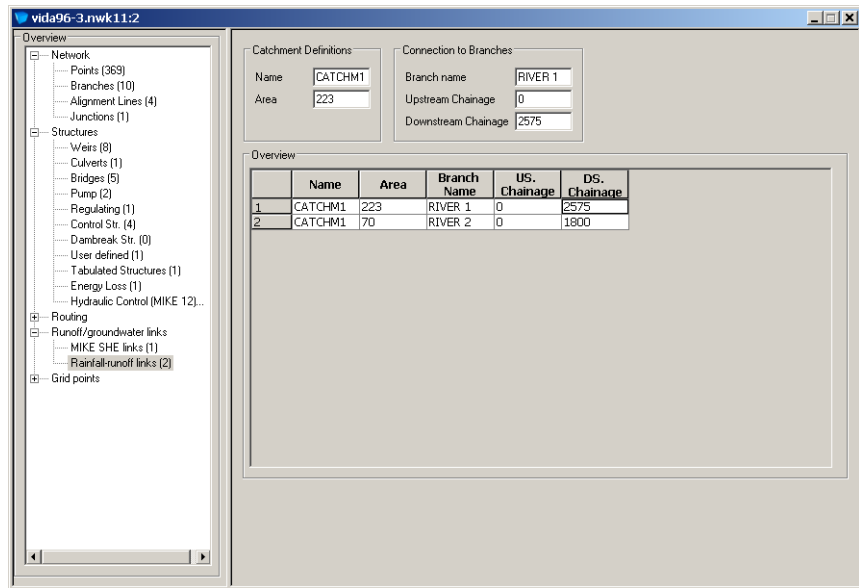


Figure 2.103 Rainfall-runoff links dialog

Catchment discharge can be calculated by the Rainfall Runoff Module and input as lateral inflows to the hydrodynamic module. The property page is used to specify the lateral inflow locations on the river network.

### Catchment Definitions
**Name**: Name of input catchment.

**Area:** Catchment area. The area may be used to distribute Runoff from one catchment to multiple reaches.

### Connection to Branches
**Branch Name**: Name of the river branch for catchment inflow.

**Upstream and Downstream Chainage**: The catchment inflow can be uniformly distributed along a river branch by specifying the upstream and the downstream chainage. Inflow will occur at a single point in the case of equal upstream and downstream chainage.

### Overview

The dialog supplies a tabular overview of the catchment and connection details, which have been entered manually one by one.

## 2.6    Tabular View: Grid Points
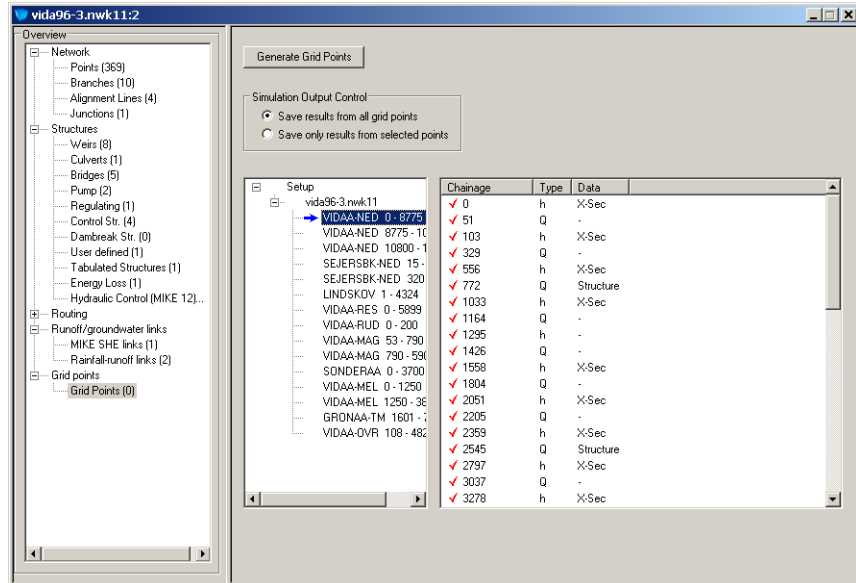


Figure 2.104   Grid points dialog

### Purpose

The page has two specific purposes:

1.  The page presents summary information on the computational network (or grid) points prior to the simulation.

2.  The page can be used to limit the number of computational points saved in result files. (e.g. for large models it is desirable to save only those grid points required and to discard remaining results thus preventing result files from becoming too large).

The page has no influence on the simulation results and is only for information purposes (i.e. the user is not required to the press Generate Grid Points button prior to a simulation. However if changes are made to the model setup (e.g. the location of cross sections or the maximum delta-x in a branch is altered) then the Generate Grid Points button can be pressed to update the tabular information presented.

All generated grid point information is displayed in the Graphical View of the network. To view the grid point information in the graphical view you must ensure the correct options are selected in the Network Settings Dialog.

## Control of Output

When reduced output is selected only those grid points highlighted with a check mark in the right hand side tree view, will be saved. The three levels in the tree view are model setup, model branch and model grid points. These are described below.

Table 2.4    Description of the three levels of output in the tree view

| Level in tree view (right hand part) | Content of the list view (left hand part) | |
|---|---|---|
| Setup | Branches: | Total number of branches |
| | h: | Total number of h-points |
| | Q: | Total number of Q-points |
| | $h^*$: | Total number of selected h-points |
| | $Q^*$: | Total number of selected Q-points |
| Filename.nwk11 | Branch Name: | Name of the branch |
| | US Chn: | Chainage of the upstream end of the branch |
| | DS Chn: | Chainage of the downstream end of the branch |
| | Length: | Length (m) of the branch |
| | h: | Number of h-points in the branch |
| | Q: | Number of Q-points in the branch |
| | $h^*$: | Number of selected h-points in the branch |
| | $Q^*$: | Number of selected Q-points in the branch |
| Branches | Chainage: | Chainage of the grid point. A check mark before the chainage indicates that the grid point is selected. |
| | Type: | h or Q. |
| | Data: | Several types of information are possible: |
| | | The "-" symbol in an h-point row indicates that no cross section is present at this location (i.e. the h-point is generated by interpolation between neighbouring cross sections to fulfil the maximum delta-x criteria). |

Table 2.4    Description of the three levels of output in the tree view

| Level in tree view (right hand part) | Content of the list view (left hand part) | |
| --- | --- | --- |
| | | The "-" symbol in a Q-point row indicates this is a standard Q-point where the momentum equation is solved.<br><br>The word "X-sec" in an h point row indicates that a cross section exists at this location.<br><br>The word "Structure" in a Q point indicates that a structure is located at this location. |

# CROSS SECTION EDITOR

# 3 Cross Section Editor

The Cross Section Editor manages stores and displays all model cross section information.

There are two types of cross section data; the *raw* survey data and the derived *processed* data. The raw data describes the shape of the cross section and typically comes from a section survey of the river. The processed data is derived from the raw data and contains all information used by the computer model (e.g. level, cross section area, flow width, hydraulic/resistance radius). The processed data can be calculated by the cross section editor or entered manually.

Each cross section is uniquely identified by the following three keys:

- **River Name**: The name given to the river branch. String of any length.
- **Topo ID**: Topographical identification name. String of any length.
- **Chainage**: River chainage of cross section.

Refer to one of the following sections for more Information:

Some of the features related to the Steady flow with vegetation module are implemented in the Cross Section Editor. These have been developed in cooperation with CTI Engineering, CO., Ltd., Japan. This also includes moving points parallel or by distance, the version manager, and interpolation of raw cross section data.

## 3.1 Raw Data View

The raw data view is the default and is displayed whenever a cross section file is opened or created (see Figure 3.1).
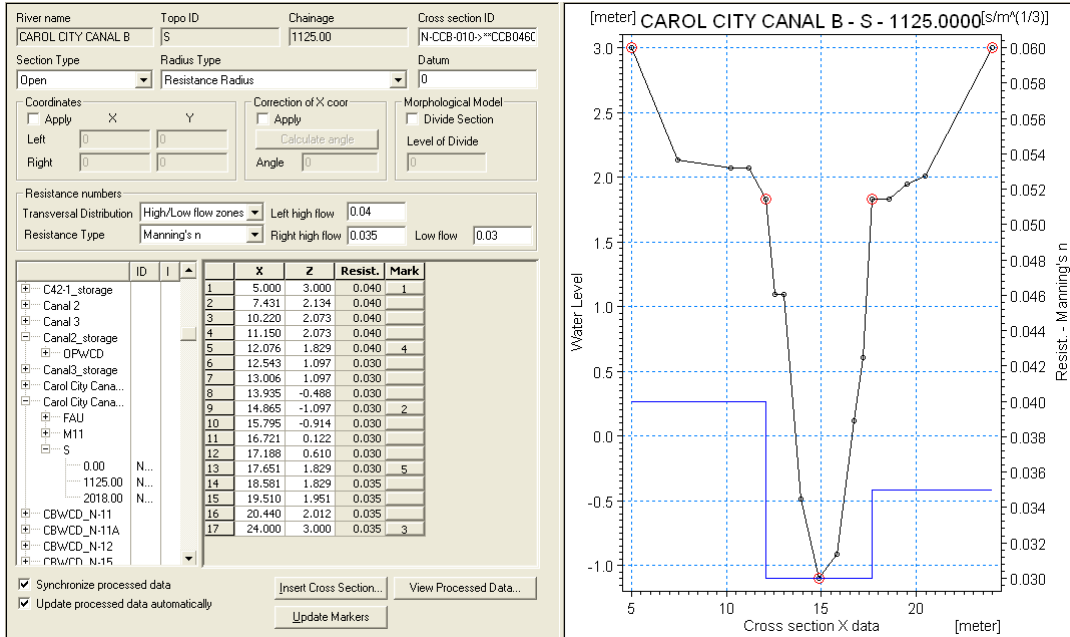
Figure 3.1    The raw data view

The raw data editor is made up by three views plus a number of additional dialog boxes:

- **Tree view**: Provides a list of all cross sections in the file. The list is displayed using a tree structure with three levels. The upper level contains river names, the second contains the Topo IDs, and the third contains cross section chainage.

- **Tabular view**: Selecting a cross section with the left mouse button will display the section information in the tabular view.

- **Graphical view**: An *x-z*-plot of the cross sectional data with markers and vegetation zones indicated (the latter only for the quasi two dimensional steady state solver with vegetation).

## 3.1.1    Cross Section header data

### River Name, Topo ID and Chainage
Non-editable information of the river name the topological identification tag and the chainage along the river. These values may be changed by selecting the appropriate level in the tree view using the rename facility (see Raw data, Tree View).

### Cross section ID

An individual identification tag for each cross section may be entered here. This tag is subsequently displayed in MIKEView and does not influence the calculations.

### Section Type

The type of section is set here. Four options are available:

- **Open section**:
  The typical setting for river cross sections.

- **Closed irregular**:
  Closed sections with arbitrary shape.

- **Closed circular**:
  Closed circular section shape where only the diameter must to be defined.

- **Closed rectangular**:
  Closed rectangular section, where Width and height must be defined.

### Radius Type

The type of hydraulic radius formulation is set here. The choices are:

- **Resistance Radius**:
  A resistance radius formulation is used.

- **Effective Area, Hydraulic Radius**:
  A hydraulic radius formulation where the area is adjusted to the effective area according to the relative resistance variation.

- **Total Area, Hydraulic Radius**:
  A hydraulic radius formulation where the total area is equal to the physical cross sectional area.

### Datum

A datum value may be entered here. The Datum is normally used for adjusting the levels of the cross sections such that they conform with a specific reference datum in the model area. The Datum value is added to all vertical coordinates in the tabular view and the levels of the calculated processed data hence, includes the Datum level as well.

## Coordinates

Plan coordinates may be entered here for the left and right end points of the cross section. If non-zero values are defined the values are used in the graphical view of the network to display the cross section width and additionally - if the 'Apply coordinates' tick-box is enabled - determines also the extent of cross sections in the HD Mapping routine (see section 6.11 Maps *(p. 335)* in HD Parameter Editor section).

## Correction of X-coor

This feature is used for determining the correction angle for the X-coordinates in the profile. The correction may be used for situations where the cross section profile isn't perpendicular to the centre line of the river.

The correction angle can be automatically calculated from the river centre line defined in the graphical network editor and the section plan coordinates by activating the 'Calculate angle' button. Note that this button requires that the simulation file is open, a network file and a cross section file are defined in the simulation editor and that georeferenced coordinates have been applied as described above.

The correction applied is simply a projection of the cross sectional profile on the normal to the thalweg of the river i.e. the correction reads

$$x_{cor} = x\cos\theta \qquad\qquad (3.1)$$

where $\theta$ is illustrated below
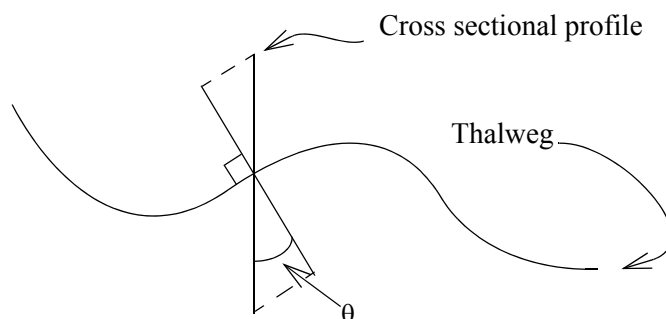


Figure 3.2    Definition sketch of the correction angle

Please note that the correction of X-coordinates is not reflected in a change of raw data x-coordinates but only reflected in the calculated processed data (adjusted widths, etc.)

## Morphological Model

A level of divide can be entered. This level of divide has two purposes:

1. To identify a level where flood plains and the main channel are separated. This feature will be used in a morphological sediment transport simulation to distinguish the morphological active main channel part from the flood plain part of the cross section where morphological changes does not occur. In such a simulation, an internal division in a main channel part and a flood plain part of the section will be made at the x-coordinates corresponding to the Divide level z-value and eventual

morphological changes will only be applied in the main channel part of the section.

2.  The second functionality of the Divide level is that it can be used for activating the flood plain Resistance feature in a hydrodynamic simulation, where a uniform resistance value can be applied in a section above the level of divide as described and specified in Section 6.15 Flood Plain Resistance *(p. 351)*.

## Resistance numbers

In this section of the raw data window the user chooses to how to deal with bed resistance. Two choices have to be made:

### Transversal distribution
This transversal distribution defines the description of resistance across the cross section. There are three choices:

1.  **Uniform**:
    A single resistance number will be applied throughout the cross section

2.  **High/Low flow zones**:
    Three resistance numbers are to be specified: 1) Left high flow resistance applying between marker 1 and 4. 2) Right high flow resistance applying between marker 5 and 3, and 3) Low flow resistance applying between marker 4 and 5. If marker 4 and 5 do not exist the low flow resistance number will apply throughout

3.  **Distributed**:
    The resistance number is to be specified for each X,Z data set in the raw data table.

### Resistance type
Multiple options exists for defining the desired type of resistance method in cross sections.

**Important notice** on selection of Resistance number type and actual resistance numbers applied in cross sections in simulation:

Resistance numbers can be defined as Global or spatially distributed (local values) through the HD Parameter Editor (see section 6.3 Bed Resistance *(p. 315)*). However, if specific resistance numbers are defined in one or more cross sections as either Manning's n, Manning's M or Chezy number values then these numbers will have first priority and overrule any resistance values defined in the HD11-file

The following choices of Resistance types are available:

- **Relative resistance**.
  The resistance is given relative to the resistance number specified in the .hd11 file (HD parameter file). Resistance numbers higher than one always corresponds to higher physical resistance than specified in the .hd11 file (i.e. independent of the choice of resistance number type in the .hd11file).

- **Manning's n**:
  The resistance number is specified as Manning's n in the unit $s/m^{(1/3)}$.

- **Manning's M**:
  The resistance number is specified as Manning's M in the unit $m^{(1/3)}/s$. (Manning's M = 1/Manning's n)

- **Chezy number**:
  The resistance number is specified as Chezy number in the unit $m^{(1/2)}/s$.

- **Darcy-Weisbach (k)**:
  The resistance is specified in the form of an equivalent grain diameter.

## Darcy Weisbach additional parameters

If the Pasche vegetation calculation module is included with the installation (license required) the user has the choice of including the effect of vegetation on the flood plains. To activate the vegetation calculation the resistance type should be chosen as Darcy-Weisbach and the small tick-box next to the resistance type should be ticked. This will activate a small edit button. This edit button will in turn grant access to a dialog from which the vegetation parameters for the left and the right flood plain may be set (Figure 3.3).
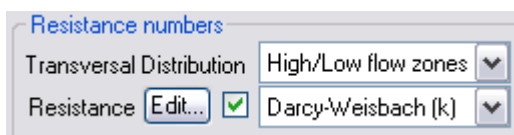


Figure 3.3    The setting of the Darcy-Weisbach resistance. The 'Edit' button is only active if the tick box next to it has been ticked

The vegetation parameters for the Pasche resistance formula are set within the 'additional resistance due to vegetation' dialog. The parameters which are required are (fig.Figure 3.4)

- The river slope

- The shape parameter (1 for a rectangular cross section, 1.2 for a trape-zoidal cross section)

Further for each of the two flood plains the following average values are needed

- The vegetation diameter (Dp)

- The distance between the vegetation elements in the flow direction (Ax)

- The distance between the vegetation elements perpendicular to the flow direction (Ay)



Figure 3.4    The additional parameters to be entered for the Pasche resistance formula

The import and export functionality from ASCII files (see section 3.3) also functions with the additional vegetation parameters.
To import the additional parameters the text file should have the line with the RESISTANCE NUMBERS augmented with the additional 8 parameters specified in the order:

Ax(left) Ay(left) Dp(left) Ax(right) Ay(right) Dp(right) slope shape

To make bulk changes (multiple cross sections) to these parameters there are two options:

1. Use the 'apply to all/selected' to change the 8 parameters
2. Use the 'Resistance number interpolation...' method.

## 3.1.2    Raw data, Tree View

The tree view presents the hierarchy of cross sections in the cross section file grouped in a tree with river names as top level, Topo ID's as secondary level and river chainages as the lower tree view level.

Selecting a river branch, Topo ID or cross section with the mouse and pressing the right mouse button will open **context sensitive pop-up menus** with different features for editing content of either individual cross sections or the entire cross section file presents the pop-up menu. Figure 3.5 presents the context sensitive pop-up menu's appearing for Branches/Topo ID's and single chainages respectively.
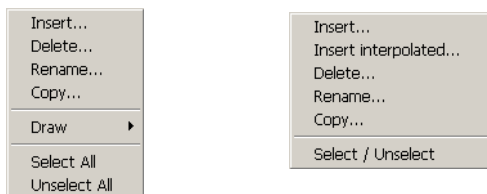
Figure 3.5    Tree-view context sensitive pop-up menus activated through the right mouse button click. A menu similar to that on the left (but also including the 'Combine' command) appears if opened at the Branch or Topo ID level, whereas the right menu appears when opened from a chainage value (individual cross section)

The following editing facilities are available from the context sensitive pop-up menus:

### Insert...
Once the 'Insert...' facility is selected from either the Branch, Topo ID or Section part of the tree view, the Insert branch dialog as presented in Figure 3.20 is activated. Through this dialog it is possible to insert either a new River Name, a new Topo ID and/or a new sections defined by a chainage of a cross section.

### Insert Interpolated
This feature is only available when activating the pop-up menu from the cross section/chainage level of the tree view. The cross section editor gives the user the possibility of inserting interpolated cross sections in a given set-up. When selecting this feature a separate dialog appears.
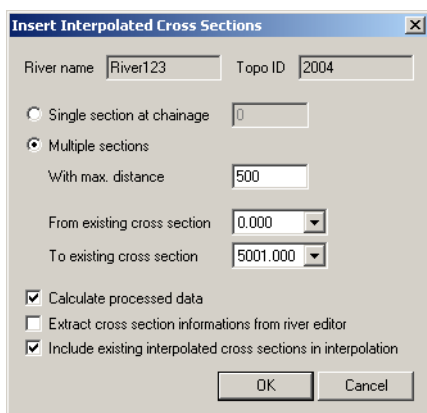


Figure 3.6    The Insert Interpolated Cross section facility from the Raw Data, Tree view

The user can either choose to interpolate a single cross section at a given chainage or multiple cross sections. In the latter case a maximum distance

between the interpolated cross sections must be defined, and additionally, the range of the interpolation need to be specified.

Finally three tick boxes gives the user additional options:

- **Calculate processed data**:
  The processed data is calculated as the cross sections are created.

- **Extract cross section informations from river editor**:
  Checking this the interpolated cross section will be updated with respect to marker positions and zone classifications according to the alignment line information in the network editor. Corresponds to the button called "Update Zone Classification" in the raw data dialog.

- **Include existing interpolated cross sections in interpolation**:
  This box should not be ticked in case the linear interpolations are to be based on the original data only.

### Delete...

From the pop-up menu it is possible to delete either an entire river branch, a single Topo ID within a river or a single cross section. Once activated, the delete-feature will present a confirmation message-box where the user must confirm the deleting of selected item by pressing Yes.
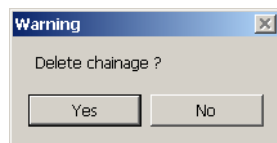


Figure 3.7    Confirmation box for deletion of cross section

If one or more sections have been selected, a different option dialog appears where the user must define whether the Delete action shall include e.g. only the selected sections within a branch or all of the sections within that branch as illustrated in Figure 3.8.
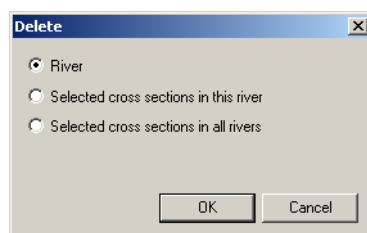


Figure 3.8    Delete sections option dialog from Raw data, tree view

## Rename...

It is possible to rename River names and Topo ID's or even rename/change the chainage of a section through the 'Rename' facility. When activating the Rename dialog from the pop-up menu in one of the tree view levels it is possible to change either name strings for the river name and/or Topo ID or chainage-values for a river section, see Figure 3.9.
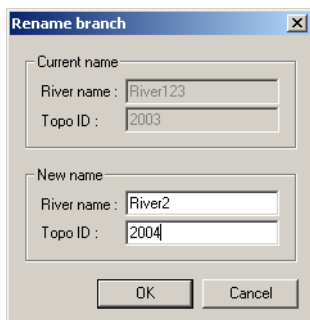


Figure 3.9     Rename feature dialog from Raw data, tree view

## Copy...

The Copy facility makes it possible to copy a single cross section, a Top-ID or a an entire River branch to a different name/location. The copy dialog requests a Topo ID, branch name and chainage before copying the cross section.

If one or more sections have been selected it must be selected in the Copy dialog whether the copy-function shall include all sections or only the selected ones as illustrated in Figure 3.10.
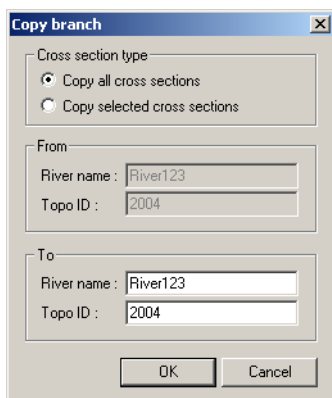


Figure 3.10    Copy facility dialog from Raw data, tree view

## Combine...

The combine dialog is used to combine two river branches of the same name but with differing Topo ID. The combination is saved as a new river branch of the same name and a specified Topo ID. The facility is designed for combining cross sections at chainage locations where two sources of cross section data exist. The dialog which appears once Combine is selected is presented in Figure 3.11.
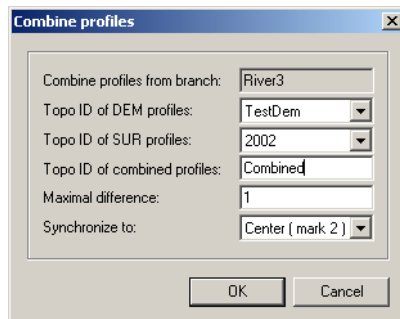


Figure 3.11    Combine cross section profiles dialog from Raw Data, tree view

A typical example occurs when combining survey cross sections (SUR) with digital elevation model sections (DEM). A DEM is typically used to extract sections from broad flood plains while survey data is used to obtain accurate topographical data for river sections. The combine feature will produce a composite section including both data-types and the combined sections can be saved with a new, user-defined Topo ID.

The top non-editable edit-field presents which river branch has been selected for combining sections. The following information shall be defined in the dialog.

- **Topo ID of DEM profiles**:
  Topo ID of the DEM data or alternative first sections to be applied in combined in new generated Topo ID.

- **Topo ID of SUR profiles**:
  Topo ID of the SUR data or second part of sections to be plied in combined in new generated Topo ID.

- **Topo ID of combined profiles**:
  Topo ID of the combined sections. New combined section will only be created at locations where corresponding chainages exist in both the DEM and SUR Topo IDs.

- **Maximal difference**:
  The Maximal difference is the tolerance limit within which sections in DEM and SUR Topo ID are considered to correspond.

- **Synchronize to:**
  Specifies the method for combining sections. Currently only one option is available: Centre (Mark 2) as illustrated in Figure 3.12.
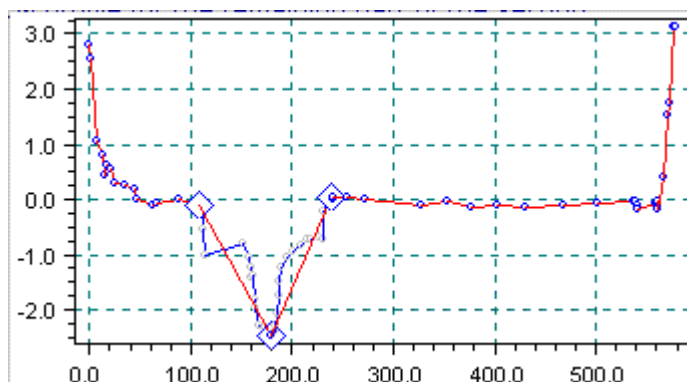


Figure 3.12    Centre (mark 2) The plot shows a DEM section (indicated by the blue/dark dots) and a SUR section (indicated by the grey dots). The combined section will include the DEM section from x=0 to the left marker (the diamond) of the SUR profile, then the SUR profile to the right marker (third diamond from the left) of the SUR profile and finally the DEM profile for the remaining part of the section

## Select / Unselect

The tree view in the raw data dialog provides a feature for selecting cross sections. Most features such as deleting, renaming, copying, processing and plotting can be applied on either all cross sections in the file or on selected cross sections only. Cross sections which are selected are marked in the tree view with chainage in **bold**. Individual cross section can be selected in five different ways:

1.  Double-click on the chainage in the tree view.

2.  <Ctrl>+click on the chainage in the tree view.

3.  Press space bar while the chainage in the tree view is in focus.

4.  Choose "Select/Unselect" in the context sensitive pop-up menu appearing for Section chainage level in the tree view.

5.  The pop-up menu at River name or Topo ID level in the tree view contains items for selecting or unselecting all cross sections in a River name or Topo ID.Raw data, Tabular view

The tabular view is only visible if the section type is set to open or to closed irregular and may in such a case consist of up to six columns given by:

## X

This column contains the transversal coordinates of the raw data.

## Z

The vertical coordinates of the raw data.

X and Z pairs can be typed in manually, or can be imported from a text file as described in section 3.3 below. A third alternative is to Copy and Paste the two columns from a spreadsheet (either together or one by one); you need to highlight the column header X or Z before pasting.

## Resist.

This column is used for setting relative resistance or it may be used for setting local values of Manning's $M$ or $n$, Chezy number or the equivalent grain diameter. The selection depends on the setting in the resistance combo box (see Figure 3.13).
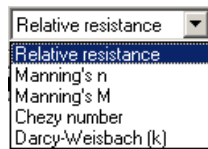


Figure 3.13     Resistance combo box

## Mark

The column is used for setting the markers 1 to 7 and eventual additional user defined marks. Clicking an element in the 'Mark' column opens a marker dialog as shown below from which a requested marker number can be assigned for the selected raw data coordinate set (x,z).
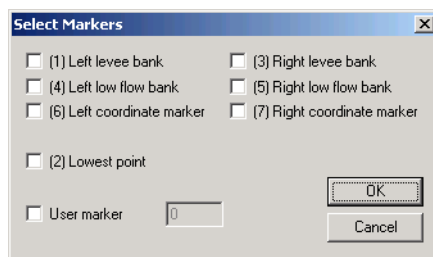


Figure 3.14     Select marker dialog box

A number of markers may be set in this dialog:

- **Left and right levee bank (Markers 1 & 3)**:
  Defines the extend or the active part of the cross section used for the calculations. Default placement of Marker 1 and 3 is to apply marker 1 in the very first point in the raw data ($x_1,z_1$), and Marker 3 at the very last point of the raw data ($x_{last}$, $z_{last}$). However, placing any of these markers at different locations will limit the extent of the active part of the cross section such that only the part of a cross section in between markers 1 and 3 are included in the simulation (that is, Processed data are only calculated for cross section data in between these markers).
  **Note:** Marker locations must be defined such that Marker 1 is defined before Marker 3 in the raw data table.

- **Left and right low flow bank (Markers 4 & 5)**:
  Defines the extent of the low flow channel. The markers have an effect on the calculation of the processed data integration. The section is internally divided into three major 'slices' at markers 4 and 5 positions and the resulting processed data for such a section is a sum of integration results of three sub-parts of the section instead of calculating a result from one single, large section.

- **Left and right coordinate markers (Markers 6 & 7)**:
  Defines the points in the cross section corresponding to the coordinates used for determining the correction angle.

- **Lowest point (Marker 2)**:
  The lowest point of the river may be set using this marker. Marker 2 settings does not affect the calculations at all.
  Marker 2 is primarily used through the network editor for presentation of longitudinal profiles as well as placement of cross section lines if section coordinates are not defined. Additionally, marker 2 is used for Longitudinal profiles and other post-processing features of results in MIKE View and hence, it is recommended to define the correct position of marker 2 in all sections.

- **User marker**:
  Any number above 7 may be used as a user marker. User markers do not impact the simulation results. They are an option for indicating a specific point in a cross section e.g. the location of the measurement gauge. To remove a user marker set the numeric value to 0 and deselect the apply flag.

### Zone

This field and the following are **only** of concern in conjunction with the quasi two dimensional steady flow with vegetation module.

The type of zone is set here by clicking an element whereby a selection combo box is displayed with the following choices:

- Normal: A normal zone.

- Dead water: A user defined dead water zone.

- Vegetation zone: A vegetation zone not at the bank.

- Bank vegetation: A vegetation zone adjacent to the bank.

Please note that the calculation kernel of MIKE 11 does not allow vegetation zones to be defined on vertical sections. The simulation will terminate if this is violated.

### Veg. h.

If a zone is set to either vegetation or bank vegetation this field becomes active. The vegetation height is set here and the average vegetation height for the corresponding panel is displayed in the graphical view.

## 3.1.3   Raw data, Graphical View

The graphical view presents either a single plot of a selected cross section or eventually a number of plots from different sections ('active' and 'passive' cross sections). The graphical plot represents the values defined in the tabular view and eventual changes in the tabular view for a section is immediately presented graphically as well.

### Toolbar icons

Two toolbar-icons for graphical editing are available once the raw data graphical view is activated, See Figure 3.15. Simply activate the graphical view by clicking with the mouse in the view.



Figure 3.15    Raw data Graphical view, Toolbar Icons

Listed from left the icons presented in Figure 3.15 are as follows:

- **Zoom In**:
  Enables zoom in of graphical view

- **Zoom Out**:
  Zoom out to full extent of all sections visible in view

- **Previous zoom**:
  Zoom to previous

- **Next zoom**:
  Zoom to next (active if Previous zoom has been activated one or more times)

- **Draw Grid**:
  On/Off switch for the drawing of a grid in view

- **Undo**:
  Undo facility for the last graphically edited point

- **Select**:
  Selector of point in the graphical view. When clicking on a point in the graphical view the line in the tabular view containing values for the actual point will be highlighted

- **Move points**:
  Moves a point in the graphical view. x- and z-values in the tabular view is automatically updated from the moving of the point.

- **Parallel move**:
  Moves a selected number of points graphically. To use this feature, first select a group of points by clicking at the first point to move and the last point to move. Thereafter the group of points within the selection can be moved by click and drag in the graphical view.

- **Insert points**:
  Insert additional points in a section. By point to a line segment of a cross section and activating the insert points a new point will be inserted in between two existing points on that section reach.

- **Delete points**:
  Deletes points from the graphical view, and automatically points will be removed from the tabular view as well.

## Right mouse pop-up menu

To control the settings and appearance of the graphical view, a number of facilities are available through a right mouse pop-up menu in the graphical view. To open the pop-up menu point to the graphical view with the mouse cursor and press the right mouse button. A pop-up menu as presented in Figure 3.16 will appear.
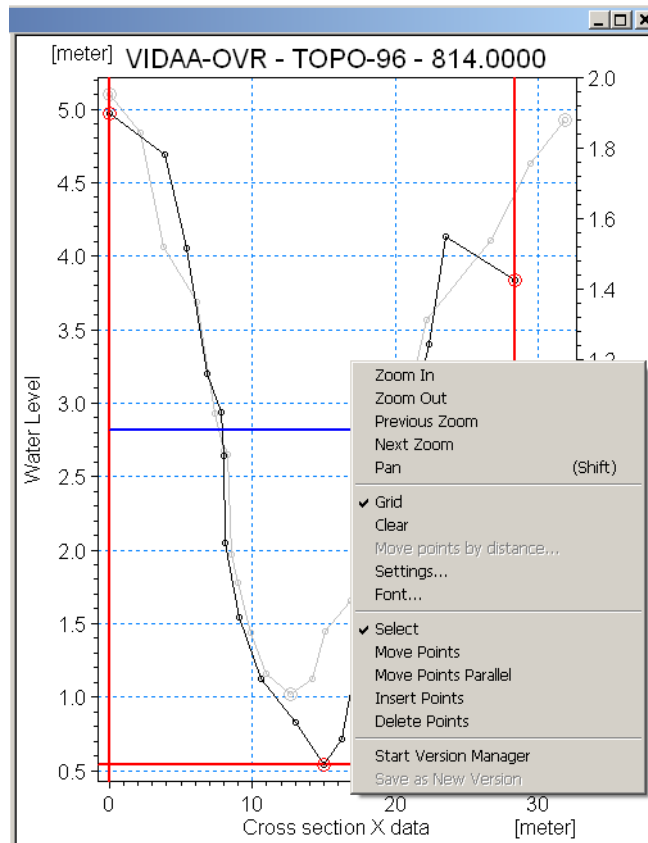
Figure 3.16    Raw data, Graphical view: Right mouse pop-up menu

The pop-up menu includes the following four feature groups:

1.   The first group of features are the zooming facilities.
     From here the zoom in, zoom out (to full extent!) and the previous zoom
     facilities as described under toolbar icons above can be activated (simi-
     lar functionality as when an icon has been activated).


2.   The second group of features variable settings for the appearance of the
     graphical view.

     These features include:

     –    **Grid:** Switch for activating the grid on the graphical view
     –    **Clear:** Clears the view such that only the active section is presented
          and additionally zooms out such that the actual cross section uses
          the entire view.
     –    **Move points by distance:** A feature for enabling a moving of a
          group of points by a user-defined distance (in both x- and z-direc-
          tion)

–   **Settings:** Opens the Cross Section Settings dialog as described in Section **3.1.6** below. Through this dialog the general appearance of the graphical plot of sections can be modified (colours, point-type and -size, etc.)

–   **Font**: opens a separate standard Font settings dialog where type, size and colours of fonts can be modified.

3.   The third group of features are different facilities for making graphical editing of the cross section raw data.
These facilities include;
**Select points,**
**Move points,**
**Move points parallel,**
**Insert points**
**Delete points.**
This list of facilities is identical to the graphical editing toolbar icon facilities as described above. Similar functionality if a feature is activated through the pop-up menu as compared to activating a toolbar icon.

4.   The last group of features in the menu is a version manager facility as described in more detail below.

## Version Manager

The version manager is a facility which enables a storing of multiple versions of one or more cross sections. It is possible through the version manager to save different versions of the same cross section which eventually undergoes different changes in e.g. a design-optimisation project.
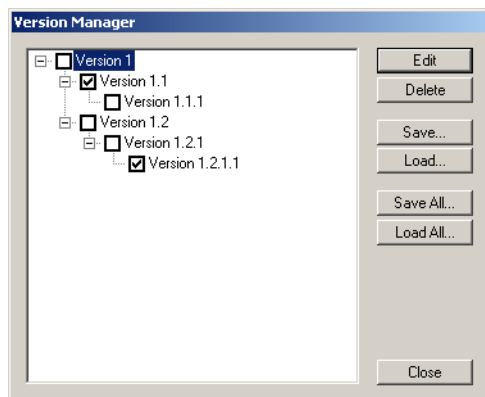


Figure 3.17    Version manager, Raw data graphical view

The version manager as presented in Figure 3.17 saves the different sections in a tree-view format which enables the user to keep track of different changes the section has undergone. The Version manager data can be saved to an ASCII file by use of the 'Save' buttons - and can be re-loaded by the 'Load' buttons.

### 3.1.4 Additional features of the Raw Data editor

#### Additional tick boxes

At the bottom left corner of the editor two tick boxes are present as presented in Figure 3.18.
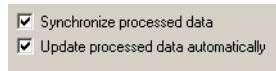


Figure 3.18     Additional tick boxes, Raw data editor

#### Synchronize processed data
By ticking this box the processed data and raw data views are synchronized i.e. if both views are open the data displayed corresponds to the same cross section.

#### Update processed data automatically
Ticking of this box ensures automatic updating of processed data for all sections that are activated in the raw data tree view.

#### Additional buttons

Additional buttons with different functionality is present in the lower part of the editor under the raw data table, see Figure 3.19.
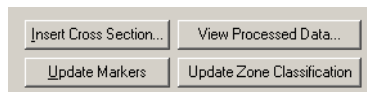


Figure 3.19     Additional buttons, Raw data editor

#### Insert cross section
Pressing this button activates a pop up dialog as shown below.
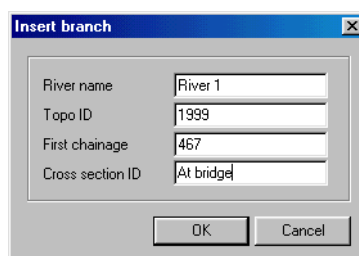


Figure 3.20     The Insert branch dialog

In this dialog the appropriate information for a new section in a new or existing river branch must be specified and thereafter press OK.

### View processed data
This button opens the separate processed data view.

### Update markers
This button updates markers 1, 2 and 3 in the actual section as the extremes of the cross section (left bank limit, lowest point and right bank limit respectively).
Note, this facility overwrites eventual user defined settings of these three markers unless the appropriate boxes in the dialog under 'Settings –> Cross section.... –> Update Markers' have been unticked. That is, markers are only updated with this button if they are activated in this settings dialog.

### Update zone classification
Only used in conjunction with the quasi two dimensional steady flow with vegetation module. Note that this button is only active and visible in case this module is present in the actual MIKE 11 license.

Used for updating the zone classifications in the cross section.

## 3.1.5   'Cross-sections' pull down menu

When the cross section editor is active the cross section pull down menu may be activated as presented in Figure 3.21.
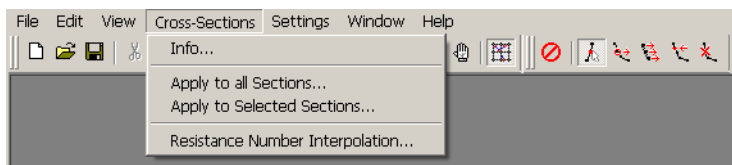


Figure 3.21    Cross-Sections pull down menu

This menu has four items:

### Info

This simply gives an overview of data in the cross section data base:

- Number of Rivers
- Number of Topo IDs
- Number of cross sections in actual Topo ID
- Number of X, Z in actual profile.

## Apply to all sections

This option activates a dialog with a number of options see Figure 3.22.



Figure 3.22    Apply to all cross sections dialog

To change any of the cross section parameters listed in this dialog the respective 'change' tick box must be selected.

### Raw data - Radius Type
The user can choose to change the radius type of all cross sections in the set-up.

### Raw data - Datum
The global datum can be changed here.

### Raw data - Section Divide
A global level of divide can be set here.

### Raw data - Resistance
A global change of transversal distribution, resistance type definition and resistance values can be set here.

### Processed data - Level selection method
The global settings for the selection of the water levels used for calculating the processed data may be set here.

### Processed data - Number of levels

The global number of levels used for determining the processed data is set here.

### Chainages

Activate the 'Calculate' tick-box to enable the Chainages calculation tool.

Two options are present in this tool:

- **From end coordinates and branch line**
  Update chainages of cross sections by using the cross section end coordinates and the resulting cross section line crossing with the branch line.

- **New Chainage = Old Chainage \*C1 + C2**
  Update chainages of cross sections as a function of existing (old) chainages and the coefficients C1 and C2.

### Raw data - XZ Data

An option for inverting cross section raw data coordinates. Generally, cross section raw data must be entered as data entered from left to right when looking downstream. However, data can be received and hence entered in the opposite direction and this option allows for inverting the entire section.

### Markers

An option for either deleting or changing/updating marker positions globally from one of the suggested options in the drop-down selection box.
The 'apply' tick-boxes determines which markers should be updated as only selected marker boxes will be updated when processing the sections.

### Action to be done

A number of options to be applied to all the cross sections in the set-up are available:

- Update zone classification.

- Update correction angle.

- Recompute all

- Activate Apply Coordinates
  Enables and checks Apply under Coordinates-group, so Left and Right (X, Y) can be edited.

### The OK button

Press the OK button to execute the section settings selected in the dialog.

## Apply to Selected Sections

Activating this feature enables the same possibility for modifying cross section parameters as listed above in the 'Apply to all sections' but the adjustments will **only** be performed **on sections selected in the tree view**.

**Please note:** Applying any of the above will overwrite any user edited settings/data. There is no undo feature so make sure to save the cross section data before activating the OK button.

## Resistance Number Interpolation...

When calibrating/changing resistance numbers in a large number of cross sections where resistance number distribution is high/low flow zone or uniform, it may be feasible to use this tool to interpolate resistance numbers. The tool works such that the user specifies the resistance number at a number of locations and when pressing the OK button the resistance numbers in all cross sections between specified locations will be linearly interpolated.

| | River name | Topo ID | Chainage | Left High Flow | Low Flow | Right High Flow | Uniform | Vegetation |
|---|---|---|---|---|---|---|---|---|
| 1 | | | 0.000 | 1.000 | 1.000 | 1.000 | 1.000 | 1.00 |

Add all sections    Add selected sections

Figure 3.23    (Parts of) Resistance Number interpolation dialog

The information in the resistance number interpolation table is saved in a separate file with the same name as the cross section file, but with extension .xns11r. This is an ASCII file in PFS format. In this way the auto calibration tool can be used to calibrate resistance numbers in cross sections.

## 3.1.6   'Settings' pull down menu

The Settings pull down menu offers options for customizing the view and performance of the cross section Raw Data Editor. The pull down menu is opened through the main menu bar as presented in Figure 3.24

File  Edit  View  Cross-Sections  Settings  Window  Help
Cross-Section...
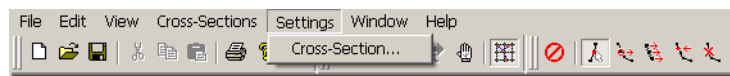
Figure 3.24    Cross section, Settings pull down menu

## Cross Section settings; Graphics

The graphics settings consists of a tree structure view of all possible settings for the graphical elements of the cross section editor, see Figure 3.25. The desired elements are ticked and the properties are set using the right hand side of the dialog box.
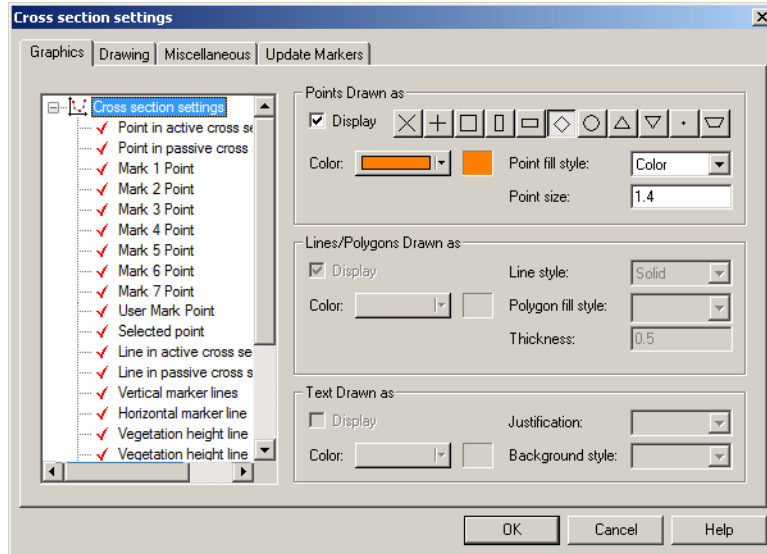


Figure 3.25    Settings -> Cross Sections.dialog, Graphics page

## Cross Section Settings; Drawing
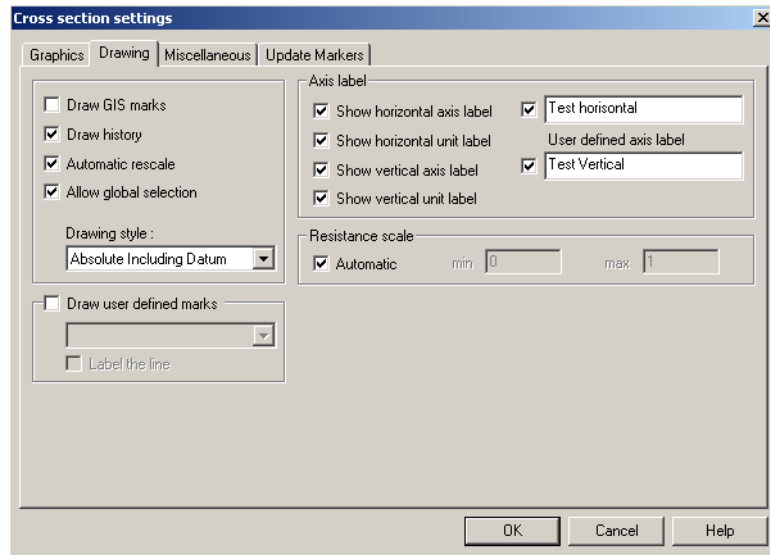


Figure 3.26    Settings -> Cross Sections dialog, Drawing page

A number of settings are available (Figure 3.26):

**Draw GIS marks**: Marks the locations on a cross section where data has been extracted from GIS images.

**Draw history**: Creates a *watermark* as a history of previous cross sections drawn on the graphical view. The current cross section and previous cross sections can be presented in different colours dependent on settings in the Graphics dialog page. This feature allows comparison of multiple cross sections on a single scale.

**Automatic rescale**: Automatic re-scaling of the graphical view when raw data is being displayed. This prevents plotting of cross sections outside of the display area.

**Allow global selection:** Allows previous cross sections displayed as watermarks to be selected from the graphical view using the mouse pointer. If this function is off, the mouse pointer will only select from the current cross section displayed in black.

### Drawing style
The drawing style controls the *Z* axis display in the graphical view. There are three options available:

1. Absolute Including Datum: The displayed *Z* values include the datum factor.

2. Absolute Excluding Datum: The displayed *Z* values exclude the datum factor.

3. Relative to Bottom: The Z values are displayed relative to the lowest point in the cross section regardless of the datum. (i.e. all cross sections will be displayed with the lowest point set to 0 metres.).

### Axis label
The axis label feature allows for user-defined axis labelling and selection whether the graphical plot should include units or not.

### Resistance scale
The resistance scale option defines whether a fixed or automatic scaling of the Resistance axis is presented. In case of a fixed scale selected, the minimum and maximum value of the axis scale must be defined.

## Cross Section Settings; Miscellaneous

The Miscellaneous page contains different options for the graphical and tabular view, see Figure 3.27.
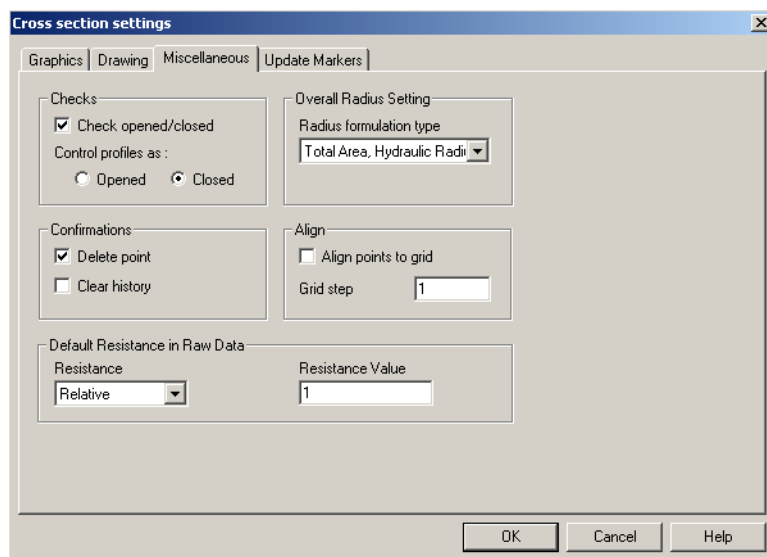


Figure 3.27    Settings -> Cross section dialog; Miscellaneous page

### Checks
A feature for checking raw data to ensure that sections are open or closed. If data does not pass the chosen check-option then a message will occur.

### Overall Radius setting

The default setting of the radius type may be altered here.

### Confirmations

The user can specify whether a confirmation dialog box should appear when deleting points or clearing history in the graphical view.

### Align

A snap to grid feature in the cross section editors graphical view.

### Default Resistance in Raw Data

Used for defining which of the default type and value of the resistance column in the tabular view of the cross section editor. Default values are applied when a new cross section is inserted.

## Cross Section Settings; Update Markers
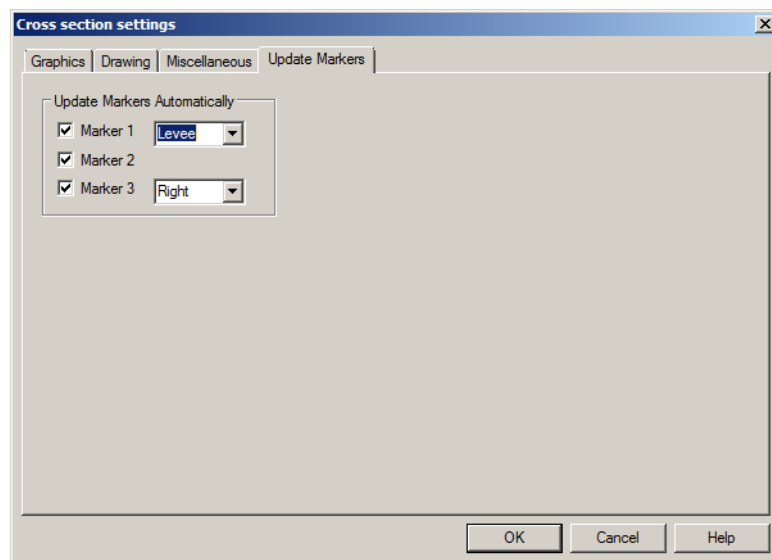


Figure 3.28    Settings -> Cross Sections dialog; Update Markers page

This dialog is used for defining which of the markers 1, 2 and 3 should be automatically updated. Only the selected markers in this dialog will be updated by eventually adjusting its position in a cross section when activating the update markers button. For markers 1 and 3 the user can select to use either the full extent (left and right selected) of the cross section or alternatively the left and the right levee (levee selected for marker 1 and 3). The levee is defined as the highest point in the left and the right part of the cross section respectively. Thus by selecting levee for the left and the right part of the cross section the highest points in the left and the right part

# 3.2    Processed Data View

Selecting the *View Processed Data* button on the Raw Data View *(p. 169)* activates the processed data view.

Processed data are essential in MIKE 11 as these table-values are used directly in the calculation such that values for Area, Radius, Width and Resistance Factors are extracted by interpolation from the processed data tables depending on the water level on a given time step as an entry for interpolation. Hence, eventual problems in calculations for specific ranges of water levels in specific points can often be analysing and adjusting/correcting the processed data for the actual calculation points in the problematic water levels.

**Note on exception:** when utilizing the quasi two dimensional steady state with vegetation module the processed data **does not** reflect the values used in the calculation. In the calculation kernel of this module the X, Z - coordinates of the individual cross sections are used for determining the hydrodynamic parameters of the individual panels.

The processed data view is similar to the raw data display. A tree view exists on the left where the required cross sections can be selected. A tabular view presents all processed data and a graphical view on the right hand side displays the processed data graphically (see Figure 3.29).
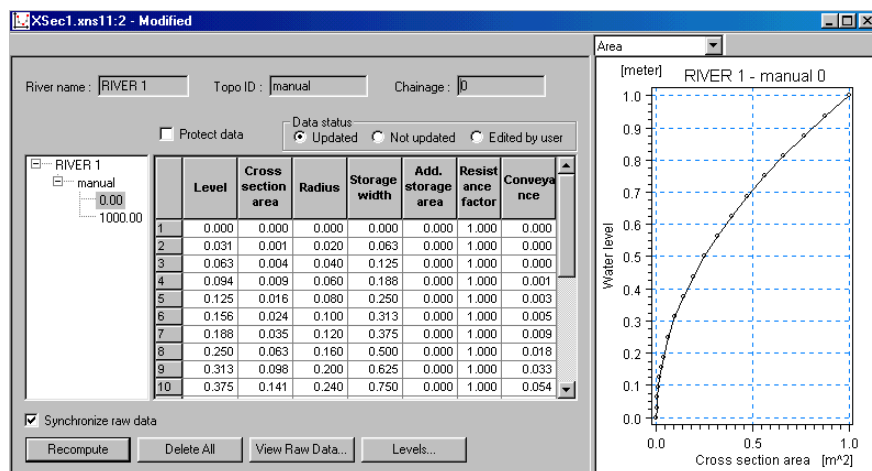


Figure 3.29    The processed data view

## 3.2.1    Processed data, Tree View

Selection of cross sections to view/analyse by use of the mouse. No context sensitive pop-up menus are available in the processed data tree view as all

modifications to the cross sections in terms of naming, location etc. all must take place in the Raw data editor.

## 3.2.2    Processed data, Tabular View

The processed data is calculated from the raw data and contains the following parameters as presented in the tabular view:

### Level
Levels for which processed data are calculated in the cross section. Default levels definition range from the lowest z-value and up to the highest z-value in the raw data table for the actual section
The method of determining processed data levels can be selected using the Levels Dialog activated by the *Levels* button (Section 3.2.4).

### Cross section area
Effective Cross sectional flow area calculated from the Raw Data. Effective Area is determined from the total flow area adjusted by eventual raw data relative resistance values different from one (see Reference manual, Resistance for further details)

### Radius
A resistance or hydraulic radius depending on the selected type in Raw Data Editor (Resistance Radius or Hydraulic Radius).

### Storage width
Top width of the cross section at each processed data level.

### Add. storage area
The surface area of additional storage to be added at a cross section. This is useful for representing small storage's associated with the main branch such as a lakes, bays and small inlets.
Additional storage areas are always user-defined - they will never be given a value from the automatic processing of the raw data.
Additional storage area is interpreted by MIKE 11 as a storage area covering only the distance (dx) determined from the actual cross section (dx/2 upstream + dx/2 downstream of section).

### Resistance factor
This factor can be used to apply a level dependent, variable resistance in the section cross section.The Resistance factor can contain two types of values depending on the Resistance Type definition in the raw data;

1.  Raw data Resistance type defined as Relative Resistance
    In this case, the Resistance factor is interpreted as a multiplication factor which during the calculation will be multiplied to the resistance numbers defined in the HD Parameter editor (HD11) in order to establish a level dependent resistance in the section. That is, the resistance factor works as a level dependent resistance scaling factor in the actual section.

It is important to notice in the case of Relative Resistance Type, this factor is always multiplied to the resistance value defined in the HD Parameter Editor. Dependent on the type of resistance number applied it is therefore important to be careful when editing the resistance factor values. Since they are multiplied to the resistance number these factors must decrease to values smaller than one in order to increase resistance if Manning's M type of resistance number is applied. However, if Manning's n (=1/M) is applied then it is required to define resistance factors higher than one to activate an increase in resistance with depth.

2. Raw data Resistance type defined as specific Resistance number (Manning's n, Manning's M or Chezy number)

In this case, the Resistance factor column contains the actual resistance number applied in the simulation as resistance numbers defined in the HD11-file is not applied for sections and calculation points which have any of these Resistance Types defined (Cross section Resistance values override any other resistance definitions in this case).
The Resistance factor column can therefore have values of either Manning's M, Manning's n or Chezy numbers in this case.

## Conveyance

The Conveyance values are not used in the simulation but is primarily displayed as part of the processed data for the purposes of checking and verifying that the conveyance relationship is monotonously increasing with increasing water level, which is one of the key assumptions for the open water hydraulics.

**Note**: a situation where the conveyance column is not monotonically increasing with water levels can relatively easily occur - especially in the case of some closed sections or in situations where the section geometry includes a sudden width increase and the Radius formulation has been selected to Hydraulic Radius.

Should this situation occur, then it is strongly recommended (not to say a strict requirement) that time is spent on adjusting the section characteristics such that a monotonically increasing Conveyance curve is present. If not, there is a very significant risk of obtaining instabilities in the simulation for water levels in the range where the non-increasing conveyance values are present.

Typical options for optimising the cross section characteristics in the situation of an open section using the Hydraulic Radius option is to manually subdivide the section into several 'slices' by adjusting the relative resistance numbers in the raw data at locations where the section significantly changes shape (e.g. changing a relative resistance value from 1.000 to 1.001 'forces' the processed data calculator to divide the integration of the processed data

into several slices and the non-monotonically increasing conveyance curve can normally be resolved from this.

Additional notice on Conveyance numbers presented:
It is important to notice, that the Conveyance numbers presented in the Conveyance column is in fact not 'True' Conveyance values!
Depending on the choice of Resistance Type in the Raw Data, resistance numbers used in the calculation are extracted from the HD11-file and to avoid a strict dependency to a specific HD11-file during editing and processing of cross sections, it has been decided to present Conveyance values, which does not include the Resistance number (as the True conveyance number naturally will do). Consequently, the conveyance does not reflect the true conveyance, but is primarily offered as a possibility for analysing the 'conveyance trend' as a function of water levels in the cross sections. And these should be monotonically increasing with water levels to secure a healthy output from your simulations!

## 3.2.3 Processed data, Graphical View

The graphical view presents a specific item of the processed data table as a water level curve dependent graph.

### Toolbar Icons

A set of toolbar icons are active with the graphical view, see Figure 3.30



Figure 3.30    Toolbar Icons for Processed Data, graphical view

Listed from left the icons presented in Figure 3.30 are as follows:

- **Zoom In**: Enables zoom in of graphical view

- **Zoom Out**: Zoom out to full extent of all sections visible in view

- **Previous zoom**: Zoom to previous

- **Next zoom**: Zoom to next (active if Previous zoom has been activated one or more times)

- **Draw Grid**: On/Off switch for the drawing of a grid in view

### Right mouse pop-up menu

As in all other graphical views and editors in MIKE 11 a right mouse pop-up menu is available with a number of facilities present.
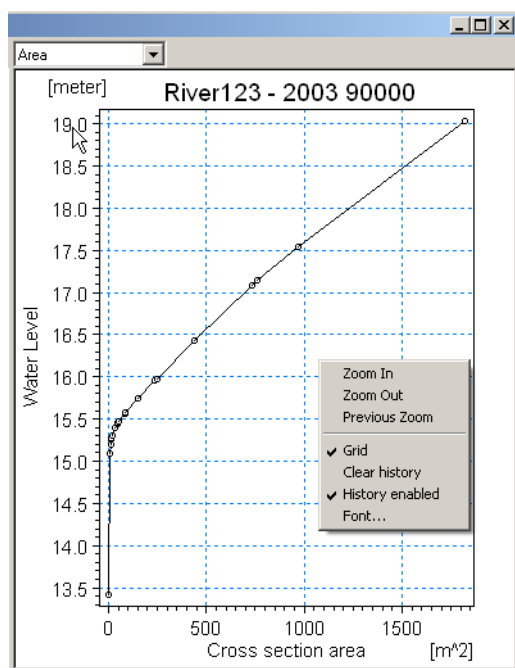
Figure 3.31    Right mouse pop-up menu for Processed data, Graphical view

The features included in the pop-up menu are as follows:

1.  The first group of features are the zooming facilities.
    From here the zoom in, zoom out (to full extent!) and the previous zoom
    facilities as described under toolbar icons above can be activated (simi-
    lar functionality as when an icon has been activated).

2.  The second group of features variable settings for the appearance of the
    graphical view.

    These features are:

    –   **Grid**: Switch for activating the grid on the graphical view
    –   **Clear history**: Clears the view such that only the active graph is
        presented and additionally zooms out such that the actual cross
        section data graph uses the entire view.
    –   **History enabled**: Enables the drawing of multiple sections in the
        view which scrolling the different sections in the tree view. If this is
        switched off then always only one graph will appear in the view.
    –   **Font**: opens a separate standard Font settings dialog where font-
        type, -size and -colour can be modified.

## Graphical plot items selector

It is possible to present the graphical plot of any of the hydraulic parameters listed in the tabular view columns. From a selection box in the top of the graphical view the requested data-type for presentation in the plot can be selected as illustrated in Figure 3.32.
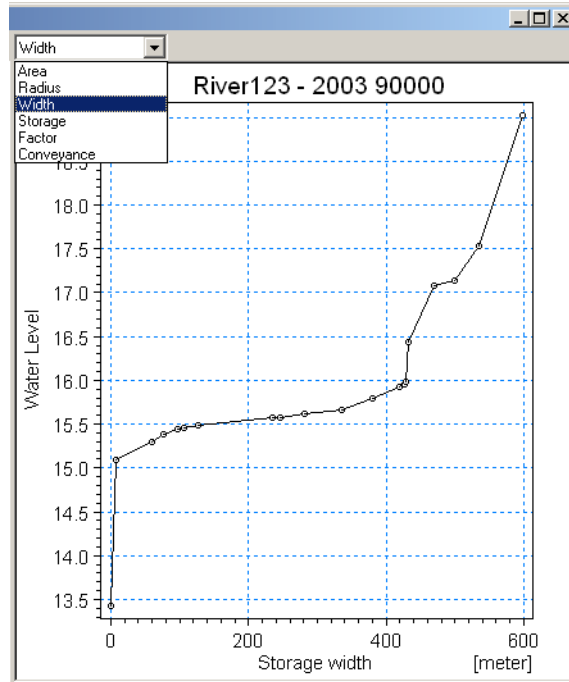


Figure 3.32    Drop-down selection box for plot of processed data types

## 3.2.4    Processed Data, Levels button

When activating the 'Levels...' button a 'Levels for Processed Data' dialog appears (Figure 3.33).

Figure 3.33    Processed data, Levels... dialog

The Levels dialog controls the method for calculating processed data in sections as well as user-defined indications for number of processed data levels as well as eventual definitions of minimum and maximum levels in the processed data.

### Level Selection Method

There are three methods by which the levels can be selected:

1. **Automatic:**
   The levels are selected automatically.
   In case Resistance Radius is applied in the Raw Data Editor, levels are selected according to variations in section flow width.
   In case Hydraulic Radius is applied, levels are selected according to variation in the section conveyance.

2. **Equidistant:**
   The levels are selected with equidistant level difference determined from the number of levels specified. If required, the Minimum level can be adjusted to control the bottom level of the processed data table. However, it is generally recommended to include the full cross section definition and hence, leave the Minimum and Maximum tick-boxes unchecked.

3. **User defined:**
   The levels can be fully or partially user-defined. The selected levels are entered to the levels table on the dialog. If the number of defined levels is less than required by the *Number of Levels* specification the remaining levels will be selected automatically.

### Minimum Level

The minimum calculation level to be presented as the first level in the processed data table. Default is the lowest point in the section defined in the Raw Data table.

### Maximum Level

The maximum calculation level to be presented in the processed data. However, it is strongly recommended not to apply this feature as it is important to have the cross section properly processed up to the highest point as defined in the Raw data.

### Number of Levels

The desired number of calculation levels. The automatic level selection method may not use the full number of level specified. This will occur when a fewer number of levels is sufficient to describe the variation of cross sectional parameters.

A minimum of two levels is required. There is no upper limit to the number of levels.

### Table of Levels

This section of the dialog is only applicable if the level selection method is user defined. The required levels are entered into the table manually. Levels can be added by pressing the Tab key while positioned at the bottom of the table. Levels can be deleted by selecting the row number and pressing the Delete key.

## 3.3 Importing Cross Sections using File Import

Via File –> Import (Figure 3.34) it is possible to read and import cross-section data (raw or processed) from a text file into MIKE 11.
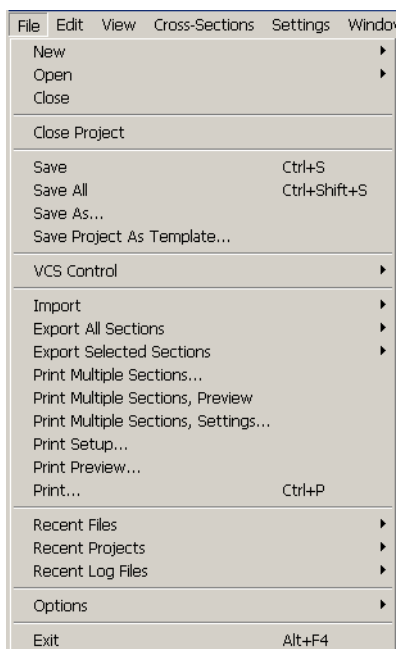
Figure 3.34    Cross section Main menu File dialog including options for Import, Export and Print of cross sections

This facility can be used to read cross-sectional data stored in an external data base format. The cross-sections are then imported via a temporary text file created as a medium between the external data base and the MIKE 11 data base.

From the text file MIKE 11 can load the data and change them to MIKE 11's internal data base format. The text file formats must correspond to one of two types, depending on whether raw or processed data is to be read.

## 3.3.1    Import Raw Data

Selecting File –> Import –> Import Raw Data it is possible to import raw data into MIKE 11's cross section data base. The File format must conform to the following format:

```
topo id
river name
                0.000
COORDINATES
     0
FLOW DIRECTION
     0
DATUM
      0.00
RADIUS TYPE
     0
DIVIDE X-Section
0
SECTION ID
      cross section ID
INTERPOLATED
     0
ANGLE
     0.00    0
PROFILE          n
     X(1)       Y(1)       r(1)       <#1>
     X(2)       Y(2)       r(2)       <#0>
      |          |          |          |
      |          |          |          |
     X(n-1)     Y(n-1)     r(n-1)     <#0>
     X(n)       Y(n)       r(n)       <#4>
*********************************
```

Figure 3.35   File format of ASCII file used for importing data into MIKE 11

In Figure 3.35 Topo ID is to be understood as the topological identification tag of the river. River name is self explanatory, the chainage should be entered in meters. The coordinates of the centrepoint of the cross section may be entered here for use in the network editor, if this is not required zero should be entered. The flow direction is set to one if the positive flow direction is to be entered else it is set to zero, again this is only for use if the information is to be imported into the network editor. The datum is entered in meters and the type of radius used is set.The DIVIDE X-section is either set to OFF (0) or to ON (1) if the latter is the case the level of divide should be entered in meters proceeding the switch indicator. The cross-sections topological identification tag follows. The section INTERPOLATED is set to OFF(0) or ON(1). If a correction angle of the cross section is to be used this may be entered here. After PROFILE the number of points (n) in the cross section should appear. Following this a table of values of X, Z, $r_r$ and markers are required.

    Xx-coordinate

    Zz-coordinate

    $r_r$relative resistance

The markers are set according to:

| | |
|---|---|
| **<#1>** | **marker 1** |
| <#2> | marker 2 |
| <#4> | marker 3 |
| <#8> | marker 4 |
| <#16> | marker 5 |

| | |
|---|---|
| **<#1>** | **marker 1** |
| <#32> | marker 6 |
| <#64> | marker 7 |

**Note** that if a point has two or more markers the number after # is found as a summation, for example: '<#6>' indicates that the point represents marker 2 and 3.

Each type of information in the ASCII file starts with an explanatory text line followed by one or more lines containing numerical information. This text line must start with three fixed characters, depending on the type of data:

- Horizontal coordinates

  Text line: COORDINATES
  Numerical line: 1    27.43    13.293

  where the first value is a coordinate data type indicator, which can be defined as either '0', '1' or '2'.

  The coordinate data type options are:
  "0": The rest of the line will be ignored.
  "1": The x- and y-coordinates will follow.
  "2": The $x_1$, $y_1$ and $x_2$, $y_2$ coordinates of the section ends follow

- **Positive current direction**

  Text line: FLOW DIRECTION

  Numerical line:  1    270

  where the first value is a flow direction type indicator, which can be defined as either '0' or'1'.

  The flow direction type options are:
  "0": The rest of the line will be ignored.
  "1": The direction will follow

- Datum adjustment

  Text line: DATUM

  Numerical line: (-)12.22

  The datum adjustment will be added to the z-coordinates from   the profile

- Closed section

  Text line: CLOSED SECTION.

  If this text line does not occur the section will be taken as open

- Radius formulation

  Text line: RADIUS TYPE

  Numerical line: 0

  where the numerical value is a radius type indicator, which can be defined as either '0', '1' or '2'.

  The radius type options are:
  0 - Resistance radius
  1 - Hydraulic radius using effective area
  2 - Hydraulic radius using total area

  The default is 0 (Resistance radius) except for closed sections, where the default is 2.

- x-z coordinates

  Text line: PROFILE

  Numerical lines: At least three lines containing corresponding values of x and z and optionally the relative resistance and/or markers. If the relative resistance is omitted 1.0 will be used. The x-values must always be increasing except for a closed section.

- End of a cross-section

  Text line: ***************************

Small or capital letters can be used. It is optional to specify the above information except the x-z coordinates (profile).

There are no limits on the number of cross-sections allowed in the text file. Numbers can be entered in a 'free format'; i.e. with any number of decimal places.

If there is an error in the text file, the loading will be terminated and information will be given regarding the erroneous line.

If data for a particular cross-section already exists in the data base, the data in the text file will be ignored.

Selecting File –> Import –> Import Raw Data and Recompute it is possible to import raw data into MIKE 11's cross section data base and recompute the processed data automatically.

## 3.3.2    Import Processed Data

Selecting File –> Import –> Import Processed Data it is possible to import processed data into MIKE 11's cross section data base. The configuration of a text file containing processed data must conform to the following format:

```
Topo-ID
River name
        chainage
COORDINATES
    0
FLOW DIRECTION
    0
PROTECT DATA
    0
PROCESSED DATA
    Level       Cross sec   Hydraulic    Width      Add. fl.      Resist.
    (m)         area  (m2)  radius (m)    (m)        areas (m2)    factor
    L(1)        A(1)        R(1)         W(1)        Afl(1)        Rf(1)
    L(2)        A(2)        R(2)         W(2)        Afl(2)        Rf(2)
     -           -           -           -            -             -
     -           -           -           -            -             -
     -           -           -           -            -             -
     -           -           -           -            -             -
     -           -           -           -            -             -
    L(M)        A(M)        R(M)         W(M)        Afl(M)        Rf(M)
********************************
```

Figure 3.36    Format used for importing processed data.

The first three lines are the same as for raw data: Topo ID, River Name and River Chainage. As for raw data format, it is hereinafter possible to specify information about:

- horizontal coordinates (as for raw data)

- positive current direction (as for raw data)

- Protection of Data
  If this is set to '1', then the processed data will be imported with the 'Protect Data' option enabled.

- processed data
  The explanatory text line (see raw data) initiating the processed data must start with PROCESSED DATA.
  After this line, two text lines (headings), followed by M number of lines with individual levels and the hydraulic parameters defined.
  The processed data for each cross-section must finish up with a line containing: *********.

Selecting File –> Import –> Import and overwrite Processed Data it is possible to import processed data into MIKE 11's cross section data base and overwrite the existing processed data. This facility is often used if for example additional storage areas have been added to the processed data and these data are copied into another data base.

### 3.3.3 Import Coordinates of Levee Marks

Selecting File -> Import -> Import Coordinates of Levee Marks it is possible to import X, and Y coordinates for right and left levees into MIKE 11's cross section database.

The format of the ASCII text-file containing Levee marks coordinates is: River Name, Topo ID, Chainage, Left X, Right X, Left Y, Right Y (items can be divided by 2 or more spaces or 1 or more tabs). One line for each series of coordinates.

Example:

Donau<Tab>2005<Tab>0.00<Tab>LeftX1<Tab>RightX1<Tab>LeftY1<Tab>RightY1
Donau<Tab>2005<Tab>750.00<Tab>LeftX2<Tab>RightX2<Tab>LeftY2<Tab>RightY2
Donau<Tab>2005<Tab>2000.00<Tab>LeftX3<Tab>RightX3<Tab>LeftY3<Tab>RightY3
--
--

## 3.4 Exporting Cross Sections using File Export

Via File –> Export (Figure 3.34) it is possible to export cross-section data (raw or processed) from the MIKE 11 data base and write these to a text file.

Export of cross sections can be done either for all sections through '*Export All Sections*', which will export the entire content of cross section file to the text-file, or for Selected cross sections only ('*Export Selected Sections*').

There are three possibilities:

1. Export All...:
   Both raw and Processed data is exported to the same text file.

2. Export Raw...:
   Only the raw data is exported to a text file.

3. Export Processed...:
   Only processed data is exported to a text file.

## 3.5    Plotting Multiple Cross Sections

In addition to printing the actual content of the graphical view of the raw data dialog using File->Print... a feature for multiple cross sections plots is available.

To use this feature make sure that:

1.   One or more cross section is selected.
2.   Click in the graphical view (such that in comes in focus).

Now, one of following items in the File menu relating to multiple cross section plotting becomes available:

### Print Multiple Cross Sections...
If the output device is selected as the printer in the settings dialog, this will open the print dialog. If a metafile is selected as output cross sections plots will generated in metafiles and no dialog will appear.

### Print Multiple Cross Sections, Preview
This will open a preview dialog allowing the user to inspect the result of the settings or to view the cross sections on screen rather than in hard copy. Using <Page Up> and <Page Down> will jump to the next and previous page with multiple cross section plots.

### Print Multiple Cross Sections, Settings...
This will open the dialog with settings for the multiple cross section plotting, see Figure 3.37.

Figure 3.37    Dialog with settings for multiple cross section plotting

The settings dialog allows for controlling the following:

### Nb. of plots on each page.

Each page is composed by number of individual cross section plots ordered in rows and columns. The user specifies the number of plots in the vertical and the horizontal direction.

### Margins

The horizontal and vertical margins and the horizontal and vertical distances between the plots can be controlled in this section.

### Horizontal and vertical scale options

The horizontal and vertical scale options are equivalent. In the following the horizontal scale options are explained.

**Automatic and individual on each section**: The minimum and maximum of the axis is selected automatically corresponding to the minimum and maximum values in each data set.

**Fixed for all sections**: User defined values for minimum and maximum of the axis will be applied for all cross section plots.

**Automatic minimum and fixed width**: All cross sections will be plotted on an axis with the same width (maximum minus minimum). The minimum value of the vertical axis will change for each plot according to the minimum value in the data set.

**Fixed scale**: The scaling of the axis will be selected according to the user defined ratio between the physical cross section size and the printed size. I.e. all cross sections will be plotted on an axis with the same width (maximum minus minimum). The minimum of the scale can be controlled as either a fixed offset below the data minimum for each cross section or as a fixed value applied for all cross sections.

## Design profile

Each plot will normally contain one cross section data set. However, selecting this option it is possible to have another data set from the same location (river name and chainage) drawn in each plot. The Topo ID for the second cross section to be drawn is specified by the user. The second cross section will be drawn using the graphical settings for "passive" cross sections, and a legend for both cross section lines can optionally be drawn.

## Output

The plots can either be routed to the printer or saved as meta graphics in a number of metafiles. Each metafile can only contain one page. I.e. the number of selected cross sections requires more than one page several metafiles will be written. The filenames are generated automatically by adding _01, _02, _03 and so on to the filename specified by the user.

BOUNDARY EDITOR

# 4 Boundary Editor

The boundary editor is used to specify boundary conditions to a MIKE 11 Model. It is used not only to specify common boundary conditions such as water levels and inflow hydrographs but also for the specification of lateral flows along river reaches, solute concentrations of the inflow hydrographs, various meteorological data and certain boundary conditions used in connection with structures applied in a MIKE 11 model.

## 4.1 Overview of the Boundary Editor

Figure 4.1 shows the initial layout of the boundary editor when opened for the first time in an application. It consists of three split windows.

The top split window contains the overall details of boundary conditions defined in the model. Each boundary condition is defined as one row in the Boundary Table and the table therefore, contains all boundaries included in a model set up. There is no limit to the number of boundaries (= table rows) that can be included in a model.

The view and contents of the second and the third split window depends on the specifications of the selected boundary identified by the highlighted row in the upper window (active row is highlighted) in the Boundary Table. Additional information needed to specify the boundary conditions are entered in the second and third split windows.
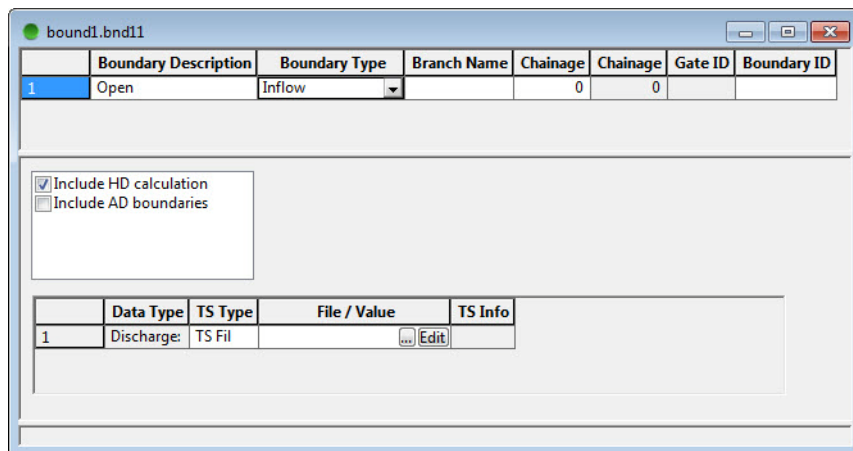


Figure 4.1 Layout of the boundary file when opened for the first time in an application. Note that the third split window is empty

## 4.1.1    The Boundary Table - Upper Split Window

The Boundary Table shown in the first split window gives an overview of the boundaries defined in the model set up.

Each boundary definition requires the following details:

- *Boundary Description*,
- *Boundary Type,*and
- *Boundary Location (River Name and Chainage)*.
- In addition a *Boundary ID* can be entered, although this is optional. Specifying an ID can be convenient for identifying the boundary, but it has no effect on the calculation.

Select the actual *Boundary Description* and the *Boundary Type* by placing the cursor at the right end of the edit field in question and left-clicking the mouse. A drop-down list appears from which the appropriate boundary type can be selected, see Figure 4.2.

| | Boundary Description | Boundary Type | Branch Name | Chainage | Chainage | Gate ID | Boundary ID |
|---|---|---|---|---|---|---|---|
| 1 | Open | Inflow | | 0 | 0 | | |
| | Open | | | | | | |
| | Point Source | | | | | | |
| | Distributed Source | | | | | | |
| | Global | | | | | | |
| | Structures | | | | | | |
| | Closed | | | | | | |

Figure 4.2    Drop down list for selection of the Boundary Description

To insert new boundaries (= rows in the Boundary Table in the first split window), press the "Insert" button on the keyboard or use the <Tab> key. A boundary (row) can be moved up or down in the table by selecting a row (clicking in the left column) and dragging to the desired location. It is also possible to sort the boundaries alphabetically by double clicking the column headers. This operates for all the column headers in the top split window.

*Boundary Description* describes the nature of the boundary (see Figure 4.3).

There are six different types of Boundary Description:

- Open;
- Point Source;
- Distributed Source,
- Global;
- Structures;
- Closed.

The boundary description options are explained in detail below.

***Boundary Type*** specifies the kind of data required for the boundary. For each *Boundary Description* there are a number of valid *Boundary Types*. Once a *Boundary Description* has been selected only the valid choices of *Boundary Type* are displayed.

There are a total of 26 possible combinations of *Boundary Description* and *Boundary Type*, as shown in Figure 4.3.



Figure 4.3     Possible combinations of Boundary Description and Boundary Type

## 4.1.2    Specifying the Boundary Description

A description of the options available in the *Boundary Description* column of the *Boundary Table* is given in the following.

### The Open Boundary
An *Open* Boundary can be specified at the free upstream and downstream ends of the model domain. When the *Open* option is selected in a *Boundary Description* cell, a branch name and chainage are also needed in order to

identify the location of the boundary. An Open boundary condition has the fol-
lowing valid *Boundary Types*:

- *Inflow* is specified when a time-varying or constant flow hydrograph con-
dition is required (for the HD model). An inflow boundary can be defined
with or without a solute component (for the AD model);

- *Water Level* is specified when a time-varying or constant water level con-
dition is required (for the HD model). Water level boundaries can be
defined with or without a solute component (for the AD model);

- *Q-h* is specified when the relationship between the discharge and the
water level is known (HD model) and used with or without a solute com-
ponent (the AD model);

- *Bottom Level* is specified for ST models, where the variation of the bot-
tom (river bed) level is required as a constant value or a function of time;

- *Sediment Transport* is specified for ST models, when a variation of the
inflow of sediment is required either as a constant value or as a function
of time;

- *Sediment Supply* is specified for ST models, when neither the bottom
level nor the sediment transport is known. Instead the inflow of sediment
is computed by the sediment transport engine and set equal to the sedi-
ment transport capacity. No other information is needed for this type of
boundary.

## The Point Source Boundary

The *Point Source* boundary condition is used at locations within the model
domain where time-varying or constant lateral inflows (or outflows) occur.
When the Boundary Description is selected as *Point Source* a branch name
and a chainage are required to identify the location.

A *Point Source* Boundary Description has the following valid types of Bound-
ary Type:

- *Inflow* is specified when a time-varying or constant lateral inflow condi-
tion (for the HD model) is required with or without a solute component
(for the AD model);

- *Sediment Transport* is specified for ST models, when a variation of the
lateral inflow of sediment is required as a function of time.

**Please Note:** Point sources takes the sign of a hydrograph into account such
that positive numbers for a boundary hydrograph is considered an inflow to
the model, whereas negative discharges are considered as abstractions of
water from the model.

## The Distributed Source Boundary

The *Distributed Source* boundary condition is used along river reaches within the model domain where time-varying or constant lateral distributed inflows (or outflows) need to be specified, or where meteorological boundaries apply.

When the Boundary Description is selected as *Distributed Source* a branch name and two chainages need to be specified. The two chainages represent the up- and downstream ends of the river reach along which the distributed boundary applies. The order in which the chainages are specified is not important. The *Distributed Source* Boundary Description allows the following Boundary Types:

- *Inflow* is specified when a time-varying or constant lateral inflow condition (for the HD model) is required with or without a solute component (for the AD model). The inflow will be divided equally between each computational h-point lying in the specified chainage range;

- *Evaporation* is specified in river reaches where loss of water by evaporation affects the water balance (HD model). *Evaporation* can also be specified globally.

- *Rainfall* is specified in river reaches where the inflow of rainfall affects the water balance and where any rain borne components affect the AD modelling. *Rainfall* can also be specified globally.

- *Heat Balance* is specified when the advanced heat balance module is activated. Three different boundaries must be specified: The temperature, the relative humidity and the solar radiation. *Heat Balance* can also be specified globally.

- *Resistance factor* is specified when a time varying resistance factor applies along a river reach. *Resistance Factor* can also be specified globally.

- *Wind Field* is specified when wind induced stress on the surface needs to be accounted for. Two boundaries must be specified: The wind velocity and the wind direction. The direction of the wind is in degrees in clock wise direction from north, see Figure 4.4. Inclusion of wind shear stress in the computation is specified in the Hydrodynamic Parameter file (.HD11). The user can reduce the effect of the wind shear stress by applying a topographical wind factor in certain reaches in the .HD11 file. *Wind Field* can also be specified globally.
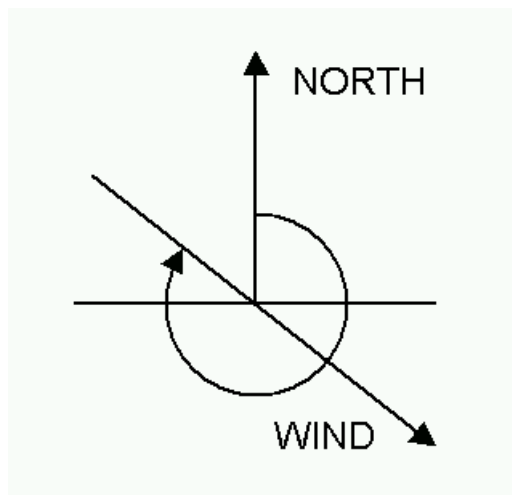
Figure 4.4     Definition of wind direction

## The Global Boundary

The *Global* boundary condition is applied when a certain boundary conditions are valid over the entire model domain. In such cases it is not necessary to specify any location.

The valid *Boundary Types* are: Evaporation; Rainfall; Heat Balance; Resistance Factor and Wind Field. These Boundary Types are used in the same manner as *Distributed Sources*.

It is possible to specify both a globally applicable boundary condition and a distributed boundary condition of the same Boundary Type. The global boundary will then be applied over the entire model area - except at those locations where distributed boundaries have been specified.

Figure 4.5 shows an example in which both global and local wind boundaries are applied. The globally defined wind stress will be applied all over the model except in the branch Main between chainage 0 - 10000 where a different time series (wind speed and direction) has been applied



| | Boundary Description | Boundary Type | Branch Name | Chainage | Chainage | Gate ID | Boundary ID |
|---|---|---|---|---|---|---|---|
| 1 | Distributed Source | Wind field | Main | 0 | 10000 | | |
| 2 | Global | Wind field | | 0 | 0 | | |

| | Data Type | TS Type | File / Value | | TS Info |
|---|---|---|---|---|---|
| 1 | Wind Velocity:  [m/s] | TS File | TS-Input.dfs0 | ... Edit | Wind vel |
| 2 | Wind Direction:  [degree] | TS File | TS-Input.dfs0 | ... Edit | Wind |

Figure 4.5     Example of the application of both global and local boundary conditions.

*EU functions*. This Boundary Type can only be used as a global boundary and is used for eutrophication models. Two boundaries must be specified: Temperature and solar radiation.

## The Structures Boundary

The *Structures* boundary condition can be used in combination with three different Boundary Types:

- *Dam* is specified when a discharge time series must be applied at the end of a stratified branch (MIKE Reservoir model). Besides the discharge boundary it is also necessary to specify the level, width and height of the extraction point. (This is done in the third split window.)

- *Dam Break* is specified for time varying conditions in connection with a dam break. Three items must be defined for such boundaries: The Dam Breach Level, the Dam Breach Width and the Dam Breach Slope;

- *Regulating Structure*, is specified to describe the discharge at a regulating structure.

- *NWS DAMBRK Piping*, is specified to describe the temporal development of a dam breach using the NWS piping breach equation. The second dam breach level (fourth parameter) to be specified is the top level of the pipe. Once the pipe collapses the top level is neglected in the simulation and the three remaining parameters will be used for describing the trapezoidal breach.

For all four types the location must by specified by giving a branch name and a chainage. For the Dam Break, NWS DAMBRK Piping and the Regulating structure, it is also **required to specify the GATE ID** which corresponds to the structure ID as defined in the Network file.

Please note also, that the temporal development of a **Dambreak** must be specified in a time series file with **Relative Time axis** (equidistant or non-equidistant time axis can both be applied). Further please ensure that the extent of this file covers the full the simulation period.

When using the NWS methods the terminal breach level is defined by the last value in the corresponding time series.

## The Closed Boundary

The Boundary Description *Closed* is used at free ends points of the model domain where a zero flux condition across the boundary is applicable. It can be used for HD, AD and ST simulations. For the HD model it corresponds to a zero discharge boundary and for AD and ST models it corresponds to zero transport across the boundary. No additional information is required except the location described by branch name and a chainage.

## 4.1.3 Specifying the Boundary Type, Data Type and File/Values

In this section the specification of the *Boundary Type* and associated Data Types are described. This additional information is given in the second and third split windows.

The content of the second split window depends on the combination of Boundary Description and Boundary Type given in the highlighted row in the Boundary Table in the upper split window. The basic purpose of the second split window is to specify the necessary boundary conditions and in some cases select whether information should be specified for additional modules, e.g. AD boundaries, AD-RR links etc.

The content of the third split window will again depend on the specifications given in the second split window. The third split window deals primarily with boundaries for the AD and ST modules.

Boundary conditions can be specified as either a time series (TS Type) or constant values. If AD components are required, the user can choose between additional data types as: Concentration, Bacteria Concentration, Salinity, Temperature and Undefined.

If a constant boundary is specified under *TS Type* in the second split window, the user can select the type of data from a drop-down list in the *Data Type* column. If a time series is specified, the corresponding *Data Type* field cannot be edited, but will be updated based on the data type of the actual time series selected.

When the *Boundary Description* is **Closed** or the *Boundary Type* is S*ediment Supply* no additional data is required.

### The Inflow Boundary

*Open Inflow Boundary*
Open inflow boundaries are used to specify inflows at free branch ends (boundaries of the model domain) for HD, and AD simulations. The layout for the Inflow boundary for Open Boundaries is shown in Figure 4.6. (Note the AD-RR option is not available for open boundaries). The three check boxes available are:

- Include HD Calculation.
  This box must be checked if the discharge time series is to be included in the water balance in the HD calculation.

- Include AD calculation.
  This box must be checked if the discharge is to be used with a concentration to compute the mass inflow of a component in an AD simulation. When checked, the associated concentrations are entered in the third split window.
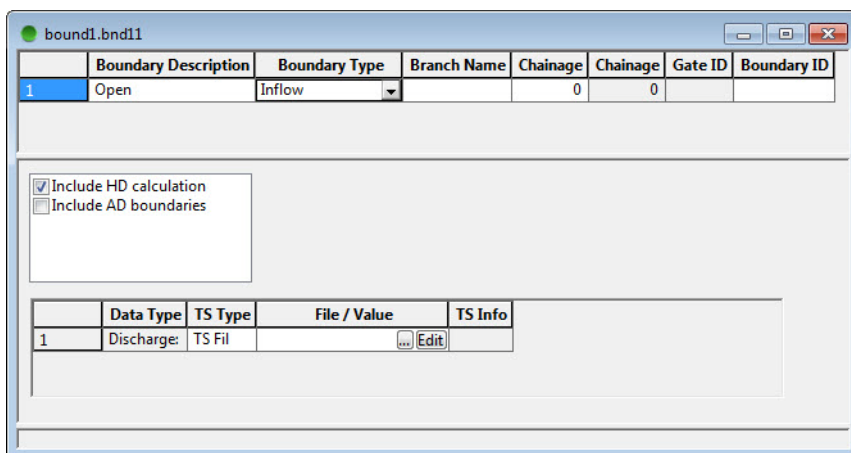
Figure 4.6     Specification of a discharge at an open inflow boundary

Figure 4.6 shows the specification of a simple discharge boundary for a HD model. The discharge (either constant or a time series) is specified in the second split window.

If the 'Include AD calculation' box is checked, additional information is needed in the second split window, see Figure 4.7. This information describes how the AD boundary condition should be processed during the simulation. There are three possibilities:

- *TS-Defined*
  This option is used when the specified concentration defined as the boundary condition (constant or time-varying) shall be applied directly as the concentration value at the boundary node.

- *Open, Concentration*.
  This option is used at locations where outflow from the model area takes place. When an outflow boundary becomes an inflow boundary during a model simulation (e.g. due to tidal conditions) the boundary condition is adjusted according to:

$$C = C_{bf} + (C_{out} - C_{bf})e^{-t_{mix}k_{mix}}$$

Where $C_{bf}$ is the concentration specified at the boundary location, $C_{out}$ is the computed concentration at the boundary immediately before the flow direction changed, $K_{mix}$ is a time scale specified in the input and $t_{mix}$ is the time since the flow direction changed.

When outflow occurs the boundary conditions is defined as

$$\frac{\partial^2 C}{\partial x^2} = 0$$

- *Open, Transport*.
  This type of boundary should be used where only inflow takes place. The transport into the model area is computed using the specified boundary concentration and the discharge computed by the HD model and hence, a well-defined amount (mass) of pollutant is introduced into the model. Consequently, the computed concentration in the boundary node can differ from the concentration specified in the boundary file. It may be used where appreciable storage and hence dilution of the inflow can take place close to the boundary.



Figure 4.7    Specification of a boundary for a combined HD and AD simulation

*Point Source or Distributed Source Inflow Boundary*

Point or distributed source inflow boundaries are used to describe lateral inflows for HD and AD simulations. Figure 4.10 shows the layout of the boundary file for the *Inflow* Boundary.

The second split window is similar to that displayed for Open inflow boundaries, with one additional facility:

- AD-RR.
  If this check box is checked AD components can be included with the inflow generated by the rainfall runoff models integrated in MIKE 11.

In Figure 4.8 only the 'Include HD calculation' box is checked. It thus represents a standard lateral inflow used in a HD simulation. Only the discharge need be specified, as either a constant value or a time series.
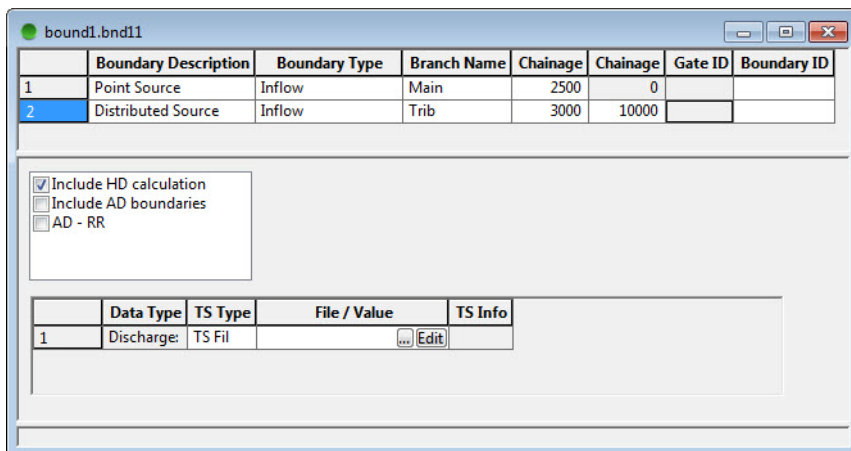
Figure 4.8    Specification of a point source lateral inflow for a HD simulation

If the 'Include AD calculation' box is also checked then the third split window becomes editable and boundaries for the different AD components can be entered (see Figure 4.9).The discharge specified in the second split window is used both in the water balance and in the AD calculation. In the AD calculation it is multiplied with the concentrations in order to calculate the mass inflow for the different components.

**Note** that if only an AD simulation was to be computed (based on a previous HD simulation), the "Include HD Boundaries" would be turned off. However the discharge would still need to be specified in order to compute the mass inflow of the components to the AD model.

If the Boundary Description was changed to Distributed Source and a second chainage were specified in the first split window, this boundary would also be valid for a distributed inflow.

Figure 4.9    Specification of a point source boundary for both HD and AD simulations

In the second split window, a discharge time series must be specified together with the level at which the inflow occurs.

Figure 4.10 shows the layout of the boundary file if we now uncheck the 'Include HD calculation' box. It is actually the same layout as if the 'Include HD calculation' box is left checked. This is because information on the discharge is still required in order to calculate the mass flux of AD components into the river branch. The only difference is that the discharge is no longer used in the water balance.
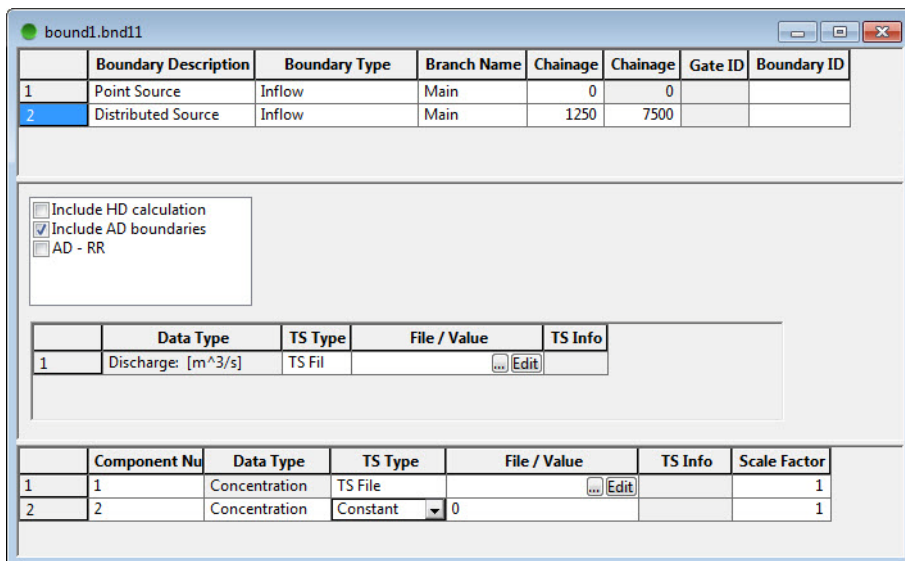
Figure 4.10    Specification of a Point Source inflow boundary for an AD simulation. The discharge is not included in the water balance when the 'Include HD calculation' is left unchecked

In Figure 4.11 the 'AD-RR' box is checked. This facility can be used where the concentration of rainfall or runoff from a NAM model are to be used in an AD simulation. All other check boxes are now invisible. Instead a section where information on the Catchment Name, Catchment Area and Runoff Type must be specified are now present.

The Catchment Name must refer to a catchment included in the MIKE 11 Rainfall Runoff model set up. The area value is used as a scaling factor meaning that runoff calculated by the rainfall runoff model is scaled *pro rata* against the catchment area specified in the rainfall-runoff editor.

The runoff type must be selected between Total Runoff, Surface Runoff, Root Zone Runoff, Groundwater Runoff or Rainfall. Note that Interflow is only available for the NAM model. Also note that the Rainfall Runoff model must run in parallel with the HD and AD models for the AD-RR facility to operate.

Figure 4.11    Specification of rainfall or runoff for input to an AD simulation. The second split window now contains information on catchment name, catchment area and runoff type

## The Water Level Boundary

The Water level boundary is valid in connection with an Open boundary. Note the "Include HD Calculation" box is not visible, as this is not an option, see Figure 4.12.The boundary is specified as either a time series or a constant in the lower window.



Figure 4.12    Specification of a simple water level boundary for a HD simulation

Where an AD simulation is to be carried out in parallel to a HD simulation, the "Include AD Boundaries" box should be checked as shown in Figure 4.13. The water level boundary is specified in the second split window together with information on the AD boundary type. An open, concentration boundary type is used because outflow occurs at the downstream end. AD boundaries are specified in the third split window.



| | Boundary Description | Boundary Type | Branch Name | Chainage | Chainage | Gate ID | Boundary ID |
|---|---|---|---|---|---|---|---|
| 1 | Open | Water Level | Main | 12500 | 0 | | |

☑ Include AD boundaries

| | Data Type | TS Type | File / Value | TS Info | AD boundaries | K-mix |
|---|---|---|---|---|---|---|
| 1 | Water Leve | TS Fil | TS-Input.dfs0 [...] [Edit] | Levels | open, concentratio | 0 |

| | Component Nu | Data Type | TS Type | File / Value | TS Info | Scale Factor |
|---|---|---|---|---|---|---|
| 1 | 1 | Concentration | Constant | 5 | | 1 |
| 2 | 2 | Concentration | Constant | 14 | | 1 |
| 3 | 3 | Concentration | TS File | TS-Input.dfs0 [...] [Edit] | Conc | 1 |

Figure 4.13    Specification of a water level boundary for a combined HD-AD simulation. Open concentration boundaries with a suitable $k_{mix}$ value are used as this is an outflow boundary

## The Q-h Boundary

Q-h (discharge-water level relation or rating curve) boundaries can be applied at Open boundaries only, and are only recommended to apply at the downstream end of a model domain, where water flows out of the model. If a Q-h relation is selected, the user will be presented with the display as shown in Figure 4.14.
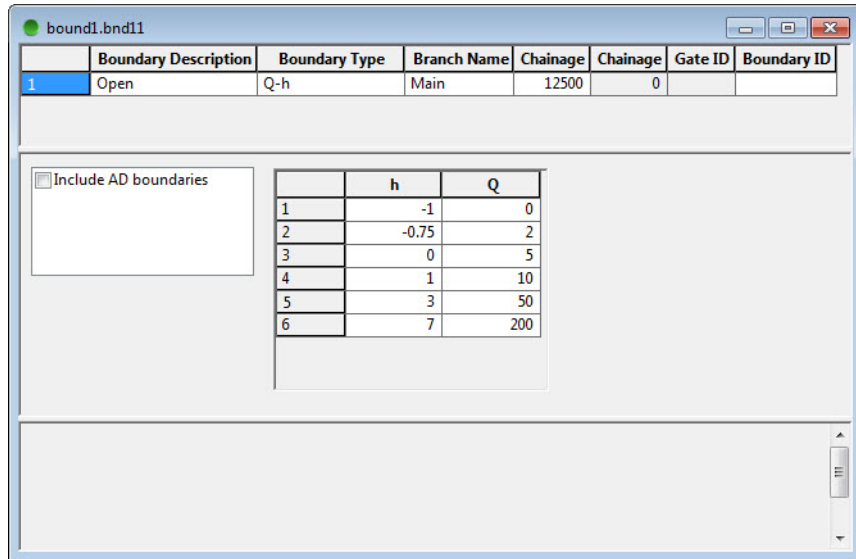
Figure 4.14    Specification of a Q-H relation at an Open boundary

The Q-h relation is given in the table in the second split window. The Q-h relation can either be entered from a known rating curve (e.g. copied and pasted from Excel) or automatically generated by selecting *Tools -> 'Auto Calculation of Q/h Table...'* in the top menu bar. If this later option is selected a new dialog appears, see Figure 4.15.
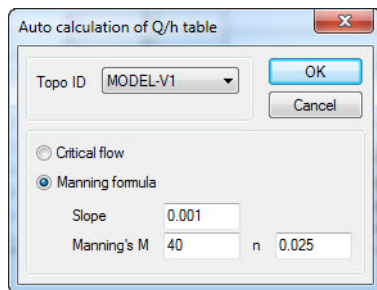


Figure 4.15    Dialog used when making automatic calculation of Q-h relation

The Q/h-relation tool calculates a Q/h-relation from the flow characteristics of the specific cross section located at the boundary location and hence, it is required to select the TOPO ID of the cross section to be used in the Q/h-calculation.

H-values in the Q/h-relation is extracted from the processed data of the actual cross section and related Q-values can be calculated from one of two assumptions; Critical flow or Uniform flow using the Manning equation. If the latter is chosen the bed slope and Manning's "n" or "M" must be specified.

Both methods will take H-values from the processed data of the specific cross section (Levels column) and Q-values is calculated from either Equation (4.1) or Equation (4.2).

In case of Critical Flow, Q is calculated from:

$$Q(h) = A(h)\sqrt{g\frac{A(h)}{W(h)}} \qquad\qquad (4.1)$$

In case of Uniform flow by Manning's formula, Q is calculated from:

$$Q(h) = Conv(h)\sqrt{I} \qquad\qquad (4.2)$$

where:

Q(h) the level depending Discharge

A(h) the level depending Area (from Cross section processed data)

W(h) the level depending Width (from Cross section processed data)

I the bed slope

Conv(h) is the level depending Conveyance calculated as a function of the resistance type definition in cross section raw data:

If raw data Resistance type = Relative Resistance:

$$Conv(h) = M_{QhTool} \cdot A(h) \cdot R(h)^{\frac{2}{3}}$$

If raw data Resistance type = Manning's M:

$$Conv(h) = M(h) \cdot A(h) \cdot R(h)^{\frac{2}{3}}$$

If raw data Resistance type = Manning's n:

$$Conv(h) = \frac{1}{n(h)} \cdot A(h) \cdot R^{\frac{2}{3}}$$

If raw data Resistance type = Chezy or Darcy Weissbach:

$$Conv(h) = C(h) \cdot A(h) \cdot \sqrt{R(h)}$$

where:

$M_{QhTool}$ is the Manning number defined in the Q/h tool dialog,

M(h), n(h) and C(h) are the respective Resistance number definitions extracted from the resistance factor column in the in the cross section processed data view.

As data from the specific cross section is required for calculating the Q/h-relation, this facility is only available when the simulation editor is open and the paths to the cross section and boundary files have been specified.

**Please note** that the location defined for the boundary and the chainage of the cross section used to calculate the Q/h-relation must be within 1mm distance of each other. If not, the Q/h-calculation tool can not find a relevant cross section and no calculations takes place.

AD boundaries may be specified together with the Q-h relation. If this is done (as shown in Figure 4.16) the user must specify the type of AD boundary (typically Open, Concentration for a downstream boundary) and enter information on the boundaries in the lower window.



Figure 4.16    Specification of a Q-h boundary to be used in a combined AD-HD simulation

## Dam Break Boundary

Dam Break boundaries need to be specified when a dam break structure with a time-dependent breach formation in defined in the river network file. The specification of the boundaries is quite simple as illustrated in Figure 4.17.

The second split window indicates that three time series must be specified: Dam Breach Level; Dam Breach Width; and Dam Breach Slope.

**Important notice concerning GATE ID:** For Dambreak structures (and Regulating structures) which utilises a boundary description to control the characteristics of the structure, it is Required to specify the 'GATE ID' in the boundary file. 'GATE ID' must be identical to the defined STRUCTURE ID as defined in the Network file for the individual structures. Hence, it is also required to define a Structure ID for all structures defined in the network file which utilises a boundary condition as a control definition!
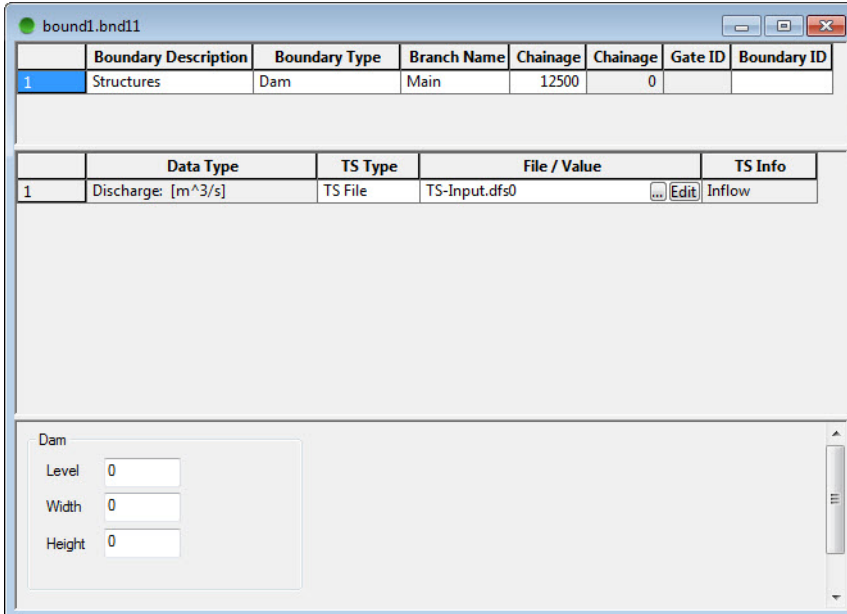


Figure 4.17    Specification of a Dam Break boundary. Note the absence of the third split window, which is not necessary for this, and the other combinations of Boundary Description and Boundary Type shown

### Dam Boundary

Dam Boundaries are used in connection with stratified branches (MIKE Reservoir model) when extraction from the dam needs to be specified. The discharge value or time series is given in the second split window, while the Level, Width and Height of the discharge point (extraction) should be given in the third split window, see Figure 4.18.

Figure 4.18    Specification of a Dam Boundary for a stratified branch (MIKE Reservoir model). Discharge is specified in the second split window and the geometrical data of the extraction point is specified in the third split window

## Rainfall and Evaporation Boundary

Figure 4.19 shows the layout for a Rainfall boundary, which can either be applied globally or as a distributed source. When applying a rainfall boundary to a HD computation, the rainfall is converted to a lateral discharge by multiplying the rainfall depth with the actual flooded area associated with each computational water level point. The actual flooded area is in turn computed during the simulation from the current cross section storage width (and if applicable, additional flooded area) and the cross section spacing.

If an evaporation time series is specified, the lateral "inflow" will be negative, i.e. an outflow. Note that if a NAM result is used as input to a HD simulation, the incorporation of rainfall and evaporation is handled automatically without the need for a separate boundary file.

If the rainfall input is to be used as a source for an AD model, the "Include AD boundaries" should be checked. In this case the concentration of components in the rainfall are specified in the third split window. Component numbers must match those in the AD parameter file. A boundary for component number '0' will be applied to all components if not otherwise specified.
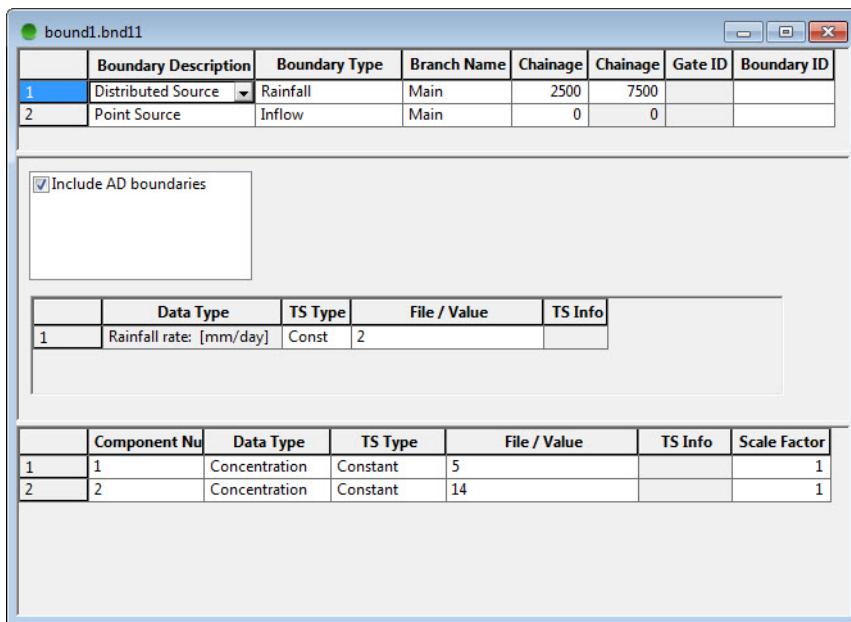
Figure 4.19    The layout of the boundary file when the Boundary Type is chosen to be rainfall. The second split window now contains a check box used to specify if AD components should be included. AD components must be specified in the third split window

### Sediment Transport Boundary

Figure 4.20 and Figure 4.21 show the layout of the boundary file for Sediment Transport, which can be specified for either an open boundary or a point source.

The second split window holds information on the time series needed and also requires information about the type of sediment transport included in the computations: single or multiple (graded) sediment fractions.

In Figure 4.20, data for only one sediment fraction is needed and the relevant time series is assigned in the second split window.
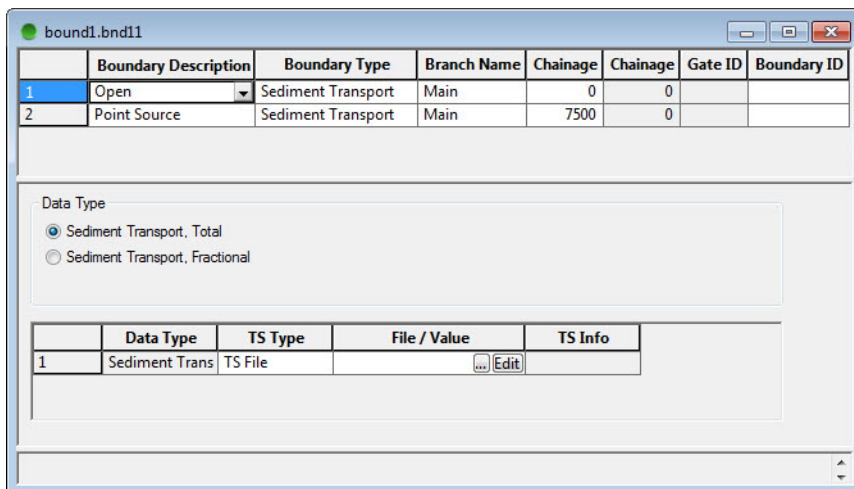
Figure 4.20    Specification of a sediment inflow boundary for a single sediment frac-
tion (total transport)

Figure 4.21 shows the layout of the boundary file for a graded sediment
model boundary. The third split window now prompts the user for fraction
numbers. The fraction numbers refer to the different fractions defined in the
Sediment Transport Editor.

Sediment inflow boundaries (either total or graded sediment) can also be
specified as point source inflows.
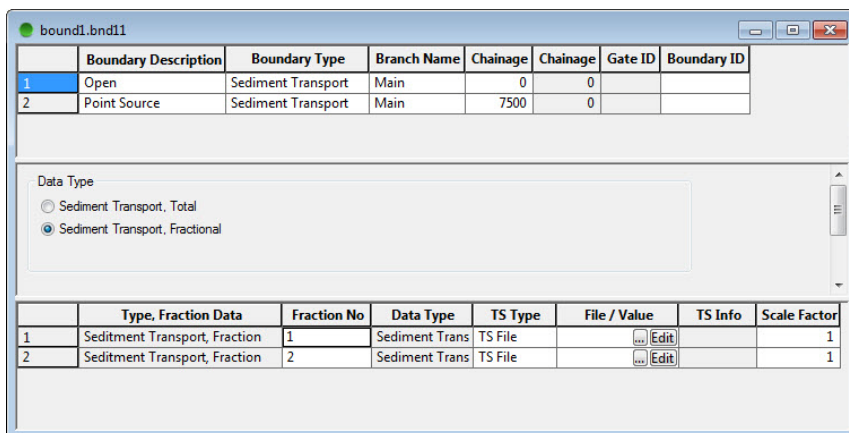


Figure 4.21    Specification of a sediment inflow boundary for graded sediment trans-
port

### The Bottom level Boundary

Bottom level (river bed level) boundaries can be specified in conjunction with the sediment transport model, at *Open* boundaries only. Figure 4.22 shows the layout for a Bottom Level Boundary. In the second split window the user specifies whether the boundary data are the absolute bottom level or the change in bottom level. The time series for both types are specified in the second split window. If the data is selected as change in bottom level the absolute bottom level is calculated during the simulation based its initial value.

If the Bottom Level Boundary Type is used in combination with a graded sediment model it is necessary to specify the relative amount of the different sediment fractions. This is done in the third split window. As indicated in Figure 4.8 it is possible to select between Fraction Value or Change in Fraction Value. This is done in the 'Type, Fraction data' column. If the "Change in Fraction Value" is set to zero, the initial distribution of the sediment fractions will apply throughout the entire simulation.



Figure 4.22    Specification of a Bottom Level boundary for ST simulations. The user selects whether the data are interpreted as absolute bottom level or change in bottom level. If the graded sediment model is used, data for the individual fractions are to be specified in the third split window

## 4.2    Boundary Editor Tools

The boundary editor includes a number of tools to assist the user in setting up complex model boundaries and quickly modifying the time series inputs.

## 4.2.1 Quick set up of Graded Sediment Boundaries

A quick method to set up boundaries for graded sediment models is to use the tool 'Make List of Fractions'. This tool is found under "Tools" in the top menu bar and is available when the lower split window is active. If the *Boundary Type* is chosen as Sediment Transport a dialog will appear, see Figure 4.23. This dialog can be used to specify several boundaries simultaneously. The boundaries can be either constant values or time series, depending on the selection made in the TS Type edit field.

If constant boundaries are chosen the number of fractions should be entered together with the value of the constant boundary in the 'Nb. Of Fractions' and 'File/Value' edit fields, respectively. If time varying boundaries are requested a time series file should be selected. When the OK button is pressed all legal time series items (time series with the requested data type) in the time series file will be inserted as boundaries in the third split window. The first legal time series will be applied for fraction number 1, the second for fraction number 2, etc.
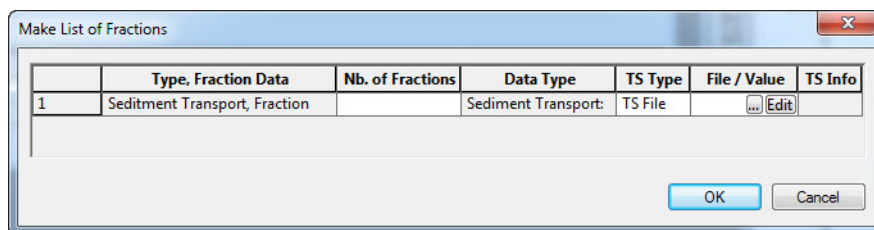


Figure 4.23    Dialog for quick specification of graded sediment inflow boundaries

If the "Make List of Fractions" Tool is used for Boundary Types equal to Bottom Level the dialog is slightly changed, see Figure 4.24. Now the user must specify if the boundaries should be 'Fraction Value' or 'Change in Fraction Value'.
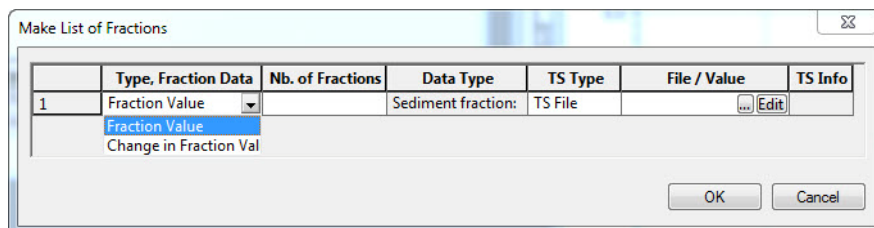


Figure 4.24    Dialog for quick specification of graded sediment bottom level boundaries

## 4.2.2    Quick set up of AD Boundaries

AD boundaries can be specified in a similar way to that described for graded sediments above. Make sure the lower split window is active, then select from the top menu bar 'Make List of Components'. The dialog that appears is shown in Figure 4.25. This dialog works in a similar manner to the dialog used to set up graded sediment boundaries. If constant boundary values are requested the user must select the Data Type and enter the number of components and the boundary value. If time varying boundaries are requested the user must select the appropriate file. All legal (i.e. concentration) time series items in the time series file will then be used as boundaries. The first legal time series is used for component number one the second legal time series for component number 2 etc.
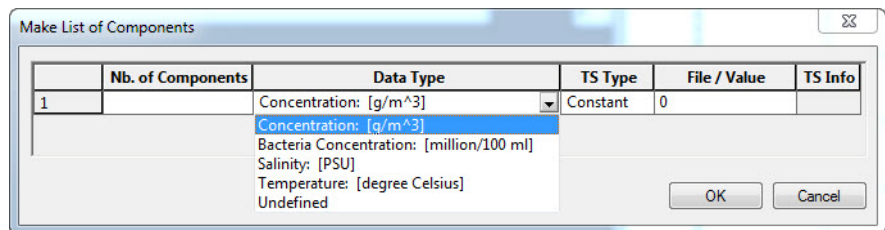


Figure 4.25    Dialog for quick specification of AD boundaries

## 4.2.3    Copying Point Source Boundaries

In order to assist the user in the creation of a boundary file containing many point sources, a tool "Copy/Paste Boundary Condition" has been implemented under "Tools" in the top menu bar. Note that the boundary to be copied must be highlighted. This facility only works for Point Sources when the AD-RR box is unchecked.

When this option is selected a dialog appears, see Figure 4.26. This dialog reflects a HD point source with a location (branch name and chainage), a boundary ID and a discharge boundary. In this dialog each row represents a new boundary. The user specifies the branch name and chainage for the new boundary in the first two columns, and optionally the boundary ID. The last two columns (with the common header Discharge) are used to specify the discharge. If they are left empty the same discharge boundary will be used for the new boundaries. If the new boundaries should use other discharges the necessary information is entered here. If constant values are requested only the File /Value edit field should be filled out. The dialog can also be filled by copying data from an Excel spreadsheet. When the dialog is closed the user is asked if the new boundaries should be pasted into the boundary editor.

If a time series is requested from a .dfs0 file with more than one item, click on the [...] button to specify which item you want.
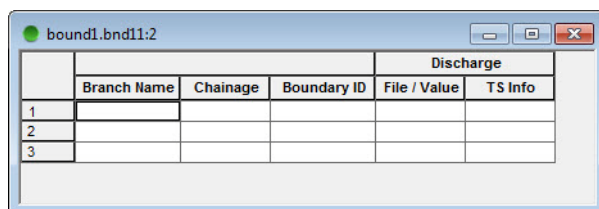
Figure 4.26    Dialog for copying a HD point source

If the point source being copied also includes AD boundaries the dialog will
also offer a possibility to change these boundaries, see Figure 4.27, in which
a point source for three AD components are being copied. The columns for
the components work in the same way as the columns used to copy the dis-
charge.
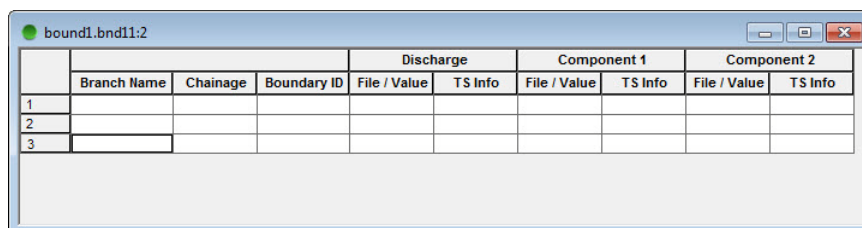


Figure 4.27    Dialog for copying an AD point source

## 4.2.4    Scale factor

In certain situations it can be useful to scale one or more of the boundaries
without changing the time series. This may be the case if a discharge hydro-
graph representing catchment runoff needs to be applied to a number of
smaller sub-catchments. Alternatively boundaries may be scaled up or down
as part of a sensitivity analysis.

The Scale Factor field is located next to the TS Info Column. The specified
Scale Factor will be multiplied with the boundary conditions value (constant or
time series). Figure 4.28 shows an AD point source with three components in
which the second component is reduced by 20 percent.

Figure 4.28     AD point source where the second component is reduce with 20 percent

An additional tool is available to quickly change scale factors. The "Change Scale Factors" tool is found under *Tools* in the top menu bar when the first split window is active. Note that this tool works for point sources of the inflow type only. The dialog is shown in Figure 4.29

The user specifies the branch name and chainage interval to which the change in scale factor(s) applies. Leaving the branch name blank corresponds to selecting all branches, but the chainage interval must still be specified. A new scale factor for the discharge boundary can be entered in the appropriate field. If this is left blank no changes will take place.

For AD boundaries, the scale factor of individual components can be changed. If the component number is left blank the new scale factor will be applied to all components. The new scale factor for the AD components must be entered; if left blank the scale factor will not be changed.
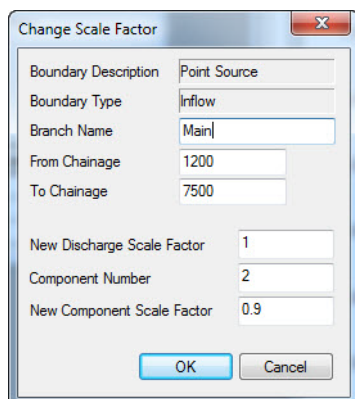
Figure 4.29    Change Scale factor dialog

## 4.3    Users Upgrading from MIKE 11 Version 2002 or Previous Versions

In version 2003 the layout of the boundary editor has been improved to make the specifications of boundary conditions easier and more intuitive. The major changes are:

- HD, AD and ST boundaries need no longer to be specified on different property pages in the boundary editor.

- The user can now choose between various boundary descriptions: open boundary, a point source, a distributed source, a globally applicable boundary, a closed boundary or a boundary related to certain structures.

- The user can now describe all boundary conditions attached to a specific location once. For example when applying the Advection-Dispersion (AD) Module the combined flow and concentration data are specified together.

- Constant boundary conditions can now be specified without the need to create a time series (.dfs0) file.

- Boundary input time series may be scaled within the boundary editor.

- Downstream Q-h boundaries can be automatically computed assuming critical or uniform flow conditions.

- When undertaking AD simulations, it is no longer necessary to specify the type of boundary in the AD editor file.

Before running a model developed with the MIKE 11 2002 version or older the old boundary file needs to be converted to the new file format. The conversion will start automatically when the old boundary file is opened in MIKE 11. When converting it is necessary to browse for the relevant network file from

which the necessary data needed for specifying the boundary type are stored. If the old boundary file contains any AD boundaries it is also necessary to specify the associated AD11file (AD parameter file) since the information regarding the type of AD boundary previously stored in this AD11 file must be transferred to the new boundary file.

# RAINFALL-RUNOFF EDITOR

# 5 Rainfall-Runoff Editor

- The Rainfall Runoff Editor (RR-editor) provides the following facilities:

- **Input and editing** of rainfall-runoff and computational parameters required for rainfall-runoff modelling.

- **Specification of time series**. Time series are specified on the Time series page within the Rainfall Runoff Editor. In other MIKE 11 modules, the time series input are specified in the boundary file.

- **Calculation of weighted rainfall** through a weighting of different rainfall stations to obtain catchment rainfall.

- **Digitising** of catchment boundaries and rainfall stations in a graphical display (Basin View) including automatic calculation of catchment areas and mean area rainfall weights.

- **Presentation of Results**. Specification of discharge stations used for calibration and presentation of results.

Some of the features in the Rainfall Runoff package have been developed in cooperation with CTI Engineering, CO., Ltd., Japan. Amongst these are additional methods for Calculation of Runoff from catchments and Calculation of Mean Precipitation of basins (method of Thiessen polygons and Isohyetal Mapping).

## Simulation

The Rainfall Runoff Editor constructs a file containing all the specified data (with extension .RR11). Once the catchments have been defined for the rainfall-runoff, and the model parameters specified in the rainfall-runoff editor, the **Simulation** is started from the MIKE 11 Run (or simulation) Editor. It should be noted that:

- It is recommended to use a time step not larger than the time step in the rainfall series and not larger than the time constant for routing of overland flow. See example in Figure 5.2.

- Simulated catchment results can be linked with the River Network. Catchment runoff/discharges are used as lateral inflows to Normal and Routing river branch types, see sections 2.5.2 and 2.4 in the River Network Editor guide.

## Results

MIKE 11 generates a variety of output types from a Rainfall Runoff simulation. These may be used for model calibration and result presentation. The result types are described in Section 5.11
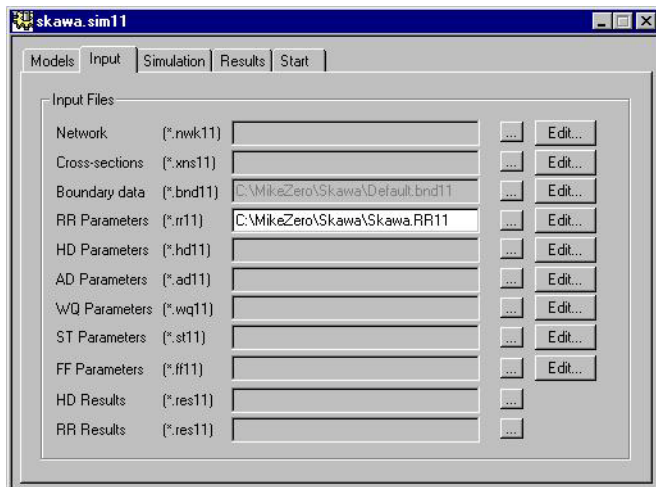
Figure 5.1    Input page to the rainfall-runoff simulation in the Simulation Editor
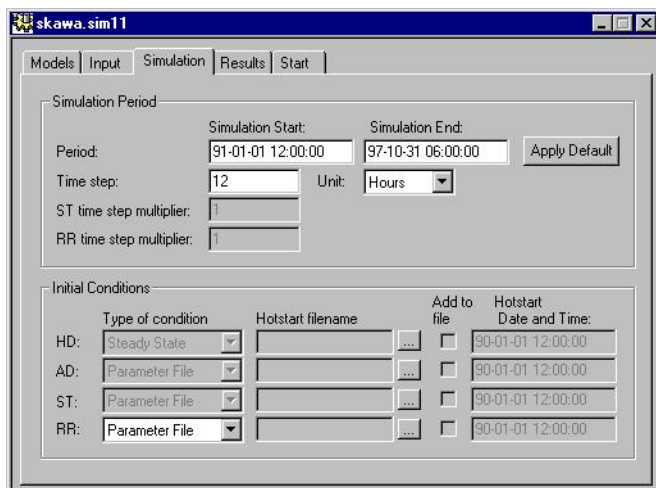


Figure 5.2    Simulation page to the rainfall-runoff simulation in the Simulation Editor.
In this example one Time step is 12 hours

## Editing using the clipboard

Tabular Overviews in the Editor shown at the bottom of each page can be
copied to the clipboard. This facility is useful, when editing a setup with many
catchments. Editing of the rainfall-runoff parameters can be carried out in a
spreadsheet after having copied the Overview to the spreadsheet via the clip-
board. After editing, the parameters are copied back to the Overview and
saved in the Rainfall Runoff Editor.

## 5.1    Specifying model Catchments

The catchment page is used to prepare the catchments to be included in the RR setup (see Figure 5.3).



Figure 5.3    The Catchment page. Additional catchments are prepared via the Insert Catchment dialog. The Example includes 2 sub-catchments and a combined catchment which includes the 2 sub-catchments

### Inserting Catchments

New catchments are defined via the Insert Catchment dialog (see Figure 5.4). A new RR parameter File is created from the MIKE Zero File dialog. Catchments are defined when pressing the button: Insert catchment.

A new catchment can be prepared as a copy with parameters from an existing catchment or with default parameters (see Figure 5.4). The copy also includes time series from the existing catchment.
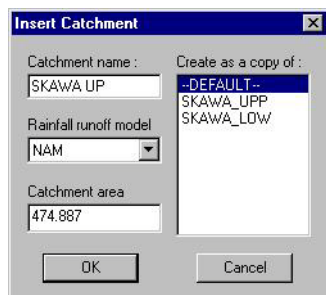
Figure 5.4    Insert Catchment dialog

## Catchments Definitions

A catchment is defined by:

### Catchment Name
Simulations can be carried out for several catchments at the same time. The catchment name could reflect e.g. the location of the outflow point.

### Rainfall Runoff Model type
The parameters required for each Rainfall-Runoff model type are specified in separate pages in the editor (see Figure 5.3). The following models can be selected:

1.  **NAM**: A lumped, conceptual rainfall-runoff model, simulating the overland-, inter- flow, and base-flow components of catchment runoff as a function of the moisture contents in four storages. NAM includes a number of optional extensions, including an advanced snow-melt routine and a separate description of the hydrology within irrigated areas. Auto calibration is available for 9 important parameters.

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

3.  **SMAP**: A monthly soil moisture accounting model.

4.  **Urban**: Two different model runoff computation concepts are available in the Rainfall Runoff Module for fast urban runoff.: A) Time/area Method and B) Non-linear Reservoir (kinematic wave) Method

5.  **Combined**: The runoff from a number of catchments, constituting parts of a larger catchment, can be combined into a single runoff series. Each of the sub- catchments must be specified separately by name, model type, parameters, etc. The combined catchment can be defined only after the sub-catchments have been created. The combined catchment is defined in the group for combined catchments, which is activated when

selecting combined catchment. The runoff from the combined catchment is found by simple addition of the simulated flow from the sub-catchments.

### Catchment Area

Defined as the upstream area at the outflow point from a catchment.

### Calibration plot

A calibration plot will automatically be prepared for catchments, where the time series for observed discharge have been specified on the Time series page and the selection of calibration plot has been ticked off. The calibration can be loaded from the Plot Composer and is saved in the subdirectory RRCalibration with the filename: Catchment-name.plc. The time series in these plots are also available in DFS0 format in the subdirectory RRcalibration with the filename: Catchmentname.dfs0. Figure 5.32 shows an example of a calibration plot.

## Calculated Areas

The Calculated area shown in the Catchment Overview is based on the digitised catchment boundaries in the Graphical display. The calculated area is activated when the Basin View has been selected, see section 5.10. The Catchment Area is shown in the edit fields for Area and Calculated Area, when transferring a catchment from the Basin View to the catchment page. The Area which is used in the model calculation can afterwards be modified manually.

## Example of a catchment setup

The catchment data included in Figure 5.3 is input data to a setup of a catchment in Poland. Rainfall Runoff parameters from this setup is used in many of the following illustrations. The setup of the catchment is further described in Section 5.12: A step-by-step procedure for using the Rainfall Runoff Editor.

## 5.2 The NAM Rainfall-runoff model

The NAM model is a deterministic, lumped and conceptual Rainfall-runoff model accounting for the water content in up to 4 different storages. NAM can be prepared in a number of different modes depending on the requirement. As default, NAM is prepared with 9 parameters representing the Surface zone, Root zone and the Ground water storages. In addition NAM contains provision for:

- Extended description of the ground water component.

- Two different degree day approaches for snow melt.

- Irrigation schemes.

- Automatic calibration of the 9 most important (default) NAM parameters.

Parameters for all options are described in the following.

## 5.2.1 Surface-rootzone

Parameters used in the surface and the root zone are described below (see Figure 5.5).



Figure 5.5    NAM - Surface Rootzone

### Maximum water content in surface storage (Umax).
Represents the cumulative total water content of the interception storage (on vegetation), surface depression storage and storage per area in the upper-most layers (a few cm) of the soil. Typically values are between 10 - 20 mm.

### Maximum water content in root zone storage (Lmax)
Represents the maximum soil moisture content in the root zone per area, which is available for transpiration by vegetation. Typically values are between 50 – 300 mm.

### Overland flow runoff coefficient (CQOF)
Determines the division of excess rainfall between overland flow and infiltration. Values range between 0.0 and 1.0

### Time constant for interflow (CKIF)

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, 2)

Determines the shape of hydrograph peaks. The routing takes place through two linear reservoirs (serial connected) with the same time constant (CK1=CK2). 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)

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 0.7% of Lmax, and the maximum values allowed is 0.99.

### Root zone threshold value for inter flow (TIF)

Determines the relative value of the moisture content in the root zone (L/Lmax) above which interflow is generated.

## 5.2.2    Ground Water

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

The Ground Water parameters are described below (see Figure 5.6).

## Overall Parameters

### Time constant for routing baseflow (CKBF)

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.

### Root zone threshold value for ground water recharge (TG)

Determines the relative value of the moisture content in the root zone (L/Lmax) above which ground water recharge is generated. The main impact of increasing TG is less recharge to the ground water storage. Threshold value range between 0 and 0.7% of Lmax and the maximum value allowed is 0.99.
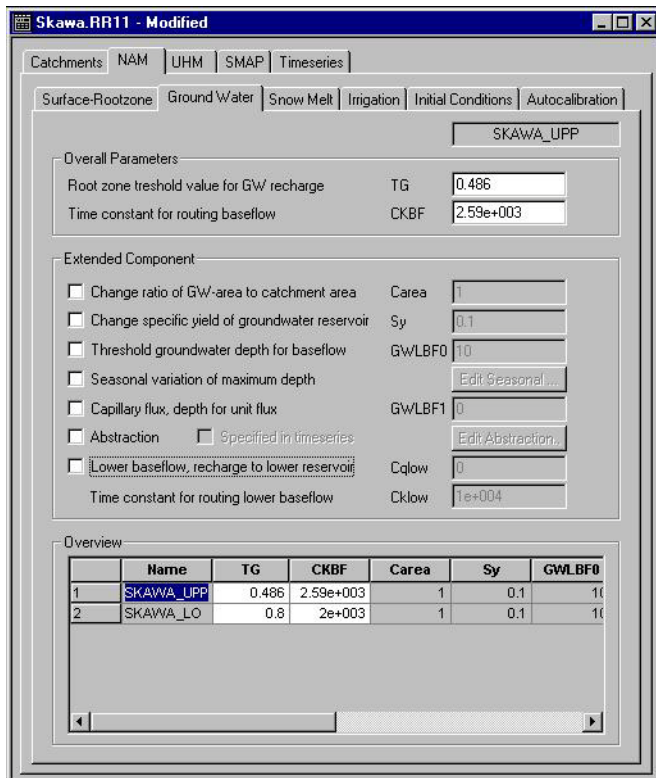
Figure 5.6    NAM - Ground Water

## Extended Ground Water Component

### Ratio of ground water catchment to topographical (surface water) catchment area (Carea)

Describes the ratio of the ground water catchment area to the topographical catchment area (specified under Catchments). 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 for the ground water storage (Sy)

Should be kept at the default value except for the special cases, where the ground water level is used for NAM calibration. This may be required in riparian areas, for example, where the outflow of ground water strongly influences the seasonal variation of the levels in the surrounding rivers. Simulation of ground water level variation requires a values of the specific yield Sy and of the ground water outflow level GWLBF0, 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 ground water depth causing baseflow (GWLBF0)

Represents the distance in metres 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, where the ground water level is used for NAM calibration, cf. Sy above.

### Seasonal variation of maximum depth

In low-lying catchments the annual variation of the maximum ground water depth may be of importance. This variation relative to the difference between the maximum and minimum ground water depth can be entered by clicking *Edit Seasonal...*

### Depth for unit capillary flux (GWLBF1)

Defined as the depth of the ground water 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.

### Abstraction

Ground water abstraction or pumping may be specified in a time series input file, in millimetres per day, or given as monthly values in mm by clicking *Edit Abstraction*.

### Lower base flow. Recharge to lower reservoir (Cqlow)

The ground water 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 ground water reservoir is specified here as a percentage of the total recharge.

### Time constant for routing lower baseflow (Cklow)

Is specified for CQlow > 0 as a baseflow time constant, which is usually larger than the CKBF

## 5.2.3   Snow Melt

The snow module simulates the accumulation and melting of snow in a NAM catchment. Two degree-day approaches can be applied: a simple lumped calculation or a more advanced distributed approach, allowing 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 simple degree-day approach uses only the two overall parameters: *a constant degree-day coefficient* and *a base temperature*.

The Snow melt module uses a temperature input time series, usually mean daily temperature, which is specified on the Time series page.

The Snow Melt parameters are described below (see Figure 5.7).
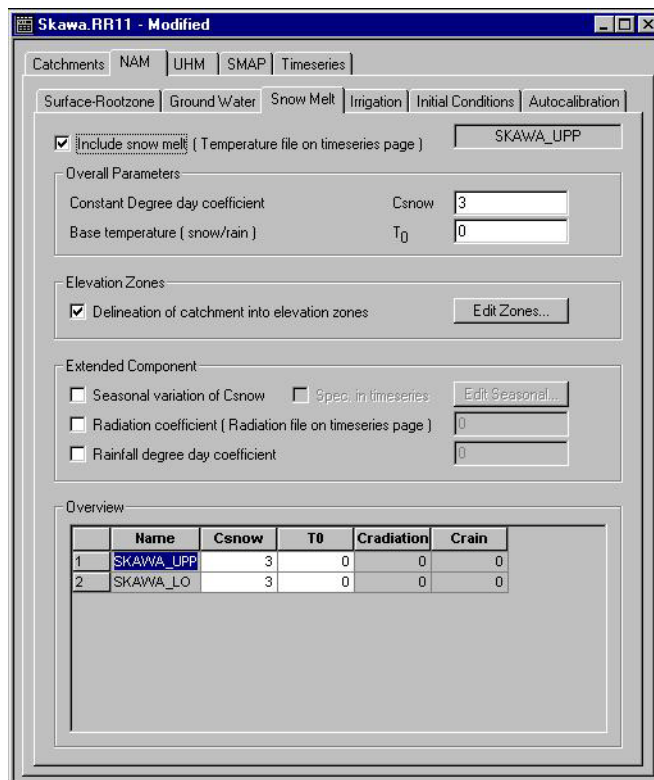
Figure 5.7    NAM - Snow Melt

## Include Snow melt

Ticked for a sub-catchments with snow melt included.

## Overall Parameters

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

### Base Temperature snow/rain (T0).
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.

## Extended Snow Melt Component

### Seasonal variation of Csnow
May be introduced when the degree-day factor is assumed to vary over the year. Variation of Csnow may be specified in a time series input file or given as monthly values in mm/day/C by clicking *Edit Seasonal*.

### Radiation coefficient
May be introduced when time series data for incoming radiation is available. The time series input file is specified separately on the Time series page. 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
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

Elevations zones are prepared in the elevation zone dialog (see Figure 5.8)

### Number of elevation zones
Defines the number of altitude zones, which subdivide the NAM catchment. In each altitude zone the temperature and precipitation is calculated separately.

### Reference level for temperature station
Defines the altitude at the reference temperate station. This station is used as a reference for calculating the temperature and precipitation within each elevation zone. (The file with temperate data is specified on the time series page).

### Dry temperature lapse rate
Specifies the lapse rate for adjustment of temperature under dry conditions. The temperature in the actual elevation zone is calculated based on a linear transformation of the temperature at the reference station to the actual zone defined as the dry temperature lapse rate (C/100m) multiplied by the difference in elevations between the reference station and the actual zone.

### Wet temperature lapse rate
Specifies the lapse rate for adjustment of temperature under wet conditions defined as days with precipitation higher than 10 millimetres. The temperature in the actual elevation zone is calculated based on a linear transformation of the temperature at the reference station to the actual zone defined as

the wet temperature lapse rate (C/100m) multiplied by the difference in elevations between the reference station and the actual zone.

### Reference level for precipitation station
Defines the altitude at the reference precipitation station (The file with precipitation data is specified on the time series page).

### Correction of precipitation
Specifies the lapse rate for adjustment of precipitation. Precipitation in the actual elevation zone is calculated based on a linear transformation of the precipitation at the reference station to the actual zone defined as precipitation lapse rate (C/100m) multiplied by the difference in elevation between the reference station and the actual zone.

**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** 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
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.

### Maximum storage in the zone
Defines the upper limit for snow storage in a zone. Snow above this values will be automatically redistributed to the neighbouring lower zone.

### Max water retained in snow
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 T0, the liquid water of the snow re-freezes with rate Csnow.

Dry temperature correction, wet temperature correction and correction of precipitation in the zone can be specified manually or calculated automatically as defined above.
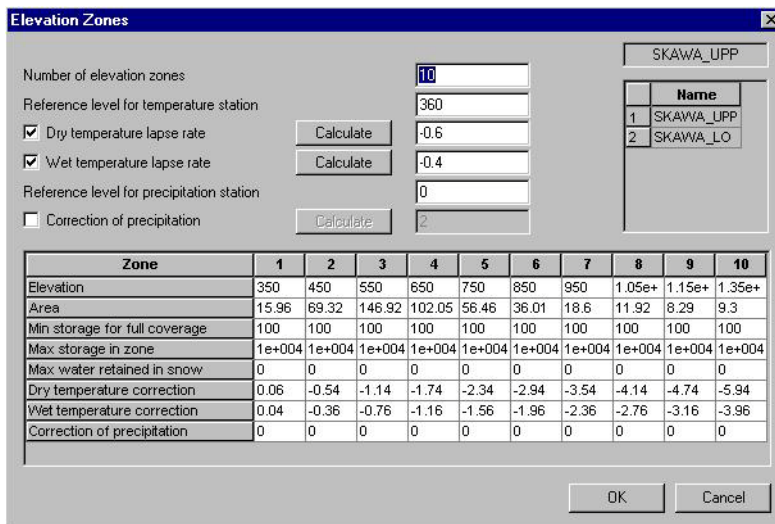
Figure 5.8    NAM - Snow Melt, Elevation Zones

## 5.2.4    Irrigation

Minor irrigation schemes within a catchment will normally have negligible influence on the catchment hydrology, unless transfer of water over the catchment boundary is involved. Large schemes, however, may significantly affect the runoff and ground water recharge through local increases in evaporation and infiltration. If the effect of an irrigation area within a catchment is to be simulated, separate NAM catchments are defined for the irrigated area and the remaining area and a *combined* catchment s defined to accumulate the runoff.

A time series of applied irrigation must be specified as a rainfall series on the time series page.

The Irrigation parameters are described below (see Figure 5.9).

Figure 5.9    NAM - Irrigation

## Include irrigation

Ticked for a sub-catchments with irrigation included.

## Infiltration Parameters

### Infiltration rate at field capacity (k0-inf)
Defines the infiltration, which is taken directly from the upper storage using a Horton-type description. This overrides the standard NAM infiltration calculation, and the overland flow coefficient CQOF and the threshold value TOF are consequently not used when irrigation is included.

## Irrigation sources

Can be local ground water, a local river, an external river, or a combination of these. Local ground water will be taken from the NAM ground water storage and irrigation water taken from a local river will be subtracted from the simulated runoff. If all the water is abstracted from an external source, outside the catchment, no subtractions are made.

## Crop coefficients and operational losses

May be specified separately. The monthly crop coefficients are applied to the potential evaporation. The operational losses, including also conveyance losses, are given in percentage of the irrigation water as losses to groundwater, overland flow or evaporation, (see Figure 5.10).



Figure 5.10    Seasonal variation of crop coefficients and losses

### 5.2.5    Initial conditions

The initial conditions are described below.

## Surface and Rootzone

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.

## Ground water

Initial values for baseflow must always be specified. When lower baseflow are included a value for the initial lower baseflow must also be specified.

## Snow melt

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

Figure 5.11    NAM - Initial Conditions

## 5.2.6    Autocalibration

Automatic calibration is possible for the most important parameters in the NAM model. A detailed description of the automatic calibration is given in the Rainfall-runoff reference manual.

The parameters used in the autocalibration are described below (see Figure 5.12).
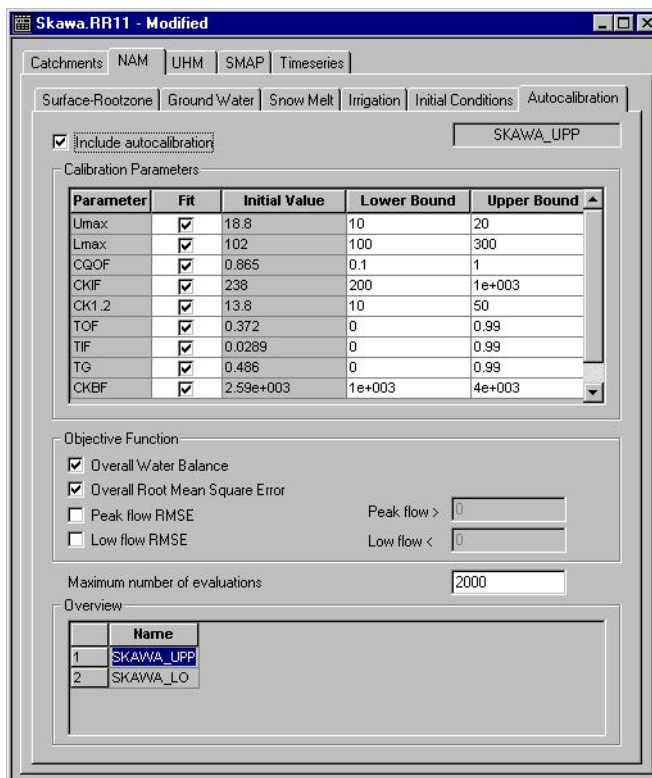
Figure 5.12    NAM - Autocalibration

## Include Autocalibration

Ticked for a sub-catchment with autocalibration included.

## Calibration parameters

The automatic calibration routine includes the 9 model parameters:

- 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 constants for routing overland flow (CK1, 2)

- Root zone threshold value for overland flow (TOF)

- Root zone threshold value for inter flow (TIF)

- Time constant for routing baseflow (CKBF)

- Root zone threshold value for ground water recharge (Tg)

The user specifies which of these parameters should be included in the auto-calibration and the minimum and maximum range for each parameter.

## Objective Function

In automatic calibration, the calibration objectives have to be formulated as numerical goodness-of fit measures that are optimised automatically. For the four calibration objectives defined above the following numerical performance measures are used:

1.   Agreement between the average simulated and observed catchment runoff: overall volume error.

2.   Overall agreement of the shape of the hydrograph: overall root mean square error (RMSE).

3.   Agreement of peak flows: average RMSE of peak flow events.

4.   Agreement of low flows: average RMSE of low flow events.

The user determined which of these objectives should be considered in the autocalibration.

## Stopping Criteria

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 evaluation is reached.

## Running the autocalibration

After preparing the autocalibration parameters the autocalibration is started as a normal simulation.

When the autocalibration is completed the message box as shown in Figure 5.13 will pop up. The Revised parameters are made available by reloading the RR-file.

A calibration plot of the results is prepared in the RRcalibration directory and can be loaded via the Plot-composer.
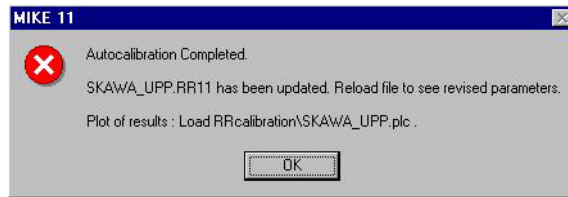
Figure 5.13    Message box after autocalibration is finished

## 5.3    UHM

### Introduction

The UHM (Unit Hydrograph) module 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 module includes a number of simple unit hydrograph models to estimate the runoff from **single storm events**. The models divide the storm rainfall in **excess rainfall** (or runoff) and water **loss** (or infiltration).

The UHM parameters are described below (see Figure 5.14).

Figure 5.14    UHM Parameters

## Areal Adjustment and Baseflow

An **area adjustment factor** (different from 1.0) may be applied if the catchment rainfall intensity is assumed to differ from the input rainfall data series.

A constant **baseflow** may be added to the runoff.

These parameters are used for all types of UHM models.

## Hydrograph

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

### SCS triangular hydrograph
The 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 hydrograph
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 hydrographs

Should be specified in their dimensionless form, i.e. Q/Qp as a function of T/Tp, as for the SCS dimensionless hydrograph above.

Six other methods for describing the hydrograph are available. These are:

- Storage Function
- Quasi Linear Storage Function
- Nakayasu
- Rational method
- Kinematic Wave (rectangular basin)
- Kinematic Wave (Non-uniform slope length)

For each of these a number of parameters are to be given. These parameters are described in more details in the reference manual.

## Loss model

### Constant loss

The infiltration is described as an **initial loss** at the beginning of the storm followed by a **constant infiltration**:

### Proportional loss

A **runoff coefficient** is specified as the ratio of runoff to the rainfall.

### The 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 further operates with three different levels of the antecedent moisture conditions **AMC**, where the initial AMC is specified.

Three other loss models are available:

- Nakayasu
- f1-Rsa
- No loss

## Lag time

Can be specified directly in hours or calculated by the standard SCS formula:

### SCS formular

Three parameters are specified: **Hydraulic Length**, **Slope** and **Curve Number**. Use the 'calculate' button to calculate the actual lag time.

# 5.4    SMAP

## Introduction

SMAP is simple rainfall runoff model of the lumped conceptual type.

It has been designed to work on the basis of monthly input data and therefore constitutes an economic alternative to the NAM model in scenarios where a daily resolution of the results is not required. This is often the case in overall water resources planning or for analyses of longterm reservoir operations. In such situations data preparation time may be saved if simulations are carried out with monthly time steps only.

The SMAP model has been tested by DHI on various dry tropical and sub-tropical catchments and has shown almost the same degree of accuracy on the simulated monthly flow as the NAM model. The model does not include a snow melt routine and is not recommended to be used in areas where snow melt has significant influence on the hydrographs.

## Model Parameters

The model accounts for the water storage in two linear reservoirs representing the root zone and the groundwater reservoirs respectively.

SMAP has five calibration parameters (see Figure 5.15):
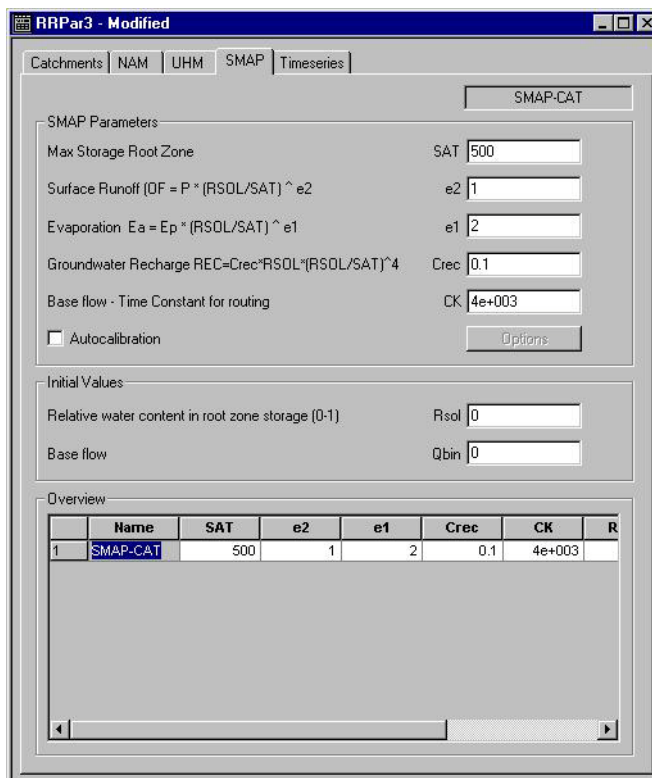
Figure 5.15    SMAP Parameters

## Max Storage Content of Root Zone (SAT)

Determines the maximum storage in the root zone storage at saturation in millimetres. The parameter determines how much water is available for evapotranspiration. The model does not account for evaporation from interception or surface depressions. Thus the magnitude of SAT is normally somewhat larger than what may be estimated from rooting depth and field capacity. Values of SAT range from 300 mm to 1500 mm. The parameter influences the total evaporation in the model and hence the overall water balance.

Similar to the NAM model many of the process descriptions in the SMAP model depends on the current saturation fraction of the root zone storage. I.e. the current storage of water (RSOL) divided by the max. possible storage (MAX).

## Surface Runoff exponent (E2)

SMAP calculates the Surface runoff (OF) as a fraction of the rainfall input during the Time step (P). The surface runoff depends both of the degree of saturation of the root zone and of the exponent E2. Note that the surface runoff will be the full rainfall amount when the root zone is saturated. Small values of E2 will increase the runoff. It is recommended to start calibration with E2 values close to 1.

### Evaporation Exponent

The actual evaporation (EA) is calculated as a fraction of the potential Evapotranspiration (EP). It depends on the current saturation degree of the root zone and the exponent E1. Small E1 will increase the Evaporation.

### Groundwater Recharge Coefficient (Crec)

Crec determines, together with the degree of saturation in the root zone, the amount of the current root zone water content (REC) to be transferred to the groundwater in each time step. Crec varies between 0 and 1.

The parameter influences the total amount of base flow generated by the model.

### Base flow Routing constant (CK)

The base flow routing constant (CK) is the time constant of the linear groundwater reservoir and is entered in the selected time unit (e.g. hours). The larger the value the slower the base flow routing. Normal interval is between 500 hours and 3000 hours.

## Autocalibration Option

Not implemented!

In addition to the above parameters the root zone content (in mm) at the start of the simulation and the initial base flow (in m3/s) needs to be specified.

## Calculation Time Step

The calculations in SMAP are non-iterative and fully forward centred. Hence, all calculations are based on the stage variables calculated in the previous time step. It is therefore recommended to perform calculations using daily calculation time steps even in situations where the rainfall input is on a monthly basis. The output (or storing) frequency can be selected on the Results page in the simulation editor and may be set to 30 days if comparison with monthly data are required. This ensures current update of the stage variables within an output interval and improves the results.

Please note that the discharge output in the main result file is in m3/s and represents an instantaneous value at the end of the last calculation time step. I.e. it is not the average discharge during the storing interval.

Values of specific discharge (in mm) accumulated over the storing interval are available in the file for additional results. This file also includes time series of other relevant parameters such as groundwater recharge, base flow and root zone moisture.

## 5.5 Urban

### 5.5.1 Introduction

Two different urban runoff computation concepts are available in the Rainfall Runoff Module:

Model A) Time/area Method

Model B) Non-linear Reservoir (kinematic wave) Method

Model A provides a hydrograph with a user-specified time of concentration suitable for larger urban catchments as well as rural catchments. Model B generally results in the hydrograph peak occurring just when heavy ranfall ceases. Model B is therefore best suited to small urban catchments.

The **Model** type (A/B) is selected in the first group box Model Parameters -> Model (see Figure 5.16 and Figure 5.17)

### 5.5.2 Urban, model A, Time/area Method

The concept of Urban Runoff Model A is founded on the so-called "Time-Area" method. The runoff amount is controlled by the initial loss, size of the contributing area and by a continuous hydrological loss.

The shape of the runoff hydrograph is controlled by the concentration time and by the time-area (T-A) curve. These two parameters represent a conceptual description of the catchment reaction speed and the catchment shape.

The Parameters for Model A are described below (see Figure 5.16).

Figure 5.16    Urban Page. Model A, Time/area Method

## Impervious Area

The Impervious area represents the reduced catchment area, which contributes to the surface runoff

## Time of Concentration

Defines the time, required for the flow of water from the most distant part of the catchment to the point of outflow

## Initial Loss

Defines the precipitation depth, required to start the surface runoff. This is a one-off loss, comprising the wetting and filling of catchment depressions.

## Reduction factor

Runoff reduction factor, accounts for water losses caused by e.g. evapo-transpiration, imperfect imperviousness, etc. on the contributing area.

## Time/Area Curve

Accounts for the shape of the catchment lay-out, determines the choice of the available T/A curve to be used in the computations.

Three pre-defined types of the T/A curves are available:

1. rectangular catchment

2. divergent catchment, widest at its outlet

3. convergent catchment, widest at its head

## 5.5.3 Urban, model B, Time/area Method

The concept of surface runoff computation of Urban Runoff Model B is founded on the kinematic wave computation. This means that the surface runoff is computed as flow in an open channel having the same area as the contributing catchment, only taking the gravitational and friction forces into account. The runoff amount is controlled by the various hydrological losses and the size of the actual contributing area.

The shape of the runoff hydrograph is controlled by the catchment parameters length, slope and roughness of the catchment surface. These parameters form the base for the kinematic wave computation (Manning equation).

Infiltration to groundwater is calculated using a modified Horton equation.

The Parameters for Model B are described below (see Figure 5.17)

Figure 5.17    Urban Page. Model B, Non-linear Reservoir (kinematic wave) Method

## Length

Conceptually, definition of the catchment shape, as the flow channel. The model assumes a prismatic flow channel with rectangular cross section. The channel bottom width is computed from catchment area and length.

## Slope

Average slope of the catchment surface used in the Manning equation for the runoff.

## Area (percentages)

The area distribution percentages allow the catchment area to be divided in up to five sub-catchments with identical geometrical, but distinct hydrological properties. The five sub catchment types are:

- impervious steep
- impervious flat
- pervious -small impermeability
- pervious - medium impermeability
- pervious - large impermeability

The hydrological properties of each of the sub-areas can be adjusted by modifying the appropriate hydrological parameters (see Figure 5.18 showing default values). The sum of the specified areas (in %) must be equal to 100%.



Figure 5.18     Model B, Hydrological Parameters for individual sub-catchments.

### Wetting loss
One-off loss, accounts for wetting of the catchment surface.

### Storage loss
One-off loss, defines the precipitation depth required for filling the depressions on the catchment surface prior to occurrence of runoff. Not applicable to 'Impervious Steep' type of catchment.

### Start infiltration
Defines the maximum rate of infiltration (Horton) for the specific surface type.

### End infiltration
Defines the minimum rate of infiltration (Horton) for the specific surface type.

### Horton's Exponent
Time factor "characteristic soil parameter" with unit of $s^{-1}$. Determines how quickly during a rainfall event the infiltration capacity rate reduces over time from the 'Start Infiltration' to the 'End Infiltration'. The actual infiltration capacity is dependent of the time since the rainfall start only.

### Inverse Horton´s Equation
Time factor used in the "inverse Horton's equation", defining the rate of the soil infiltration capacity recovery after a rainfall event, i.e. in a drying period.

### Manning's number
Describes roughness of the conceptual catchment surface, used in hydraulic routing of the runoff (Manning's formula).
Note that the Manning's number here is specified as Manning M - the reciprocal of Manning's n.

### 5.5.4   Additional Time series

#### Additional runoff

Additional runoff can be specified as a constant flow or specified as **load based on inhabitants** (PE). An additional time series for load (qload) is specified in the time series, when the flow is based on load based on inhabitants (PE>0). The flow is calculated as:

$$\text{Flow} = [\text{PE}]qload(t) \tag{5.1}$$

#### Evaporation

The Evaporation check box controls if the evapo-transpiration shall be calculated based on a time series (when checked the time series is specified on the Time series page) or based on a constant loss (equal to 0.05 mm/hour).

#### Snow melt

The Snow melt check box controls if snow melt is to be included in the calculation. The content of the snow storage melts at a rate defined by the degree-day coefficient **CSnow** multiplied with the temperature deficit above 0 Degree Celsius. Typical values for Csnow is 2-4 mm/day. When snow melt is checked a time series for temperature is specified on the Time series page.

## 5.6   Flood Estimation Handbook (FEH)

### 5.6.1   Background

The Flood Estimation Handbook (FEH) was introduced in 1999 to replace the previous Flood Studies Report (FSR) methods for flood estimation in the UK. The FEH comes in 5 volume, with 2 associated software products. The FEH set comprises:

1.   Overview

2.   Rainfall Frequency Estimation

3.   Statistical Procedures for Flood Frequency estimation

4.   Restatement and application of the Flood Studies Report rainfall-runoff method

5.   Catchment Descriptors

The implementation of MIKE FEH is mainly concerned with Vol. 4.

## 5.6.2 Methods for hydrograph Generation

The following methods for computing a hydrograph have been incorporated into MIKE FEH:

- Generation of a T-year event -Chapter 3 of the FEH handbook

- Generation of a Probable Maximum Flood (PMF) - Chapter 4 of the FEH handbook

- Generation of an observed Flood Event - Chapter 5 of the FEH handbook

## 5.6.3 T-Year Event

The steps described in Table 5.1 are used to compute a T-Year hydrograph:

Table 5.1      T-Year event

| Step | Input | Computation | Reference |
|------|-------|-------------|-----------|
| 1 | The catchment in question is identified from the FEH CD ROM and the catchment descriptors exported to CSV format.<br>The main descriptors are:<br>· AREA = catchment area (km2)<br>· DPLBAR = = mean drainage path length (km)<br>· DPSBAR = mean drainage path slope (m/km)<br>· PROPWET = proportion of time when Soil Moisture Deficit (SMD) was below 6mm in period 1961-90<br>· SPRHOST = Standard Percentage Runoff derived from HOST soil classification (%)<br>· URBEXT = extent of urban and suburban land cover<br>· SAAR = Standard Average Annual Rainfall (mm)<br>· The Depth-Duration-Frequency rainfall descriptors:<br>c, d1, d2, d3, e and f | | Vol. 5, Chap 7. |
| 2 | Compute $Tp(0)$ - Time to peak of instantaneous unit hydrograph (IUH). | This can be computed from<br>i. Catchment LAG (Vol. 4, Eq. 2.9)<br>ii. Catchment Descriptors (Vol. 4, Eq.2.10)<br>iii. Donor catchment<br>iv. Observations | Vol. 4, Chap 2.2. |
| 3 | Compute delta T, time interval of unit hydrograph. | Recommended as 10-20% of $Tp(0)$. Computed as 20% of $Tp(0)$. | Vol. 4, Chap 2.2. |
| 4 | Compute Time to peak, Tp, of unit hydrograph. | $Tp = Tp(0) + deltaT/2$ | Vol. 4, Chap 2.2. |

Table 5.1          T-Year event

| Step | Input | Computation | Reference |
|------|-------|-------------|-----------|
| 5 | Calculate Unit Hydrograph peak (Up) and time base (TB). | | Up: Vol. 4, Eq 2.6.<br>Tb: Vol. 4, Eq. 2.7. |
| 6 | Calculate design Storm duration, D. | D = Tp (1+SAAR)/1000 | Vol. 4, Chap 3.2.1, Eq. 3.1. |
| 7 | Calculate rainfall return period, TR for each flood return period TF. | If URBEXT < 0.125, then TR is determined from Vol. 4, Figure 3.2.<br>If URBEXT >0.125 and <0.50, then TR = TF | Vol. 4, Chap 3.2.2. |
| 8 | Compute the D hour, TR year point rainfall (mm) = MT-Dh<br>This is calculated from the catchment descriptors  c, d1, d2, d3, e and f.<br>Not Displayed. | Gumbel reduced variate:<br>$Y=-\ln[-\ln(1-1/TR)]$<br>(Vol. 2, Eq. 2.1)<br>The point rainfall is computed as a function of y and c, d1, d2, d3, e and f. | Vol. 2, Chap 2.3.<br>Eq. 2.2 - 2.4. |
| 9 | Compute storm depth P for catchment by scaling point rainfall depth with Areal Reduction Factor (ARF)<br>Not Displayed | ARF = 1 - bD-a<br>Where a and b are functions of the Area. | Vol. 2, Chap 3.4.<br>Vol. 4, Chap 3.2.2. |
| 10 | Derive design storm profile. There are 2 standard profiles, the winter and the summer profile.<br>The actual design profile is based on the standard one taking into account the catchment storm depth and duration.<br>Compute and write to output file in dfs0 format. | If URBEXT < 0.125, use winter profile.<br>If URBEXT >0.125 and <0.50 use summer profile.<br>The profiles are defined in Vol. 2 Eq 4.2 and shown in Vol. 4 Figure 3.5. | Vol. 2, Chap 4..<br>Vol. 4, Chap 3.2.3. |
| 11 | Compute Catchment Wetness Index (CWI). This is function of SAAR. | | Vol. 4, Chap 3.2.4.<br>Figure 3.7. |

Table 5.1        T-Year event

| Step | Input | Computation | Reference |
|------|-------|-------------|-----------|
| 12 | Compute Standard Percentage Run-off (SPR). SPR can be computed from<br>· Baseflow index (Vol. 4 Eq. 2.16)<br>· SPRHOST<br>· Transfer from donor<br>· From observations | | Vol. 4, Chap 2.3. |
| 13 | Calculate percentage runoff (PR) appropriate to the design event. This is based on PR for the rural fraction of the catchment and scaled according to URBEXT. | Vol. 4 Eq. 2.12, 2.13, 2.14, 2.15. | Vol. 3, Chap 3.3.1. |
| 14 | Compute baseflow. This can be computed from<br>· Catchment descriptors (Vol. 4 Eq 2.19)<br>· Transfer from donor catchment<br>· From observations | | Vol. 4, Chap 2.4. |
| 15 | Compute the net event hydrograph by multiplying the design rain event hyetograph by PR. | Output of Step 10 multiplied with PR. | |
| 16 | Compute the rapid response hydrograph by convoluting the net rainfall event hyetograph against the unit hydrograph (computed in step 5). | Vol. 4, Eq. 2.3 | Vol. 4, Chap 3.3. |
| 17 | Compute total response hydrograph by adding baseflow (Step 14) to rapid response hydrograph (Step 16). | | |
| 18 | Scale computed hydrograph according to Target Peak Flow. | | |

## 5.6.4    Probable Maximum Flood

PMF computations are used for e.g. reservoir and dam safety studies. The main differences between PMF and T-Year hydrograph generation described in the previous section are:

- · Unit hydrograph parameters
- · Rainfall generation
- · CWI estimation
- · Contribution of Snowmelt

- · Standard percentage runoff (SPR) and Percentage runoff (PR)

### Unit Hydrograph Parameters

Very simply, the time to peak of the instantaneous unit hydrograph, Tp(0), for a PMF computation is assume to be 0.67 times the standard value. This affects both the peak (Up) and time base (TB). See Vol. 4, Chap 4.2.1.

### Rainfall Generation

The Probable Maximum Precipitation (PMP) hyetograph is constructed directly, not via storm depth and standard profiles as in the T-year case. The user is required to construct the design hyetograph manually and store the profile in a dfso file.

### Catchment Wetness Index (CWI)

This is now a function of the estimated maximum antecedent rainfall, which in turn is a function of the storm hyetograph. The user should make the computation (given in Vol. 4 Chap 4.3.3) and enter the value directly in the menu.

### Contribution of Snowmelt

Snowmelt may contribute to both the storm depth and antecedent rainfall, and therefore the CWI. The user should define a snow melt rate (mm/h) from which both these effects can be computed. See Vol. 4, Chap 4.3.4 and example 4.1f. Output from this part is an adjusted CWI (denoted CWI') as well as a modified storm profile in dfs0 format (as for step 10, above).

### Standard percentage runoff (SPR) and Percentage runoff (PR)

If using a winter PMP, the SPR is set to a minimum of 53% to account for frozen ground. See Vol. 4 Chap 4.2.2. In addition, a revised formulation for PR is made, Vol. 4, Eq. 4.12.

Initially, the SPR is computed as for step 12, above, and subsequently checked to ensure that the value is greater than or equal to 0.53 (if a winter profile is used).

In the computation of PR, PR rural is computed from Vol. 4, Eqs. 2.13 - 2.15 and Eq. 4.12 using the SPR (just computed), the CWI (adjusted for snowmelt) and the integral over time of the design hyetograph (also adjusted for snowmelt).

## 5.6.5   Generation of an Observed Flood Event

In this case observed rainfall is used as input from which the resulting hydrograph can be computed. Computation of the CWI is also based on the rainfall observations. See Vol. 4, Chapter 5.

### Catchment Rainfall (MAR)

Catchment rainfall is provided by the user as a dfs0 file. Please note that mean area rainfall computation is done on the Time series page in the Rainfall Runoff editor. Following the specification of a catchment rainfall file, the

period (start and end) covered by the time series will be shown. Using this information the user is required to set the design storm period, which defines the storm duration and the rainfall depth.

### Storm Depth and Duration
The storm duration and depth is computed automatically from analysis of the input rainfall. Note the rainfall must start at least 5 days before the storm start time in order to compute antecedent wetness.

### Storm Profile
An option is available to allow the user to use the measured rainfall time series distribution to generate the hydrograph, alternatively one of the two standard profiles (summer or winter) may be used.

### Observed Antecedent Catchment Wetness Index (CWI)
The CWI calculation is based on the observed rainfall record in the 5 days prior to the start of the event and the observed soil moisture deficit (SMD). The procedure is described in detail in Vol. 4, Appendix A, section A.4.2. SMD is defined by the user. Subsequently, CWI is computed by MIKE FEH.

## 5.6.6   Results

The user may set the origin of the time axis of all the result files computed in MIKE FEH. This may be desirable if a hydraulic analysis (using MIKE 11 HD) is to be done afterwards.

The results include:

1. The design storm profile as interpreted by MIKE FEH
2. The unit hydrograph profile used to compute the hydrograph
3. The computed hydrograph

If, for a T-Year event, multiple return periods have been specified, the result files contain multiple items, one item for each event.

## 5.6.7   Validation

Once the user presses the Compute button on the Results page, MIKE FEH starts the validation of the provided input. If the input is accepted, the model proceeds, otherwise the validation error messages are shown in the interface.

## 5.6.8   Log Files

An excerpt of the input and the intermediate results are found in a text file with the extension .log. The file is located in the directory of the RR11 file.

From the interface, the user selects for each catchment whether or not a log should be created.

## 5.7    DRiFt

The DRiFt module (DRiFt = Discharge River Forecast) is a semi-distributed rainfall-runoff model based on a morphological approach. The model is able to consider the topography of each site analysed and the spatial variability of soil characteristics and rainfall patterns. Input data for the DRiFt model is divided into three groups:

- Surface Flow parameters

- Initial Conditions

- Rainfall / Precipitation data

The development of DRiFt was done by CIMA - Centro di ricerca Interuniversitario in Monitoraggio Ambientale - a research institution of the Universities of Genoa and Basilicata (Italy), in cooperation with ACROTEC S.r.l.

Please note that 'DRiFt' is only available in the 32-bit version.

### 5.7.1    Surface flow

Parameters for calculating the surface flow are described below (see Figure 5.19).
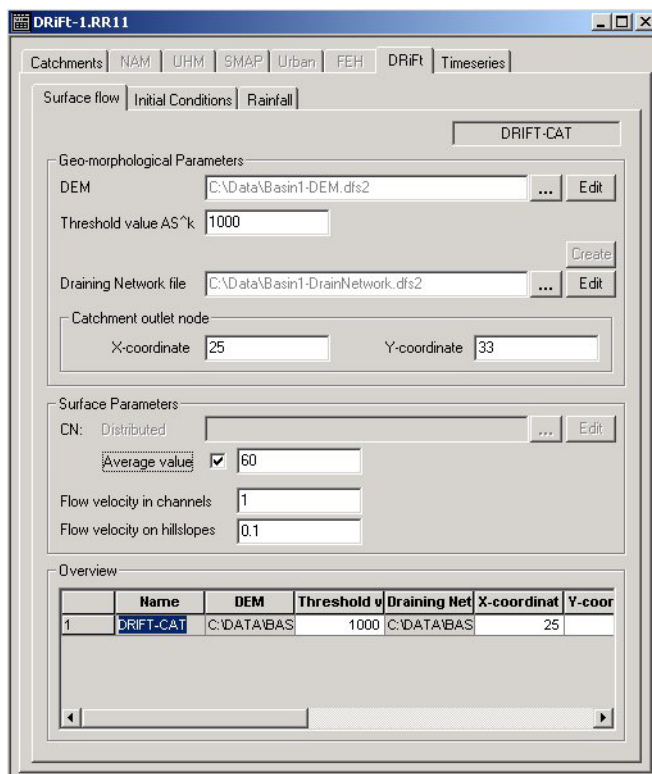
Figure 5.19    DRiFt Surface Flow parameters

## Geo-morphological Parameters

### DEM

The DEM (Digital Elevation Model) of the basin - prepared and saved in a two-dimensional grid-file (*.dfs2). For a better interpretation of the derived files (like draining network files, etc.), it is opportune to mask the useless parts of the territory (e.g. the sea) setting them to "no data value".

The DEM-file to be applied for a particular catchment must be selected by pressing the Browse button '...'. The 'Edit' button opens an already selected DEM dfs2-file for viewing and editing in the MIKE Zero grid editor.

### Threshold value $AS^k$

$AS^k$ is the threshold value for a slope-area filtering procedure applied when generating the channel drainage network in the basin from the DEM. Typical range of $AS^k$ is 100-1.000.000 m$^2$, default value for e.g. a 225m x 225m DEM resolution: 100.000 m$^2$.

### Geo-morphological exponent, k

The Geo-morphological exponent, k is the exponent used in the expression for the threshold value $AS^k$. Default value is 1.7.

### Draining Network file

The draining network file is a grid-file containing some topography information in general from the DEM and additionally, the information on the channel or draining network inside the basin. The Draining network file is the grid-file which is actually used in the calculation of surface runoff. Therefore, it is required to specify a draining network file for each DRiFt catchment prior to a simulation.

The Draining Network is created by activating the 'Create' button after a DEM-filename, an 'AS$^k$'-value and a 'k'-value has been defined. Alternatively, a pre-defined network-file can be loaded by use of the '...' button. Before the creation, a name for the destination file must be chosen by clicking on the '…' button. The suggested default name of the draining network file is "catchment_name.choice.dfs2".

Other files are automatically generated together with of the draining network file (e.g. catchment_name.pnt.dfs2 representing the slope orientation, catchment_name.area.dfs2 representing the drained upstream area). These files are used in the simulation even though they do not appear within the interface, therefore they always must be located together in the same directory. The user must pay attention to the operations which could alter this conditions (renaming, moving, deleting these files).

### Catchment outlet node

The catchment outlet node is defined as the pixel (grid point) in the draining network where outlet from the basin occurs. The outlet node is specified by the X- and Y-coordinates (or j- and k-grid coordinates as are the standard notations for dfs2-file definitions in MIKE Zero) of the grid point which contains the basin outlet location.

## Surface Parameters

### Curve Number (CN)

CN (the Soil Conservation Service Curve Number) can be specified in two ways; either as an average, constant value for the entire basin or as a distributed grid defined by a dfs2-file.

Activating the 'Average value' tick-mark will activate the average value field and a constant value must be defined. De-activating the tick-mark will require a dfs2-file selection in the 'Distributed' field by use of the browse button.

The range of this variable is from 0 to 100.

### Flow velocity in channels

Flow velocity in each area (cell) within the basin which is identified in the draining network file as 'channel'. Contributes to the computation of a total routing time.

Default value for flow velocity on channels is 1 m/s. The normal range of the variable is from 0,1 m/s to 10 m/s.

### Flow velocity on hillslopes

Flow velocity in each area (cell) within the basin which is identified in the draining network file as 'non-channel' or 'hill-slope'. Contributes to the computation of a total routing time.

Default value for flow velocity on hillslopes is 0,1 m/s. The normal range of the variable is from 0,001 m/s to 1 m/s.

## 5.7.2    Initial conditions

Initial conditions for the DRiFt model comprises the SCS Antecedent Moisture Content. The SCS-AMC value is specified in the DRiFt Initial Conditions page (see Figure 5.20).
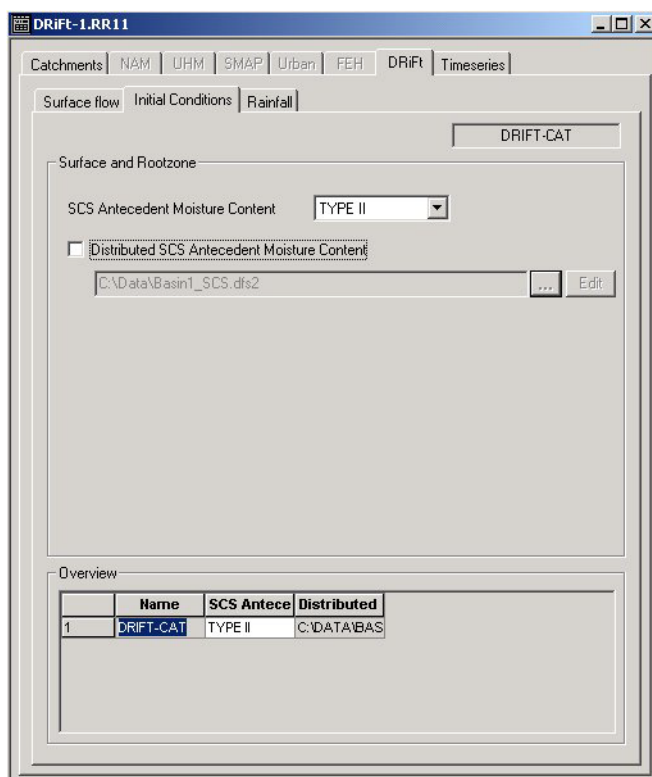


Figure 5.20    DRiFt Initial Conditions page

### SCS Antecedent Moisture content

The antecedent moisture content can be defined as either a constant value all over the catchment or a distributed value (defined in a dfs2-file). Only three values are allowed: Type I (dry condition), Type II (normal condition) and Type III (wet condition).

Default option is to use a constant value. If, however a distributed AMC-value is required, then activate the 'Distributed SCS Antecedent Moisture Content' tick-mark and select an AMC dfs2-file by use of the browse button. AMC dfs2-file must be constructed assigning to each cell numeric values 1, 2 or 3 corresponding to Type I, II or III.

## 5.7.3    Rainfall

The input for the rainfall-runoff simulation is supplied as rainfall (or precipitation) data. Rainfall data can be specified either as a constant value, as a time series or as a time-varying distributed rainfall pattern.

The rainfall input is defined in the DRiFt 'Rainfall' page (see Figure 5.21).
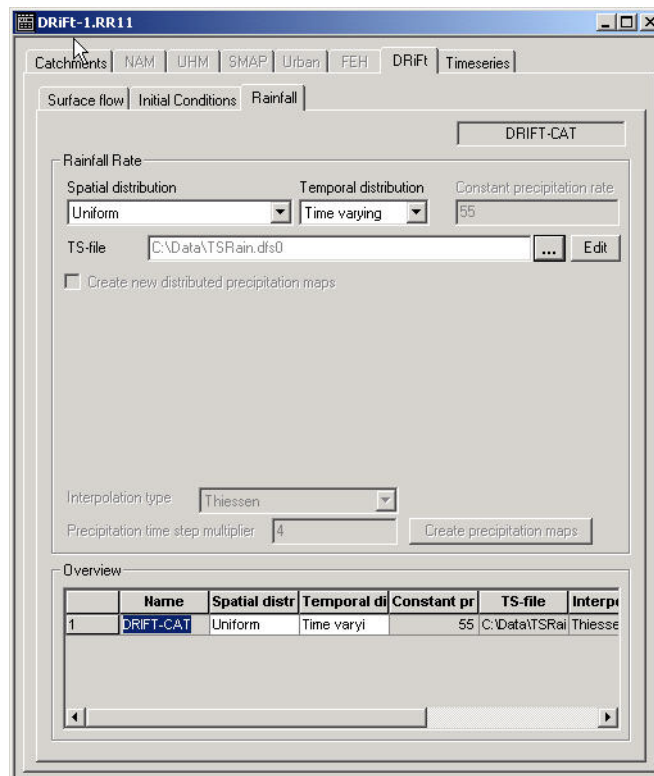


Figure 5.21    DRiFt Rainfall page (Spatial distribution = Uniform)

## Precipitation Rate

DRiFt utilises a spatial distributed map of precipitation as input to the computational part. It is therefore required to preprocess a rainfall map from either spatial and/or temporal distributed sources.

## Spatial distribution

Spatial distribution of precipitation can be made either Uniform or Distributed. Select the required option from the Spatial Distribution combo-box.

## Temporal distribution

Temporal distribution of precipitation can be made either Constant (= constant value in space and time) or Time Varying.

Based on the selected combination of spatial and temporal distribution of rainfall, different precipitation data definitions are required (see Table 5.2):

Table 5.2    Specification of precipitation data. Requirement as function of Spatial and Temporal distribution selections.

| Spatial Distribution | Temporal Distribution | Required precipitation data definition |
|---|---|---|
| Uniform | Constant | Constant value for precipitation rate [mm/hours] |
| Uniform | Time Varying | Time series file (dfs0) of precipitation as rainfall intensity [mm] or rainfall intensity [mm/hour] |
| Distributed precipitation maps | Time Varying | Time varying grid-file (dfs2) of precipitation as rainfall [mm] |

## Constant precipitation rate

Here the constant precipitation rate is defined [mm/hour].

## TS-File

With a selection of uniform spatial distribution and time varying temporal distribution as presented in Figure 5.21, it is required to select a time series file (dfs0-file) with rainfall data in the 'TS-file' filename field.

Precipitation can inserted in two different ways:

- as **"rainfall" [mm]** and **TStype = "mean step accumulated"**

- as **"rainfall intensity" [mm/hour]** and **TStype = "instantaneous"**

## Rainfall file

With a selection of distributed precipitation maps as presented in Figure 5.22, it is required to select a time varying grid-file (dfs2-file) with rainfall data in the 'Rainfall file' filename field.

Here precipitation is always treated as **"rainfall" [mm], "mean step accumulated"**

### Create new distributed precipitation maps

DRiFt includes a possibility for generating a time varying grid based precipitation input file from a number of single rain gauges observations by use of spatial interpolation. If the rainfall pattern must be distributed and no rainfall file exists then by activating this check-box, DRiFt will generate a time varying distributed file with the filename as specified by the user in the 'Rainfall File' filename field.

In case the 'Create new distributed precipitation maps' feature is enabled, it is required to specify Rain gauges definitions in the table below the checkbox.

### Rainfall station specification table

Rainfall stations (rain gauges) definitions for the spatial interpolation feature in DRiFt is given in this table. The information required for the interpolation is:

- the location of the rain gauge station in the basin defined by plan coordinates (X and Y),

- a time series file (dfs0-file) with rainfall measurements from the specific station. Use the Browse button to select the required dfs0-file.

- additional, optional information in the table is the possibility to specify a text-string identifier for each rainfall station.

To start the definition of rain gauges stations in an empty rain gauges table, click on the 'Edit' column-button and thereafter press <TAB>. Alternatively, it is possible to select from the 'Grid' option in the Main Menu Bar the 'Insert line' option (after one of the column-buttons has been activated). Thereby, a new line will be appended to the table and rain gauges specifications can be made.

Every time a new rain gauge definition must be added, it is possible to add a new line to the table by using the Tabulator when the cursor is located in the last column of the table.

An example of definition of three rain gauges stations is presented in Figure 5.22.
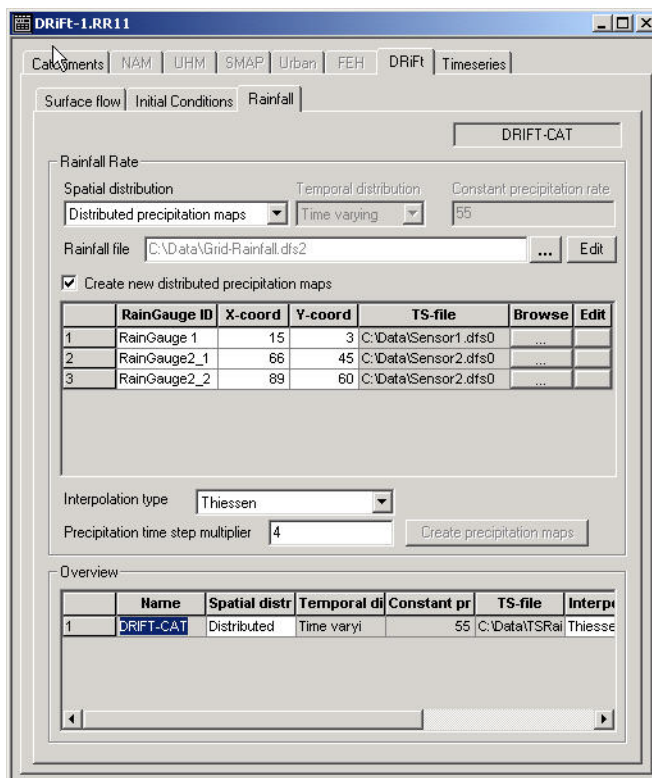
Figure 5.22    DRiFt Rainfall page (Spatial distribution = Distributed)

## Interpolation type
Interpolation options for generation of time varying distributed precipitation maps are: 'Thiessen' and 'Inverse squared distance'. Select the desired interpolation type from the Interpolation type combo-box.

## Precipitation time step
The precipitation time step is the temporal resolution of the new distributed maps [seconds]. In order to exploit the whole available information, it is recommended to set the precipitation time step on the same value of rain gauges measurements resolution.

## Create precipitation maps
This action-button creates the required distributed precipitation maps for the DRiFt runoff calculations. The distributed rain maps are required for all DRiFt calculations and it is therefore requested for the user to prepare a rain map for all catchments.

The precipitation map will be created here as  **"rainfall" [mm],  "mean step accumulated"**

The file containing the rain map is a dfs2-file with a filename depending on the selection of spatial distribution (uniform or distributed). For a selected uni-

form spatial distribution the filename will be 'Rainmap+Catchment-name'.dfs0. If a distributed spatial distribution has been selected then the Rain map will be saved in the file as specified in the 'Rainfall file' edit-field.

## 5.8    Time Series

The Time series page serves two purposes: Input of time series and calculation of weighted time series (see Figure 5.23)
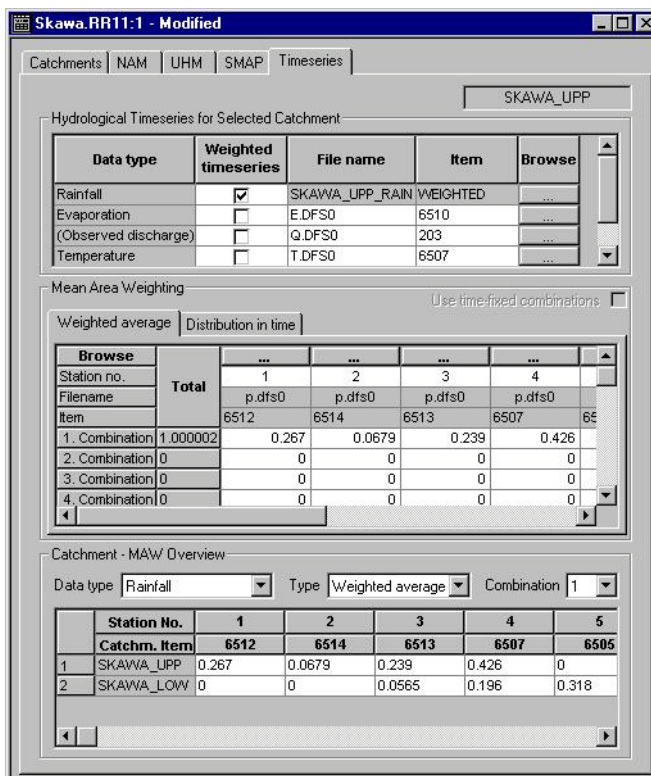


Figure 5.23    Time series page

### Input of time series

The input time series for the rainfall-runoff simulations are specified on this page. The time series are used as boundary data to a MIKE 11 simulation. The following data types are used:

#### Rainfall

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 must be the rainfall depth accumulated since the previous value.

Time series item definitions for rainfall input require:
- Item Type must be defined as 'Rainfall'
- TS-Type must be set to 'Step Accumulated', which means each value is total rainfall quantity in its time step.

## Evaporation

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

Time series item definitions for Evaporation input require:

- Item Type must be defined as 'Evaporation'
- TS-Type must be set to 'Step Accumulated'

## Temperature

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

Time series item definitions for Temperature require:

- Item Type must be defined as 'Temperature'
- TS-Type must be set to 'Instantaneous'

## Irrigation

An input time series is required to provide information on the amount of irrigated water applied, if the irrigation module is included in a NAM simulation

Time series item definitions for Irrigation require:

- Item Type must be defined as 'Irrigation'
- TS-Type must be set to 'Step Accumulated'

## Abstraction

Groundwater abstraction can be included in NAM simulations for areas, where this is expected to influence e.g. the baseflow. The data should be given in mm.

Time series item definitions for Abstraction require:

- Item Type must be defined as 'Ground Water Abstraction Depth'
- TS-Type must be set to 'Step Accumulated'

## Radiation

A time series of incoming solar radiation can be used as input to the extended snow melt routine.

Time series item definitions for Radiation require:

- Item Type must be defined as 'Sun Radiation'
- TS-Type must be set to 'Instantaneous'

### Degree-day coefficient

A time series of seasonal variation of the degree-day coefficient can be specified as input to the extended snow melt routine.

Time series item definitions for Degree-day coefficient require:

- Item Type must be defined as 'Melting Coefficient'
- TS-Type must be set to 'Instantaneous'

### Observed Discharge

A time series of observed discharge values can be specified and used for model calibration. The observed discharge must be specified when automatic calibration is included.

The selection of the observed discharge will automatically enable additional output which includes a calibration plot with comparison of observed and simulated discharge and calculation of statistical values. See Section 5.11.

Time series item definitions for Observed Discharge require:

- Item Type must be defined as 'Discharge'
- TS-Type must be set to 'Instantaneous'

## Calculation of Weighted time series

This calculation usually needs only to be made once. Once the calculation is made the result are stored in time series that can be used for subsequent rainfall-runoff modelling runs.

If the rainfall data, weights or number of catchments changes the calculation must be repeated.

The Mean Areal Weighting calculation can be performed in two ways.

1. Directly within the Rainfall Runoff Editor. From the top toolbar menu select Basin Work Area and the Calculate mean precipitation. The calculation is made without requiring a model run.

2. During a simulation. A new simulation is started in the Simulation Editor: If the weighted time series is ticked, the Mean Area weighting calculation is carried out as part of the model run.

It is recommended to use option 1.This will ensure that the available time periods of the input files are available for the simulations editor.

After having calculated the weighted time series once the calculation can be disconnected by removing the tick mark for weighted time series.

## Mean Area Weighting

### Weighted Average combinations

Where complete time series for all stations are available for the entire period of interest only one weight combination is required. Where data is missing from one or more stations during the period of interest different weight combinations can be specified for different combinations of missing data.

It is not necessary to specify weight combinations for all possible combinations of missing stations. For each calculation, the Mean Area Weighting algorithm will identify estimate weights which best represent the actual combination of missing data. In most cases only one set of weights need to be specified. The Mean Area Weighting algorithm will automatically redistribute weights from missing stations equally to the stations with data.

Alternatively, the user may specify the weight to be used for specific combination of missing data. For each such catchment, a suitable weight should be specified for the reporting stations and a weight of "-1.0" given for the non-reporting station(s), including missing data.

### 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 the distribute (desegregate) this daily rainfall in time. Different weight combinations for different cases of missing values may be applied also to this calculation of the distribution in time.

### Time fixed combinations

It is possible to specify fixed periods with different combinations. The periods are specified from the menu bar (select: Parameters | Time-fixed combinations). To enable calculation: Tick mark in the check box on the Time series page.

### Deleting stations

Stations which are not longer valid in the weight combinations are removed from the editor by deleting the station number in the editor.

### Delete values

The delete value used in the time series indicating periods with missing data is usually specified with the default delete value '1e-30'. The default delete value can be changed via the MIKEZero Data Utility tool.

## 5.9 Parameters menu

The parameters menu contains a number of items mainly relating to the UHM models Storage Function, Quasi Linear Storage Function, Nakayasu, Rational method, and Kinematic Wave.

### 5.9.1 Enlargement ratio

The rainfall specified in the Time series page can be enlarged by a factor. Three factors, each with a duration for which they should apply, can be specified.

### 5.9.2 Loss Parameters

For the Nakayasu and the f1-Rsa loss method a number of parameters can be specified. Later when specifying the loss method for a UHM catchment a set of parameters from this dialog can be selected by refering to the row number in the dialog.

### 5.9.3 Land use definitions for QLSF method

A number of parameters relating to the UHM method quasi linear storage function method can be specified. Later when defining QLSF catchments the user refers to the row number of this dialog when defining the percentage of area covered by the land use category.

### 5.9.4 Default values for specific method

A number of parameters which can be specified globally only, i.e. they apply to all catchment of the given type, are available. See technical reference for more details on each parameter.

### 5.9.5 Time-fixed combinations

The mean area rainfall calculator selects the weight combination based on the availability of the rainfall stations. However, if desired the selection of the combination may be specified fixed in time. Within this dialog, the time periods for which each of the combinations should be applied can be specified. Time-fixed combination are activated by selecting the check-box on the time-series page.

## 5.9.6 MAW merged output file.

The mean area rainfall for each catchment is saved in a separate file. If desired these files can be combined into one file. This is selected within this dialog, and the filename for the merged file is specified.

## 5.10 Basin View

The Basin View provides a graphical interface for some useful rainfall-runoff modelling tools providing facilities to:

- Digitise catchment boundaries and the location of rainfall stations
- Calculate catchment areas
- Calculate weights used for mean area rainfall calculation

The Basin View is by default not activated when a Rainfall Runoff file is opened or created. It is not required to activate the Basin View for preparation of the RR-file.

### 5.10.1 Activating the Basin View

To activate the Basin View within MIKE 11 select View and Basin View from the top menu bar. When opening a new Basin View the extent of the basin area is defined in the Define Basin Area dialog.
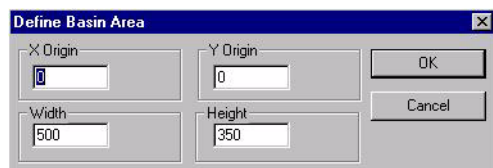


Figure 5.24    Define Basin Area Dialog.

When opening a new Basin View at least one catchment (usually the default) must exist in the Rainfall Runoff Tabular View, which must be open at the same time as the Basin View. The default catchment can afterwards be deleted from the catchment page in the Tabular View, such that the catchments in the Basin View and in the Tabular View are the same.

### 5.10.2 Importing Layers

The layer management tool is used to import a graphical image as background in the Basin View (select "Layers | Layer management" from the menu bar). The graphical image is georeferenced in the image coordinates dialog when importing the layer.

## 5.10.3 Basin Work Area

The Basin Work Area dialog selected from the top menu bar contains the following facilities (see Figure 5.25).
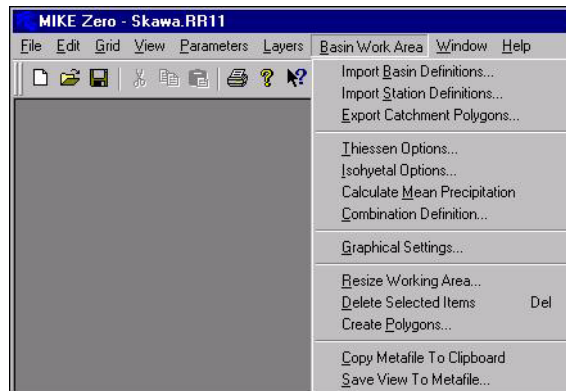


Figure 5.25    Basin Work Area.

### Import Basin Definitions

Import of predefined Basin Definitions from a file with the format:

1 n1

X1 Y1

X2 Y2

.... Xn1 Yn1

2   n2

X1 Y1

X2 Y2

.... Xn2 Yn2

#

Line 1: Index of catchment boundary section and number of pairs of (x,y)-coordinates.

Line 2 - Line4 : (x,y) -coordinates for first catchment boundary section.

Line 5: Index of catchment boundary sections and number of pairs of (x,y)-coordinates.

Line 6- 8: (x,y) -coordinates for second catchment boundary section.

LIne 5 to 8 are repeated for the following sections. The # marks the end of all sections

## Import Station Definitions
Import of predefined Location of Rainfall Stations from a file:

| 1 | 478.2 | 98.0 |
| 2 | 488.5 | 110.1 |
| 3 | 462.5 | 113.2 |
| 4 | 425.0 | 151.9 |

Line 1 - 4: Station index, (x,y)-coordinates.

The lines are repeated until the last station.

## Export Catchment Polygons
Export of Catchment boundaries to a file.

## Thiessen Options
Preparation of Thiessen Weights takes place within the "Thiessen Option"-dialog. Select number 1 for the first combination and press OK (see Figure 5.26). Thiessen weights have now been prepared (see Figure 5.23, Time series page in the Rainfall Runoff Editor).
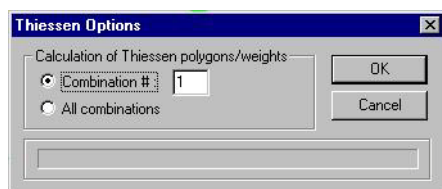


Figure 5.26    The Thiessen Option dialog.

Apply the weight "-1.00" (for stations with missing data) on the Time series page before calculating other combinations.

To show Thiessen polygons for a catchment on the Basin View do the following:

1. Press the Thiessen icon ( ) on the Basin View toolbar.
2. Right click on the basin view.
3. Select the combination number and left click on the catchment.

## Isohyetal Options
The Isohyetal Option is used as a post processing tool to calculate average catchment rainfall for a fixed period based on isohyetal lines. The tool has no

link to data on the Time series page in the Rainfall Runoff Editor. It should therefore be noted that the Isohyetal Option <u>cannot</u> be used to prepare weights and time series of mean area rainfall used as input to the rainfall-run-off calculation. Select the Isohyetal Options to activate the Isohyetal Option dialog (see Figure 5.27). The dialog has the following pages:

1.  Preparation of periods.

2.  Grid Interpolation

3.  Isoline Options

4.  Calculated catchment rainfall based on interpolated isolines

To see the Isolines on the Basin View: Press the Isoline icon on the Basin View toolbar ( ).
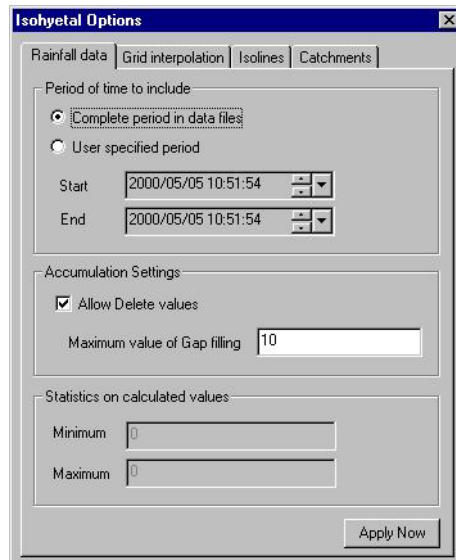


Figure 5.27    Isohyetal Options dialog

## Calculate Mean Precipitation

After having prepared the Thiessen weights (see Figure 5.23, Time series page in the Rainfall Runoff Editor), this option is used to calculate the weighted time series used as catchment mean rainfall for a Rainfall-runoff calculation.

## Combination Definitions

Options used to View different Thiessen Polygons on the Basin View.

## Graphical Settings

Graphical Settings can be modified from the Graphical Settings Dialog (see Figure 5.28).

The "Graphics" page is used to adjust display options for the following graphical objects:

- Basin Web Objects (active when editing or deleting objects)
- Catchment Objects
- Station Objects
- Thiessen Objects

The page "Mouse" is used to adjust the digitizing distance and the Mouse sensitivity for digitizing on the screen.
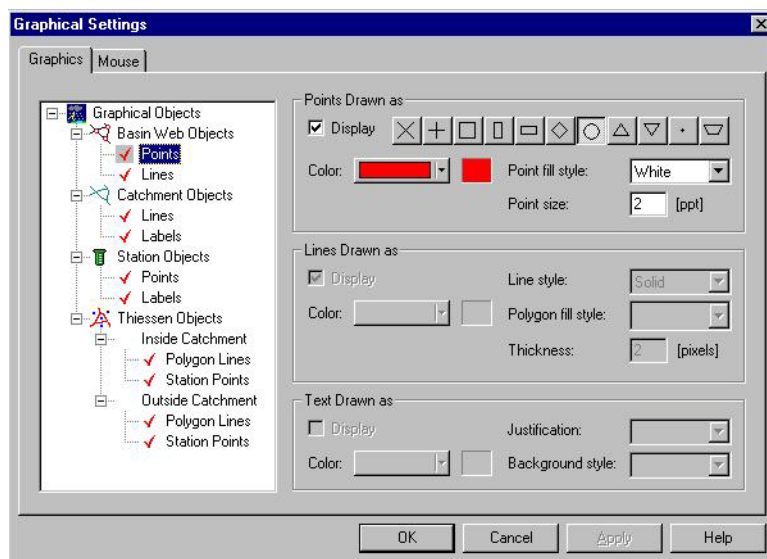


Figure 5.28    Graphical Settings

### Resize Working Area
The working area on the Basin View can be resized from this option.

### Delete selected items
After selection a catchment boundary (press the delete boundary icon:  and click on the actual boundary) or selecting a rainfall station (press the default mode icon:  and click on the actual station) the item can be deleted either by using this option or by pressing the delete button.

### Create Polygons
After having digitized the catchment boundaries this option is used to create catchment polygons (alternatively press the Create Polygon Catchments icon: ). Each catchment will be created in the Rainfall Runoff Editor, including an automatic calculation of the area.

### Copy Metafile to clipboard
The Basin View is copied to a the clipboard.

### Save View to Metafile
The Basin View is saved as a Metafile (*.emf). Afterwards this Metafile can be used as background image in the River Network Editor.

## 5.10.4  Preparing Catchments

### Defining Catchment Boundaries
Defining and editing boundaries is mainly undertaken using the add catchment boundary button ( ) from the Basin View toolbar. The first catchment boundaries are defined as a set of points connected by straight lines forming a polygon. To define the boundaries press the add catchment boundary button and start digitising the first catchment boundary. To close the first catchment boundary polygon double click on the mouse. Digitising of additional boundaries is initiated when selecting the add catchment boundary, clicking on the mouse with the cursor placed close to an existing boundary point. The first boundary line for the second catchment is therefore from the closest existing boundary points to the cursor points.This boundary is closed when double clicking on the mouse close to an existing boundary point.

### Deleting Catchment boundaries
Existing catchment boundaries can be deleted as follows:

1. Press the "Delete Boundary"-icon ( ).
2. Click on the actual boundary to be deleted.
3. Press the delete button.

### Testing catchment
After having prepared the catchment boundaries the "Test fill catchment"-icon ( ) can be used to test the validity of the digitized catchment polygons.

### Create Polygons
Catchments are created within the Rainfall Runoff Editor using the "Create Polygon Catchments"-icon ( ) after having digitized the catchment boundaries. Each catchment will be created in the Rainfall Runoff Editor, including automatic calculation of the area. Catchment names can be modified in the Rainfall Runoff Editor.

## 5.10.5  Inserting Rainfall Stations

### Defining Stations
New rainfall stations are created with the "Create New Stations" icon ( ). Click in the Basin View on the Station Location and use the "Edit Station"-dialog to select the time series and select the name for the Rainfall station (see Figure 5.29).
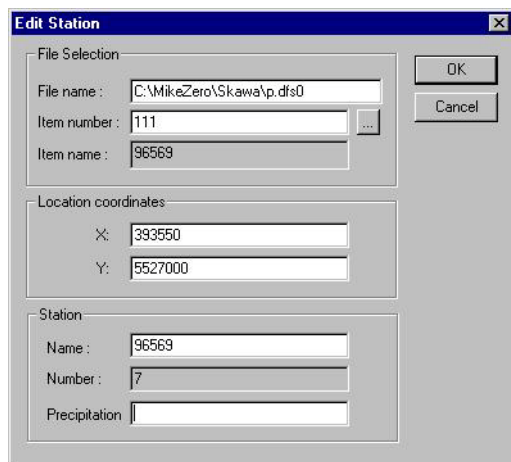
Figure 5.29     Edit Station dialog

### Deleting Stations

Rainfall stations are deleted from the Basin View as follows:

1.  Press the default mode icon: 

2.  Click on the actual station.

3.  Press the delete button.

### Editing Stations

Stations are modified in the "Edit Station" dialog as follows:

1.  Press the default mode icon: 

2.  Right click on the actual station and select "Edit Station".

## 5.10.6   Preparing Thiessen weights

Thiessen weights are prepared from the menu bar (Basin View | Thiessen Options).

Figure 5.30 shows an example of a Basin View for two catchment showing the catchment boundaries, 7 rainfall stations and the Thiessen polygons for all 7 stations.
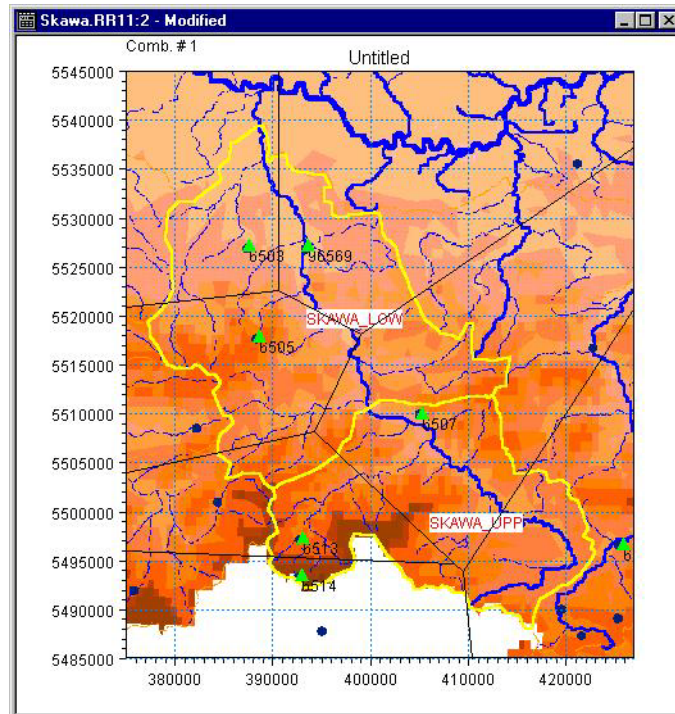
Figure 5.30    Basin View with catchment boundaries, rainfall stations and Thiessen polygons

## 5.11    Result Presentation

### Results

MIKE 11 generates two Rainfall Runoff **Result files**. The first result file contains simulated runoff and net precipitation. The second, additional result file (RRAdd) contains time series of all calculated variables, such as the moisture contents in all storages, the baseflow etc., and can be very useful during model calibration. The results of the simulation can be generated in two formats, either as RES11 or DFS0 filetype (for MIKE 11) or alternatively as RES1D files. The format of the result file should be selected before running the simulation. Three facilities are available to plot and analyse the **results** of a rainfall-runoff simulation:

**1. MIKE View.** To apply MIKE View for result analysis during calibration, use RES11 as result file type. Plot layouts can be generated (and saved) in MikeView for comparing simulated and observed flow while displaying e.g. the Root Zone storage variation, the snow storage, the rainfall, etc.

**2. MIKE Zero Time series Editor** The time series editor can also be used to view and compare simulated and measured results and to export results to e.g. a spreadsheet for further processing. The result file should then be given a DFS0 extension.

**3. MIKE Zero Plot Composer**. The MIKE Zero Plot composer, which also uses DFS0 files, is suitable for arranging final plots for presentation in reports and can also be used in the calibration procedure.

## Summarised output

MIKE 11 generates as standard a table with yearly summarised values of simulated discharge. The table is stored as the textfile "RRStat.txt" in the current simulation directory. The table is extended with observed discharge for catchments, where the time series for observed discharge have been specified on the Time series Page. This includes a comparison between observed and simulated discharge with calculation of the water balance error and the coefficient of determination.

The output from a NAM catchment is extended with summarised values from other components in the total water balance for a catchment. Figure 5.31 shows an example on the content of summarised output.

```
SIMULATED PERIOD : From: 1991/ 1/ 1  12:00   To: 1997/10/31   6:00

TIMESTEP   : 12.00 HOURS

(Accumulated values in mm)

Catchment: SKAWA_UPP, Area=  474.89 km2
---------------------------+-----------------------+-----------------------
Period                     |  Q-obs   Q-sim %diff  |Rainfall PotEvap ActEvap
---------------------------+-----------------------+-----------------------
1991/ 1/ 1 - 1992/ 1/ 1 |    418.2    482.6 -15.4 | 1089.1    551.6    529.1
1992/ 1/ 1 - 1993/ 1/ 1 |    408.7    391.3   4.3 |  825.0    671.5    488.1
1993/ 1/ 1 - 1994/ 1/ 1 |    331.3    316.8   4.4 |  771.8    600.1    502.9
1994/ 1/ 1 - 1995/ 1/ 1 |    458.4    412.6  10.0 |  985.1    656.3    540.0
1995/ 1/ 1 - 1996/ 1/ 1 |    470.3    421.9  10.3 |  944.8    539.5    496.0
1996/ 1/ 1 - 1997/ 1/ 1 |    653.8    685.3  -4.8 | 1127.7    467.5    454.7
1997/ 1/ 1 - 1997/10/31 |    536.5    550.4  -2.6 | 1031.5    519.3    491.3
---------------------------+-----------------------+-----------------------
1991/ 1/ 1 - 1997/10/31 |   3277.3   3260.9   0.5 | 6775.0   4005.9   3502.1
---------------------------+-----------------------+-----------------------
Coefficient of determination: R2 =   0.839
```

Figure 5.31   Example of contents of summarised output from a NAM catchment with observed discharge included.

## Calibration Plot

A calibration plot will automatically be prepared for catchments, where the time series for observed discharge have been specified on the Time series page and the selection of calibration plot has been ticked on the catchment page. The calibration can be loaded from the Plot composed and is saved in the subdirectory RRCalibration with the filename: Catchment-name.plc. The time series in these plots are also available in DFS0 format in the subdirectory RRcalibration with the filename: Catchmentname.dfs0. The plot shows the following results (see Figure 5.32):
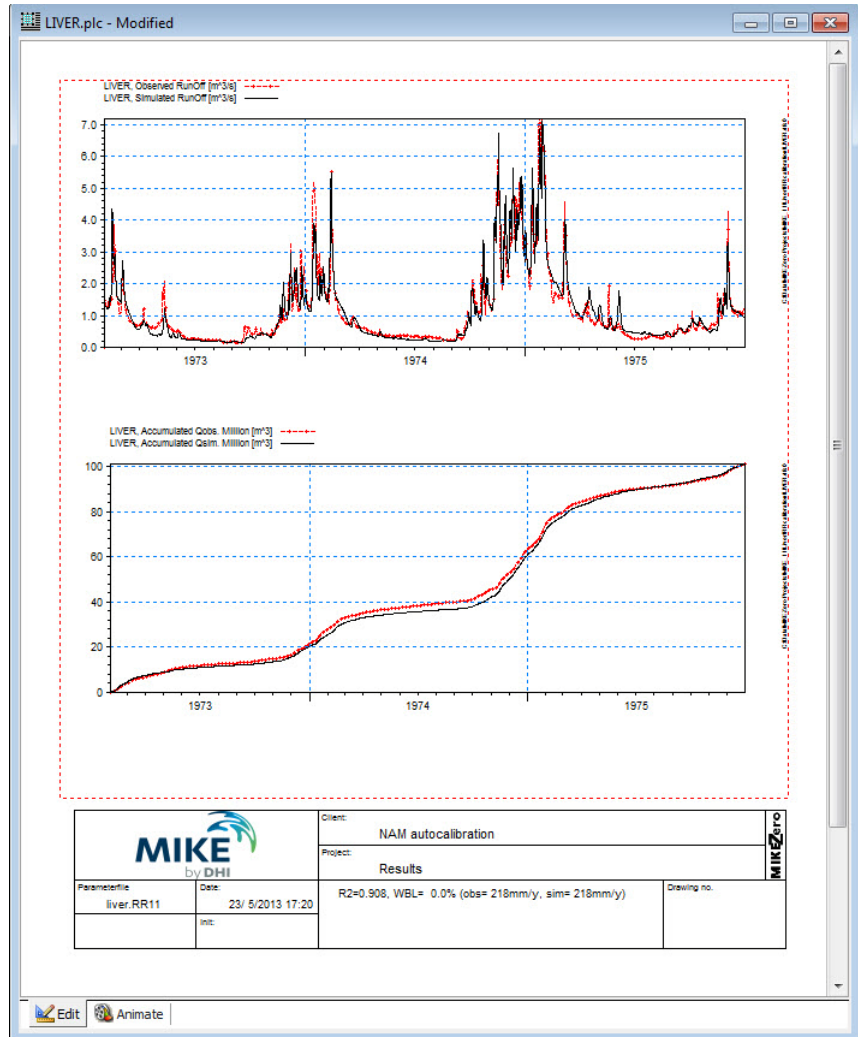
Figure 5.32    Example on a Calibration Plot

- Comparison between observed and simulated discharge.
- Comparison between accumulated series for observed and simulated discharge.
- Values for water balance error and coefficient of determination.

It should be noted that the calibration plot requires the results saved for each simulation time step (See Simulation editor, Results Page).

A combined catchment has no input time series and is therefore not represented on the Time series page. The observed discharge for a combined catchment is therefore included as the observed discharge for the previous catchment on the Time series Page.

## 5.12 A Step-by-step procedure for using the RR-Editor

This section illustrates the steps required to create a rainfall-runoff model setup, and then carry out an auto calibration and model simulation The example is based on the Skawa catchment, which is located in the Upper Vistula Basin in Poland. The figures presented in this chapter describing the Rainfall Runoff Editor are taken from this example. The following steps were performed:

1. **Opening a new MIKE 11 RR - Parameter file**. A catchment must be defined in the first "Insert Catchment"-dialog (see Figure 5.4). This catchment is used to initialize the Rainfall Runoff Editor for the Basin View.

2. **Activating the Basin View** (select View | Basin View).

3. **Importing a background images** (select Layers | Layers management). The imported image was prepared and geo-referenced from an ArcView application.

4. **Digitising catchment boundaries**. The catchment was subdivided into two sub-catchments defining the Upper and Lower part of the Catchment (see Section 5.10.4).

5. **Creating polygon catchments** (see Section 5.10.4), which includes the preparation of the two NAM sub-catchments in the Rainfall Runoff Parameter file with automatic calculation of the catchment areas. Default catchment names are automatically assigned to each catchment. The names of the two catchment were modified to SKAWA_UPP and SKAWA_LOW and the default catchment was deleted from the Catchment Overview.

6. **Setting up a combined catchment**. A Combined catchment was defined as the sum of the two sub-catchments (see Figure 5.3).

7. **Inserting the rainfall station**s. Stations included in the calculation of catchment rainfall were included in the Basin View (see Section 5.10.5).

8. **Preparing Thiessen Weights** (see Section 5.10.3, Thiessen Options). The calculated Thiessen Weight Polygons are shown on Figure 5.30, which also shows the two sub-catchments. The weights which were automatically transferred to the Time series page are shown in Figure 5.23.

9. **Calculating Mean Precipitation**. The Weighted time series were calculated based on the weights prepared as described in the previous step.

10. **Setting up other input time series**. Input Time series for Evaporation, Temperature and Observed Discharge were included on the Time series page (Figure 5.23).

11. **Setting up NAM snow melt parameters**. The Skawa catchment is located in the mountains ranging from 200-1500 m above sea level. and the runoff is therefore influenced from snow melt for part of the year. The NAM setup was prepared with the extended snow melt component including elevation zones (see Figure 5.7). Areas of the elevation zones were prepared from a Digital Elevation Model. Areas were afterwards copied to the "Elevation zone" dialog via the clipboard. Temperature corrections were finally calculated using a fixed temperature lapse rate (see Figure 5.8).

12. **Initial Conditions**. The simulation starts from the beginning of a year with relative high moisture content in the soil. The Initial Conditions for the Upper and the Lower zone were therefore estimated to respectively 100% and 90% of maximum capacity (see Figure 5.11).

13. **Setting up Autocalibration for the upper catchment**. Estimation of parameters in the upper catchment were based on the NAM auto calibration routine. The auto calibration was based on minimising the Overall Water Balance error and the Overall Root Mean Square error with a maximum of 2000 iterations (see Figure 5.12).

14. **Starting the simulation editor**. After having saved the Rainfall Runoff Parameters, a MIKE 11 simulation editor was opened. The Input page includes the RR Parameter file (see Figure 5.1). The Simulation period was prepared from the Apply default button and a time step of 12 hours was found appropriate for the simulation (see Figure 5.2).

15. **Estimating RR-parameters for the lower catchment**. The parameters for the lower catchment were estimated based on results from the auto calibration of the upper catchment and the knowledge of a lower response and higher storage capacity for a catchment close to the flood plains compared to the more hilly upper catchment. Parameters in the Surface-Rootzone and Ground water are shown in Figure 5.5 and Figure 5.6. The values for the 3 most important parameters are (in bracket values for the upper catchment): Maximum water content of rootzone: 200 mm (100 mm), Runoff coefficient: 0.7 (0.83) and Time Constant Overland flow: 13.6 hours (20 hours).

16. **Presenting Results**. Results from the simulation were finally compared in tables and in plots. Figure 5.31 shows an example of summarised output from the upper catchment, while Figure 5.32 shows the calibration plot for the upper catchments.

# HYDRODYNAMIC EDITOR

# 6 Hydrodynamic Parameters Editor

The Hydrodynamic parameters editor (HD-editor) is used for setting supplementary data used for the simulation. Most of the parameters in this editor have default values and in most cases these values are sufficient for obtaining satisfactory simulation results. The HD parameter editor contains a number of tabs as presented in Figure 6.1 below.
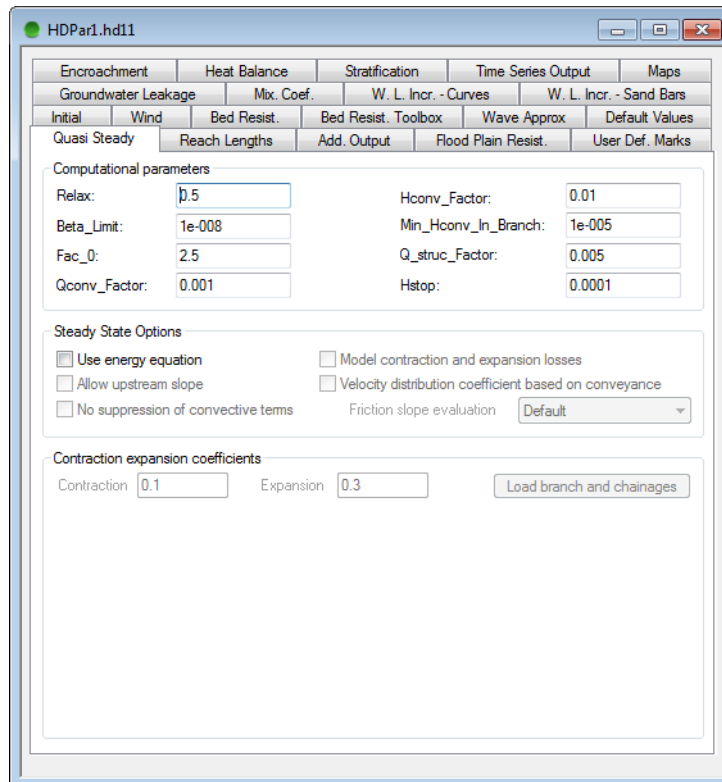


Figure 6.1     The HD Parameters Editor - opening view

A list of the tab-page available in the Editor are presented below and described in the following: paragraphs.

- Initial *(p. 312)*
- Wind *(p. 313)*
- Bed Resistance *(p. 315)*
- Bed Resistance Toolbox *(p. 318)*
- Wave Approximation *(p. 321)*
- Default Values *(p. 323)*
- Quasi Steady *(p. 325)*

- Heat Balance *(p. 328)*
- Stratification *(p. 330)*
- Time Series Output *(p. 333)*
- Maps *(p. 335)*
- Groundwater Leakage *(p. 345)*
- Groundwater Leakage 345
- Reach Lengths *(p. 346)*
- Add. Output *(p. 347)*
- Flood Plain Resistance *(p. 351)*
- User Def. Marks *(p. 352)*
- Encroachment *(p. 354)*
- Mixing Coefficients *(p. 359)*
- W. L. Incr.- Curve *(p. 360)*
- W. L. Incr.- Sand Bars *(p. 363)*

**Note:** The HD Parameter editor is configurable and the number of tab-pages presented when opening or creating a new HD Parameter file depends on the content of your MIKE 11 license! Therefore, the view as presented in Figure 6.1 can differ in the sense that some tabs are visible to you and some are not. This is an intended behaviour and you should therefore not be concerned about obtaining different Editor-appearance on different license-configurations.

## 6.1    Initial

Initial conditions for the hydrodynamic model are specified on this page. The initial values may be specified as discharge and as either water level or water depth. The radio button determines whether the specifications are interpreted as water level or depth.The global values are applied over the entire network at the start of the computation. Specific local values can be specified by entering river name, chainage and initial values and if defined, local values will override the global specification.

New lines in the Local Values table are created either by placing the cursor in the last column and pressing <TAB> or through the 'Grid' option in the main menu bar;.

An example on applying a combination of Global and Local values are presented in Figure 6.2. The global water level and discharge have been specified as 5.0 and 1.4 respectively and Local values have been specified in the branch "RIVER 1".

This definition will result in a local varying initial water depth from 5.70 to 5.00 with a linear relationship between chainage 0 and 3000 and the discharge also varies between 1.000 and 1.400 with a linear relationship between chainage 0 and 3000. At any other locations in the model, Global values will be applied as the initial value.
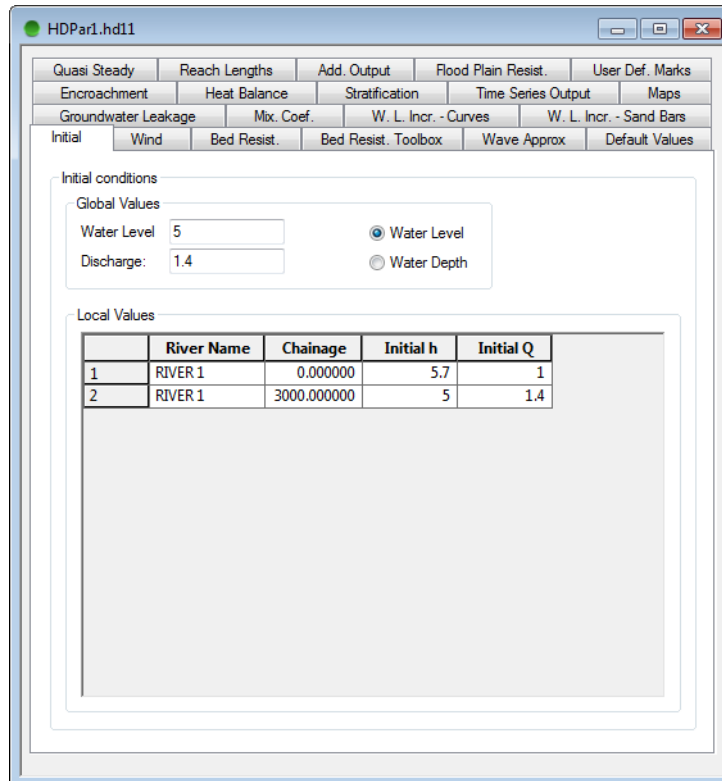


Figure 6.2    Initial value tab

Note: Linear interpolation will be performed automatically between the defined locations (river name + chainage) in the Local Values table. Hence, it is not required to manually define initial condition values in every single calculation point in the model. Local Values should only be defined at locations where change in values should be applied.

## 6.2    Wind

Wind friction on the water surface can be accounted for by adding wind shear stress in the simulation. Wind fields can be applied to the entire model network using the wind property page.

The property page contains an "on/off" switch to either include or exclude wind shear stress in the simulations (default is 'off') and the possibility to define Global as well as local wind factors.

MIKE 11 includes wind in simulations by combining the factors defined in the present page with a wind field defined by wind speed and wind direction. Wind fields are defined as boundary conditions in the Boundary Editor.

The wind factors defined in the present page is applied to the simulations in the following way:

- *Topographical Factor*:
  The topographical factor is a multiplication factor which is multiplied to the Wind Velocity, and hence by optional use of Local Factors it is possible to adjust the Wind Field applied with different values at different locations.

- *Friction Factor:*
  Friction Factor is a global parameter used in the calculation of exchange between vertical layers and as such, the Friction factor is only applicable to Stratified flows or MIKE 12 applications.

An example of applying the Wind friction is presented in Figure 6.3:

To activate Wind friction, remember to activate the 'Include Wind' tick-mark. The global wind factor is set to 0.70 and local variation of wind factors are defined as a linear change from 0.70 to 0.30 in the branch named "RIVER 1" from chainage 0 to 5000.
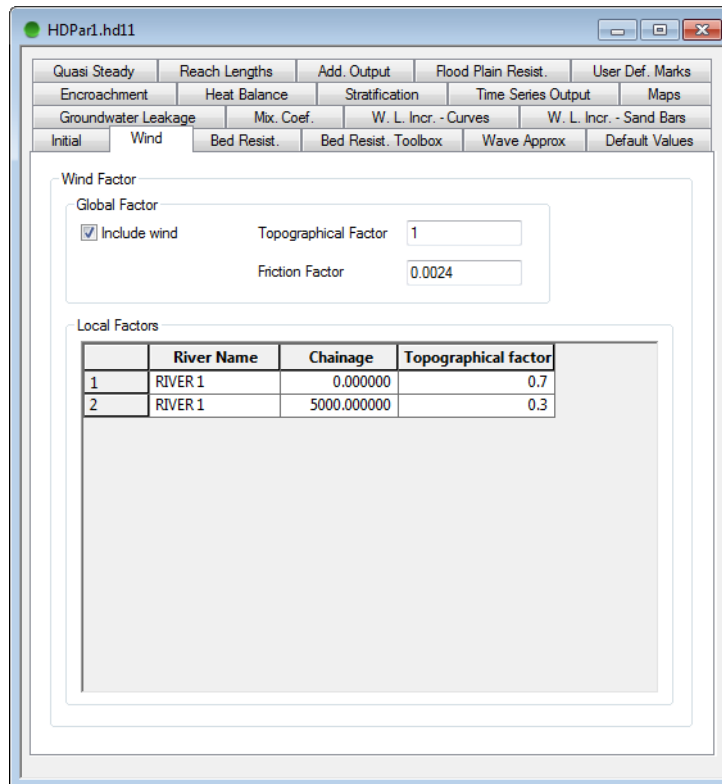
Figure 6.3    Wind tab

## 6.3    Bed Resistance

Two approaches may be applied for the bed resistance. Either a uniform or a so-called triple zone approach.

### 6.3.1    Uniform approach

The bed resistance is defined by a Resistance Formula and a corresponding global value. Local values are entered in the Local Values table at the bottom of the page

Three resistance type options are available:

1.  Manning's $M$ (unit: $m^{1/3}$/s, typical range: 10-100)
2.  Manning's $n$ (reciprocal of Manning's $M$, typical range: 0.010-0.100)
3.  Chezy number.

After defining a resistance formula and the Global resistance and Local resistance values if required, the resulting bed resistance number applied in

the simulation is the defined resistance numbers multiplied by the water level depending 'Resistance factor' which is specified for the cross sections in the cross section editor (.xns11 files).

Note that the resistance is only used if relative resistance has been selected in the corresponding cross sections.

Example (Figure 6.4): A global resistance (Manning's *M* type) of 30 is specified. In the branch "RIVER 1" local resistance numbers are specified between chainages 0 and 21000 m. The resistance number at intermediate chainage values is calculated linearly.



Figure 6.4    Uniform approach for implementation of the bed resistance

## 6.3.2    Triple zone approach

The Triple Zone Approach offers a possibility for the user to divide the river sections in three zones with different bed resistance values. These zones represent the vegetation free zone in the bottom of the profile, a vegetation zone on banks etc. and a zone for description of flow over banks and flood plains etc. as indicated in Figure 6.5
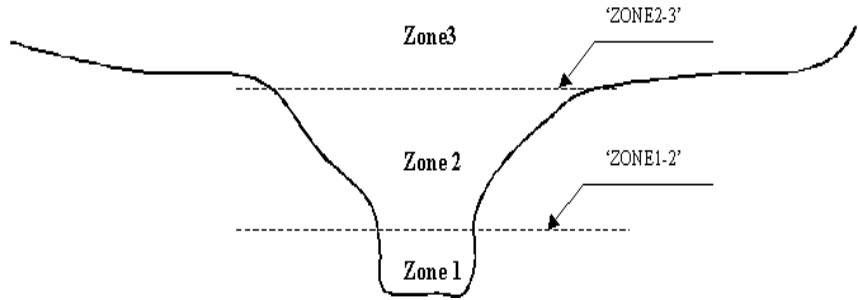
Figure 6.5     Triple Zone division of cross section

Zone separator lines must be defined in the User Defined Markers page (see description in Activation of Bed resistance Triple Zone Approach *(p. 353)*).

Global and local values of bed resistance for each zone can be specified as described for the Uniform approach (see Figure 6.6).



Figure 6.6     Triple Zone approach for Bed Resistance

Due to the special description in the friction term in the higher order fully dynamic wave description the triple zone approach is only available for fully dynamic and diffusive wave descriptions.

### 6.3.3 Vegetation and bed resistance

It is evident that the Bed Resistance - amongst other parameters - depend on vegetation growth and well-defined relationships between vegetation growth and flow conditions are required.

However, only few detailed investigations have been made to establish relationships between flow resistance and vegetation growth. A quantitative evaluation of the influence of vegetation on the flow resistance has been performed in a few Danish gauging-programmes. These are referred to in Appendix A.1 Flow Resistance and Vegetation *(p. 491)*.

## 6.4 Bed Resistance Toolbox

The bed resistance toolbox offers a possibility to make the program calculate the bed resistance as a function of the hydraulic parameters during the computation by applying a Bed Resistance Equation. The Bed resistance toolbox page is presented in Figure 6.7 below.
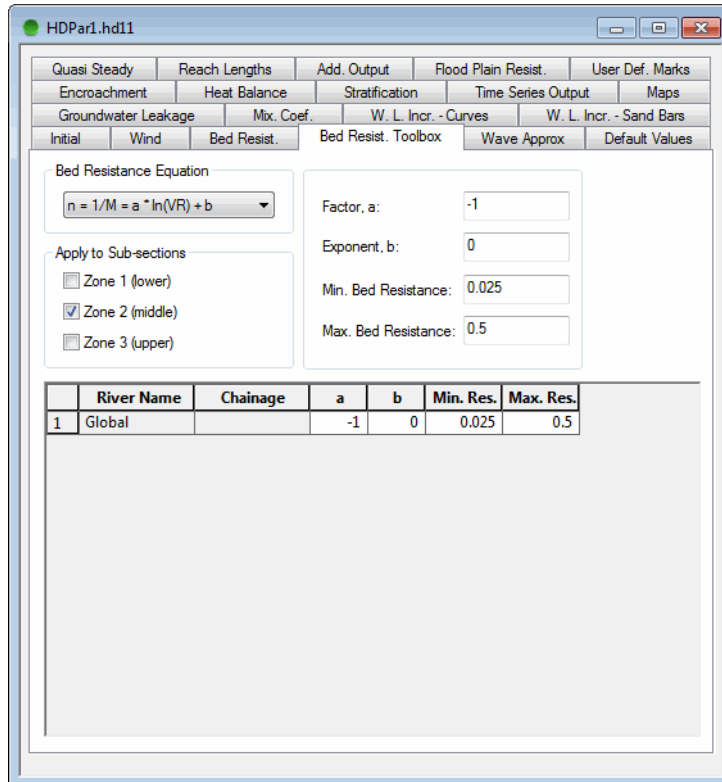
Figure 6.7    The Bed Resistance Toolbox property page

Five options are available in the Bed Resistance Equation combo box:

- **Not Active**

  Bed resistance values used in the computation are those specified in the Bed Resistance page (Uniform or Triple zone approach)

- $n = 1/M = a*\ln(VR)^b$

  The bed resistance is calculated as a function of ln(velocity*Hydraulic Radius).

- $n = 1/M = a*D^b$

  The bed resistance is calculated as a function of the depth.
  Please note that the Depth is calculated as an average depth (or 'Hydraulic Depth') to avoid local deep parts of a section to control the resistance for the entire cross section. D is therefore calculated as: D = Area / Width.

- $n = 1/M = a*V^b$

  The bed resistance is calculated as a function of the velocity.

- Table (Velocity, Resistance value)

  A User defined table of resistance value as a function of actual velocity can be defined. The bed resistance value applied in the simulation will be the interpolated value from this table, depending on the actual velocity.

**Note**: To define the first line in the table, click the 'Velocity' bar in the upper half of the page. Thereafter, press the <TAB> button and a new line will be present in the grid in the upper part of the page.

All features (equations and table) can be defined both globally and locally.

If a Triple Zone Approach is applied, it can be specified for which zones the bed resistance should be based on the toolbox definitions and which zones the bed resistance number should be taken from the Bed Resistance page. Activate the 'Apply to Sub-sections' check-boxes to specify that for a specific zone the bed resistance values must be determined from the toolbox definitions.

If one of the equations has been applied, the user must define values for the coefficient, a, and exponent, b. Additionally, a minimum and a maximum value must be specified to control, that bed resistance values calculated from the equations are inside the interval considered reasonable by the user for the specific setup.

Note that when using a Chezy or Manning description the maximum bed resistance requires the smallest Manning's M or Chezy's C. Similar for the minimum bed resistance requiring the highest resistance number.

Further due to the special description in the friction term in the higher order fully dynamic wave description. The bed resistance toolbox is only available for fully dynamic and diffusive wave descriptions.

## 6.5    Wave Approximation
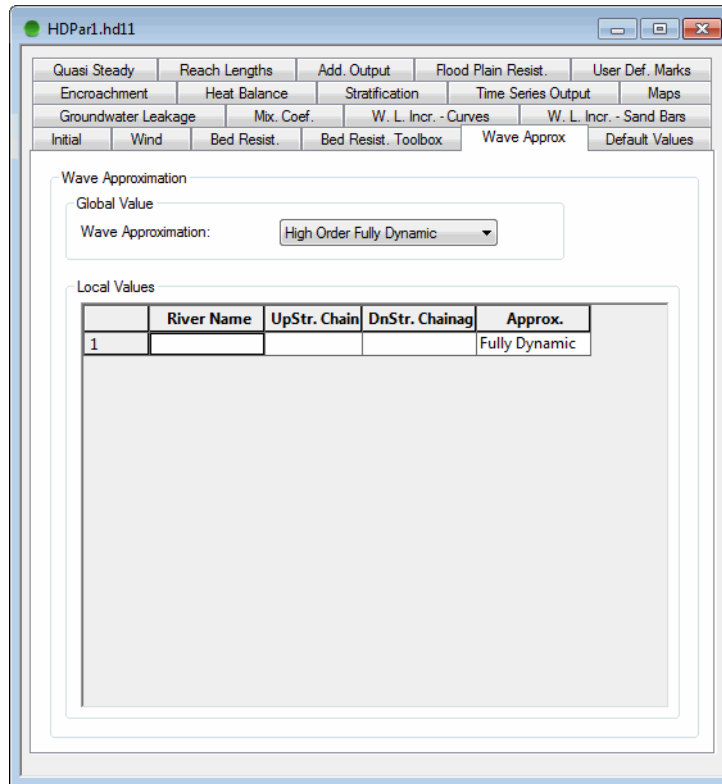


Figure 6.8    The Wave Approximation property page

There are four possible flow description available in MIKE 11. The flow descriptions can be selected globally for the system and/or locally for individual branches. Locally specified flow descriptions must be specified for the whole branch.

In general it is recommended to use the 'fully dynamic' or the 'high order fully dynamic' flow descriptions. Only in cases where it can be clearly shown that the 'diffusive wave' or the 'kinematic wave' are adequate should they be used. The latter two flow descriptions are simplifications of the full dynamic equations. These are provided to improve the computational efficiency of models in specific circumstances. They should only be used when the simplifications/assumptions upon which they are based are valid (see below).

### 6.5.1 Fully Dynamic and High Order Fully Dynamic

The 'fully dynamic' and 'high order fully dynamic' flow description should be used where the inertia of the water body over time and space is important. This is the case for all tidal flow situations and in river systems where the water surface slope, the bed slope and the bed resistance forces are small.

The 'high order fully dynamic' flow description contains specific high order and upstream centred friction terms in the momentum equation. This modification typically allows simulations to be performed at longer time steps than the 'fully dynamic' description.

### 6.5.2 Diffusive Wave

The diffusive wave description is a simplification of the full dynamic solution and assumes that there are no inertial forces (i.e. the inertial terms are dropped from the momentum equation). It is suitable for backwater analysis slow propagating flood waves and for cases where the bed resistance forces dominates. It is not suitable for tidal flows.

### 6.5.3 Kinematic Wave

The kinematic wave approach assumes a balance between the friction and gravity forces on the flow. The description is suitable for relatively steep rivers without backwater effects.

## 6.6    Default Values
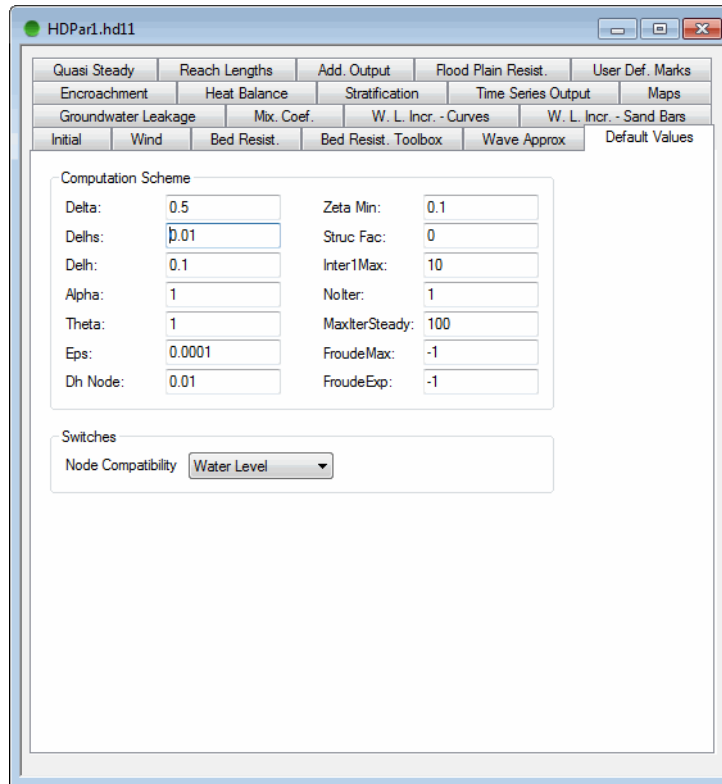


Figure 6.9     The Default Values property page

The default value property page contains various parameters related to the computational scheme. These parameters are essential for the simulation and have been given default values. The parameters can be modified if required.

In the following, a brief descriptions are provided of the different parameters (see also section Coefficients, HD default parameters in the MIKE 11 Reference Manual).

### 6.6.1    Computation Scheme

#### Delta
The time-centring of the gravity term in the momentum equation. A value of 0.5 will produce the most accurate calculations provided there are no numerical instabilities, but instabilities are less likely with higher Delta values. For MIKE FLOOD models incorporating MIKE 11 channels, the smaller time steps are usual and Delta of about 0.85 generally provides accurate calculations and stable computations. Large value of Delta (towards 1.0) has a sig-

nificant dissipative effect which can significantly influence model dynamics and high values of Delta should therefore be avoided especially for applications with strong and systematic dynamics such as in e.g. a tidal range application.

### Delhs
The minimum allowable water level difference across a weir. To obtain a steady solution for differences below this limit a linear flow description is used.

### Delh
The Delh factor controls the dimensions of an artificial 'slot', which is introduced to a cross section to prevent 'drying out' of the section. The artificial slot is a small void introduced at the base of the section and allows a small volume of water to remain in the section preventing computational instabilities at low flows. The slot is inserted at height Delh above the river bottom and extends to a depth of 5·Delh below this level.

### Alpha
The velocity distribution coefficient used in the convective acceleration term of the momentum equation.

### Theta
A weighting factor used in the quadratic part of the convective acceleration term of the momentum equation.

### Eps
The water surface slope used in the diffusive wave approximation. If the water surface slope becomes greater than EPS, the computational scheme will become fully forwarded upstream. The parameter can be used to control the stability of the computation.

### Dh node
Not used.

### Zeta min
The minimum head loss coefficient allowed in the computation of flow over structures.

### Struc Fac
Not used.

### Max Iter Struc (Inter1Max)
The maximum number of iterations permitted at each time step to obtain a solution at a structure.

### No. Iterations (NoIter)
The number of iterations at each time step, generally 0, 1 or 2. Therefore, a number of 0 here results in zero iterations, but naturally one calculation on

each time step is performed, and a number of 2 will result in 3 calculations of the hydrodynamic simulations on every time step.

### Max Iter Steady

The maximum number of iterations used to obtain a steady state water level profile at the start of a simulation. Only used when the initial conditions for the simulation are either 'steady' or 'steady + parameter'. If the simulation type is 'Quasi steady' then the parameter is used at each time step.

### Froude Max and Froude Exp

'Froude Max' is the parameter 'a' in the enhanced formulation of the suppression term applied to the convective acceleration term in the momentum equation. Similarly 'Froude Exp' is the parameter 'b' in the enhanced formulation. By default the values are -1, indicating that the traditional formulation is used. For situations with high Froude numbers combined with small grid spacing the enhanced formulation can be applied, see section Suppression of convective acceleration term in the Reference Manual.

## 6.6.2    Switches

### Node Compatibility

This switch should be set to water level since the energy compatibility is not available.

## 6.7    Quasi Steady

Various parameters required for the quasi steady simulation to be carried out are set here.
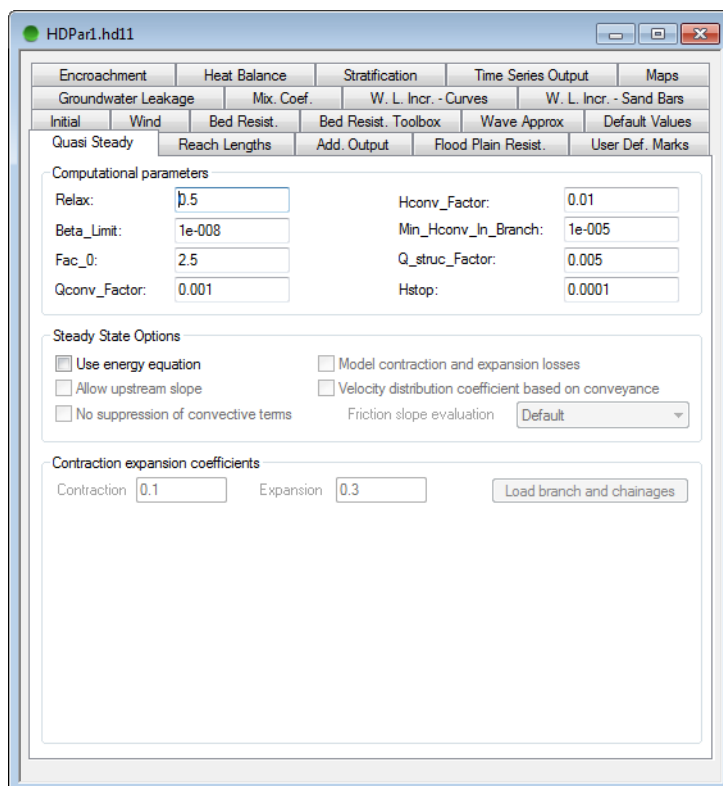
Figure 6.10    The Quasi Steady property page

## 6.7.1    Computational parameters

In order to optimize the convergence parameters with respect to accuracy and computational time it is recommended that the parameters be adjusted to obtain a satisfactory solution for low flow conditions. This will lead to accurate results for higher flow conditions as well.

The optimization is carried out by running the hydrodynamic model for constant low flow conditions until steady conditions are obtained. These results can then be compared with those obtained using the quasi-steady model. It is emphasized that the parameters are 'model specific', i.e. each model setup and associated flow condition requires individual parameter optimization.

### Relax
Weighting parameter used in the quasi-steady solution. For single branches without bifurcation the value should be 1. In more complex systems the value should be less than 1.

### Beta_Limit
Factor used to avoid underflow in 'horizontal' branches.

### Fac_0
Factor used to control the stop criteria for the discharge convergence test.

### Qconv_factor
Q convergence factor used in the stop criterion for the backwater computation iterations.

### Hconv_factor
H convergence factor used in the stop criterion for the backwater computation iterations.

### Min_Hconv_In_Branch
Minimum stop criterion to avoid underflow.

### Q_struc_factor
Q structure factor: Used to determine the discharge at structures where a slot description is introduced due to zero flow conditions.

### H_stop
Stop criteria in the water level convergence test. Used also by the quasi two dimensional steady state solver with vegetation as the convergence criteria in the outer loop.

## 6.7.2 Steady state options

The steady state options are accessed by setting the switch 'Use energy equation'. This also indicates that the options are only available for steady state flow situations using the energy equation as the governing equation.

### Allow upstream slope
This switch allows solutions where the water surface is sloping in the opposite direction of the flow. If this switch is off the solver will project the downstream water level to the upstream location and add 1 mm in situations where the water level is sloping in the opposite direction of the flow. Only turn off in situations where the calculated water level does not seem to be within an acceptable range.

### No suppression of convective term
In some cases where the flow is in or close to the super critical range the solution algorithm may have trouble converging. This is handled by MIKE 11 through the introduction of a suppression term which varies with the Froude number. So that for full super critical flow the convective terms in the governing equation are fully suppressed. If this suppression is not desired please set this switch to on.

### Model contraction and expansion losses
This switch will allow the inclusion of expansion and contraction losses in the energy equation. When this is activated the lower part of the page should be used to input the contraction and expansion loss coefficients. Note that the

contraction/expansion loss criteria is based on the difference in the velocity head upstream versus downstream.

### Velocity distribution coefficient based on conveyance

By default the velocity distribution coefficient used by MIKE 11 is 1 (user defined under default values). Using this switch the velocity distribution coefficient will be calculated based on the conveyance distribution in the cross section.

### Friction slope evaluation

This option allows the user to select the method for calculation of the friction slope. Five options are available which are documented in the reference manual.

## 6.7.3 Contraction and expansion loss coefficients

If the user has selected to model contraction and expansion losses the coefficients must be specified. The user may choose to only give global values which are given in the top two fields above the table. If the user would like to specify either values throughout or at selected locations the lower table should be used. Note that by the use of the button on the right "Load branch and chainages" the table can be populated with all h-point locations in the set-up, the user then simply edits the parameters to be used at the different locations.

## 6.8 Heat Balance

The property page used for setting up heat exchange simulations is illustrated in Figure 6.11.

Figure 6.11    The heat balance property page

The information needed for the heat exchange calculation are (information is also needed in the boundary-editor):

## Latitude (N pos.)
Latitude of the considered area. Used in solar radiation calculation.

## Longitude (W pos.)
Longitude of the considered area.

## Time meridian zone (W pos.)
The standard longitude for the time zone.

## Displacement in time
Summertime correction: +1hour if the clock is 1hour ahead.

## Light attenuation
Attenuation of solar radiation in the water column. Used to distribute the incoming solar radiation over the different layers.

### Constant in Beer's law

The incoming solar radiation is distributed over the layers by use of the following formula:

$$E_{fac} = \frac{I_{sun}}{I_{tot}}(1 - \beta)\exp(-a(D - z))$$
(6.1)

where $I_{sun}$ is the solar radiation, $\beta$ is constant in Beer's law, $(D\text{-}z)$ is distance from surface and $a$ is light absorption.

### Radiation Parameter *A*

Daily radiation under cloudy skies is determined by:

$$\frac{H}{H_0} = A + B\frac{n}{N_d}$$
(6.2)

where $n$ is sunshine hours and $N$ is the day length.

### Radiation Parameter *B*

See above.

### Vaporization Parameter *A*

Vaporative heat loss is determined by:

$$q_e = LC_e(A + BW_2)(Q_W - Q_a)$$
(6.3)

where $L$ is latent heat of vaporization, $C_e$ is the moisture coefficient, $W_2$ is the wind speed 2 m above surface, $Q_w$ is the vapor density close to the surface, and $Q_a$ is the vapor density close to the surface.

### Vaporization Parameter *B*

See above.

## 6.9    Stratification

The property page used for setting up stratified flow simulations is illustrated in Figure 6.12.
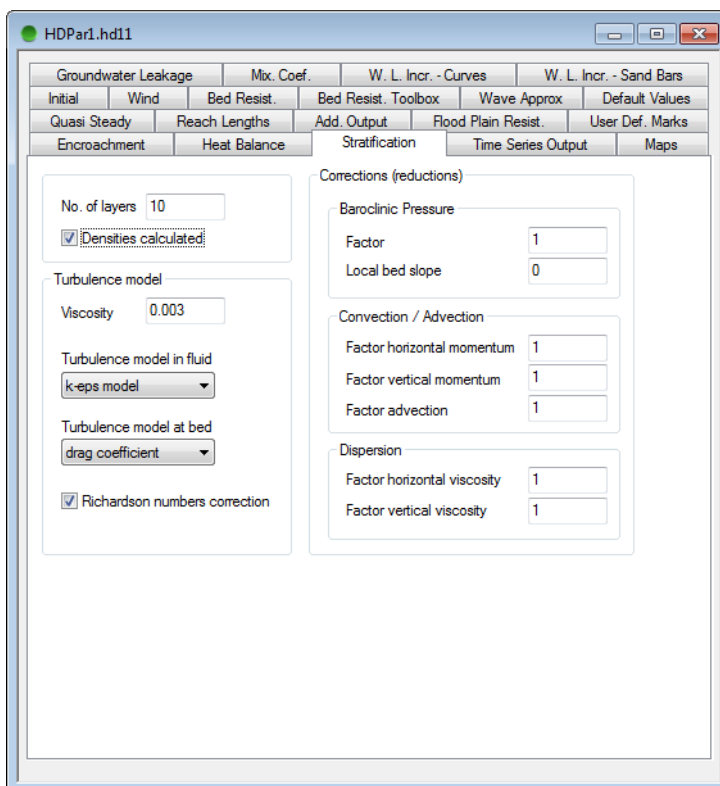
Figure 6.12    The stratification property page

Note that if stratified flow is to be simulated then the specific branch must be defined as being stratified.

The information needed for the stratified branches are:

### No. of layers
Number of layers in the stratified branches. The same number of layers is assumed in all stratified branches. The thickness of a layer is equal to the local depth divided by the number of layers.

### Density calculated
Tick means yes and no tick means no. If densities are calculated it is done on the basis of the simulated water temperatures, and if not density is assumed to be 1000 kg/m3.

## Turbulence model

### Viscosity
In the case one chose a constant viscosity under turbulence model it is the used viscosity in the calculations.

### Turbulence model in fluid

It is possible to chose between constant viscosity, mixing-length, $k$-model and $k$-$\varepsilon$ turbulence models. It is recommended to use the $k$-$\varepsilon$ model.

### Turbulence model at bed

Presently only drag coefficient can be chosen. The Chezy or Manning number specified is used to calculate the bed friction, see scientific documentation.

### Richardson number correction

Tick means yes and no tick means no. If Richardson numbers correction is active the turbulence is dampened in stable stratified areas.

## Corrections (reductions)

### Baroclinic pressure: Factor

A factor multiplied on the Baroclinic pressure. Default is 1, whereby the correct equation is solved. If the factor is 0 the Baroclinic pressure term is removed in the momentum equation.

### Baroclinic pressure: Local bed slope

The higher the number the less the Baroclinic pressure term is taken into account in areas with steep bed gradients.

### Convection /Advection: Factor horizontal momentum

A factor multiplied on the horizontal exchange of horizontal momentum ($uu$). Default is 1, whereby the correct equation is solved. If the factor is 0 the term is removed in the momentum equation.

### Convection /Advection: Factor vertical momentum

A factor multiplied on the vertical exchange of horizontal momentum ($uw$). Default is 1, whereby the correct equation is solved. If the factor is 0 the term is removed in the momentum equation.

### Convection /Advection: Factor advection

A factor multiplied on the advection terms in the transport equation. Default is 1, whereby the correct transport equation is solved. If the factor is 0 the advection of matter is removed in the transport equation.

### Dispersion: Factor horizontal viscosity

A factor multiplied on the turbulent viscosity to get the horizontal diffusion in the transport equation.

### Dispersion: Factor vertical viscosity

A factor multiplied on the turbulent viscosity to get the vertical diffusion in the transport equation.

## 6.10 Time Series Output

On this property page request is made for time series output files to be generated during the simulation. This output is in addition to the regular and the additional *.res11 or *.res1d output file. Time series output can be saved in .dfs0 or ASCII files. Time series output files are typically requested in stead of manually extracting time series data in selected grid points from the *.res11 or *.res1d file after the simulation has been completed. This is often useful for automatic or manual calibration or when running production simulations. The time series output page is shown in the following figure:



Figure 6.13    Time series output property page in HD parameter editor

### 6.10.1  Generating Time Series Output Files

Selecting this option, the time series output files will be generated during the simulation. For each row in the grid control one time series output file will be generated. The format column gives three choices for the file format:

- Dfs Time series Bridge: This will generate a .dfs0 file in the standard DHI file format for time series. Files can be loaded into for instance the time series editor, plot composer, MIKE View or used as simulated data for the auto calibration tool.

- ASCII format 1 Time series Bridge. This will generate an ASCII file with a column based format. There will be one time column and one column for grid point item selected for output. The number of lines will equal the number of saved time steps.

- ASCII format 2 Time series Bridge. This will generate an ASCII file with a table based format. There will be one table for each time step saved. Each table will have rows corresponding to the number of selected grid points, and columns corresponding to the number of selected items.

## 6.10.2  Text File Settings

These check boxes only apply for the two ASCII file formats.

- Minimum: A separate table at the bottom of the file will show the minimum value of the output items.

- Maximum: A separate table at the bottom of the file will show the maximum value of the output items.

- Time of minimum: A separate table at the bottom of the file will show the time of minimum value of the output items.

- Time of maximum: A separate table at the bottom of the file will show the time of maximum value of the output items.

- Delimiter in output file: Between each column a special character like semicolon, comma etc can be requested. This makes import to for instance Excel easier.

- Width of columns: The desired minimum width of each column can be specified.

## 6.10.3  Time Period in Output

Selected, this check box makes it possible to define the period for which the time series output is generated. If not selected, the full simulation period will apply. The saving frequency is always equal to the saving frequency for the regular *.res11 or *.res1d output file as specified in the simulation editor.

### 6.10.4   Items in Output

Time series output is produced for the selected items of which some are dynamic and some are static. Dynamic items are items between "Water level" and "Bed shear stress". In the dfs0 files only dynamic items will be saved. In ASCII items both dynamic and static items will be saved.

A user defined name for the item can be specified. If not specified, the default name will apply.

The number of decimals will only apply for output in ASCII files.

### 6.10.5   Grid Points in Output

The grid points from which the time series output is to be generated are specified here. Using the radio button to the right, the user may choose to generate time series output from all, selected or all-but-selected grid points. The selection of grid points is done through specifying a number of reaches (river name, upstream chainage and down stream chainage). Single grid points can be selected by setting the upstream chainage equal to the downstream chainage.

Time series output is only generated at h-points.

## 6.11   Maps

MIKE 11 may be used to produce two dimensional maps based on the one-dimensional simulations. The maps are made as dfs2-files (rectangular grids) and constructed through interpolation in space of the grid point results. Thus the maps constructed in this way should be viewed as a two dimensional interpretation of results from a one-dimensional model. For MIKE 11 simulations producing a map as output, the maps will therefore be calculated only within the extent of cross sections defined in the mapping area as illustrated in Figure 6.14.
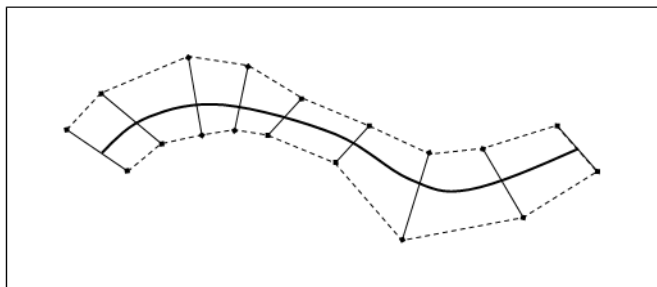
Figure 6.14    MIKE 11 Mapping feature; Illustration of area included in map calcula-
tion. Thick black line represents river line, cross section lines along the
river and the dashed polygon represents the area for the MIKE 11 simu-
lation and hence, the active area where simulation results will be trans-
ferred into a map representation

No calculation of map-values will take place outside the cross sections
extent, and the maps obtained, therefore represents exactly what the model
calculates during the actual simulation.

An example of a flood map produced through the map-feature is presented in
Figure 6.15:



Figure 6.15    An example of a flood inundation map presented in the Result Viewer

Various simulation results may be mapped. The complete list (selected from
the 'Item' column in the property page) consists of:

● water level

● water depth

- velocity

- velocity times depth

- advection dispersion component

- (h, p, q)

The maps may be of three types:

- minimum values
  the overall minimum value throughout the simulation period for each grid cell included in the map.

- maximum values
  the overall maximum value throughout the simulation period for each grid cell included in the map.

- dynamic
  Time-varying grid-file - for animations etc.

Only dynamic maps require a time span to be specified.

Through the mapping page, the user specifies the area that is to be mapped, the type of map to be produced and the desired result item. Depending on the user selections additional information may be required such as the advection dispersion component, the time span for which the map is to be produced and the storing frequency.

The option named '(h, p, q)' produces a map-file containing the three results-items; Depth, P-flux and Q-flux. This map-type is primarily targeted the creation of 'seamless dfs2 result-files' with results from a MIKE FLOOD simulation (coupled 2D- and 1D-simulation) such that inside river cross sections coverage, the results in the map are calculated from MIKE 11 and the results outside the river sections coverage are calculation results obtained from MIKE 21 and transferred to the mapping creation routine during simulation of the coupled model.

Additionally the mapping interface may also be used to generate a digital elevation model (DEM) for the river bed. This is done by selecting the DEM option. The DEM option only requires information for the area to be used. This option may further be used for introducing surveyed cross sections into a DEM covering the surrounding area. Figure 6.16 shows an example of a DEM generated using the map capabilities of MIKE 11. The DEM consists of the coarse background DEM (input topography) which has been superimposed with the topography found through interpolation of the cross sections in the MIKE 11 set-up.

Please refer to the documentation of the bathymetry editor on how to generate a dfs2 bathymetry file.
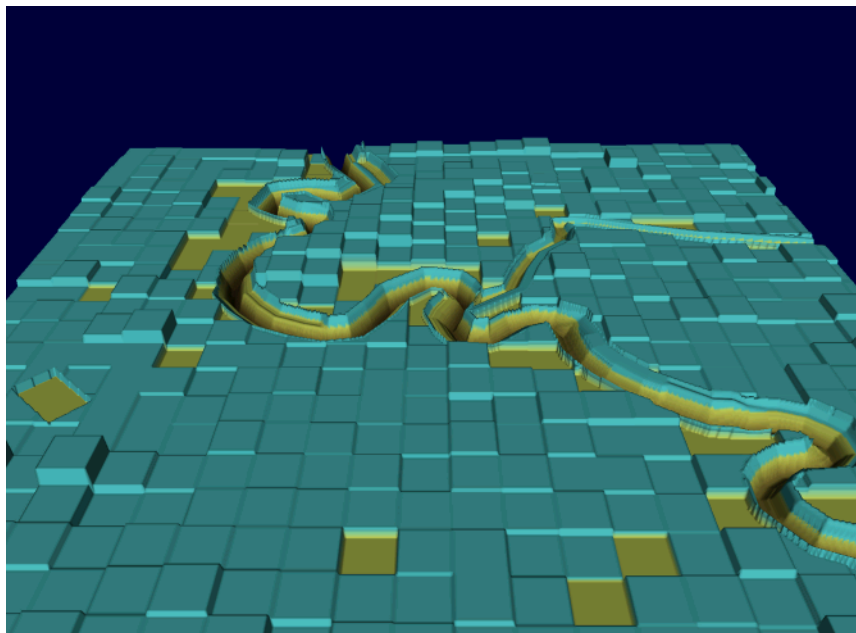
Figure 6.16    An example of generating a DEM using MIKE 11. The DEM is constructed based on a background DEM in the form of a dfs2 file and surveyed cross sections found in the MIKE 11 set-up. To visualize the DEM MIKE Animator has been used

**Important:** Creating maps can be rather time-consuming and hence increasing the simulation time significantly. Especially maps of the types; 'Velocity', 'Velocity * Depth' or '(h, p, q)' are heavy on processing and it is therefore recommended to save as limited time steps as possible. Use the save frequency field to decrease the number of mapping time steps being processed and saved to the map-file to improve the overall computational performance.

The maps definition property page is defined in Figure 6.17 below:
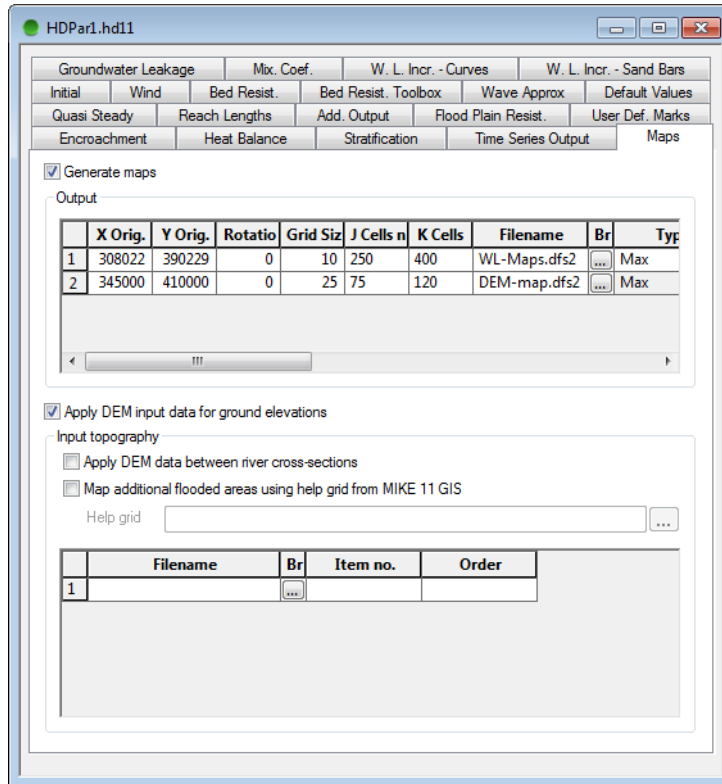
Figure 6.17    The Maps property page

To activate the Map Generation in a simulation you must select the 'Generate maps' tick-box. This will open the property page for editing of data.

User-definitions for map-generations includes:

- 'X Orig.' and 'Y Orig.'
  These are the coordinates for the Origo of the grid-area in which the map will be produced

- 'Rotation'
  The orientation of the grid at origo is defined as the angle from map projection North to the y-axis of the map-grid (defined as Clock-wise positive direction). Note that map projection North differs from true North by the Convergence angle.
  Rotation angle is defined in either Radians or Degrees - depending on the settings of the Unit Base Group settings. Default unit is Radians but the fly-by text when placing the cursor on top of the rotation edit-field will inform you which angle unit is active with your installation.

- 'Grid Size'
  The grid cell size of the maps produced. Maps generated can only be squared (equal spacing of dx and dy of the grid)

- 'J cells no.' and 'K Cells. no.'
  Number of grid cells in the J- and K-direction respectively. J-direction is equivalent to the X-axis and K-direction is equivalent to the Y-axis)

- 'Filename'
  The filename for the map you are creating.
  Note: The map-file generated is a dfs2-file and hence, you should ensure that the filename you specify has an extension named '.dfs2'.
  use the 'Browse' button ('...') to search for existing filename or define a new name for the map to be generated.

- 'Type'
  The map-type to be generated; Minimum, Maximum or Dynamic

- 'Item'
  The Result Item to be included in the Map (Water Level, Water Depth, Velocity, Velocity * Depth, AD Component, DEM, (h,p,q)).

- 'Component'
  If AD Component has been selected as the Map-Item it is required to define which component number (or State Variable) the maps shall be produced.

- 'Storing Frequency'
  The frequency between time steps saved in a dynamic map.
  The time step in the generated map-file will be defined as the 'Storing Frequency' multiplied with the specified result saving frequency in the simulation file. (example; a simulation storing frequency of 2 hours and a Map Storing Frequency of 6 will result in a map grid-file which will contain map-results for every 12 hours).

- 'Default Period'
  The Default Period check-box relates only to Dynamic type of maps.
  Activate the Default Period box to create dynamic maps throughout the entire simulation period.
  Deactivate the check-box to create dynamic maps for only a limited period within the simulation period.
  Reducing the time in which the maps are created have two main purposes: a) reducing the overall simulation time as map processes are only activated in a limited period of time, and b) reducing the file-size of a dynamic map.

- 'Start time' and 'End time'
  The Start and End times are applicable for 'Dynamic' Type maps and if the 'Default Period' is unchecked.
  Define the period for which the maps should be generated by the start and end times respectively. Both dates defined here must lie within the total time span of the simulation.

An additional option generating maps is to include information from external grid-files including DEM data

Activate the 'Apply DEM input data for ground elevations' tick-box to activate this feature.

Input Topography options:

1. 'Apply DEM data between river cross sections'
   This feature allows the usage of one or more pre-defined Digital Elevation Models to determine the Input Topography used by the mapping routine.

   – As previously stated, the mapping routine creates maps with grid-values within the extent of cross sections. The input topography encapsulated by the polygon as illustrated in Figure 6.14, is normally determined from interpolation between the defined cross sections, but more details can be obtained by using the present option, where topographical data are taken from externally defined dfs2 grid-files containing DEM-data.

   – Multiple DEM-input files can be defined in the table in the bottom of the page in a prioritised order such that you can decide which DEM-value to take in case of overlapping Input maps.

2. 'Map additional flooded areas using help grid from MIKE HYDRO River '
   This feature can be used only in connection with applying an underlying DEM, which covers the additional flood areas to be mapped.

   – MIKE HYDRO River is an extension to ESRI Arc-GIS which amongst others, includes a feature, where potential flooded areas outside of the area covered by the cross sections in the model can be digitizes as a polygon.
   In MIKE HYDRO River, the digitized polygon is assigned a linkage to a specific cross section and thereafter exported to a dfs2 grid-file. The dfs2-file contain the cross section number (defined as a number in the Cross Section ID) in all cells which represents the additional flooded area.

   – During simulation and map generation, flood maps for the area covered by the helping grid are generated by using the calculated water level in the actual cross section and eventual flooding in the additional area is included in the generated map.
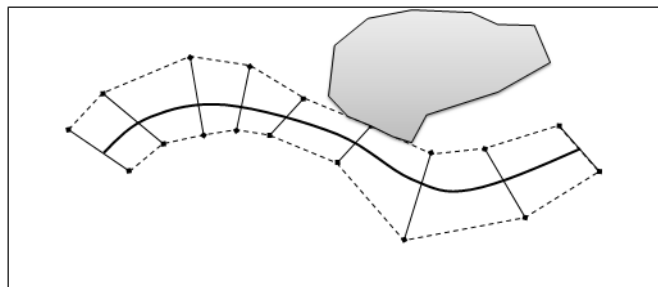
Figure 6.18     Map area including additional Area outside of cross section defined mapping area

## 6.11.1  A step by step guide to generating two-dimensional maps

1. Mark the 'Generate map' tick box.

2. Locate the area where the map is to be produced. The area is given by the origo, the orientation, the cell size and the number of cells in the two directions (see Figure 6.19).

3. Select the cell size to be used. Remember the finer the resolution the larger the result file.

4. Specify a name for the output file including the extension dfs2. Note that the browse button may be used to select the directory in which the file is to be saved.

5. Open the network file to rectify that the map outline is correct.
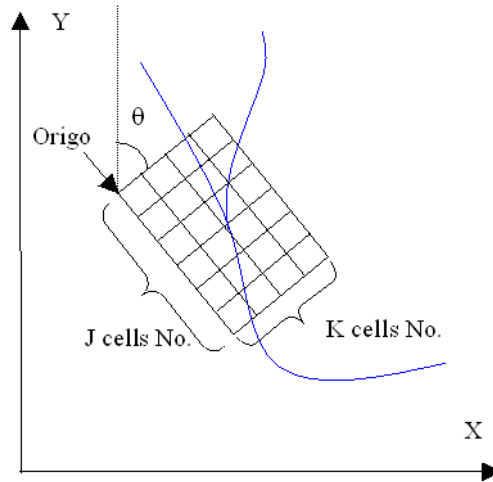
Figure 6.19    Definition of grid used for mapping. The rotation angle is positive clock-wise. The definition of origo is also shown. The two blue lines indicate the location of the river system
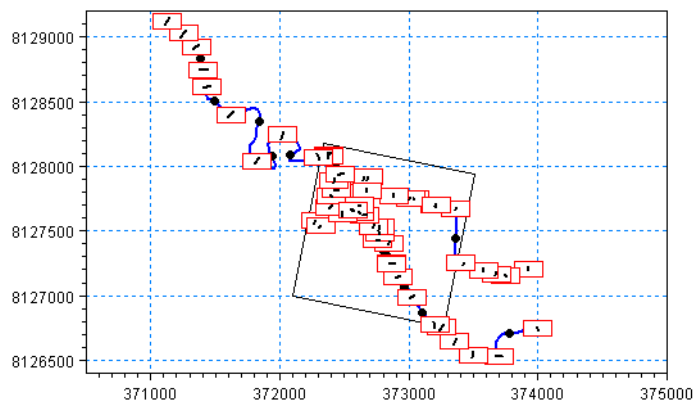


Figure 6.20    Overview of the location of the map in the network file. Note that the map is rotated with respect to the working area coordinates

6.   Select the item to be mapped. Note that if an advection dispersion component is to be mapped then the component number also needs to be supplied.

7.   Specify the type of map which is to be generated (minimum, maximum or dynamic).

8.   If a dynamic map has been selected then the storing frequency and the mapping period need to be specified. Note that as an alternative the full simulation period may be used. The latter is achieved by activating the

tick-box 'default period'.

9.  The storing frequency is a multiplier of the storing frequency specified in the simulation editor. Thus the mapping storing frequency is found as the general simulation editor storing frequency times the storing frequency specified in this menu.

10. If you have one or more additional digital elevation models in the form of a dfs2 files that you wish to include then continue to the next step otherwise go to step 16.

11. Please refer to the documentation of the bathymetry editor on how to generate a dfs2 bathymetry file.

12. The topographical information from the cross sections may be augmented or overridden with additional topographical information located in one or more dfs2 files. The dfs2 files needs to be in the same projection as the working area and contain an item describing the elevation.

13. To activate the use of the an input topography use the 'Apply DEM data between cross sections' tick box.

14. Browse for the file(s) containing the additional topography and set the item number of the elevation.

15. If there are multiple background topography files then the ranking order needs to be specified. The lowest order number is given the highest priority.

16. Run the simulation.

17. The output files may now be viewed using the Result Viewer or MIKE View. MIKE Animator if installed may also be used to view the results (see Figure 6.15).

## 6.11.2  A step by step guide to generating Digital Elevation Models (DEM)

1.  Follow steps 1 through 5 in section 6.11.1 A step by step guide to generating two-dimensional maps *(p. 342)*.

2.  Select the map item as DEM

3. Please refer to the documentation of the bathymetry editor on how to generate a dfs2 bathymetry file.

4. The topographical information from the cross sections may be augmented or overridden with additional topographical information located in one or more dfs2 files. The dfs2 files need to be in the same projection as the working area and contain an item describing the elevation.

5. Do not select the 'Apply DEM data between cross sections' tick box.

6. Browse for the file(s) containing the additional topography and set the item number of the elevation. Please refer to the documentation for the bathymetry editor on how to generate a dfs2 bathymetry file.

7. If there are multiple background topography files then the ranking order needs to be specified. The lowest order number is given the highest priority.

8. Run the simulation.

9. The output DEM may now be viewed using the Result Viewer or the grid editor. If installed MIKE Animator may also be used to view the generated DEM (as presented in Figure 6.16).

## 6.12   Groundwater Leakage

The Groundwater Leakage page defines leakage coefficients such that an additional loss of water from the river (to the groundwater) can be included in the simulation.

The Leakage coefficients page is presented in Figure 6.21.
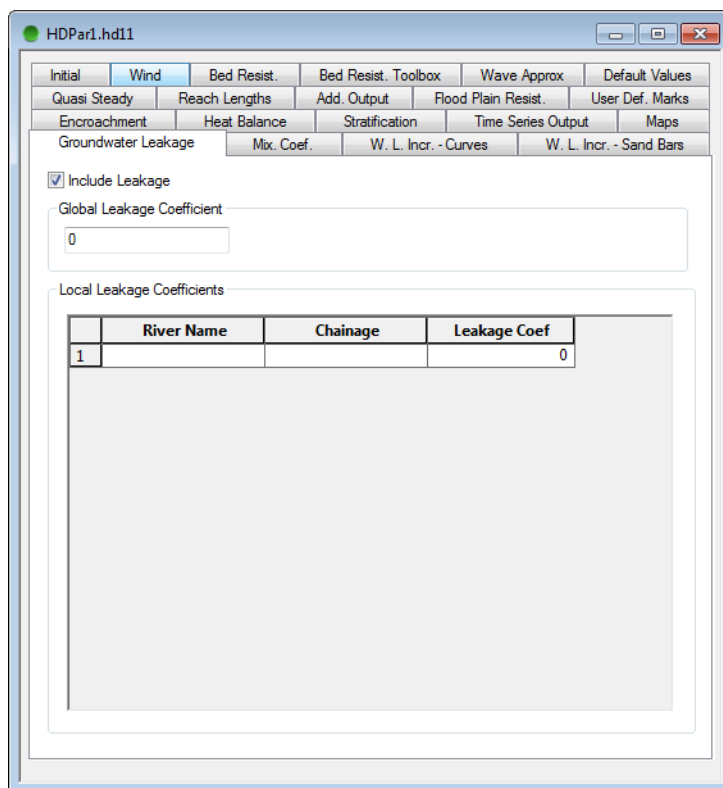
Figure 6.21    Groundwater Leakage page

Include the Leakage in the simulation by activating the 'Include Leakage' tick-mark on the top of the page.

Leakage coefficients are defined as in many other pages in the HD editor by an overall Global Leakage coefficient and local coefficients which overrides the Global value.

Hence, if leakage should be included only at some reaches, then a Global value of 0 can be defined and the local reaches defined in the Local Leakage Coefficients table.

## 6.13   Reach Lengths

The Reach lengths page is ONLY for use with the steady state energy equation switch found under the Quasi Steady State page. The reach lengths are used in evaluation of the friction loss from one cross section to the next. For unsteady simulations the reach lengths are ignored.

In MIKE 11 cross sections are viewed looking downstream. Downstream is per definition the direction of increasing chainage. This definition of down-

stream is independent of the flow direction and is used throughout by the Graphical User Interface.

On the Reach lengths page the user specifies the reach lengths of the left and the right overflow banks in the downstream direction (increasing chainage). Note that the reach lengths are to be specified at all cross section locations except for the cross section with the highest chainage in a branch (the area downstream of such sections is either a node/junction or beyond the model area). The reach lengths are defined as the distances to the next downstream cross section (next chainage). Please refer to fig. 1 below. As an aid to the user the "Load branch and chainages" button will populate the table with all rivers and h-point locations. Please note that the functionality of this button is only available if the simulation file is open and the data from the cross section and network editors are accessible. Since the use of reach lengths is based on raw data the method requires that there exist cross sections at all h-points in the grid. If this is not the case please insert cross sections or increase max-dx in the network editor to avoid the code generating interpolated h-points at run time. The grid generating function in the network editor can be used to check whether cross sections are present at all h-points
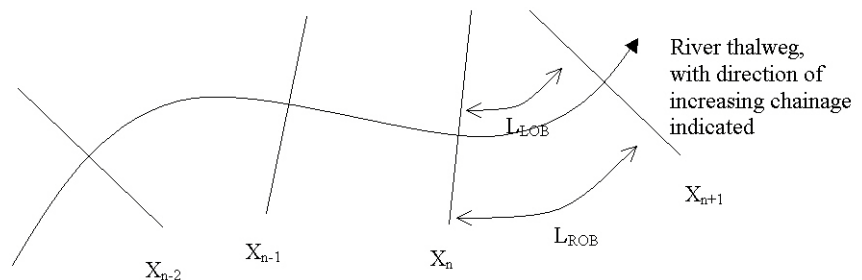


Figure 6.22    Reach length definition

The reach lengths to be specified at cross section with chainage $X_n$ is shown in Figure 6.22. The reach lengths indicate the distance on the flow banks from the current cross section ($X_n$) to the next cross section (increasing chainage). $L_{LOB}$: Length Left Overflow Bank, $L_{ROB}$: Length Right Overflow Bank

## 6.14   Add. Output

A number of simulated parameters can be selected for storage in an additional output result file (with the filename extension '.res11' or '.res1d'). The parameters are saved for each save step at each $h/Q$ point of the river system. Time series and longitudinal profiles of the parameters can be viewed in the same way as normal MIKE 11 result files.
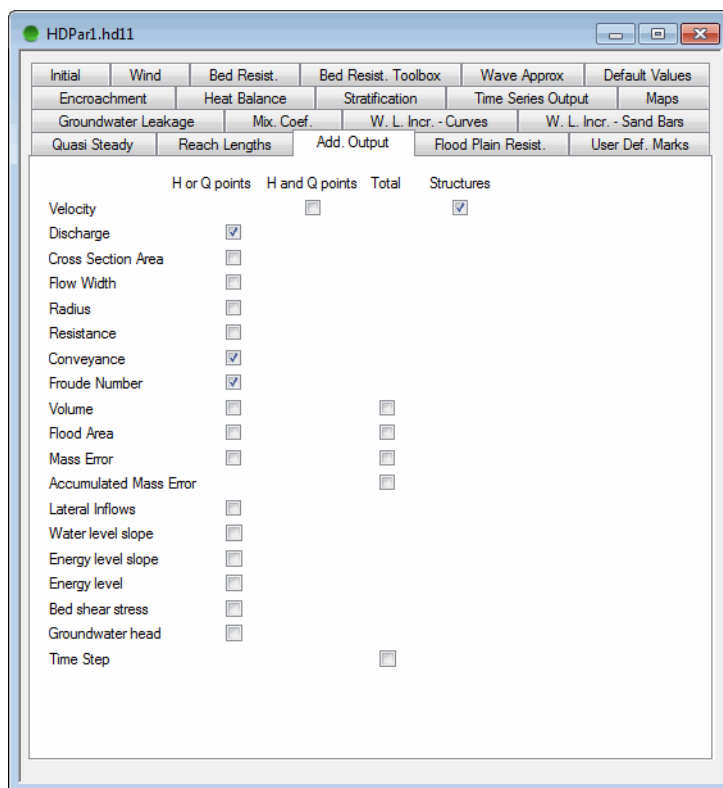
Figure 6.23    The additional output property page

## Structures
Structure flow, area and velocity. In case of control structures the gate level is also stored.

## Velocity
Velocities are calculated as the discharge divided by the cross sectional area.

## Discharge
The discharge calculated at *h*-points is a weighting of up- and downstream discharges calculated at *Q*-points.

## Slope
The free water surface slope.

## Cross section area
The area of flow in the cross section. At computational H-points where no cross section is present the area is linearly interpolated from upstream and downstream areas.

## Top width
The channel width at the free surface level.

### Radius
The resistance radius.

### Resistance
The cross-sectional resistance (resistance number multiplied by the resistance factor).

### Conveyance
The conveyance

### Froude number
Defined as:

$$F = \frac{Q}{A\sqrt{g\dfrac{A}{b_s}}}$$  (6.4)

Where $F$ is the Froude number, $Q$ the discharge, $A$ the cross sectional area, $g$ the acceleration due to gravity and $b_s$ the channel width at the surface.

### Volume
The volume calculated around the H-grid point.

Total: The total water volume for the river system.

### Flooded Area
H-points: The flooded area of the water surface between two neighbouring Q-points.

Total: The total surface water area for the river system.

### Mass Error
The mass error is defined as the difference between the volume calculated in the model and the true volume. At nodal points with more than two connections the mass error is distributed uniformly between each connection.

Total: The total mass error for the river system.

### Accumulated Mass Error
The sum of the 'Mass error' in time and space. Generally, the mass error can be reduced by increasing the number of iterations per time step, reducing the time step, and or by increasing the resolution of the cross-sections.

**NOTE!** Some cross-sections can cause mass-balance problems due to large contractions. These problematic cross-sections can be detected by selecting the mass error item calculated for each grid point.

### Lateral Inflows

Lateral inflows due to boundary conditions, catchment runoff, Flood forecasting updating or coupling to MIKE SHE, MIKE 21 or MIKE URBAN.

### Water level slope

Water level slope at discharge points.

### Energy level slope

Energy level slope at discharge points.

### Energy level

Energy level at water level points.

### Bed shear stress

The bed shear stress $\tau$ at water level points given as

$$\tau = \rho g R \frac{V^2}{M^2 R^{4/3}}$$

where $\rho$ is the fluid density, $g$ the acceleration of gravity, $M$ the Manning number, R is the radius (approximated equal to the flow depth, h) and V is the flow velocity.

An alternative option for Bed Shear Stress saved to the additional result file is available. The MIKE11.ini file contains a parameter; 'AddOutput_ShearStress_UseEnergyLevelSlope' which controls whether the bed shear stress shall be calculated in the most common form using the friction slope as indicated above - or if the bed shear stress should be calculated for using the Energy level slope.

When the parameter is set to ON, the following equation is used.

$$\tau = \rho g R \frac{\mathrm{d}E}{\mathrm{d}x}$$

where $E$ is the energy level and $x$ is the grid spacing between calculation points along the river.

## 6.14.1 Additional output for QSS with vegetation

Note that when utilising the quasi two dimensional steady state with vegetation module, additional output is based on the processed data which does not take the effect of dead water zones or vegetation zones into account. Additional data for these calculations can be obtained by setting the following switch in the "MIKE11.ini" file:

CREATE_QSSVEG_VELOCITY_FILE=ON

With this setting eight .txt-files are generated and saved in the working directory i.e. where the simulation file is stored.

The files are titled:

- QSSVEG_velocities: Velocity and area of the individual panels.

- QSSVEG_velocities_add1: Energy slope, low water channel width, high water channel width, Radius, wetted perimeter and Manning's n of the individual panels.

- QSSVEG_velocities_add2: Height of water/water interface, water/vegetation interface of the individual panels.

- QSSVEG_velocities_add3: Mixing coefficients of the individual panels.

- QSSVEG_velocities_add4: Shear forces of the individual panels normalised with $\rho$.

- QSSVEG_junctions: The appropriate parameters used for obtaining the water level increment due to the junction and the water level increment in the channels meeting at the junction.

- QSSVEG_sandbars_curves: Water level increments due to sandbars and river curvature.

- QSSVEG_bridges: The water level increments due to bridges.

## 6.15   Flood Plain Resistance

The flood plain resistance numbers are applied above the 'Level of divide' specified in the raw cross section data (.xns11 files).

The global resistance number is applied on all flood plains unless local values are specified. Local values are linearly interpolated at intermediate chainage values. The resistance number value -99 indicates that the flood plain resistance should be calculated from the raw data in the cross-section data-base.

Example (Figure 6.24): In 'RIVER 1' the resistance on the flood plains is globally calculated on the basis of the raw cross section data. However, between chainage 5000 m and 10000 m an alternative flood plain resistance is applied. The resistance number on the flood plains in this reach varies linearly between 25 and 30.
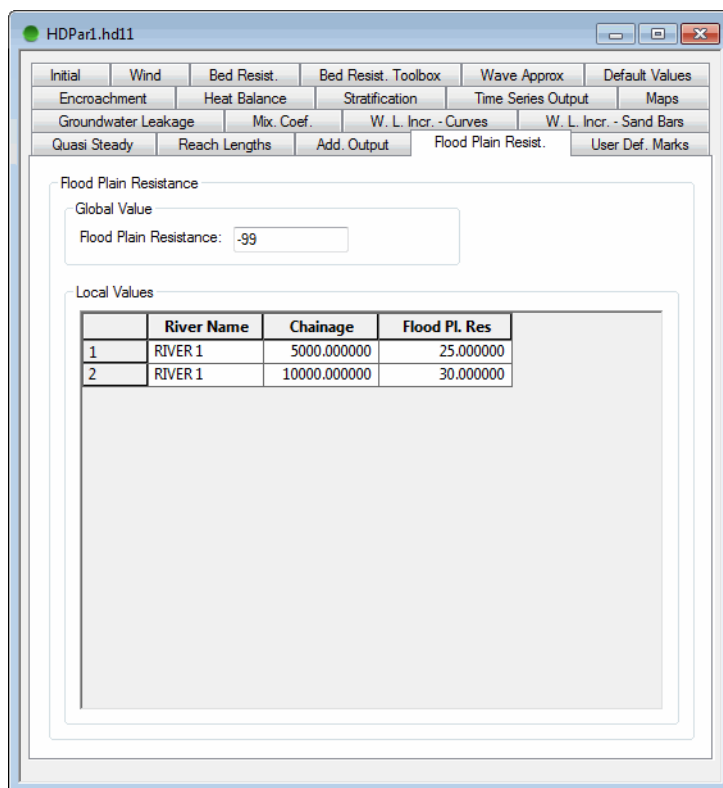
Figure 6.24    The Flood Plain Resistance property page

## 6.16    User Def. Marks

The User Defined Markers page offers a possibility for the user to define spe-
cial markers/points in the river network by defining the location and the top
level of the item. Items defined as user defined markers can be presented on
a longitudinal profile in the result presentation programme; MIKEView. Mark-
ers could be the location of an important hydraulic structure, a gauging sta-
tion or other significant items in the modelling area.

**Note**. To define the first Marker in an empty page, click the 'Mark title' bar in
the upper half of the page. Thereafter, press the <TAB> button and a new line
will be present in the grid in the upper part of the page as well as a new col-
umn is introduced in the 'location grid' in the lower half of the page. Write the
name of the marker in the empty line in the upper grid, and this name will
automatically be transferred as the name of the column.

Markers can be defined as single points only and as markers defined along a
river stretch. The 'Interpolate' column must be checked in case a linear inter-
polation is requested on stretches between chainages and marker levels
defined in this page. In case a user defined mark should be presented on the

longitudinal profile as a single point (e.g. a bridge location or flood mark indicator) the Interpolate check-box must be unchecked.

## 6.16.1 Activation of Bed resistance Triple Zone Approach

The Bed resistance Triple Zone approach is activated by defining two markers with the names; 'ZONE1-2' and 'ZONE2-3'. Marker names can not differ from these names if they are to be used for defining zone-separators for the triple zone approach.

After defining the marker names, the zone-separator levels must be defined as two levels defined in stations along the river stretches in the setup where the separation between Zone 1 and 2, and Zone 2 and 3 are present. That is, a longitudinal profile/line should be defined for each of the two zone-separators.

**Note:** In case the Triple Zone Approach has been activated and zone separator lines are not defined for the entire setup, MIKE 11 uses the uniform bed resistance values in the points where separator lines are not defined. The resistance value used at these points is the value (global or local) defined for the lower zone.

Figure 6.25 shows an example where a single point marker has been defined ('Main Bridge' at RIVER1, chainage 1500) and triple zone separator lines has been defined in RIVER1 in the reach from chainage 0.0 to 5000.
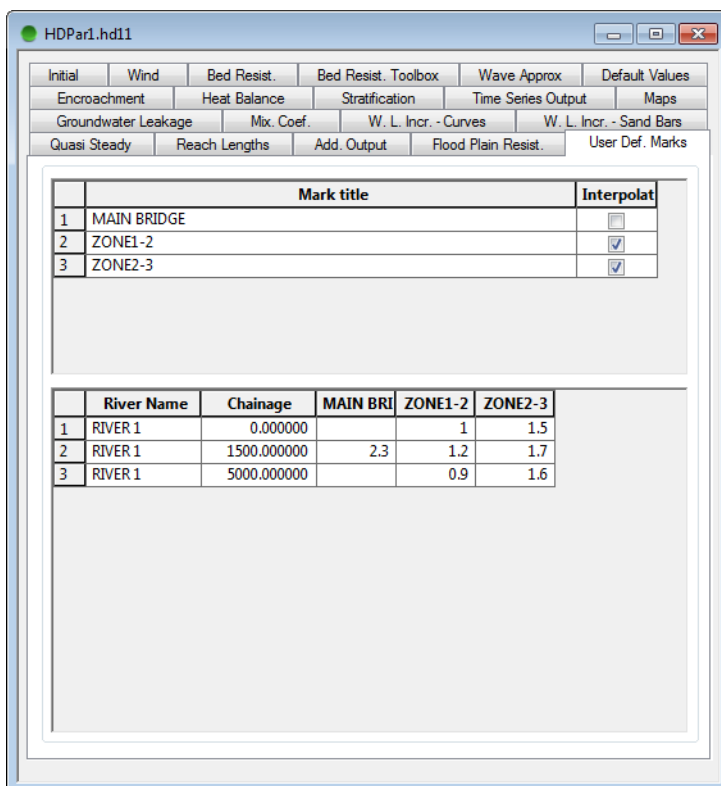
Figure 6.25    Example of defining User Defined Marks

## 6.17   Encroachment

Encroachment simulations are setup through this page. An encroachment simulation consist of two or more simulations. The first simulation acts as a reference simulation to which all other results are compared. The reference simulation is set up as any ordinary steady state simulation.

Based on the reference simulation a number of encroachment simulation may be carried out. Each of these are specified as a line in the Encroachment simulation overview. Hereby one can evaluate different encroachment strategies through the same setup. The parameters used for defining the encroachment simulations are described below.

**Note** that only MIKE 11's default steady state solver may be used. Further since the encroachment module utilizes the steady state solver, the installation of MIKE 11 should include a steady state module to make the encroachment module function.
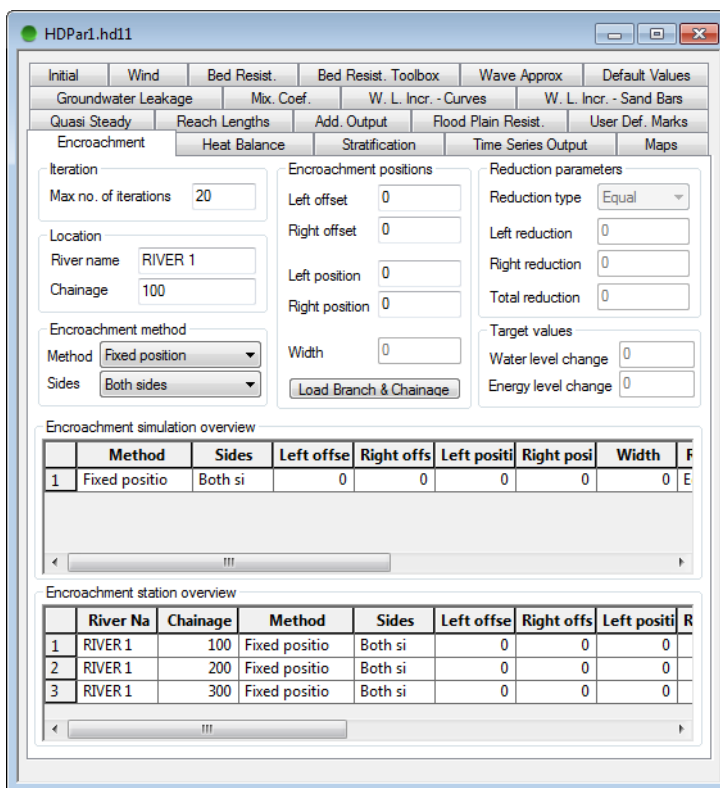
Figure 6.26    The encroachment property page

## 6.17.1  Iteration

### Max no. of iterations
The maximum number of iterations allowed when obtaining encroachment positions. The default setting is 20. If a non-valid number (<2) is entered the code will use the value to 2. **Note** that this parameter is global for all encroachment simulations and stations.

## 6.17.2  Location

The location of the encroachment station is entered here through a river name and a chainage. If a location is entered for which no corresponding cross section exists a warning is issued at run time and the station will be ignored in the subsequent simulation.

## 6.17.3  Encroachment method

### Method
A total of five different methods are available:

1. Fixed position: The position of the encroachment stations are user specified.

2. Fixed width: The position of the encroachment stations are found through a user defined width.

3. Conveyance reduction: The encroachment stations are found through user specified conveyance reductions.

4. Target water level: The position of the encroachment stations are determined by ensuring that the conveyance of the encroached cross section at the user defined target water level is equal to the conveyance of the undisturbed cross section at the reference water level.

5. Iteration: The encroachment positions are found through an iterative procedure where steady state simulations are evaluated. The objective of the evaluations are to reach a user defined target water level or energy level.

### Sides

It is possible for the encroachment to take place on both sides of the main channel or only on one of the sides. For this purpose the sides combo box may be used.

**Note**: If the method chosen for encroachment is 'Fixed width' then the sides switch is automatically set to both sides. Since a fixed width encroachment only makes sense if both sides are to be encroached.

## 6.17.4 Encroachment positions

### Left and right offset

The user may specify a left and a right offset for the encroachment positions. These specify the minimum distance between the position of the encroachment and the river bank. The latter being defined by markers 4 and 5.

### Left and Right position (only encroachment method 1)

For the fixed position encroachment method the user should here specify the position of the left and the right position as the distance from the river bank.

### Width (only encroachment method 2)

The width used for the fixed encroachment width method is entered here.

## 6.17.5 Reduction parameters (only encroachment methods 3 to 5)

### Reduction type

The way that the conveyance reduction should be accomplished is specified here. Three possibilities are available:

- Equal: The conveyance reduction is accomplished by reducing the conveyance equally on both flood plains.

- Relative: The conveyance reduction is accomplished by reducing the conveyance relative to the conveyance distribution in the reference simulation.

- Specified: The user may specify the conveyance reduction for each of the flood plains.

The above-mentioned settings are only meaningful if the 'Sides' switch is set to 'both sides'. If the latter is not the case the reduction type switch should be set to specified.

### Left and Right reduction

These are only used if the reduction type is set to 'specified'. The conveyance reduction is entered in percentage of the total conveyance.

### Total reduction

If the reduction type is set to either 'Equal' or 'Relative' this field becomes active. The total required conveyance reduction is entered here in percentage.

## 6.17.6 Target Values

These fields are only of importance if the encroachment method is chosen as either 4 or 5. In method 4 a water level target is used to determine the encroachment. In method 5 the simulation tries to determine the encroachment stations such that the water level or the energy level found through simulation is equal to the target water level or energy level respectively.

### Water level change

The target water level used in the simulation is the reference water level plus the user specified water level change.

### Energy level change (only encroachment method 5)

The target energy level used in the simulation is the reference energy level plus the energy level change.

### Encroachment strategies using method 5

If method 5 is utilised there are three possible strategies.

**Water level target**: The encroachment may be carried out so that only a water level target is considered. This strategy is achieved by setting the water level change to a non-zero value and the energy level change to zero.

**Energy level target**: The encroachment is carried out so that only an energy level target is considered. This strategy is achieved by setting the water level change to zero and the energy level change to a non-zero value.

**Water level target and energy level target**: The encroachment is carried out so that a water level target is met. Once this has been achieved the energy level is checked. If the energy level is above the energy level target the code will reconsider the encroachment and try to satisfy the energy level request instead. This strategy is achieved by setting both the water level change and the energy level change to non-zero values.

Please note that the position of the encroachments are found through an iterative procedure. This procedure considers each cross section individually starting downstream and working upstream. To ensure that this method is successful do not use method 5 for river reaches which form part of a loop in a network. Further method 5 is designed for encroaching river reaches where the discharge distribution can be determined a priori, thus the method will be less successful for networks having river bifurcations in a downstream direction as opposed to bifurcations in upstream directions. Finally it should be mentioned that not all user specified targets can be reached. If this is the case the code will issue a warning and return the encroachment which is closest to the requested target.

### 6.17.7  Encroachment simulation overview

Each row in this overview represents an encroachment simulation. The parameters set here are used as default values for all the stations entered subsequently in the Encroachment station overview. Thus the number of rows is equal to the number of encroachment simulations which are to be carried out.

### 6.17.8  Encroachment station overview

Each row in this overview represents a location along a river reach. For each location all of the above parameters may be set individually (except max no. of iterations).

### 6.17.9  General guide lines for carrying out encroachment simulations

Since MIKE 11 uses preprocessed data for the simulation it is important to have a fine resolution in the cross sectional processed data. Further the

encroachment module only allows equidistant level selection for the cross sections used for encroachment. If the latter is not the case an error message will be displayed and the simulation stopped.

For encroachment simulations only the initial start time in the simulation editor is used. This start time is used for determining the boundary values in the river set-up. Note that constant boundary conditions in MIKE 11 are specified by the use of non-varying boundary conditions in the boundary editor.

The choice of encroachment method depends on the application. Please note that methods 1 to 4 all analyse the individual cross section without considering the rest of the network. For instance method 4 seeks a water level change with the same conveyance as the reference level and thus only considers the individual cross section from a point of view of flow taking place at the natural depth. The actual steady state simulation carried out may not give rise to the required water level change. To obtain the latter method 5 should instead be used.
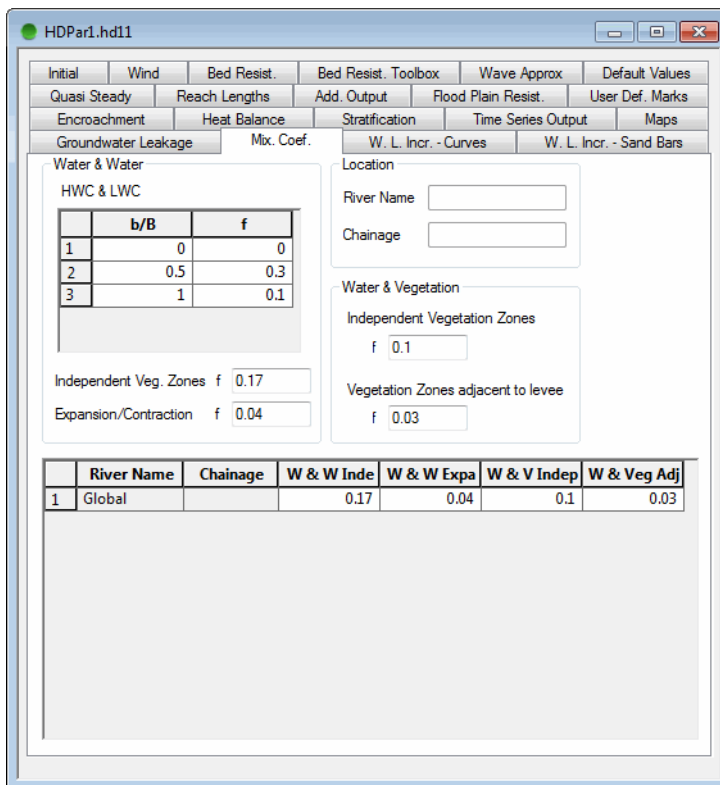
## 6.18   Mixing Coefficients



Figure 6.27    The Mixing Coefficients property page

Used **only** in conjunction with the quasi two dimensional steady state vegetation module. This menu is used for setting the mixing coefficients between adjacent panels in the river cross sections. Both global and local values may be set here.

## 6.18.1 Water & Water

### HWC & LWC

In this box the mixing coefficients between the low water channel (LWC) and the high water channel (HWC) are set. The data is entered as a function of the ratio between the width of the low flow channel and the total width of the river (*b/B*). Linear interpolation is used to obtain intermediate values.

**Important!** The table should start with *b/B*=0 and end with *b/B*=1 and all intermediate values of *b/B* must be monotonically increasing. If the table does not meet this criteria a warning is issued and the default settings are used.

**Note**. To define the first line in the table, click the 'b/B' bar in the upper half of the page. Thereafter, press the <TAB> button and a new line will be present in the grid in the upper part of the page.

### Independent Veg. Zones f

The mixing coefficients at a water/water boundary at an independent vegetation panel and a normal panel.

### Expansion/Contraction f

The mixing coefficients at a water/water boundary at a dead water interface.

## 6.18.2 Location

The river name and location (chainage) is displayed here.

## 6.18.3 Water & Vegetation

The mixing coefficients at water/vegetation boundaries are set here.

### Independent Vegetation Zones

Mixing coefficient at independent vegetation zones.

### Vegetation Zones adjacent to levee

Mixing coefficient at vegetation zones adjacent to levee.

## 6.19 W. L. Incr.- Curve

Used **only** in conjunction with the Quasi Two Dimensional Steady State vegetation module. This menu is used for setting the parameters which are used

for determining the increment of the water level due to the presence of river curvature.

The tab is illustrated in Figure 6.28 with all the different features all of which are described below.
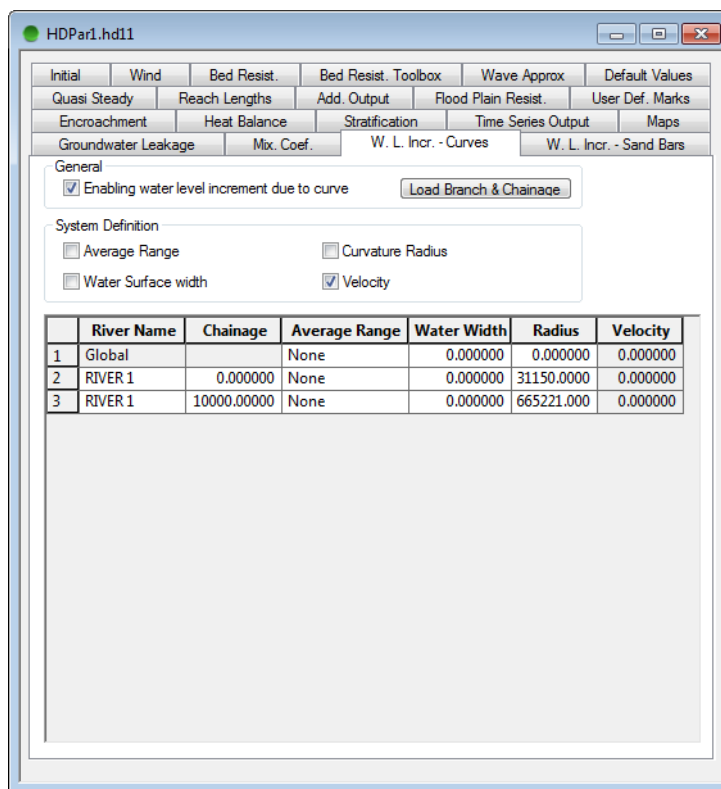


Figure 6.28    Water level increment due to curves

## 6.19.1  General

### Enabling water level increment due to curves
If the effect of the river curvature on the water level is to be included in the calculations this box should ticked.

### Load Branch and Chainage button
This button activates a window with three choices

- Load the Branch and Chainage from Cross Section editor.

- Load the Branch, Chainage and Radius from Cross Section editor and Network Editor.

- Load the Branch, Chainage, Radius and Channel Width from Cross Section editor and Network Editor.

Tick the appropriate choice and click OK.

**Important!** To successfully activate the second or third choice (Radius/Radius and Width) it is required that the network file is open. If the NWK11 file is not open, then go to the Simulation Editor (Input page) and press the 'Edit...' button to open the network-file. Thereafter, it is possible to extract the Radius and/or Width values.

At the bottom of the editor a table is displayed with river name, chainage and the four parameters appropriate for the determination of the water level increment. The parameters which are not greyed may be edited.

## 6.19.2  System Definition

In this box the user may tick the appropriate parameters which should be user defined or system defined. The parameters which are subsequently used in the calculations are:

1.  Average Range.
2.  Curvature Radius.
3.  Water Surface Width.
4.  Velocity.

**Note**! If either 2 or 3 is ticked the velocity is also automatically ticked.

## 6.19.3  Tabular view

The editor displays a tabular view of the parameters which will be used in the determination of the water level increment. The user should edit these values appropriately.

In the column "Average Range" the user can control the calculation of the curvature radius. If the average range is set to "None" no water level increment due to curvature applies. For other values of average range a curvature radius is initially calculated or assigned (depending on the what's selected in the group box "System Definition") individually in each h-point. If the average range equals "Single" the curvature radius is kept unchanged, otherwise this is averaged over a number of h-points. Consecutive h-points with the same average range setting is lumped together when calculating an average curvature radius. "Multiple 1/2" is used for h-points to be included in both the upstream and downstream averaging reach.

## 6.20    W. L. Incr.- Sand Bars

Used **only** in conjunction with the Quasi Two Dimensional Steady State vegetation module. This menu is used for setting the parameters which are used for determining the increment of the water level due to the presence of sand bars.

The tab is illustrated in Figure 6.29 with all the different features all of which are described below.
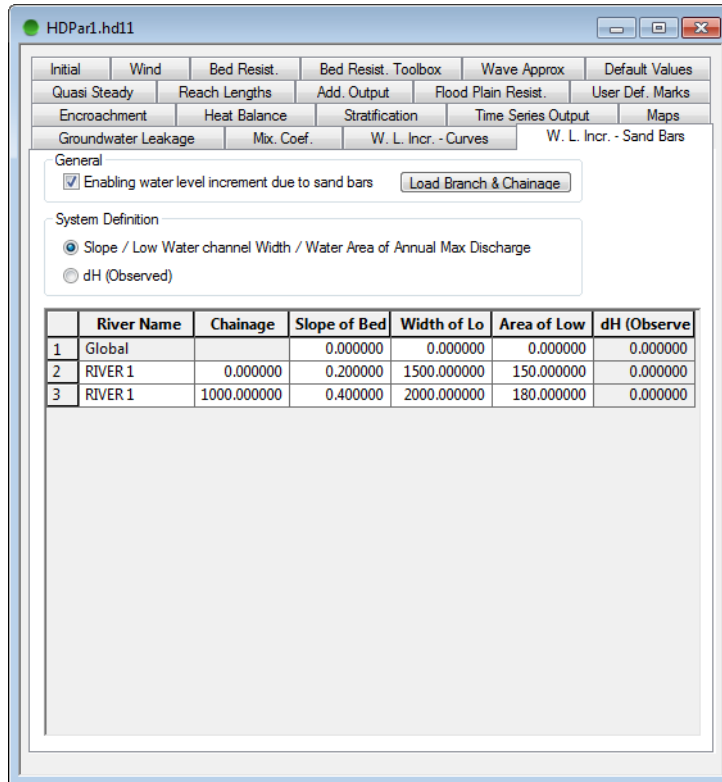


Figure 6.29    Water level increment due to sand bars

### 6.20.1  General

#### Enabling water level increment due sand bars
If the effect of sandbars on the water level is to be included in the calculations this box should be ticked.

#### Load Branch and Chainage button
This button loads the branch name and chainage from the cross section editor (remember to have the simulation editor open).

At the bottom of the editor a table is displayed with river name, chainage and parameters appropriate for the determination of the water level increment.

## 6.20.2 System Definition

In this box the user may tick the appropriate parameters which should be user defined or system defined. The parameters which are subsequently used in the calculations are either

- Bed slope, low water channel width and water area of annual maximum discharge

or

- An observed water level increment.

## 6.20.3 Tabular view

The editor displays a tabular view of the parameters which will be used in the determination of the water level increment. The user should edit these values appropriately.

# ADVECTION-DISPERSION EDITOR

# 7 Advection-Dispersion Editor

The AD Editor is used in conjunction with the following modules:

- Advection Dispersion module (pure AD)
- Water Quality simulations (using MIKE ECO Lab)
- Cohesive sediment transport module
- Advanced cohesive sediment transport module

A brief description of each of these modules is provided below followed by a description on the dialogs included in the AD Parameter Editor.

## 7.1 Module Description

### 7.1.1 Advection-Dispersion module (AD)

The advection-dispersion (AD) module is based on the one-dimensional equation of conservation of mass of a dissolved or suspended material, i.e. the advection-dispersion equation. The module requires output from the hydrodynamic module, in time and space, in terms of discharge and water level, cross-sectional area and hydraulic radius.

The Advection-Dispersion Equation *(p. 368)* is solved numerically using an implicit finite difference scheme which, in principle, is unconditionally stable and has negligible numerical dispersion. A correction term has been introduced in order to reduce the third order truncation error. This correction term makes it possible to simulate advection-dispersion of concentration profiles with steep fronts.

## 7.1.2 The Advection-Dispersion Equation

The one-dimensional (vertically and laterally integrated) equation for the conservation of mass of a substance in a solution, i.e. the one-dimensional advection-dispersion equation reads:

$$\frac{\partial AC}{\partial t} + \frac{\partial QC}{\partial x} - \frac{\partial}{\partial x}\left(AD\frac{\partial C}{\partial x}\right) = -AKC + C_2 q \tag{7.1}$$

where

$C$: concentration

$D$: dispersion coefficient

$A$: cross-sectional area

$K$: linear decay coefficient

$C_2$: source concentration

$q$: lateral inflow

$x$: space coordinate

$t$: time coordinate

The equation reflects two transport mechanisms:

- Advective (or convective) transport with the mean flow;
- Dispersive transport due to concentrations gradients.

The main assumptions underlying the advection-dispersion equation are:

- The considered substance is completely mixed over the cross-section, implying that a source term is considered to mix instantaneously over the cross-section.

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

- Fick's diffusion law applies, i.e. the dispersive transport is proportional to the concentration gradient.

## 7.1.3 Cohesive Sediment Transport module (CST)

The cohesive sediment transport (CST) module also forms part of the AD module. In contrast to the non-cohesive sediment transport (NST) module, the sediment transport cannot be described by local parameters only

because the settling velocity of the fine sediments is very low. The cohesive module uses the AD module to describe the transport of the suspended sediment. Erosion/deposition is modelled as a source/sink term in the advection-dispersion equation. The erosion rate depends on the local hydraulic conditions whereas the deposition rate depends on the concentration of the suspended sediment and on the hydraulic conditions.

The module can also be used when resuspension of sediment affects water quality. This is because the resuspension of cohesive sediment often gives rise to oxygen depletion due to the high organic content and associated oxygen demand (COD) in the cohesive sediment. Likewise resuspension of cohesive sediment can give rise to heavy metal pollution since heavy metals adhere to the sediment.

### 7.1.4 Advanced Cohesive Sediment Transport module (ACST)

The Advanced cohesive sediment transport module provides an alternative, more complex, process description than the simple CST module. This module is especially useful in situations where a mass balance of cohesive sediment is required in order to simulate the accumulation of sediment. Then, knowing the exact location of sediment pools, it is possible to estimate the siltation in navigation channels, waterways, harbours etc.

The advanced cohesive sediment transport module is part of the advection-dispersion (AD) module. As for the standard formulation, the sediment transport is described in the AD-model through the transport of suspended solids. Erosion and deposition of cohesive sediment is represented in the AD-model as a source/sink term. Whereas the erosion rate depends only on local hydraulic conditions (bed shear stress), the deposition rate also depends on the suspended sediment concentration.

## 7.2 Components

Component names and numbers must be specified in this dialog.

The components are user defined. Each component is modelled using a defined concentration unit and component type. In Figure 7.1 below an example is presented where four components are defined. One 'Normal type' salinity component type, two sediment components of one single layer and one multi-layer Cohesive Sediment type and one normal component at the end.

Please note that the combination presented in the figure is primarily for demonstration purpose. In a real application it would not be recommended for several reasons to include two different types of Cohesive Sediment component types in the same simulation.

Please Note: For Water Quality simulations using the MIKE ECO Lab Framework it is not required to define any Components in this property page. Com-

ponent definitions (names) in a MIKE ECO Lab (Water Quality) simulation are taken from the 'State Variables' definition in the applied MIKE ECO Lab11-file.

Components are only required to be defined in this page when AD-simulations without any MIKE ECO Lab file are to be conducted.
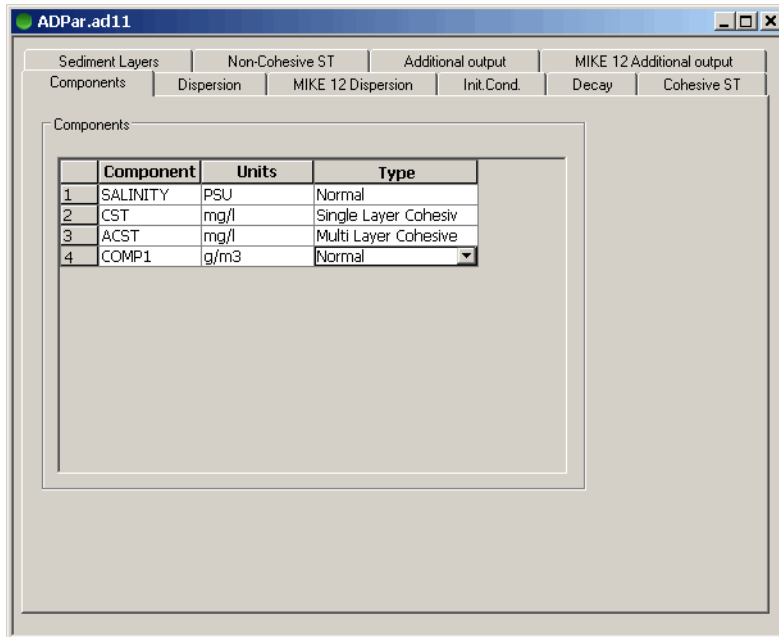


Figure 7.1    The component property page

The components definition includes:

## Component
Here one or more component names are defined by a user-defined string.

## Units
The requested unit for each of the components is selected through the drop-down selection box. Options available:

- **my-g/m$^3$**: Microgram per cubic meter.
- **mg/m$^3$**: Milligram per cubic meter
- **g/m$^3$**: Gram per cubic meter
- **kg/m$^3$**: Kilogram per cubic meter
- **my-g/l**: Microgram per litre.
- **mg/l**: Milligram per litre.
- **g/l**: Gram per litre.
- **Deg. Cel**: Degrees in Celsius.

- **1/100 ml**:

- **PSU**: Practical Salinity Units

- **Counts x 1E6/100 ml**: Bacterial counts.

<span style="color:#4a90b9">Type</span>

- **Normal**: A component used for standard Advection-Dispersion simulations.

- **Single layer cohesive**: A component used only in the single layer cohesive sediment transport model (the CST module).

- **Multi cohesive**: A component used in the multi layer cohesive sediment transport model (the ACST module).

- **Non-cohesive**: A component defined as a Non-Cohesive is actually not simulated using the Advection-Dispersion equation - but rather using traditional sediment transport equations.
Please be informed that this option for simulation a non-cohesive component within the AD Parameter Editor Framework is actually a leftover from earlier versions of MIKE 11 where combined Sediment Water Quality interaction was simulated through this interface. However, now WQ-simulations are made using the MIKE ECO Lab Framework and the non-cohesive component type is therefore rather unuseful in the present release.
Simulating a Non-cohesive sediment component through the AD Parameter Editor does Not include morphological updating and as such it is not very useful.
For Non-cohesive sediment transport simulations, please apply the ST-Module instead.

## 7.3    Dispersion

The dispersion coefficient, *D*, is described as a function of the mean flow velocity, *V*, as shown below.

$$D = aV^b \tag{7.2}$$

Where *a* is the dispersion factor and *b* the dispersion exponent. Typical value ranges for *D*: 1-5 m$^2$/s (for small streams), 5-20 m$^2$/s (for rivers). The velocity is given in m/s.

Both the 'dispersion factor' and the 'dispersion exponent' can be specified. If the dispersion exponent is zero then the dispersion coefficient *D* becomes constant (equal to the dispersion factor). By default the dispersion is zero (i.e. there is only advective transport and no dispersion). The 'Minimum dispersion coefficient' and the 'Maximum dispersion coefficient' parameters are used to control the range of the calculated dispersion coefficients.

Figure 7.2    The dispersion property page

## Global values

The Global dispersion values are defined in this group of fields. These values will be defined in all branches and calculation points unless Local Values are defined additionally.

### Dispersion factor

The dispersion factor equivalent to parameter *a* in equation (7.2) is defined here.

### Exponent

The dispersion exponent equivalent to parameter *b* in equation (7.2) is defined here.

### Minimum disp coeff.

When using (7.2) to calculate the dispersion coefficient it is depending on the velocity that will vary during the simulation. To limit the interval in which the dispersion coefficient will vary the lowest allowable value of the dispersion coefficient during the entire simulation period can be entered here.

### Maximum disp coeff.

When using (7.2) to calculate the dispersion coefficient it is depending on the velocity that will vary during the simulation. To limit the interval in which the dispersion coefficient will vary the highest allowable value of the dispersion coefficient during the entire simulation period can be entered here.

### Local Values

MIKE 11 will use the values specified under global values except for those places were local values have been specified.

#### River Name
Name of the river with local dispersion values.

#### Chainage
Chainage in river with local dispersion values

#### Dispersion factor
Local value of the dispersion factor

#### Exponent
Local value of the dispersion exponent

#### Minimum disp coeff.
Local value of the minimum dispersion coefficient.

#### Maximum disp coeff.
Maximum value of the dispersion coefficient

### Example

In Figure 7.2 both global and local values are entered. In 'RIVER 1' the dispersion coefficient is globally set to 10 m$^2$/s (independent of the flow velocity because $b$ equals 0). Local values are defined in the reach between chainages 1000 m and 2500 m, where the dispersion coefficient is dependent on the velocity ($D = 15V$, $5 < D < 25$)

## 7.4    Init. cond.

Initial component concentrations are defined on this property page. If an initial concentration is not specified a default value of zero will be applied throughout the model. Global and local values of initial concentrations can be specified for each component. Local values are specified by entering the river name, chainage and concentration in the local values table. Initial concentrations are not used if the AD simulation is started with a hotstart file.

Figure 7.3    The initial conditions property page

## Initial conditions table

### Component
Here the component in question is selected. It is possible to choose between the components defined in the Components property page by selecting from the drop-down selection box present in the 'Component' column.

### Concentration
The Initial concentration-value for the specific component is defined here

### Global
This box must be checked if the value entered in the Concentration field should be used as a global value. If it is left unchecked the value will be used as a local value.

### River name
The name of the river with the local initial value.

### Chainage
The chainage in the river with the local value.

## Example

In Figure 7.3 initial conditions are defined for the component 'SALINITY'. The initial, Global concentration of SALINITY is set to 35 PSU for the entire river network. However, the initial concentration of SALINITY varies from 20 PSU

to 33 PSU along the reach defined as river branch named 'RIVER 1' at the chainages 5000m to 115000 m. Linear interpolation is used between specified values

## Initial conditions - stratification table

Initial conditions for AD-simulations where Stratified river branches are included are given in this section. (Stratified river branches are defined in the River Network Editor -> Table View -> Branches -> Branch Type = Stratified).

### Component
Here the component in question is selected. Presently only temperature can be selected.

### Conc. S
Temperature at the surface.

### Conc. 2
Temperature at layer k2 above the bottom.

### Conc. 3
Temperature at layer k3 above the bottom.

### Conc. B
Temperature at the bottom.

### k2
Layer number above the bed

### k3
Layer number above the bed.

### Global
This box must be checked if the value entered in the Concentration field should be used as a global value. If it is left unchecked the value will be used as a local value.

### River name
The name of the river with the local initial value.

### Chainage
The chainage in the river with the local value.

## 7.5    Decay

This page contains information for non-conservative components. These components are assumed to decay according to a first-order expression:

$$\frac{\mathrm{d}C}{\mathrm{d}t} = KC \tag{7.3}$$

Where $K$ is a decay constant. $C$ is the concentration. Both global and local values of the decay constant $K$ can be specified.

**IMPORTANT NOTE**: If the components selected are used for a water quality simulation (WQ) then decay constants should not be specified in this dialog as process-equations defined in the Water Quality (or MIKE ECO Lab) template normally includes all processes affecting the State variables. Therefore, adding an additional decay in this AD Parameter Editor will normally interfere negatively with the defined equation framework of the Water Quality model.

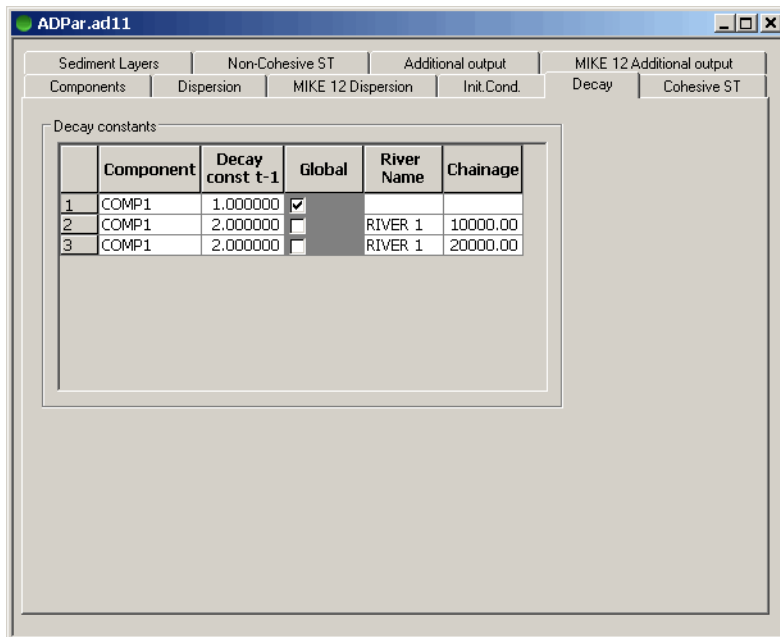The decay page is presented in Figure 7.4 below.



Figure 7.4    The Decay property page

### Component
Here the component in question is selected. Select from the Components drop-down box present in the Components column

### Decay const

Here the value of the decay constant is entered (default unit is [per hour])

### Global

This box must be checked if the value entered in the Decay const. field should be used as a global value. If it is left unchecked the value will be used as a local value.

### River name

The name of the river with the local initial value.

### Chainage

The chainage in the river with the local value.

## Example

In Figure 7.4 the component COMP1 has been selected to be non-conservative. The decay constant is 1.00 globally in the river network and has a value of 2.00 in RIVER 1 between the chainages 10000 m and 20000 m.

## 7.6 Cohesive ST

Data used for each of the two cohesive sediment transport models available are entered on this page. When using the cohesive sediment transport models (either the simple or the advanced) all components specified in the AD editor must be defined as 'Single layer cohesive' or 'Multi layer cohesive' in the Components dialog.

The cohesive sediment transport parameters can only be accessed when a component type on the 'Components' page is defined as either single or multi layered.

Global and local parameter values can be specified as required.

## 7.6.1    Single Layer Cohesive Model



Figure 7.5    The Cohesive sediment property page when a single layer model is selected

Below the parameters that apply to the 'Single layer cohesive' sediment transport model are described.

### Fall Velocity

#### w0
The free settling velocity.

### Deposition

#### Critical shear stress/velocity for deposition
Deposition occurs for shear stresses or velocities lower than the critical value. The user can select which one to use. The typical range is: 0.03 - 1.00 N/m$^2$.

#### Time centring
This centring factor used in the deposition formula. Typical range is: 0.5-1.0.

## Erosion

### Critical shear stress/velocity for erosion

Erosion occurs for shear stresses or velocities larger than the critical value. The user can select which one to use. The typical range is: 0.05 - 0.10 N/m$^2$.

### Erosion coefficient

The erosion coefficient is applied linearly in the erosion expression. Typical range: 0.20 - 0.50 g/m$^2$/s.

### Erosion exponent

The erosion exponent describes the degree of non-linearity in the rate of erosion. Typical range: 1-4.

## Overview

At the bottom of the property page a overview table is shown.

### Global

If this box is checked the entered parameters are used globally.

### River Name

The name of the river for which the data applies.

### Chainage

The chainage of the river for which the entered data applies.

## 7.6.2 Multi Layer Cohesive Model



Figure 7.6    The cohesive sediment property page when a multi layer model is selected

Below the parameters that apply to the 'Multi layer cohesive' sediment transport model are described.

### Fall velocity

#### C-offset
Concentration limit for flocculation affected settling velocity. For higher concentrations the settling velocity is affected by hindered settling.

#### g
Exponent used in the settling velocity expression. Typical range: 3 - 5.

#### m
Exponent in the settling velocity expression for concentrations below C-offset.

#### w0
Free settling velocity. Typical range: 0.0025 - 0.01 m/s.

#### swi
Sediment volume index used in the settling velocity expression.

## Deposition

### Critical shear stress/velocity for deposition

Deposition occurs for shear stresses or velocities lower than the critical value. The user can select which one to use. The typical range is: 0.03 - 1.00 $N/m^2$.

### Time centring

This centring factor used in the deposition formula. Typical range is: 0.5-1.0.

## Erosion

### Instantaneous erosion of layer 1

Instantaneous re-suspension of layer 1 occurs when the computed bed shear stress is greater than the critical shear stress for erosion of layer 1.

### Critical shear stress/velocity for erosion

Erosion occurs for shear stresses or velocities larger than the critical value. Typical ranges are: 0.05 - 0.10 $N/m^2$ for layer1 and 0.20 - 0.50 $N/m^2$ for layer 2 and 3.

### Erosion coefficient

The erosion coefficient is applied linearly in the erosion expression. Typical range: 0.20 - 0.50 $g/m^2/s$.

### Erosion exponent

The erosion exponent describes the degree of non-linearity in the rate of erosion. In case that 'Instantaneous erosion' of layer 1 is selected the erosion exponent is not applicable for layer one. Typical range: 1-4.

## Consolidation

### Transition rates

The consolidation of the sediment layers is described by transition rates between the layers. The transition rates include hindered settling and consolidation. Typical ranges:

layer 1 -> layer 2: 2.35 - 3.11 $g/m^2/s$,

layer 2 -> layer 3: 0.10 - 0.20 g/m2/s

### Sliding friction coefficient

Coefficient used in the formulation for sliding of sediment. Typical range: 3 - 7 $m^{1/2}/s$.

## Overview

At the bottom of the property page an overview table is shown.

### Global

If this box is checked the entered parameters are used globally.

### River Name

The name of the river for which the data applies.

### Chainage

The chainage of the river for which the entered data applies.

## 7.6.3 Cohesive Sediment transport dynamics; Description

### Single Cohesive Layer Model - Deposition

Deposition of suspended material occurs when the mean flow velocity is sufficiently low for particles and sediment flocs to fall to the bed and remain there without becoming immediately resuspended. Particles and flocs remain on the bed if the bed shear stress is less than the critical shear stress for deposition.

The rate of deposition can be expressed by:

$$S = \frac{wC}{h^*}\left(1 - \frac{\tau}{\tau_{cd}}\right), \ \tau \leq \tau_{cd} \tag{7.4}$$

where,

$S$ is the source term in the advection dispersion equation

$C$ is the concentration of the suspended sediment (kg/m³)

$w$ is the mean settling velocity of suspended particles (m/s)

$h^*$ is the average depth through which the particles settle

$\tau$ is the critical shear stress for deposition (N/m²)

$\tau_{cd}$ is the bed shear stress (N/m²)

The bed shear stress can be given by the Manning formula (as an example):

$$\tau = \rho g \frac{V^2}{M^2 h^{1/3}} \tag{7.5}$$

where,

$\rho$ fluid density (kg/m³)

$g$ acceleration of gravity (m/s²)

$M$the Manning number (m$^{1/3}$/s)

$h$flow depth (m)

$V$flow velocity (m/s)

Substituting the bed shear stress into the deposition equation results in the following equation:

$$S = \frac{wC}{h^*}\left(1 - \left(\frac{V}{V_{cd}}\right)^2\right), \ V \leq V_{cd}$$ (7.6)

where,

$V_{cd}$critical deposition velocity.

## Single Cohesive Layer Model - Erosion

The resistance against erosion of cohesive sediments is determined by the submerged weight of the individual particles and by the interparticle electro-chemical bonds which must be overcome by the shear forces before erosion occurs.

$$S = \frac{M^*}{h}\left(1 - \frac{\tau}{\tau_{ce}}\right)^n, \ \tau \leq \tau_{ce}$$ (7.7)

where

$S$source term in the advection dispersion equation

$\tau$bed shear stress (N/m$^2$)

$\tau_{ce}$critical shear stress for erosion (N/m$^2$)

$M^*$erodibility of the bed (g/m$^2$/s) (= erosion coefficient)

$h$flow depth (m)

$n$erosion exponent

Using the Manning formula as described in the deposition section above, the following expression for the erosion rate can be derived:

$$S = \frac{M^*}{h}\left(1 - \left(\frac{V}{V_{ce}}\right)^2\right)^n, \ V \geq V_{ce}$$ (7.8)

where

$V_{ce}$ critical erosion velocity

## Multi Layer Cohesive Model - Deposition

Deposition occurs when the bed shear stress is smaller than a critical shear stress for deposition. In the advanced cohesive model the rate of deposition ($S_d$) is given by:

$$S_d = w_s\left(1 - \frac{\tau_b}{\tau_{c,d}}\right)c , \ \tau < \tau_{c,d}$$  (7.9)

where

$w_s$ mean settling velocity of suspended particles (m/s)

$S_d$ rate of deposition (kg/m²/s)

$\tau$ critical shear stress for deposition (N/m²)

$c$ suspended sediment concentration (kg/m³)

All deposited material is added to sub-layer 1.

The model concentration $c$ is weighted in time according to the following expression:

$$c = (1-\theta)c_j^n + \theta c_j^{n+1}$$  (7.10)

where:

$j$ spatial index

$n$ time index

$\theta$ the time centring for deposition

## Multi Layer Cohesive Model - Erosion

The erosion process can be described as either instantaneous or gradual. Instantaneous erosion occurs when the bed shear stress exceeds the critical shear stress for erosion of the sediment. This implies that all sediment is resuspended instantaneously.

The gradual erosion is described by an erosion rate assumed to be a non-linear function of the excess stress:

$$S_E = E_o(\tau_b - \tau_{c,e})^n , \ \tau_b > \tau_{c,e}$$  (7.11)

where,

$S_E$rate of erosion (kg/m$^2$/s)

$E_o$erosion coefficient (kg/s/N)

$\tau_{c,e}$critical shear stress for erosion n: erosion exponent

$n$exponent

Both instantaneous and gradual erosion formulations can be applied to sub-layer 1. Gradual erosion is automatically applied for sub-layers 2 and 3. Thus, it is possible to describe each sub-layer separately through the parameters $E_o$, and $n$. The erosion rate can be specified in terms of velocity or shear stresses.

## 7.7    Sediment Layers

Initial conditions for the sediment layers may be defined on the Sediment Layers page. Selection drop-down menus are available for the component types 'Single cohesive', 'Multi cohesive' or 'Non cohesive' and hence, only components defined as one of these types can be selected in this page.



Figure 7.7      The Sediment Layers property page

## Location

The Location specification includes the definition of locations of initial values of sediment layers as well as the values for sediment layer definitions.

### Component

Select from the drop-down box one of the allowed component types: Single Layer Cohesive, Multi Layer Cohesive and Non-Cohesive. If no components have been selected as one of these types, the selection box will appear empty.

### Layers

Only available when Component is chosen as a Multi Layer Cohesive component. The user can select between Upper, Middle and Lower representing the three layers in the Multi Layer Cohesive model. Parameters must be specified for each of the layers.

### Table

Only applicable for Multi Layer model components. Instead of giving the initial conditions in Height *(p. 386)*, a more detailed initial condition can be specified using a width-Height table, see Parameters *(p. 386)*.

### Height

Although the header says 'Height' the initial data should be entered as volume of sediment per length of river. In order to convert this initial data into an amount MIKE 11 uses the porosity and the relative density specified in the Non-cohesive ST *(p. 389)* property page.

### Density

The density of the layer.

### Pot. fac. (Potency factor)

The Potency factor is a left-over from previous Sediment Water Quality interaction simulations performed through the AD Parameter Editor and not used in the present version of MIKE Zero.

### Global

If this box is checked the entered parameters are used globally.

### River Name

The name of the river for which the data applies.

### Chainage

The chainage of the river for which the entered data applies.

## Parameters

For multi layer components a volume width relation can be entered. The width in this relation is the width of the cross section, the volume is the volume of

sediment per length of the river. It is hereby possible to vary the thickness of the sediment layer along the transverse of the cross section.

## Single layer cohesive component.

When the single layer model is used only one sediment layer is displayed. The sediment layer initial conditions are defined by the following parameters:

### Pot. fac. (Potency factor)

Initial amount of BOD attached to the sediment (kg BOD / kg sediment). The Potency factor is a left-over from previous Sediment Water Quality interaction simulations performed through the AD Parameter Editor and not used in the present version of MIKE Zero

## 7.7.1 Initial layer thickness values from external file

Please note that this option applies only for the advanced, 3-layer cohesive sediment transport module (ACST).

As an alternative to the manual definition of initial layer thickness values through the AD Parameter Editor as described above it is possible to define initial layer thickness values through an ASCII text file. The calculation engine reads and utilises this file during pre-processing of the model simulation.

The ASCII file must fulfil a number of requirements to be used as input to the simulation:

- The filename is 'LayerInp.00?' where the last '?' in the file extension refers to the Component number for which the layer thickness shall be defined. That is, if an ACST component is defined e.g. as component number 2 in the list of defined components, then the filename must be; 'LayerInp.002'

- The ASCII files must be located in the simulation folder. That is, the folder where the .sim11 file for the actual simulation is located.

- If the simulation engine detects a file with the requested name, a message box will appear as presented in Figure 7.8 below. From the dialog it can now be chosen whether the initial layer depths should be read from the external file (Press Yes) or using the definitions from the AD Parameter file (Press No).
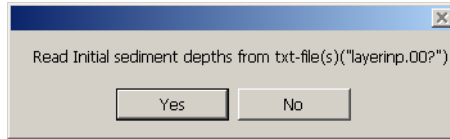
Figure 7.8 Confirmation dialog for reading initial sediment layer depths from external ASCII file

- The ASCII file contains information on levels, widths and layer depths (in m3/m) for 3 layers. Levels and Widths must be identical to the values for each cross section and h-points in the model domain. The format is presented below. Data for different cross section points are divided with a separator string of stars ('*********************')

```
Component number :  River branch name :  Gridpoint chainage [m] :  Branch nr ic:
          1 ;           CALI ;             0.0000 ;           0
   level [m]      width [m]      layer1 [m3/m]     layer2 [m3/m]    layer3 [m3/m]
    54.760  ;       0.000 ;       0.0000 ;        0.0000 ;        0.0000
    54.897  ;      18.285 ;       0.0000 ;        0.0000 ;        0.0000
    55.033  ;      34.372 ;       0.0000 ;        0.0000 ;        0.0000
    55.175  ;      41.557 ;       0.0001 ;        0.0020 ;        0.0030
    55.311  ;      95.534 ;       0.0002 ;        0.0020 ;        0.0030
    55.448  ;     176.989 ;       0.0003 ;        0.0020 ;        0.0030
    55.585  ;     264.186 ;       0.0004 ;        0.0020 ;        0.0030
    55.721  ;     366.189 ;       0.0006 ;        0.0020 ;        0.0030
    55.858  ;     480.683 ;       0.0007 ;        0.0020 ;        0.0030
    55.995  ;     557.637 ;       0.0010 ;        0.0021 ;        0.0030
    56.131  ;     589.673 ;       0.0009 ;        0.0020 ;        0.0030
    56.267  ;     639.047 ;       0.0006 ;        0.0020 ;        0.0030
```

Figure 7.9 File format of ASCII input file for initial cohesive sediment layer depths

## Auto-generated ASCII file with Initial conditions layer values

Evidently, it is difficult to manually prepare an input file with the correct format and content, so the calculation engine automatically generates an ASCII file for you, which can serve as a template for defining initial conditions for sediment layers for the simulation.

Running a simulation with a ACST component, the calculation engine automatically creates an ASCII file, which contains the initial sediment layer depths as distributed functions of water level and width for all calculation points in the model. This ASCII file is named; 'LayerOut.00?' where the '?' in the extension name represents the component number from the defined list of model components. You can use this file to define your own initial layer values by simply editing the file with your required initial layer values and rename it to 'LayerInp.00?'. Thereby, the simulation engine will use your values as layer initial conditions.

Note that the LayerOut.00? file(s) always contains the sediment layer values as defined in the AD Parameter, so the content of this file will not change regardless of which layer input file you eventually use for the simulation.

## Auto-generated ASCII file with layer values at End of Simulation

As an alternative to using the above described 'LayerOut.00?' files as initial conditions an additional ASCII file is generated with terminal layer values.

This ASCII file is named; <ADResultFileName>.CST' and it is located in the result-file folder for the AD Result file.

This CST-file will contain the values for the sediment layers at the time where the simulation ended, which may be useful initial conditions for consecutive simulations.

### Regular hotstart including ACST components

Please be informed that the standard Hotstart facility also includes the hot-start of sediment layers such that a previous AD-result file can also be used as initial conditions.

If Hotstart is used as initial conditions for the AD-simulation then the Layer-Inp.00? file option is not used and consequently, the message-box (Figure 7.8) will not appear for AD-simulations using hotstart.
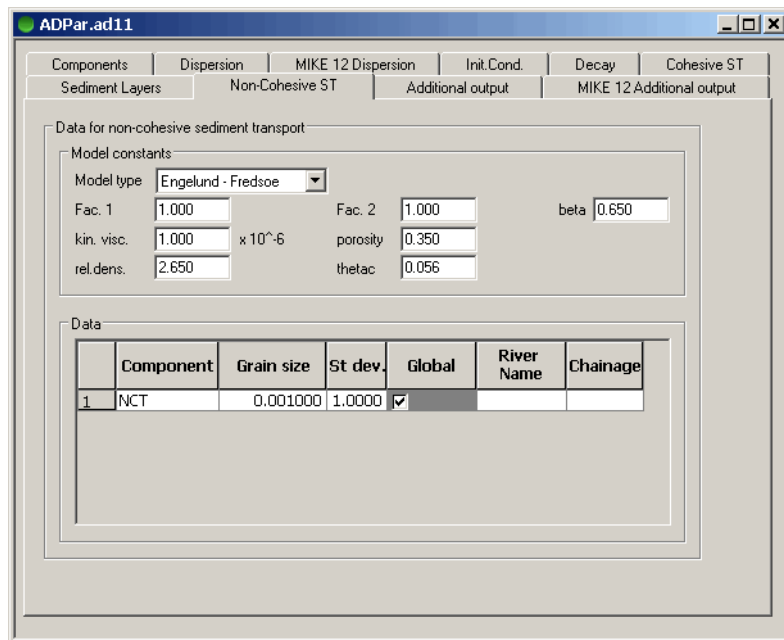
## 7.8 Non-cohesive ST



Figure 7.10    The Non-Cohesive property page

This page contains input parameters for Non-Cohesive components. A non-cohesive component is defined using the data section at the bottom of the page.

Please note (as mentioned also in the Components section):

This option for simulation a non-cohesive component within the AD Parameter Editor Framework is actually a left-over from earlier versions of MIKE 11 where combined Sediment Water Quality interaction was simulated through this interface. However, now WQ-simulations are made using the MIKE ECO Lab Framework and the non-cohesive component type is only kept for backward compatibility.

Simulating a Non-cohesive sediment component through the AD Parameter Editor does not include morphological updating and as such it is not appropriate for describing the physics.

For Non-cohesive sediment transport simulations, please apply the ST-Module instead.

## Model constants

### Model Type
A pop down menu provides a choice from two types of sediment transport formulations; the Engelund-Fredsøe and the van Rijn model.

### Fac.1
Calibration factor for bed load transport. The calculated bed load is multiplied by the calibration factor.

### Fac.2
Calibration factor for suspended load transport. The calculated suspended load is multiplied by the calibration factor.

### Beta
Dynamic friction factor used in the Engelund-Fredsøe model.

Typical range: 0.50 - 0.65.

### Kin.visc.
The kinematic viscosity of water.

### Porosity
The porosity of the sediment.

### Rel.dens.
The relative density of the sediment.

### Thetac
Shield's critical parameter. Typical range: 0.04 - 0.06.

### Data

#### Component
Here a Non-cohesive component is selected.

#### grain size
The $D_{50}$ value.

#### st dev.
Standard deviation in the grain size distribution.

#### Global
If this box is checked the entered parameters are used globally.

#### River Name
The name of the river for which the data applies.

#### Chainage
The chainage of the river for which the entered data applies.

## 7.9    Additional Output

The additional output page contains check boxes which can be used to store internal model parameters in an additional result file. (The filename is: '<AD filename>ADAdd.res11').

#### Mass
The mass in the system. Given in the units specified on the 'Components' property page. Total and total accumulated as well as grid and grid accumulated values can be selected.



Figure 7.11    The additional output property page

#### Mass balance
The mass balance is given in o/oo (per thousands). Total and total accumulated as well as grid and grid accumulated values can be selected.

### 1. order decay

The 1st order decay is given in the units specified on the 'Components' property page, per second. Total and total accumulated as well as grid and grid accumulated values can be selected.

### Mass in branches

The mass in river branches given in the unit specified on the 'Components' property page. Total and total accumulated values can be selected.

### Transport, total

The total transport is given in the unit specified on the 'Components' property page, per second. Grid and grid accumulated values can be selected.

### Dispersive transport

The dispersive transport is given in the unit specified on the 'Components' property page per second. Grid and grid accumulated values can be selected.

### Convective transport

The convective transport is given in the unit specified on the 'Components' property page, per seconds. Grid and grid accumulated values can be selected.

## 7.10 AD Simulations, Additional features

This section includes a number of additional features for the AD simulation, which can be activated through other options than the Parameter Editor.

### 7.10.1 Alternative Solution scheme; The ULTIMATE method

An option for changing the solution scheme from the traditional Finite Difference scheme to the so-called Ultimate solution scheme is available.

The AD simulation engine can be instructed to utilise the ULTIMATE solution scheme by placing an ASCII text-filenamed 'ULTIMATE.TXT' in the folder where the sim11-file is located. Should a file with this name be found in the simulation folder, then the Ultimate method is automatically activated.

So why use the ULTIMATE method?

The ULTIMATE method is very strong and efficient at resolving very sharp concentration gradients at neighbouring calculation points, where the standard Finite Difference method typically makes a solution with significant numerical 'wiggles'. The Ultimate method is able to keep the sharp gradients without generating wiggles as illustrated in Figure 7.12.
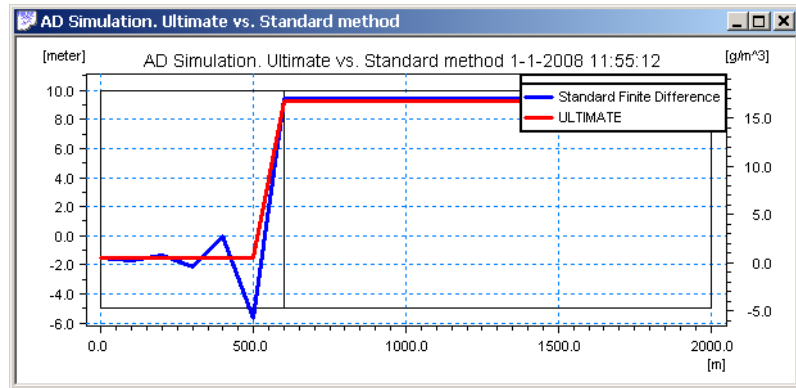
Figure 7.12    Simulation results, longitudinal profile of AD-simulation with abrupt change in concentration due to high-concentration tributary inflow to main river. Blue line presents concentration profile from calculation of standard Finite Difference method and Red line presents results obtained from the ULTIMATE method

**Important**: Though the ULTIMATE solution method is a very efficient and strong method which supplies accurate results also at locations with very sharp concentration gradients there is a significant reason why this methodology is not applied as a default method!
The ULTIMATE method is derived from a fundamental assumption that the Courant Number must never exceed 1! That is, if you have locations where the Courant Number exceeds 1 then the ULTIMATE method can simply not be applied. This often sets such a high restriction on the combination of grid-spacing (dx) and time step (dt) to keep the Courant Number below 1 that it is not feasible in all applications.

## 7.10.2  AD simulation features activated through the MIKE11.ini file

Located in the 'bin'directory of the MIKE Zero installation you will find a MIKE11.ini file which is used to activate and control specific behaviour and performance of the calculation engine.

The MIKE11.ini file also contains a section specifically for AD-simulations (section named '[AD_SIMULATION]') and a few of the parameters possible to alter in the ini-file are mentioned below.

Please note that it is recommended to copy the MIKE11.ini file from the bin-folder to your project simulation folder (must be located where you have your sim11-file). The simulation engine searches for the MIKE11.Ini file locally first, and secondly for the global file in the bin-folder. By copying the file to the local folder, you ensure, that the changes you make in the ini-file only concerns the present project, which is often required as changes in the ini-file are possibly

required and beneficial for one project, whereas different values or parameters must be altered in other projects.

Variables from the ini-file of interest here include:

### AD-Variables no 1:
'ADLOG=ON'

Automatic creation of an Error log-file from AD-Simulations containing Courant number warnings can be disabled by changing the variable.

### AD-Variables no 3:
'MORPH_UPD_IN_COHE_3LMODEL=OFF'

Morphological updating of bed topography due to erosion/deposition of cohesive sediments can be activated (for the ACST, 3-layer cohesive model only) by adjusting this variable. Setting this variable to ON also activates bed level as an additional output variable in the AD-result-file from the ACST simulation.

WQ ECO LAB EDITOR

# 8    WQ ECO Lab Editor

MIKE ECO Lab is a numerical lab for Ecological Modelling. It is an open and generic tool for customizing 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 tailormade 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 pre-defined 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).

## 8.1    Model Definition

The Model Definition page consists of the selection of the template (.ecolab) file, which will form the basis of the Water Quality simulation. The .ecolab file contains the definitions of the MIKE ECO Lab model. The content of the remaining MIKE ECO Lab Dialogs depends on the contents of this file as parameters, forcing functions, output variables etc. are extracted from the MIKE ECO Lab file and presented in the dialog pages of the MIKE ECO Lab Editor. The MIKE ECO Lab file can be selected by choosing the "From File..." item in the combo-box and browsing to the location of the file. This procedure is normally adopted for selecting user-defined .ecolab files.In case your MIKE 11 Installation includes one or more of the pre-defined MIKE Water Quality models, they will be listed in the combo-box as well - or alternatively, then can be found in the Template folder under the installation (typically; '\MIKE Zero\Templates\ECOLab').

Having selected the .ecolab file, a brief summary of the contents of the model is shown in the Dialog as presented in Figure 8.1.
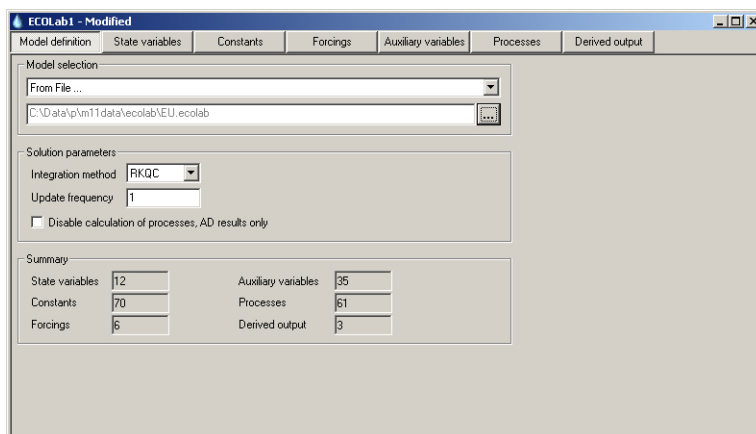


Figure 8.1    Menu for Water Quality Model Definition

Please note that every time a new .ecolab file is selected, the specifications of all the remaining MIKE ECO Lab Dialogs are reset to default values.

The specification of the Solution Parameters includes selection of the Integration Method for the coupled ordinary differential equations defined in the .ecolab file. At present the following three methods are available (please consult the MIKE 11 Reference Manual for details on the listed options available for solving the coupled linear differential equations in the MIKE ECO Lab framework):

- Euler integration method
  A very simple numerical solution method for solving ordinary differential equations.

- Runge Kutta 4th order
  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.

- Runge Kutta 5th order with quality check
  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.

Finally the Update Frequency has to be specified. This parameter is the number stating 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 used for the HD- and AD-simulation and therefore determines the frequency for simulating water quality processes on top of the standard Advection and Dispersion processes which are calculated at every time step. The Update Frequency must be an integer above zero. Definition of the frequency should be based on careful considerations. The user should recognise that the dynamics of the advection dispersion is comparable to the dynamics of the process descriptions. Thus it is strongly recommended to use an update frequency of 1.

The selection of the Time Step of the MIKE ECO Lab model, and hereby the Update Frequency, has to be based on considerations of the time scales of the processes involved. Please notice that this selection can be rather decisive for the precision of the numerical solution as well as for the CPU time of the simulation. A large Update Frequency will decrease the precision as well as the CPU time. It is therefore advisable to perform a sensitivity analysis on the Update Frequency before making the final selection.

## 8.2 State Variables

The State Variables Dialog shows a summary of the State Variables defined in the MIKE ECO Lab model. The Description, Unit and Transport type of each State Variable are given ("No Transport" refers to a fixed State Variable and "'Transport" refers to a State Variable, which is transported by Advection-dispersion).
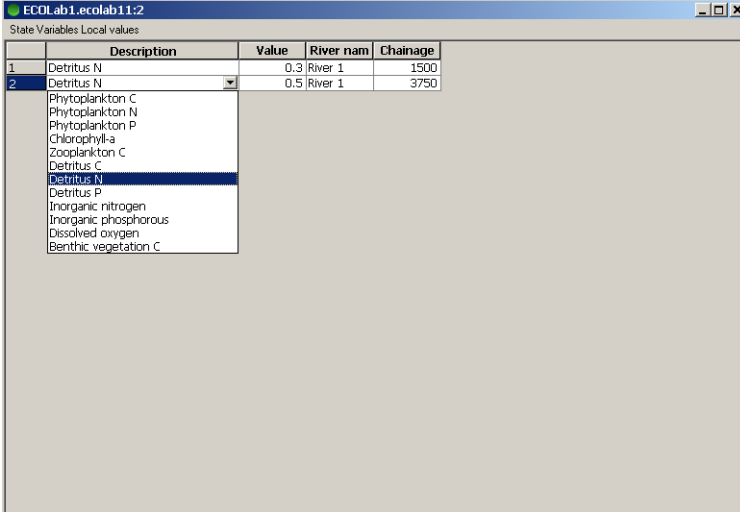


Figure 8.2    The State Variables tab

For each State Variable its initial value within the model area should be specified. It can be specified in one of two ways: As a constant value applied to all points in the area through the 'Value' column or with local exceptions. Local

exceptions for parameter values are defined by opening the local variables sub-dialog. Press the [___] button in the 'Local' column and a local values sub-dialog will open as presented in Figure 8.3.



Figure 8.3    State Variables, Local Value definition page

To generate table-lines where local values can be defined, use the <TAB> key at the last row of the table or press the <Insert> key. Local values can be defined for one or multiple of the state variables in this page. Select the variable for which the local variables must be defined from the drop-down selection box containing all the model state variables in the 'Description' column and specify river location (Rivername and Chainage) and the local value of the specific parameter.

Note that if you want to specify local variations at the same locations for different state variables, you must define this in individual lines for each State Variable.

## 8.3    Constants

The Constants are defined as any input parameter (physical constant, coefficient, rate, etc.) in the MIKE ECO Lab model, which is constant in time. The Constants are essentially divided into two groups:

- Built-in Constants and
- User-specified Constants

The built-in Constants are automatically provided by the model system during execution, whereas the user-specified Constants have to be specified in the present Dialog.

Depending on the Spatial Variation of the Constant, as defined in the MIKE ECO Lab model, it can be specified as a "Constant value" or as local values. Please note that a Constant, which is defined as a built-in Constant in the MIKE ECO Lab model, will appear as a user-defined Constant in case it is not supported by MIKE 11.



| | Description | Unit | Value | Local |
|---|---|---|---|---|
| 1 | Latitude | Degrees | 55 | |
| 2 | Pelagic paramenters: Growth rate diatoms | per day | 0.5 | |
| 3 | Pelagic paramenters: Growth rate green algae | per day | 1 | |
| 4 | Pelagic paramenters: Sedimentation rate <2m | per day | 0.25 | |
| 5 | Pelagic paramenters: Sedimentation rate >2m | meter/day | 0.5 | |
| 6 | Pelagic paramenters: Max. grazing rate | per day | 1.5 | |
| 7 | Pelagic paramenters: Death rate phytoplankton | per day | 0.02 | |
| 8 | Pelagic paramenters: Zooplankton death rate 1st ord | per day | 0.1 | |
| 9 | Pelagic paramenters: Zooplankton death rate 2nd ord | per day | 12 | |
| 10 | Pelagic paramenters: Oxygen reaeration constant | per day | 1.5 | |
| 11 | Pelagic paramenters: Detritus C mineralisation rate | per day | 0.05 | |
| 12 | Pelagic paramenters: Detritus C settling rate <2m | per day | 0.05 | |
| 13 | Pelagic paramenters: Detritus C settling rate >2m | meter/day | 0.1 | |
| 14 | Pelagic paramenters: Light extinction constant phytop | m2/g Chl-a | 20 | |
| 15 | Pelagic paramenters: Light extinction background con | m2 | 0.35 | |
| 16 | Pelagic paramenters: Light extinction detritus C | m2/g detritus | 0.1 | |
| 17 | Pelagic paramenters: Light extinction constant macro | m2/g macroal | 0.02 | |
| 18 | Pelagic paramenters: Light extinction constant suspe | m2/g SS | 0.1 | |
| 19 | Macroalgea parameters: Sloughing rate at 20 degree | per day | 0.01 | |
| 20 | Macroalgea parameters: Production rate at 20 degree | per day | 0.25 | |
| 21 | Macroalgea parameters: Respiration rate at 20 degre | per day | 0 | |
| 22 | Simple sediment description: Proportional factor for s | dimensionles | 1 | |
| 23 | Simple sediment description: Proportional factor for N | dimensionles | 1 | |
| 24 | Simple sediment description: Proportional factor for P | dimensionles | 1 | |
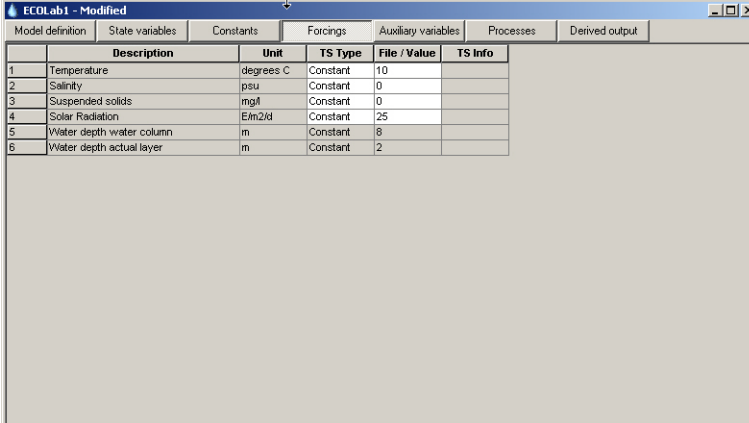
Figure 8.4    Menu for Constants

Constants can be declared as spatial varying in the MIKE ECO Lab template file and for these, it is possible to specify local variation of the parameter values through the 'Local' column. Definition of local values are made in the same was as described for defining local values of State Variables.

## 8.4    Forcings

The Forcings are defined as any input parameter (physical property, rate, etc.) in the MIKE ECO Lab model, which is varying in time. Examples of a Forcing are: Temperature, salinity, solar radiation and water depth. The Forcings are essentially divided into two groups:

- Built-in Forcings and
- User-specified Forcings

Figure 8.5    Menu for Forcing Functions

The built-in Forcings are automatically provided by the model system during execution, whereas the user-specified Forcings have to be specified in the present Dialog. Depending on the Spatial Variation of the Forcing, as defined in the MIKE ECO Lab model, it can be specified as a "Constant value" or a "Type 0 data file".

Please note that a Forcing, which is defined as a built-in Forcing in the MIKE ECO Lab model, will appear as a user-defined Forcing in case it is not supported by MIKE 11.

Alto note that the forcings called "Water depth " and "Water layer height" both hold the actual hydraulic radius as a measure for the water depth rather than for instance the maximum or the average depth across the cross section.

## 8.5    Auxiliary Variables

In case Auxiliary variables (or help processes) has been defined in the MIKE ECO Lab template file, these can be stored as output in an additional output result-filenamed '<AD filename>WQAdd.res11'. The author of the MIKE ECO Lab file has decided which of the auxiliary variables described in the MIKE ECO Lab file that the user can select and store as additional output. Simply tick the auxiliary variables you want to save.
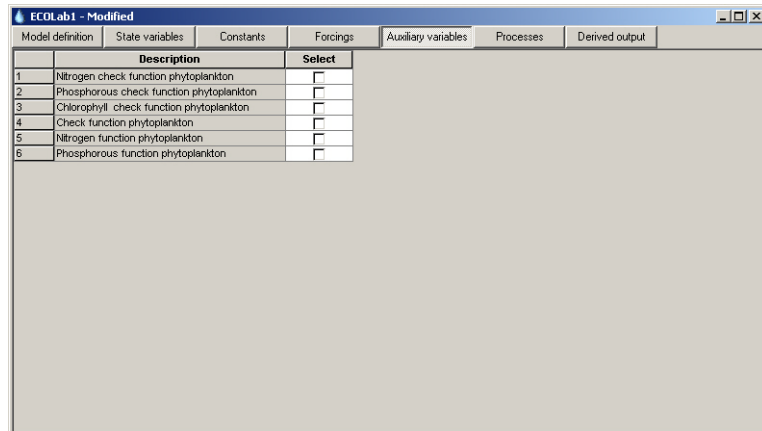
Figure 8.6     The Auxiliary Variables tab

## 8.6     Processes

Processes defined in the MIKE ECO Lab file, and at the same time marked as OUTPUT, can be saved in the additional output file named '<AD file-name>WQAdd.res11'. Also for the processes the author of the MIKE ECO Lab file has decided which of the processes that you can store as additional output. Simply tick the processes you want to save.
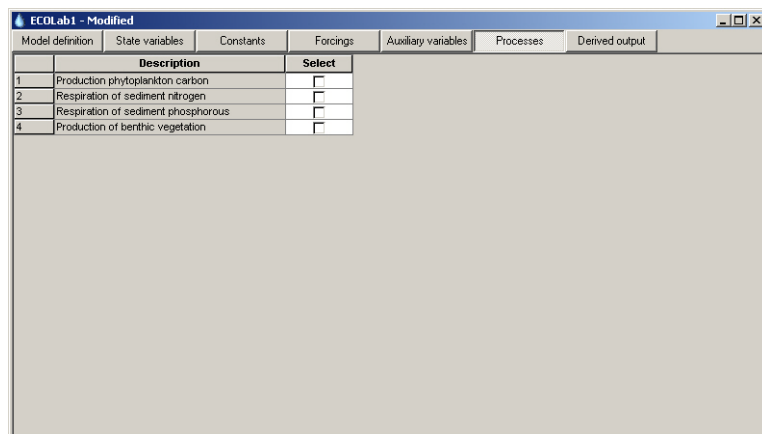


Figure 8.7     The Processes tab

## 8.7     Derived Output

Derived output defined in the MIKE ECO Lab file could be the sum of various state variables (e.g. Total N = Organic N and Inorganic N) that are useful to save in the additional output file ('<AD filename>WQAdd.res11') to avoid for instance any manual post-processing of the main model results. The author

of the MIKE ECO Lab file may have chosen other types of processed model results from which you can select the derived output. Simply tick the derived output you want to save.
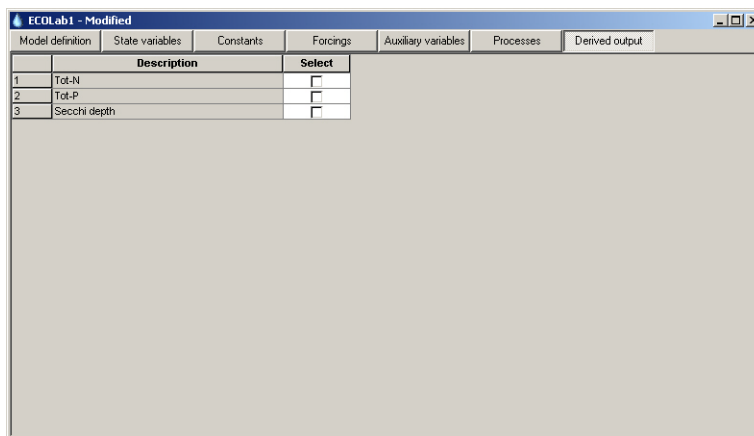


Figure 8.8    The Derived Output tab

# SEDIMENT TRANSPORT EDITOR

# 9 Sediment Transport Editor

The MIKE 11 non-cohesive sediment transport module (NST) permits the computation of non-cohesive sediment transport capacity, morphological changes and alluvial resistance changes of a river system.

Input data concerning non-cohesive sediment properties are defined in the ST Parameter Editor which contains the following tabs (property pages):

- Sediment Grain Diameter *(p. 409)*
- Transport Model *(p. 410)*
- Calibration Factors *(p. 417)*
- Data for Graded ST *(p. 418)*
- Preset Distribution of Sediment in Nodes *(p. 419)*
- Passive Branches *(p. 421)*
- Non-Scouring Bed Level *(p. 422)*
- Initial Dune Dimensions (Engelund Fredsoe Model) *(p. 423)*

Opening view of the ST Parameter editor is presented in Figure 9.1below.



Figure 9.1    ST Parameter Editor, Opening view

Some of the sediment transport formulas and other features of the Non Cohesive Sediment Transport module have been developed in cooperation with CTI Engineering CO., Ltd., Japan.

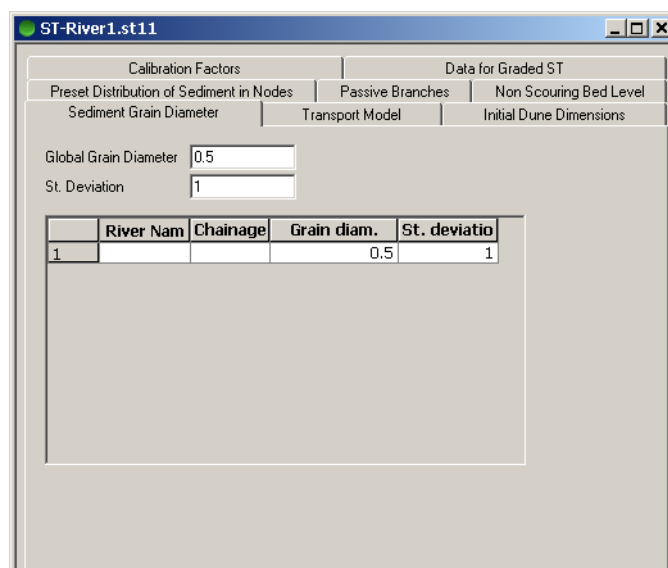### 9.0.1 Sediment transport simulations; Simulation mode

#### The explicit sediment transport mode

In the explicit mode, the sediment transport computations are based either on the results from an existing hydrodynamic result file or from a hydrodynamic computation made in parallel using characteristic transport parameters. The sediment transport is calculated in time and space as an explicit function of the hydrodynamic parameters (i.e. discharge, water levels etc.) previously calculated. Note that there is no feedback from the sediment transport calculations to the hydrodynamics. Results are volume transport rates and accumulated volumes of deposition or erosion.

The explicit mode is useful where significant morphological changes are unlikely to occur. An estimate of the sediment budget can then be obtained economically (in terms of computer time) using this mode.

The explicit sediment transport mode is active if the check box; 'Calculation of Bottom Level' is unchecked (in the 'Transport Model' page).

#### The morphological mode

Sediment transport computations made in the morphological mode are made in parallel with the hydrodynamic computations. The morphological mode is activated through the 'Transport Model' tab page by activating the check-box; 'Calculation of Bottom Level'.The sediment transport is calculated in time and space as an explicit function of the corresponding values of the hydrodynamic parameters calculated in tandem. The sediment transport module solves the sediment continuity equation and determines the updating of bed resistance, transport rates, bed level changes and dune dimensions (depending on the transport relationship adopted), so that changes in flow resistance and hydraulic geometry due to the sediment transport can be included in the hydrodynamic computations.

The morphological simulation mode requires considerably more computation time than the explicit mode but is more representative of the dynamic alluvial processes.

### 9.0.2 The transport models

A variety of transport models are available. Some of the transport models determines the total sediment transport and others distinguish between bed load and suspended load. Following transport models are available:

- Engelund - Hansen (Total load)
- Ackers - White (Total load)
- Smart - Jaeggi (Total load)

- Engelund - Fredsøe (Bed load and Suspended load)

- Van Rijn (Bed load and Suspended load)

- Meyer Peter and Muller (Bed load)

- Sato, Kikkawa and Ashida (Bed load)

- Ashida and Michiue Model (Bed load and Suspended load)

- Lane-Kalinske (Suspended load)

- Ashida, Takahashi and Mizuyama (ATM) (bed load)

All of the transport models can be used for both explicit and morphological mode computations.

No general guidelines can be given for the preference of one model over another, as the applicability of each depends on a number of factors. Further details can be found by consulting the NST Reference Manual.

Sediment transport is a highly non-linear function of the flow velocity. Depending on the model used, the transport is proportional to the velocity raised to the 3rd or 4th power. Instabilities may occur in certain cases even when the hydrodynamic computation is stable. Special care must be taken in the determination of initial conditions and time step selection to avoid instability problems.

Features and usage of the ST Parameter Editor pages are described below.

## 9.1    Sediment Grain Diameter

Sediment grain diameter(s) and standard deviation(s) of grain size to be used in the sediment computations are specified in this page. The grain diameter and standard deviation may be specified as being applicable globally and locally. If grain diameters and standard deviations are specified for a local application, these values are used instead of any globally specified values.

The Standard Deviation defined in this page is only applied for two of the Transport Models available;

- the Van Riin model utilises the standard deviation for calculating the mean diameter and particle size diameter used in the equation set of the Van Rijn model, and

- the Engelund-Fredsoe model utilises the standard deviation in the Suspended Load calculation to determination the grain size diameter applied in calculating the effective fall velocity

- The other transport models available does not include the standard deviation in the respective set of equations and hence, adjusting the standard deviation when applying any of these other transport models will not affect the simulation results.

Figure 9.2 shows an example where the sediment grain diameter is globally set to 0.5 mm. This value will be used in the entire river network except for the reaches 'RIVER1' between 1000 m and 2500 m, where the local grain diameter varies linearly between 1.2 and 1.5 mm. and between 2500 m and 4400 m where the grain diameter varies linearly between 1.5 and 1.1 mm. At the same chainages, the standard deviation varies linearly between 1.2, 1.2 and 1.0.
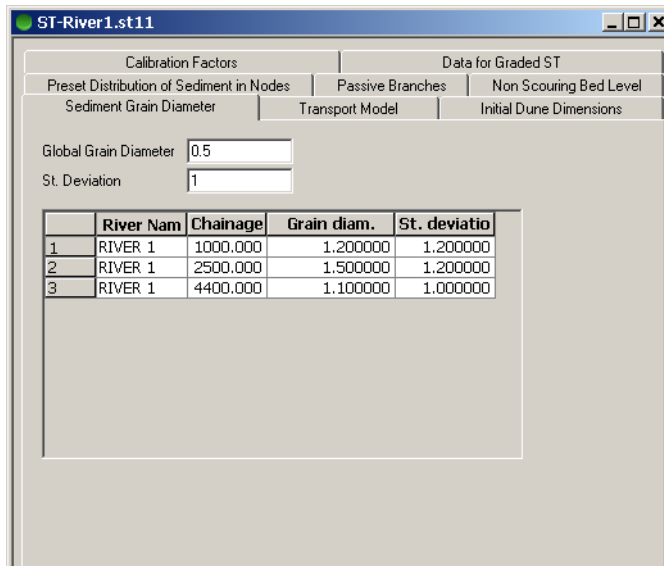


Figure 9.2    Example of implementation of local grain diameter

## 9.2    Transport Model

Selection of sediment transport model as well as editing the model specific parameters are essential for the calculation of the sediment transport. This page should therefore always be visited and verified by the user to set the model type (Total load or bed load/suspended load model) and the appropriate transport model(s) as well as adjusting transport parameters if required prior to making any ST-simulation.

Figure 9.3 shows an example of how to set the transport model type and appropriate parameters in the dialog. In this example, the bed load transport will be calculated using the 'Engelund and Fredsoe' model and the suspended load transport calculated using Van Rijn formula. Morphological computation is selected as the check box for 'Bottom Level' is activated, but there will be no computing of the bed shear stress.
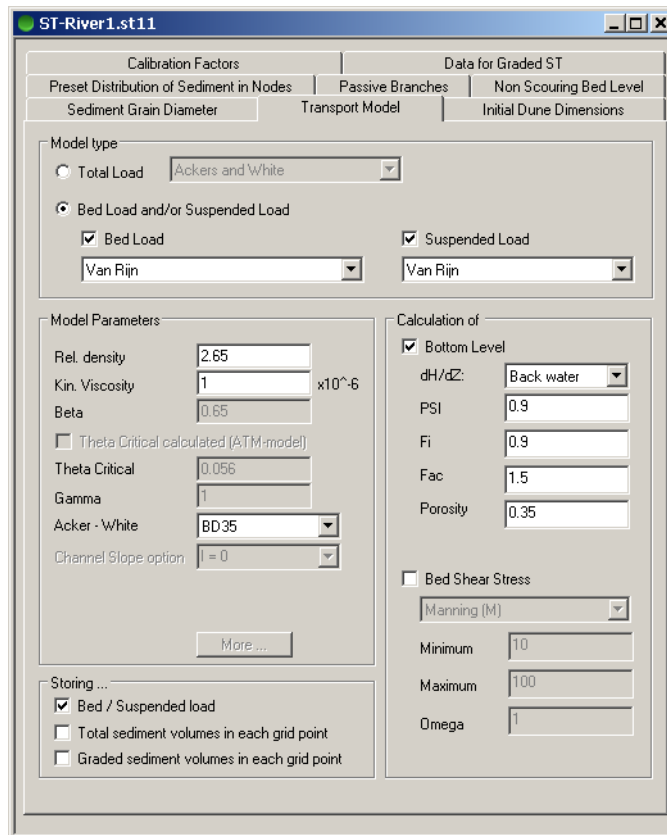
Figure 9.3    Sediment transport model parameters. Selection of bed load and suspended load models. Morphological updating is included but Bed Shear stress updating excluded in this example

**Note**: When selecting Transport models, which divides sediment transport into Suspended Load and Bed load respectively, it is generally recommended - if possible - to apply the models of the same author (e.g. Van Rijn or Engelund-Fredsoe) to secure that identical assumptions are used in the formulation of - and division between - suspended and bed load characteristics.

## 9.2.1    Model parameters

The transport model parameters are divided into three sub-groups: Model parameters appears visible as a function of the selected transport models.

### Rel. density

Relative density (or Specific gravity) of the sediment. Relative density determined as; $\rho_{sediment}/\rho_{water}$.

### Kin. Viscosity

Kinematic viscosity of water.

Note that the Kinematic Viscosity must be specified as 'value · $10^{-6}$' m$^2$/s (for SI-Units). That is, if a value of 0.000001 m$^2$/s should be used, in the dialog, you must specify 1.0.

### Beta

Dynamic friction coefficient used in the Engelund-Fredsoe model. Please consult the MIKE 11 Reference Manual for further details.

### Theta Critical

Critical Shields' parameter for incipient motion of sediment.

In case of Ashida, Takahashi and Mizuyama (ATM) bed load model a selection box is made visible and the user can hereby select whether Theta Critical should be calculated or if the constant value from the input page should be applied. If 'Theta Critical calculated' has been selected, the value of Theta Critical will be calculated from Iwagaki's model and the modified Eqiazaroff model (for further details, see the MIKE 11 NST reference manual; section of 'Incipient Motion Criteria').

### Gamma

Calibration parameter applied to suspended load with the Engelund-Fredsoe model when calculating the height of sand dunes. For further details please consult the MIKE 11 Reference Manual.

### Ackers-White

Switch used in the Ackers-White model indicating whether the applied grain size represents $d_{35}$ or $d_{65}$.

### Channel slope option (ATM bed load model only)

The Ashida, Takahashi and Mizuyama bed load model takes into account effects of channel slope on the calculated bed load. Different options are available for representing the channel slope 'I': I=0, I=Energy Grade and I= bed level slope.

### Storing

- Bed / Suspended load

- Storing of suspended load and bed load as individual result items in the ST result file from a simulation. This feature is only applicable for those of the transport models which separates the sediment transport into bed load and/or suspended load components.

- Total sediment volumes in each grid point

- Storing of total sediment volume in each grid point - accumulated over time. That is;

- Graded sediment volumes in each grid point

- Storing of sediment volume of each fraction in each grid point.

## Parameters used for morphological computations

### Calculation of Bottom Level
If Morphological updating of bed levels is required in the sediment transport simulation, then activate this check-box. If the check-box is left non-active, the sediment transport calculation will be performed in the so-called Explicit sediment transport mode.

### dH/dZ
Calculation parameter for the morphological model.

### PSI
Centring of the morphological computation scheme in space.

### FI
Centring of the morphological computation scheme in time.

### FAC
Calibration parameter for computation of derivatives in the morphological model.

**Note** that this parameter implicitly defines the step length for a number of numerical derivatives. For this reason the parameter must be greater than unity. If this is not the case MIKE 11 sets its value equal to 1.01 internally.

### Porosity
Porosity of the sediment (pore volume/total volume).

## Parameters used if updating of bottom shear stress is included

### Bed Shear Stress
A check box is provided to include or exclude bed shear stress updating during the simulation.

### Resistance type combo box
The user is given the option to select which shear stress / resistance type formulation to be used for defining minimum and maximum limits of resistance number applied in the ST simulation (Manning's M, Manning's n or Chezy). Values of resistance number limits must be defined in the selected type units.

### Omega
Calibration parameter for the resistance number. (ResistanceST = OMEGA * ResistanceHD). Note that Omega is applied to the resistance number, which is internally in the MIKE 11 engine calculated as Manning's M. Hence, an increased value of Omega will result in a smaller overall resistance (higher

value of M) and a reduced value of Omega will result in a higher overall resistance.

### Minimum/Maximum

Minimum/maximum limits for the calculated resistance number in the computations.

**Please Note:** If calculation of the bottom shear stress is selected in a morphological computation, the updated shear stress values are used in the hydraulic computations as well. Thus, the Chezy or Manning number specified in the cross-section data base may differ from the value(s) applied in the hydrodynamic computations.

Also, note that only few of the authors of sediment transport formulas has considered formulations for updated alluvial resistance. Hence, MIKE 11a applies alternative formulations for these formulas as presented in Table 9.1

Table 9.1    Methods applied for Updating bed shear stress in sediment transport formulas available

| Transport formula | Bed Shear Method |
|---|---|
| Ackers-White | Ackers White formulation |
| Engelund-Hansen | Engelund-Hansen method |
| Smart & Jaeggi | Engelund-Hansen method |
| Engelund-Fredsoe | Engelund-Fredsoe method |
| Van Rijn | No method implemented |
| Meyer, Peter & Mueller | Engelund-Hansen method |
| Sato, Kikkawa & Ashida | Engelund-Hansen method |
| Ashida & Michiue | Engelund-Hansen method |
| Lane-Kalinski | Engelund-Hansen method |

Please note that for most of the methods there are no specific formulation for a bed resistance update method and the Engelund-Hansen method has been adopted in these formulations.

However, It is IMPORTANT to pay extremely much attention to the results obtained if the update bed resistance is activated for ANY of the formula which does not have a specific update method available. In most cases it would probably NOT be recommendable to activate the bed resistance update for these methods

### 9.2.2 Special features for specific transport models

#### Engelund-Fredsoe model

When selecting the Engelund-Fredsoe transport model, dune height and dune length are computed - if calculation of Bed Shear Stress is included. Therefore, an additional property page; 'Initial Dune Dimensions' is made visible in the ST Editor when either the bed load or suspended load transport model is chosen as Engelund and Fredsoe (see Section 9.8).

#### Smart-Jaeggi model

When selecting the Smart-Jaeggi transport model, the model parameters must be edited as for all other transport models. Additionally, user-defined coefficients and exponents used in the Smart-Jaeggi formulation can be specified. Therefore, when selecting the transport model for Total Load as 'Smart and Jaeggi' values for coefficients and exponents can be edited in a separate dialog.

The Smart - Jaeggi Factors dialog is activated by pressing the [More ...] button, which can be activated as soon as the transport model selected is 'Smart and Jaeggi'. as shown in Figure 9.4.



Figure 9.4    Additional dialog for defining Smart and Jaeggi model factors

Coefficients and exponents are essential for the Smart and Jaeggi transport model and a simulation should therefore not be performed until this dialog has been edited.

### 9.2.3 Bottom level update methods

A choice of method for updating bottom level exists. The default method is an assumption that the whole cross section is moved undistorted up in the case of deposition and down in the case of erosion.

If required, an alternative method for updating bottom levels can be defined through the MIKE11.ini file (ST_Parameters section; 'BOTTOM_LEVEL_UP-DATE_METHODS' parameter). The value of this parameter determine which approach is selected for cross section updating through the entire sediment transport simulation.

Update methods available are:

- Method no 1.

  Deposition in horizontal layers from the bottom. Erosion proportional with depth below bank level

- Method no 2

  Deposition and erosion uniformly distributed below the water surface. No deposition and erosion above.

- Method no 3

  Deposition and erosion proportional with depth below water surface. No deposition and erosion above.

- Method no 4

  Deposition and erosion uniformly distributed over the whole cross section (i.e. below the bank level).

- Method no 5

  Deposition and erosion proportional with depth below bank level

An illustration of the different methods available is presented in. More details on the calculation of bottom levels is given in the NST Reference Manual.
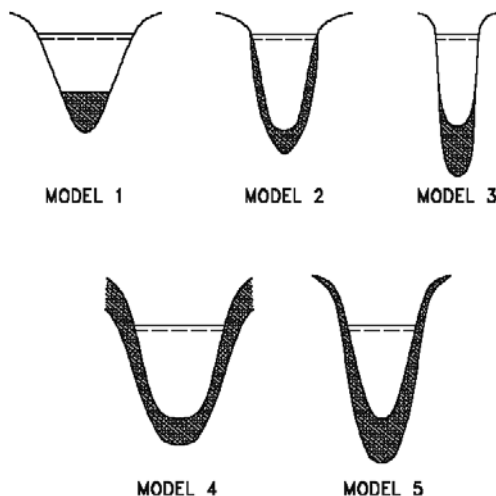
Figure 9.5    Options available for updating of bed levels and cross sections parameters

Please note that it is not unfortunately not possible to extract and present the updated cross sections shapes due to morphological updating. from any result-file. Analysis of impacts on cross section parameters (Area, Width, Radius) can be made if these parameters are saved through the HD additional output feature. If saved to the HD Additional Result-file then the Area, Width and Radius will all contain the updated parameter values as calculated during the sediment transport calculation.

## 9.3    Calibration Factors

The factors 'Factor 1' and 'Factor 2' can be applied to the calculated transport rates as correction factors. Both are simple multiplication factors used to either up- or downscale the calculated sediment transport.

The calibration factors are applied as follows:

- Factor 1: Multiplication factor for Total Load and Suspended Load.
- Factor 2: Multiplication factor for Bed Load.

Hence, If the sediment transport is calculated as total load (e.g. Engelund-Hansen, Ackers-White and Smart-Jaeggi models) 'Factor 1' is used as the correction factor and 'Factor 2' is not applied at all, whereas for other models distinguishing between bed load and suspended load both factors are applied.

Calibration factors can be specified globally and locally as shown in Figure 9.6, where 'Factor 1' and 'Factor 2' are globally defined as 1.0, but varies linearly with values different from the global in the river reach 'RIVER1' chainage 1000 to 4000 m.
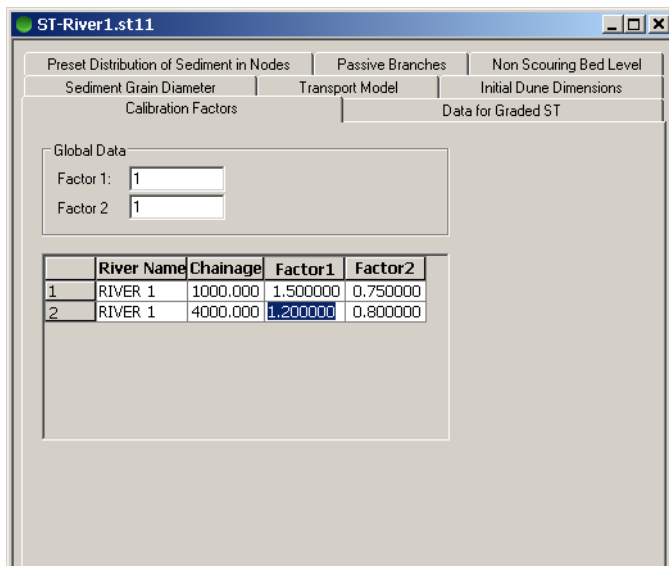


Figure 9.6     Calibration factors dialog

## 9.4    Data for Graded ST

The required input data for the simulation of graded sediment transport and sediment sorting are specified on this property page.

The bed material is represented by two layers, an active layer in which sediment transport occurs overlying an in-active, passive layer. Each layer is divided into an equal number of fractions (or classes) specified by the user. A mean grain size for each fraction and the percentage distribution for both the active and the passive layers must be specified. The fraction mean grain sizes are global but the initial percentage size distributions may be specified globally or locally. The sum of the initial percentage distributions for both the active and the passive layers must equal 100%.

It is possible to specify a lower limit for the active layer depth ('Min. depth active layer') and an initial depth for the passive layer.

The effects of shielding can also be included by setting a check mark in the 'Shielding of particles' checkbox.

The percentage contribution and transport rate of each fraction can be stored in the ST result file by activating the a check boxes; 'Save fraction values' and

'Save sed. transport each fraction'. If the result file is to be used as a hot start file, these values must be saved.

Global and local values can be specified.

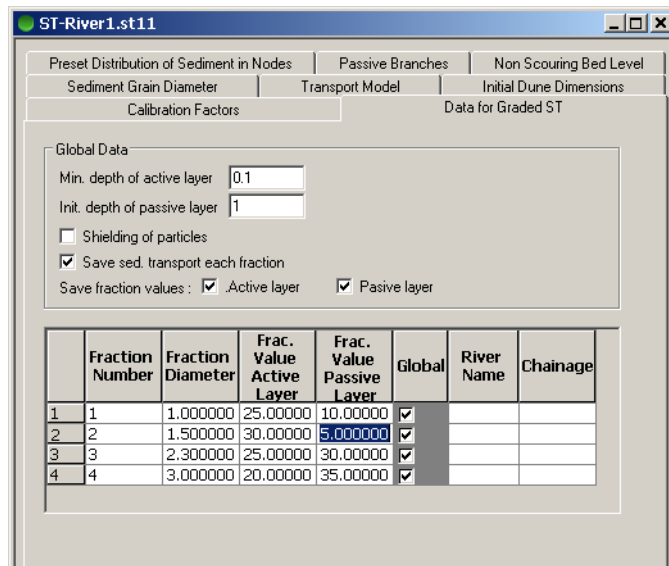An example of defining 4 fractions (global defined fractions only) is shown in Figure 9.7.



Figure 9.7      Example of specifying Graded ST data (4 fractions)

**Please note** that Graded Sediment Transport option is **not** available for the Meyer, Peter & Mueller sediment transport formula.

## 9.5    Preset Distribution of Sediment in Nodes

The property page enables the addition and editing of a preset distribution of sediment in calculation nodes.

The default sediment distribution in a node is carried out as the ratio of flow discharges where outflow from the node in a specific branch is related to the

total inflow of the considered node. The default formulation for distribution of the sediment into branch 'm' reads:

$$QSt_m^{(n+1)} = \sum_{up} QSt^n \cdot \frac{Q_m}{\sum_{down} Q_i}$$

(9.1)

Where:

$QSt_m^{(n+1)}$    Sediment transport at time step (n+1) for outflow branch '*m*'

$\sum_{up} QSt^n$    Acc.sediment transport in upstream branch-ends (= outflow from node

$Q_m$    Discharge from node into branch '*m*' (time step n+1)

$\sum_{down} Q_i$    Acc. inflow to node (downstream branch-ends) at time step n+1.

An alternative distribution can be specified on this property page by providing the distribution coefficients (A) and exponents (B) through the following relationship:

$$QSt_m^{(n+1)} = \sum_{up} QSt^n \cdot \frac{A_m \cdot Q_m^{B_m}}{\sum_{down} A_i \cdot Q_i^{B_i}}$$

(9.2)

Where

Index '*m*' refers to branch '*m*' and index '*i*' refers to inflow branches to the specific node.

The coefficients and exponents are given for each branch, specified by its upstream and downstream chainage as linked to the node as illustrated in the figure below, where 'River 1' discharges into a node (defined by the 'Downstream' type) and 'River 2' and 'River 3' respectively receives water from the node (= outflow branches in the node. Defined by the 'Upstream' type).

Figure 9.8     Preset Sediment Distribution in calculation nodes

## 9.6     Passive Branches

It is possible to exclude Sediment transport calculations in selected branches. Branches in which sediment transport should not be calculated are specified in this property page by the River Name and the upstream and downstream chainages as shown in Figure 9.9, where two branches are defined as being passive. A branch will only be defined as passive in case both the upstream and downstream chainages are recognised internally in MIKE 11 as being the start and end point of a branch.

Sediment can be transported into a passive branch, but the sediment eventually transported into a passive branch is 'trapped', as no sediment can be transported inside and out of a passive branch.
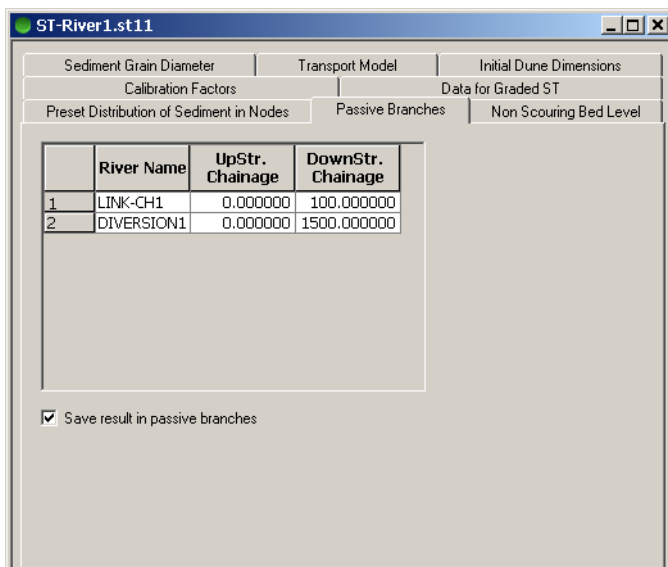
Figure 9.9    Passive branches property page

The 'Save result in Passive Branches' check-box enables saving of sediment transport results to include also calculation points in branches which have been defined as 'Passive'. Enable this check-box to include only all calculation points in which MIKE 11 actually calculates the sediment transport in order not to start analysing further on results in points where no sediment transport calculation has actually taken place. However, do note that in case the result-files you are producing shall eventually be used as hotstart-file for additional simulations, then it is actually required that the result-files contains results from all calculation points in the model, and therefore, in this case, this check-box must be kept de-active.

## 9.7    Non-Scouring Bed Level

The Non Scouring Bed Level page offers the possibility of defining two parameters; thickness of active layer and a non scouring bed level.

The Thickness of active layer is used in the Graded sediment transport calculations. Default formulations in MIKE 11 define the thickness of active layer as half the dune height, but this value can also be user-defined through this property page. The value must be given as a depth. That is; a height above bottom of river bed.

**Please note:** Setting the Thickness of active layer to a value of -99 switches back the formulation to the previous default formulation in MIKE 11 (thickness equal half the dune height).

The Non scouring bed level item gives a possibility for the user to define levels (global and/or locally) where a non-erodible surface is present. If, during a

morphological simulation, bed erosion occurs and the bottom of the bed reaches the defined Non scouring bed level, no further bed erosion will take place.

**Important:** the non scouring bed level must be defined as a actual level - and not a height!



Figure 9.10      Non scouring Bed Level property page

Figure 9.10 shown an example where the global values of Thickness of active layer is defined to 0.1 m and Non scouring bed level is set to -1.5 m. These values are used in the entire river setup except for specific reaches in 'RIVER1' and 'RIVER2' where local values are specified. Linear interpolation will be used to define Layer thickness and Non scouring level at calculation points in between the local stations defined in the dialog.ue must be given as a depth. That is; a height above bottom of river bed.

**Pleas note** that in 'RIVER2' from chainage 0 to 4000 a value of -99 has been defined which means, that the previous default formulation for defining thickness of active layer in MIKE 11 will be activated for this river reach.

## 9.8      Initial Dune Dimensions (Engelund Fredsoe Model)

When selecting the Engelund-Fredsoe transport model the dune height and length are computed when calculation of bottom shear stress is included. The dune dimensions can be specified as applicable globally and locally. If dune dimensions are specified for local application, these values will be used instead of any globally specified values.

Figure 9.11 shows an example where the global dune height has been set to 0.25 m, and the global dune length has been set to 12.50 m. These values will be used in the entire river network, except in the reach 'RIVER1', between chainage 5.000 km and 10.000 km, where the dune height varies linearly between 0.25 m and 0.40 m.
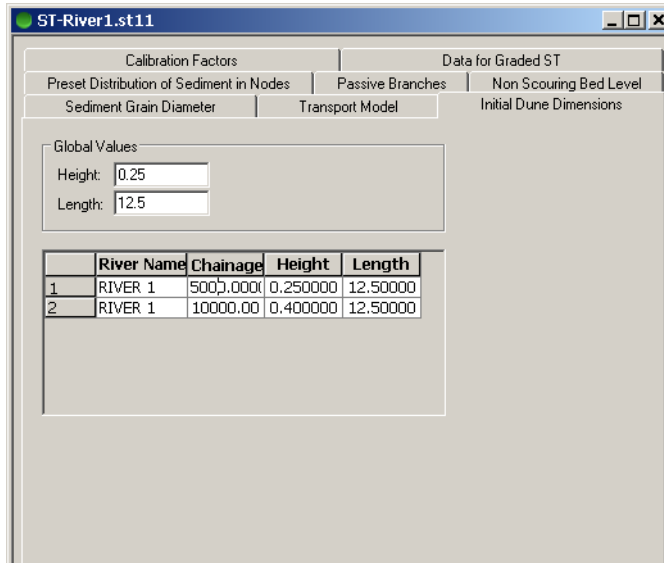


Figure 9.11    Example of an implementation of local initial dune dimensions

If no dune dimensions are given, or the dune height and length equals zero, then the dune height will be calculated as the water depth divided by 6 with a dune length of 15 times the water depth.

## 9.9    ST Simulations; Additional Features

This section includes a number of additional features for the ST simulation, which can be activated through other options than the Parameter Editor.

### 9.9.1    Transport layer depth options through ASCII file; TrsDepth.Inp

This option is valid for the Graded Sediment Transport module only.

The default MIKE 11 formulation for defining the transport layer depth (active layer) is the water depth divided by (D/6). However, this formulation only gives realistic results in case of slowly varying flow and similar water depths in the entire modelled river reaches.

An option for selecting alternative formulations for the transport layer depth is available through the use of a ASCII text-file named; 'TrsDepth.inp'. This text file must be placed in the simulation folder where the sim11-file for the project

is located. If MIKE 11 locates an ASCII file with this name a message-box will appear prompting you whether you wants to read values from this text-file or not.

The format of the TrsDepth.Inp file is presented below:

```
Transport Depth Type  (1 or 2)   //line no. 1    Descriptor line
         1                       //line no. 2    Transport depth type
Transport Depth Factor   (TDF)   //line no. 3    Descriptor line
       0.2                       //line no. 4    Transport depth factor
```

Figure 9.12    File format of ASCII file 'TrsDepth.Inp' used to apply alternative methods for initial transport depth in the Graded Sediment transport module.

Two parameters are defined through the TrsDepth,inp file;

- Line 2: Transport Depth Type (options available: '1' or '2')
- Line 4: Transport Depth Factor (TDF)

The alternative method applied for the transport layer depth depends on the defined Transport Depth Type.

The following methods are available:

1. Transport Depth Type = 1
   Transport depth defined as a constant throughout the entire model area.
   That is: Transport Depth = TDF

2. Transport Depth Type = 2
   Transport depth defined as constant ratio of water depth.
   That is: Transport Depth = Radius * TDF

## 9.9.2 Shielding option defined through ASCII file; Shield.inp

This option is valid for the Graded Sediment Transport module only.

The standard Egiazaroff formulation for description of sheltering is over-ruled if the ASCII file named 'Shield.Inp' is located in the simulation folder (same folder as where the sim11-file is located).

The Shield.Inp file consists of two lines only as presented below:

```
Shielding type   Shielding Factor  //line no. 1 Descriptor line
       2                0.80       //line no. 2 Shielding Type and Shielding factor
```

Figure 9.13    File format of ASCII file 'Shield.Inp' for defining alternative options for shielding (hiding/exposure) of grains

1. Line 1 is a header line (i.e. could be any text-string)

2. Line 2 contains two numbers; Shielding Type (1 or 2) and a Shielding factor

Following alternative formulations are used depending on the Shielding Type definition:

Shield Type = 1: $\tau_{c,shield} = \tau_c \cdot \left(\dfrac{\ln(sf)}{\ln(sf \cdot d_i / d_m)}\right)^2$

Shield Type = 2: $\tau_{c,shield} = \tau_c \cdot \left(\dfrac{d_m}{d_i}\right)^{sf}$

where:

$\tau_{c,shield}$ is the critical shear stress adjusted with sheltering effects

'*sf*' is the Shielding Factor (from the Shield.Inp file)

'*di*' is the grain size diameter of fraction i

'*dm*' is the mean diameter

### 9.9.3 ST simulation features activated through the MIKE11.ini file

Located in the 'bin' directory of the MIKE Zero installation you will find a MIKE11.ini file which is used to activate and control specific behaviour and performance of the calculation engine.

The MIKE11.ini file contains also a section specifically for ST-simulations (section named '[ST_SIMULATION]') and a few of the parameters possible to alter in the ini-file are mentioned below.

Please note that it is recommended to copy the MIKE11.ini file from the bin-folder to your project simulation folder (must be located where you have your sim11-file). The simulation engine searches for the MIKE11.ini file locally first, and secondly for the global file in the bin-folder. By copying the file to the local folder, you ensure, that the changes you eventually make in the ini-file only concerns the present project, which is often required as changes in the ini-file are possibly required and beneficial for one project, whereas different values or parameters must be altered in other projects.

Selected variables from the ini-file mentioned here includes:

### ST-Variables no 1:
BOTTOM_LEVEL_UPDATE_METHODS = 4

Defines which updating method is used for calculating morphological changes in cross sections (see section 9.2.3)

### ST-variable 13-17:
COUPLED_ST_AD=OFF
CORRECT_ST_AD=OFF
NO_AD_ST_FRACTIONS=1
AD_COMP_NO=1
ST_FRAC_NO=1

Variables affecting coupled Non-cohesive and Cohesive sediments through the graded sediments feature.

# FLOOD FORECASTING EDITOR

# 10 Flood Forecasting Editor

The MIKE 11 Flood Forecasting Module (MIKE 11 FF) has been designed to perform the calculations required to predict the variation in water levels and discharges in river systems as a result of catchment rainfall and runoff and inflow / outflow through the model boundaries.

The MIKE 11 FF module includes:

- Definition of basic FF parameters

- Definition of boundary conditions in the forecast period (Forecasted boundary conditions)

- Definition of Forecast stations

- An updating routine to improve forecast accuracy. The measured and simulated water levels and discharges are compared and analysed in the hindcast period and the simulations corrected to minimise the discrepancy between the observations and model simulations.

## 10.1 Basic Definitions

### 10.1.1 Simulation Period and Time of Forecast

The Time of Forecast (ToF) is defined in relation to the Hindcast and the Forecast Period in Figure 10.1.The Hindcast Period defines the simulation period up to ToF and is specified in the simulation file or calculated by the system; see Chapter 10.1.2, Simulation Mode. The length of the Forecast Period is always specified in the Forecast Menu, see section 10.2.1
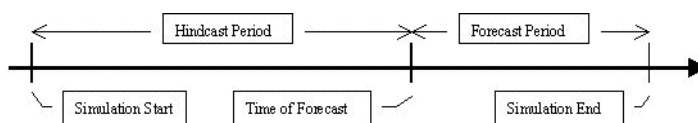


Figure 10.1    Definition of ToF

### 10.1.2 Simulation mode

#### Real-time mode

*Real time mode* defines a condition where MIKE 11 FF is used to execute simulations applying real-time hydrometeorological data as boundary conditions. The common time span of the boundary data defines the hindcast period, see Figure 10.2. As real-time hydrological and meteorological data are often captured and supplied by a telemetry network, pre-processing of

these data is usually required for a specific (user defined) Hindcast Period and Time of Forecast.
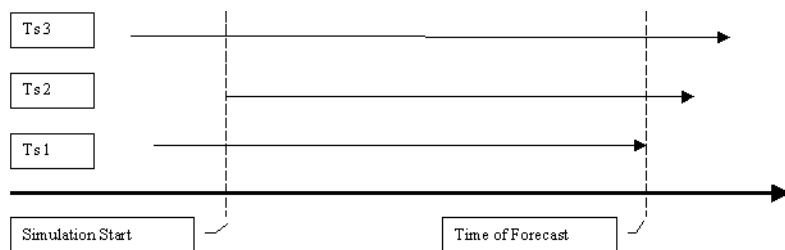


Figure 10.2    Definition of Hindcast Period and ToF

## Historical mode

While real-time telemetry data form the boundary conditions in an operational forecasting mode, historical hydrometeorological data are applied as boundary conditions in the calibration and validation phase of forecast modelling.

When MIKE 11 FF runs in historical mode, the hindcast period is defined via the Simulation Menu in the sim11 editor. The Hindcast Period is defined from Simulation Start to Simulation end i.e. Simulation end is interpreted as ToF.

In the example shown below in Figure 10.3 the hindcast period starts on the 4 January 1999 at 12:00 and last up to 7 January 1999 at 12:00.
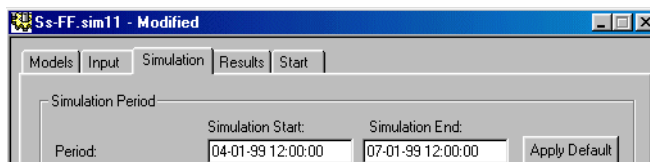


Figure 10.3    Definition of Hindcast period in historical mode

The forecast period is defined in the Forecast Menu.

## 10.2   Forecast

The main forecast parameters are specified in the Forecast Menu, Figure 10.4.

Figure 10.4    Basic Forecast Definitions

## 10.2.1  Forecast length

The Forecast length is equal to the Forecast Period (Figure 10.4). The length of the Forecast Period can be specified in **hours** or in **days**

## 10.2.2  Include updating

Tick on the appropriate check box to include the updating routine. Update points and parameters are specified on the **Update Specification** menu; see section 10.4.

## 10.2.3  Accuracy

The Boundary Conditions estimated after the Time of Forecast are obviously uncertain. The effect of a specified uncertainty level can be included in the simulations.

Tick on the 'Include uncertainty level' check box to include.

Specify either global and/or local values for the deviation. Global values are applied to all catchments or HD boundary conditions, except those which are listed in the 'Local Values' fields.

Estimated boundary conditions with Upper and Lower levels are stored in the 'Boundary Estimates' directory as described in Section 10.3.4.

## 10.2.4 Alternative modes

### Multiple forecast with historical data

To execute simulations in Historical Mode tick on the Multiple forecast check box, see Figure 10.4 or below. Additional information about simulating in Historical Mode can be found in Section 10.1.2.

In Historical Mode it is possible to execute consecutive simulations shifting the Start time and ToF of each simulation. Simulation start and ToF applied in the first simulation are defined on the simulation menu in the sim11 editor.



Figure 10.5    Selection of Historical Mode

**No of FC** defines the number of consecutive simulations to be executed and **Step** defines the interval at which multiple forecasts are made. The Time of Forecast (ToF) is moved forward Step (hours) between forecasts (see Figure 10.6).
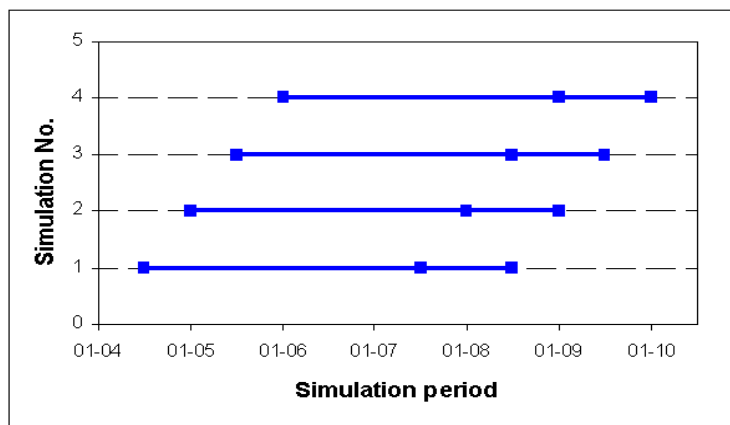


Figure 10.6    Multiple simulations in Historical Mode

Simulation no. 1 is executed according to Simulation Start and Simulation End found in the Simulation Menu in the Sim11 editor. As described in Section 10.1.2, Historical Mode, Simulation End is interpreted as ToF In each of the following simulations Simulation Start and ToF are shifted 12 hours.

### Seasonal forecasting

Not yet implemented

## 10.2.5 Location of forecast stations

Forecast points are specified as shown in Figure 10.7 below



Figure 10.7    Location of Forecast Points

Simulated water level or discharge at a forecast point is extracted from the MIKE 11 HD resultfile and stored together with the "Danger level" as individual time series files (dfs0 format), one file for each forecast point (location). These files are named according to the **Name** field in the Locations menu and are stored in a directory structure as illustrated in Figure 10.8.
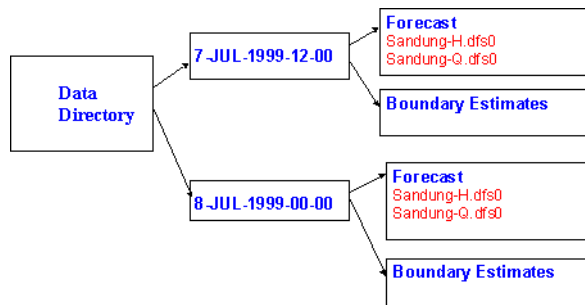


Figure 10.8    Forecast data directory structure

MIKE 11 FF generates a data sub-directory, named according to the ToF, e.g "8-jul-1999-12-00" in the example shown in Figure 10.8. The individual forecast time series are stored in a sub-directory named "Forecast"

### Save all forecasts

Tick off the "Save all forecasts" check box to avoid generating the individual forecast time series according to the specifications from the Location menu.

### Storage time step

The storage frequency of forecast results can be more or less frequent than the general MIKE 11 HD storage frequency specified in the Results menu in the sim11 editor.

## 10.3 Boundary Estimates

To simulate beyond the ToF requires boundary conditions for the forecast period i.e rainfall, evaporation and possibly temperature for each catchment in the RR simulation and water level or discharge for each of the open boundaries in the HD model.

Boundary conditions applied during the forecast period are in this manual described as Estimated boundary conditions.

Estimated boundaries can to some extent be defined by the FF module using boundary conditions from the hindcast period. Details about these options can be found in Section 10.3.3.

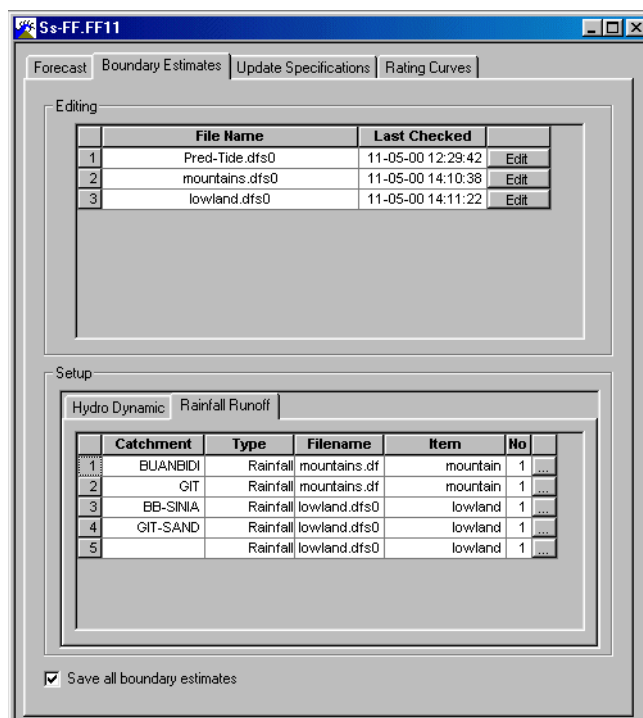Figure 10.9 shows the Boundary Estimates menu.



Figure 10.9    Boundary Estimates

### 10.3.1 Setup

Specify catchment name (RR) or River name and Chainage (HD) to locate the actual boundary

## Type

Specify the appropriate data type:

RR: Rainfall, Evaporation, Temperature, Irrigation and Abstraction

HD: Water level, discharge or gate level.

## Filename

Press the "..." button to select the appropriate time series file.

## Filetype

The Axis type for the dfs0 files applied in the forecast period can be either 'Calendar axis' or 'Relative axis'. If a dfs0 file is based on a 'Relative time axis' the start time of that particular time series will be interpreted as ToF.

### 10.3.2 Editing

All files included in the setup menu will be listed in the 'Editing' menu as seen in Figure 10.9 above. Pressing the "Edit" button will start the MIKE Zero time series editor with the actual time series loaded. In this manner it is possible to view and edit the boundary estimate time series.

### 10.3.3 Boundary data manipulation

To minimize the time spent entering and editing data related to the 'Estimated boundaries' several alternative boundary estimation methods have been implemented in the FF module. The different boundary estimation methods are summarised in Table 10.1 and their effect illustrated in Figure 10.10 through Figure 10.14.

## Omit a boundary condition.

A boundary condition time series (i.e. rainfall / evaporation or discharge / water level time series) is simply omitted in the 'Setup' list.

Table 10.1

| Case | Estimation method | Illustration |
|------|-------------------|--------------|
| Omit a boundary condition in the 'Setup' list | If data from the hindcast time series cover the forecast period, these are applied. Otherwise the hindcast value at ToF is applied. | Figure 10.10 |
| The time series covers at least the whole forecast period. | No manipulation is required. | Figure 10.11 |
| Estimated time series starts at ToF but does not cover the whole forecast period | Time series is extrapolated applying the last found value | Figure 10.12 |
| Estimated time series starts after ToF. | Time series is interpolated using hindcast data at ToF and the first entered estimated value | Figure 10.13 |
| The time series cover the whole forecast period, but there is a discontinuity at ToF | During the first 10 HD time steps the boundary data are interpolated between hindcast data at ToF and estimated data | Figure 10.14 |
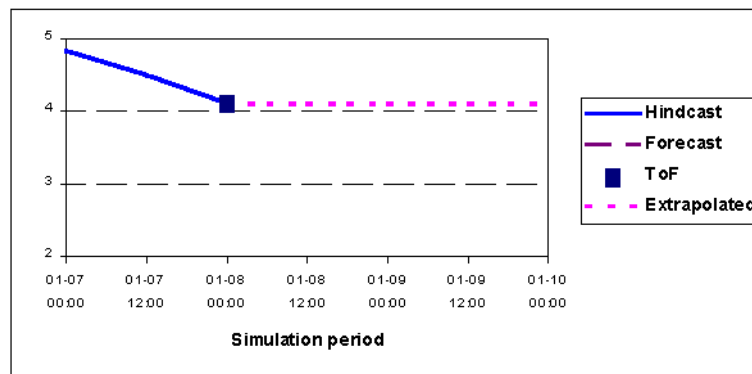


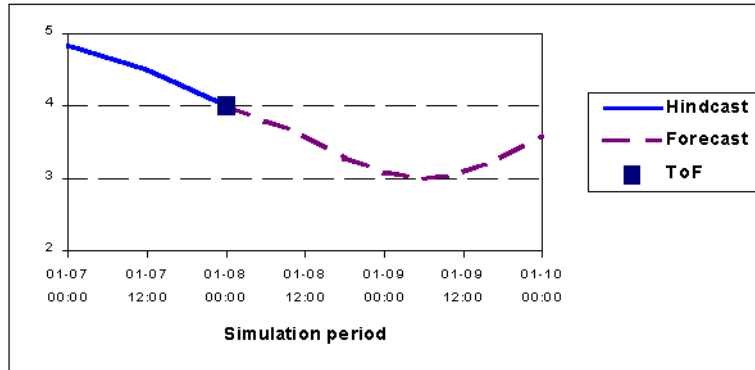Figure 10.10   Extrapolation from value at ToF

Figure 10.11   Estimated boundary conditions as specified
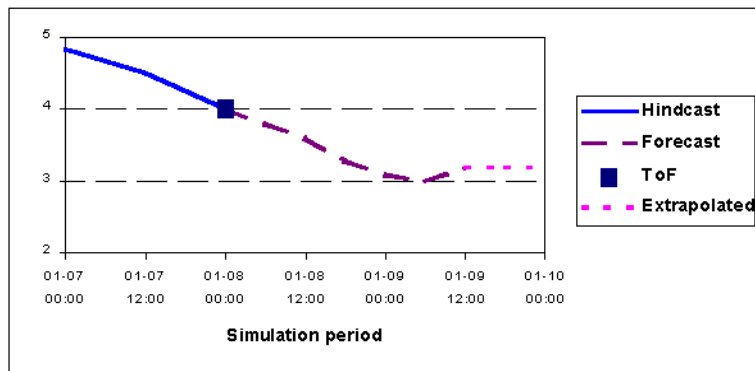


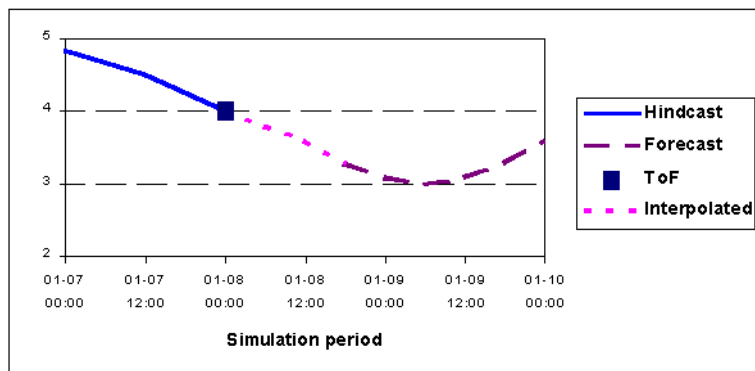Figure 10.12   Extrapolation of Estimated boundary conditions



Figure 10.13   Interpolation of Estimated boundary conditions
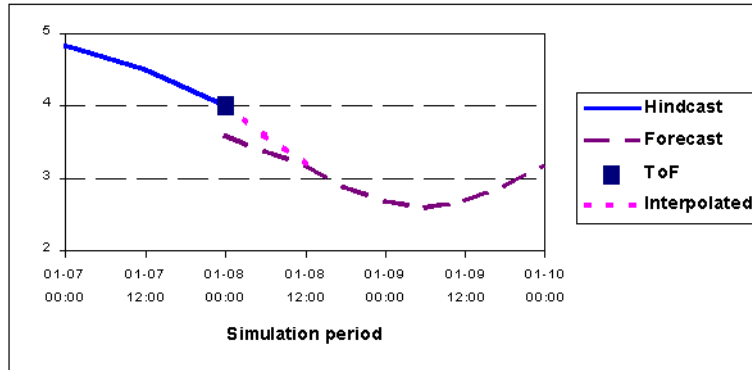
Figure 10.14  Discontinuity at ToF

### 10.3.4  Storing of Estimated boundaries

Estimated boundaries are stored for each forecast in a similar manner to the simulated levels or discharges from the forecast stations, see Section 10.2.5 and Figure 10.15 below.
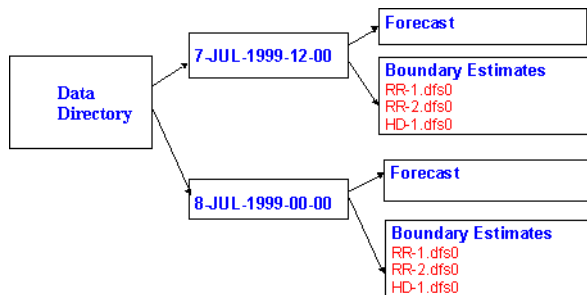


Figure 10.15  Estimated boundary directory structure

## 10.4  Update specifications

The purpose of updating is to evaluate and eliminate deviations between observed and simulated discharges/water levels in the Hindcast Period to improve the accuracy of the model results in the Forecast Period. Phase and amplitude errors are identified by the updating routine and corrections in the hindcast and the forecast period are subsequently applied.

Figure 10.16 shows the Update Specification menu.

Figure 10.16   Update Specification

## 10.4.1  Comparison

### Station

The location of the update point is defined via its River name and Chainage. If the specified chainage does not correspond to the computational network it is shifted to the nearest *h*- or *Q*-point by the FF module and a warning message is issued.

### Data type

The Data type can be specified as water level or discharge. In general, water level data should be specified at all sites where level forecasts are to be issued, and discharge at reservoir inflow points.

Discharge updating is generally preferable and should be selected at all forecasting locations where reliable discharge data are available.

### Measured time series

The updating routine compares measured and simulated data. The time series of measured water level or discharge data must be specified.

### Method

#### Iterations
See No. of Iterations

#### Implicit solution
The specified time series are applied as internal boundary conditions in the model. In the Continuity Equation $h^{n+1}$ is substituted by the observed water level and the lateral inflow $q^{n+\frac{1}{2}}$ is calculated and applied as the updating discharge.

## No. of iterations

If a river branch includes a number of update points the specified No. of iterations should be equal to or larger than this number. For large rivers with few update points it may increase the update efficiency to use an even larger number of iterations. Different numbers of iterations should be tested before operational forecasting is initiated. A larger number will increase the accuracy but also increase the required calculation time.

## Frequency

Frequency of updating, i.e. the number of MIKE 11 HD time steps between data observations in the time series used for updating.

## 10.4.2 Correction

The updating routine will calculate a correction discharge to be routed into the river system along the correction branch. The correction branch is specified by **River name**, **First chainage** and **Last chainage**.

If the specified chainages does not correspond to the computational grid they are modified by the FF module and a warning message is issued

### 10.4.3 Parameters

Table 10.2

| Parameter | Main effect | Typical value |
|---|---|---|
| Max phase error | Higher phase errors are automatically reduced to this value | Equal to AP |
| Analyse Period (AP) | Determine the period where observed and simulated data are analysed | Found by calibration |
| Time constant in AP | If less than AP, recent deviations may be given more weight | Equal to AP |
| Time constant in forecast period | Corrections at ToF are gradually decreased in the forecast period by a first order decay with this time constant. | Found by calibration |
| Adjust factor | Increasing/decreasing the calculated updating discharge | 1.0 |
| Alpha | An increase in Alpha will cause deviations to be interpreted more as amplitude errors | Found by calibration |
| Peak value | Highest expected discharge after applying the correction discharge | From observed discharge hydrographs |

## 10.5 Rating curves

Not implemented.

# DATA ASSIMILATION EDITOR

# 11 Data Assimilation editor

The Data Assimilation editor is used for specifying the parameters needed when carrying out

- Uncertainty estimation
- Model updating
- Forecasting

The uncertainty assessment and updating methods are applicable to hydro-dynamic and/or advection dispersion simulations. Uncertainty assessment and updating for NAM rainfall runoff will become available in a later release. Thus any reference to catchment modelling in the present interface should be neglected. One exception being that the uncertainty may be applied to output from catchments.

## Uncertainty assessment

The uncertainty assessment is a powerful tool for evaluating the effect of uncertainties on the boundary conditions in a river network. To carry out an uncertainty assessment a minimum of input parameters is required from the user.

## Model updating

If reliable measurements are available within the model domain, these may be utilized to improve the model results of MIKE 11 using updating. Two different updating methods are available:

- A Kalman filter procedure
- A combined weighting function and error correction procedure

## Forecasting

With the appropriate data the updating routines of MIKE 11 can improve model results prior to a time of forecast. After the time of forecast the model is corrected using forecasts of errors identified prior to the time of forecast. The type of error correction to be applied in the forecast period depends on the option chosen for model updating. If the Kalman filter is applied, estimated errors in the boundary conditions at the time of forecast are phased out according to an exponential decay. If the combined weighting function and error correction procedure is applied, errors at measurement points are forecasted. Thus, the updating algorithm has an effect also after the time of forecast. Furthermore, by using the Kalman filter procedure a stochastic or ensemble forecast may be generated.

Depending on the method applied different menus within the editor are required.

# 11.1   General



Figure 11.1   The general data assimilation model parameter tab. Note that depend-
ing on the choice made in the Module selection section one or more of
the other boxes will be made inactive

•

## 11.1.1   Module selection

The simulation mode is selected here. Either the model is run using measure-
ment update or uncertainty assessment.

The first option requires the user to choose which updating technique that
should be applied. The choices are:

• Kalman filter: The model uses the ensemble Kalman filter based on
Monte Carlo simulation techniques to estimate the updates to be applied.

• Weighting function: The model uses user-defined weighting functions to
estimate the updates to be applied.

If the uncertainty assessment option is chosen, the model uses Monte Carlo
simulation to propagate uncertainties in the boundary conditions into uncer-
tainties in the simulated model output.

## 11.1.2 Basic parameters

Depending on the simulation mode selected in the Module selection box the user is required to supply additional data.

### Ensemble size

If applying either the uncertainty assessment simulation method or the Kalman filter updating method, the user is required to supply the ensemble size. The ensemble size is defined as the number of simultaneous runs that are to be carried out to evaluate the statistical properties needed for the uncertainty assessment output and also for determining the updating parameters applied in the Kalman filter.

The quality of the statistical estimates are strongly dependent on the ensemble size. The larger the ensemble size the higher the confidence in the results. On the other hand, the ensemble size has a linear effect on the run time, that is, when the ensemble size is doubled the run-time is also doubled.

The above considerations should be taken into account when choosing the ensemble size. Recommended values are 50-200. If only reliable estimates of standard deviations are of importance, a smaller value may be chosen. When producing confidence intervals an estimate of the full uncertainty distribution is needed, thus an increase in the ensemble size is recommended for such cases.

When applying the weighting function method no stochastic element is applied in the simulation and thus the ensemble size is not required.
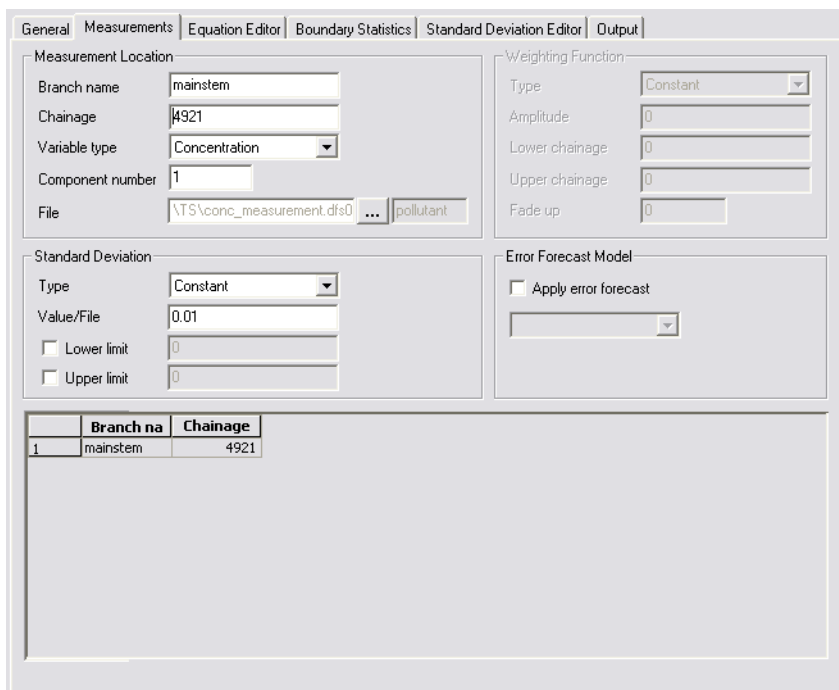
### First filtering time step

This option is only applicable when using updating. The user needs to specify at which time step the updating is to be initiated. The default value is 0 corresponding to the first time step. The use of this variable is to ensure that the stochastic process is sufficiently evolved to give good estimates of the uncertainty before the first update, thereby avoiding model stability problems.

## 11.1.3 Forecast

The forecasting parameters are only applicable to the updating methods. The time of forecast should be within the simulation period specified in the Simulation editor.

When using the Kalman filter, the user may select either a deterministic or a stochastic forecast. The first uses the ensemble mean of the state variables including corrections at the time of forecast for a deterministic run, whereas the latter continues the stochastic run omitting the updating step.

## 11.2    Measurements



Figure 11.2    The Measurements tab. Depending on the choice made in the general tab some fields are made inactive

The measurements page is used for specifying the location and parameters of the measurements that are to be used for the updating. Note that the page is only used when applying the updating option.

To add a new measurement place the cursor in the last row in the overview and press the right tab until an additional row appears.

### 11.2.1   Measurement location

The location of the measurement is specified here. The location does not have to coincide with a grid point location since MIKE 11 uses linear interpolation for determining the simulation value at the appropriate measurement location.

#### Branch Name
The branch name where the measurement is located is specified.

#### Chainage
The chainage of the measurement is specified.

### Variable Type

Presently three types of measurements may be applied:

- Water level
- Discharge
- Concentration

If a concentration measurement is selected, the component number is required. The component number is given by the order found in the Components page of the AD editor.

### File

The location of the measurement data is required. The dfs0 file may be selected by browsing.

## 11.2.2 Standard deviation

When the Kalman filter updating method is applied, the user must specify the standard deviation for the measurement. The standard deviation may be of three different types:

- Constant. The user specifies a constant value.
- Relative. The standard deviation is taken as a relative value of the measurement.
- Time dependent. The standard deviation may vary with time with the variation defined in a dfs0 file.

When applying a relative or time dependent standard deviation, the user may apply bounds on the standard deviation.

Note that for the percentage option the percentage is taken based on the absolute value for discharge and concentration. For water level the percentage is interpreted as being with respect to the water depth.

For the weighting function method the standard deviation box is not applicable.

## 11.2.3 Weighting function

When the weighting function update procedure is applied a weighting function must be specified for each measurement. This function defines how errors at the measurement location are distributed to neighbouring points in the river network. The procedure updates the parts of the model state that correspond to the measurement variable (e.g. if water levels are measured the water level grid points are updated), whereas the other parts of the model state are implicitly updated according to the numerical scheme in the next time step.

### Type

Three different types of weighting functions are available:

- Constant. In this case the error correction at the measurement location is distributed evenly over the grid points between Lower and Upper chainage.

- Triangular. In this case the error correction is linearly decreasing from the measurement location to 0 at the Lower and Upper chainage.

- Mixed exponential. In this case the error correction is decreased according to an exponential function from the measurement location to the Lower and Upper chainage to about 0.01 times the correction at the measurement location.

### Amplitude

The Amplitude specifies the fraction of the observed error at the measurement location that should be applied as error correction at that point. The Amplitude should reflect the confidence of the observation as compared to the model forecast. That is, if the amplitude equals one, the measurement is assumed to be perfect, whereas for smaller amplitudes less emphasis is put on the measurement as compared to the model forecast.

### Lower chainage

The Lower chainage specifies the lower point (smallest chainage value) in the branch where the measurement point is located for which the weighting function is applied.
Note: Lower chainage must be smaller than the Upper chainage (regardless of branch flow direction definition).

### Upper chainage

The Upper chainage specifies the upper point (highest chainage value) in the branch where the measurement point is located for which the weighting function is applied.
Note: Upper chainage must be larger than the Lower chainage (regardless of branch flow direction definition).

### Fade up

The user specifies the number of time steps used to fade up the corrections initially, starting from zero corrections at the first time step up to full corrections after the fade-up period. Employment of this function ensures that model instabilities are avoided because the desired error correction value is obtained gradually rather than abruptly.

## 11.2.4 Error forecast model

The weighting function update procedure may be combined with error forecasting at measurement points in the forecast period. In this case the Apply error forecast tick box is activated and the name of the error forecast model is

chosen from the pull-down menu. The error forecast model is defined in the Equation editor.

Up to the time of forecast the observed errors at the measurement locations are distributed to the neighbouring grid points according to the defined weighting functions. After time of forecast the defined error forecast models are used to forecast the errors at the measurement locations which are then distributed to the neighbouring points. Thus, by applying error correction the model is updated also in the forecast period. If error correction is not applied, updating is not performed in the forecast period, and the model forecast is simply a normal MIKE 11 run using the updated model state at the time of forecast as initial conditions.

The Error forecast model tick box is not applicable for the Kalman filter method.

## 11.3 Equation Editor



Figure 11.3    The Equation Editor tab

The Equation editor is used to define the error forecast models used in the weighting function update procedure. Error forecast models can be defined as general linear or non-linear functions with a one-step ahead prediction that depends on the previous errors at the measurement location, the model state

at any grid points in the river network, and the boundary conditions (or any other time series defined in a dfs0 file).

## 11.3.1  General

To add a new equation place the cursor in the last row in the Overview of Error Forecast Model Equations and press the right tab until an additional row appears.

### Name

Unique name of the defined equation. This name is shown in the pull-down menu of available equations when defining an error forecast model for the Weighting function method on the Measurements page.

### Equation

The equation editor is based on an equation parser that uses the general arithmetic operators:

- Addition (+)
- Subtraction (-)
- Multiplication (*)
- Division (/)

In addition, a number of mathematical functions are supported, see Table 11.1

Table 11.1  Mathematical functions used by the equation parser (X and Y are variable names)

| Syntax | Function |
|---|---|
| SQR(X) | Square function |
| SQRT(X) | Square root function |
| SIN(X) | Sine function. SIN returns the sine of the angle X in radians. |
| COS(X) | Cosine function. COS returns the cosine of the angle X in radians. |
| TAN(X) | Tangent function. TAN returns the cosine of the angle X in radians. |
| COTAN(X) | Cotangent function. COTAN returns the cosine of the angle X in radians. |
| ATAN(X) | ArcTangent function |
| EXP(X) | Exponential function |
| LN(X) | Natural logarithmic function |

Table 11.1 Mathematical functions used by the equation parser (X and Y are variable names)

| Syntax | Function |
|---|---|
| LOG(X) | 10 based logarithmic function |
| SINH(X) | Sinus Hyperbolic function |
| COSH(X) | Cosine Hyperbolic function |
| INTPOW(X,Y) | The INTPOW function raises X to an integer power Y, e.g. INTPOW(2, 3) = 8. Note that the result of INTPOW(2, 3.4) = 8 as well. |
| POW(X,Y) | The POW function raises X to any power Y |
| ABS(X) | Absolute value |
| SIGN(X) | SIGN(X) returns -1 if X<0; +1 if X>0, 0 if X=0. |
| TRUNC(X) | Discards the fractional part of a number, e.g. TRUNC(3.2) is 3. |
| MIN(X,Y) | Minimum of X and Y, e.g. MIN(2, 3) is 2. |
| MAX(X,Y) | Maximum of X and Y, e.g. Max(2,3) is 3. |
| E(L) | Error function with lag L (L<0), e.g. E(-1) is the error at the previous time step. |

As an example, suppose the error forecast model is to be expressed as a function of the variables X1, X2 and X3 as 5 times variable X1 minus the square of variable X2 plus 2 times the natural logarithm of X3, the Equation field should be written:

5*X1-SQR(X2)+2*LN(X3)

If one wants to apply a second order auto regressive model, the Equation field should be written:

A*E(-1)+B*E(-2)

where A and B are the auto regressive parameters.

### Estimation period

For parameters defined as values in the equation, automatic parameter estimation can be applied based on the record of observed errors. The period of the record to be used for the parameter estimation can be specified relative to the time of forecast.

This option allows parameters of the error forecast models to be updated continuously, and hence the error forecast models to adapt from one forecast to the next to the prevailing conditions at the time of forecast. For instance,

the error forecast models can be adapted to the structural differences in the model errors that are often seen for different flow regimes.

## 11.3.2 Parameter definition

To add a new parameter place the cursor in the last row in the Overview of Equation Parameters and press the right tab until an additional row appears.

### Name
Unique name of parameter specified in the equation.

### Type
Three different parameter types are available:

- Value. The parameter is assigned a numerical value.

- Time series. The parameter is assigned a time series defined in a dfs0 file. This may be a boundary condition in the HD or AD setup or a rainfall input.

- State variable. The parameter is assigned a state variable, i.e. a water level, discharge or concentration at a grid point in the river network.

### Estimated
For a parameter that is assigned a numerical value, automatic parameter estimation can be applied to estimate the value from the time series of errors. Otherwise the value should be specified by the user in the Value field.

### Value
Numerical value assigned to the parameter. If the Estimated tick box is active, this field is made inactive.

### Minimum
Allowable lower bound on the parameter when automatic parameter estimation is adopted.

### Maximum
Allowable upper bound on the parameter when automatic parameter estimation is adopted.

### Variable type
If the parameter is defined as a state variable, the variable type should be defined:

- Water level
- Discharge
- Concentration

### Component number

If a concentration variable is selected, the component number is required. The component number is given by the order found in the Components page of the AD editor.

### River name

The river name of the grid point of the state variable.

### Chainage

The chainage of the grid point of the state variable.

### File

If the parameter is defined as a time series, a dfs0 file item must be specified.

## 11.4    Boundary Statistics



Figure 11.4    The Boundary Statistics tab

This menu is only applicable to the Kalman filter updating method and the uncertainty assessment simulation modes. The structure of this page has been chosen to facilitate that the same uncertainty statistic can be applied to multiple boundaries. For instance, it may be that the user would like to apply a standard deviation of say 10 percent to all discharge boundary conditions in the set-up.

## 11.4.1 Details

The details for the boundaries where uncertainty is to be applied are specified here.

### Boundary Type

The boundary types that uncertainty may be applied to are:

- Water levels
- Discharges
- Lateral source point inflows
- Q/h-relations
- Output from catchments
- Concentrations
- Wind fields

**Note** that at present uncertainty cannot be attributed to distributed lateral discharge sources. Also note that the interface has been prepared for including updates on rainfall runoff models, though the present release does not include any rainfall runoff update capabilities.

Once the boundary type has been selected the locations where the uncertainty is to be applied is entered in the second of the upper tables. Use the tab key on the keyboard to supply additional locations. A location is specified through a branch name and a chainage. If a concentration is to be given, the component number is also required. The component number is given by the order found in the Components page of the AD editor. For rainfall/runoff, only the catchment name needs to be specified. Wind fields in MIKE 11 can be of two types either distributed or global. If a distributed wind field is to be augmented with uncertainty, then the chainage where the distributed wind source starts is required. If uncertainty is to be described for the global wind field, then the river name should be chosen as 'GLOBAL WIND' without a chainage.If the user-supplied location does not coincide with a boundary/forcing term, a warning is issued and the boundary information is ignored in the subsequent simulation.

For wind fields and catchments two standard deviations must be supplied since both of these types consist of two input variables. That is, wind velocity and wind direction for a wind field, and runoff and net rainfall for a catchment.

The specific standard deviation is defined in the Standard deviation editor and may be selected from a pull-down menu.

The lower window gives an overview of all the boundary statistics.

## 11.5  Standard deviation editor



Figure 11.5    The Standard Deviation Editor tab

The standard deviation editor is used for defining the different standard deviations that may be applied at the boundaries.

The page is structured as a table, the columns of which are described below.

- ID: For each standard deviation an ID is chosen. This ID can be any string, although it must be unique within the list of standard deviations. It is used when selecting which standard deviation to apply in the Boundary Statistics page.

- Data Type: The standard deviation is of a certain data type e.g. a standard deviation on water level requires a unit in the form of a length. The type required for the different boundary types is described in Table 11.2

Table 11.2    Boundary type and corresponding data type

| Boundary type | Data Type |
| --- | --- |
| Water Level | Water Level |
| Discharge | Discharge |
| Lateral inflow | Discharge |

Table 11.2    Boundary type and corresponding data type

| Boundary type | Data Type |
|---|---|
| Q/h relation | Water Level |
| Wind Field | Wind Direction and wind speed |
| Output from Catchment | Discharge and Rainfall |
| Concentration | Concentration |
| Input to Catchment (not available yet) | Rainfall, temperature and evaporation |

- Standard deviation: The method by which the standard deviation is to be determined is selected. The choices are constant, relative and time series. A relative standard deviation is selected as a percentage of the value found in the boundary at that specific time step. The percentage is taken based on the absolute value for discharge, concentration, wind velocity, runoff, and net rainfall. For water level and Q/H-rating curves the percentage is interpreted as being taken from the water depth.

- File/value: Depending on the choice made in the previous column the user supplies a constant value, a percentage or a time series with the temporal variation of the standard deviations to be used.

- Item: This column is only of importance when selecting a times series.

- Limits: For a relative and a time series type of standard deviation bounds may be applied. Thus the standard deviation is limited by these bounds.

- TC before and TC after TOF: Explained below.

The errors applied at the boundaries may be described through a first order auto regressive process:

$$\xi_{n+1} = \phi\xi_n + \varepsilon \qquad\qquad (11.1)$$

where

$\xi_n$ the error at time step $n$

$\phi$ the regression coefficient

$\varepsilon$ white noise

The regression coefficient should be interpreted as the 'memory' of the model error. To ensure that this 'memory' is independent of the time step used the

user is required to specify a time constant instead. The relation between the regression coefficient and the time constant is:

$$\phi = \exp\left(-\frac{\Delta t \ln 2}{T_{\frac{1}{2}}}\right)$$

(11.2)

where

$\Delta t$   the simulation time step

ln   the logarithm with base e.

$T_{\frac{1}{2}}$   the time constant TC.

The time constant TC should be interpreted as the time it takes for the correction to drop to half the initial magnitude (exponential decay). For wind fields and Catchment runoff, two values must be supplied. The numerical value of the regression coefficient must be less than unity to ensure that the variance of the model error is limited. A negative time constant results in a regression coefficient which is greater than unity. Therefore, if a negative time constant is entered, a warning is issued and the regression coefficient is set to 0 in the simulation.

As an option, the time constant used when forecasting may differ from the time constant used for updating. Thus, all in all up to four time constants should be supplied by the user.

The model errors being described through the auto regressive process are used in the forecast period to generate errors to be added to the boundaries. If the forecast is deterministic, the model errors are described through the auto regressive process without adding white noise. The model errors fade out according to an exponential decay.

## 11.6    Output

Figure 11.6    The Output selection tab

The Output page is used for specifying additional output from an uncertainty assessment analysis or a Kalman filter update. The output page is split in two; one for hydrodynamic output and one for advection dispersion output. For both types the user may select the following output:

- Standard deviation
- Corrections (the corrections of the state variables in the river network caused by updating with each of the measurements)
- Confidence intervals

The latter requires the user to specify a percentage for each confidence interval. For example in Figure 11.6 as additional output two confidence intervals (75% and 90%) along with the standard deviation have been selected for the hydrodynamic variables consisting of water level and discharge. Thus, in the additional HD result file these confidence intervals and the standard deviation in the whole network will be stored. Further, in Figure 11.6 two confidence intervals have been chosen for the advection dispersion components. These confidence intervals are stored in the additional AD result file.

## 11.7    A Step-by-step Guide to Uncertainty Assessment

Uncertainty assessment may be carried out for hydrodynamic and/or advection dispersion. Below follows a step-by-step guide for how to set-up an uncertainty assessment model.

1.  Set up the MIKE 11 model as a pure deterministic model i.e. no data assimilation.

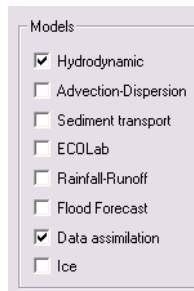2.  In the simulation editor activate the 'Data assimilation tick box'.



Figure 11.7    Selecting the Data assimilation mode in the simulation editor

3.  Create a new data assimilation file (extension '.DA11').

4.  Make sure that this file is connected with the simulation file by selecting the created DA11 file in the DA parameters box in the input tab of the simulation editor.
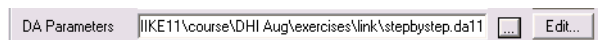


Figure 11.8    Selection of the data assimilation file.

5.  Open the DA parameter file.

6.  Select 'Uncertainty prediction' in the General menu.

7.  Choose the ensemble size. Remember that the larger this number the more accurate the uncertainty assessment, the trade-off being longer run times. A value of 100 is recommended if feasible.

8.  Consider boundary conditions applied to the model of the type

    –   Water level
    –   Discharge
    –   Q/H-relation
    –   Wind
    –   Source point lateral discharge
    –   Concentration
    –   Output from Catchment

For each of these boundaries consider which are to be assessed with respect to uncertainty.

9. Based on the selection above the boundaries are added in the DA editor. Note that the same boundary statistics may be applied to multiple boundaries of the same type. Thus, for instance a 10 percent uncertainty can be added to all discharge boundaries by selecting all the locations in the second box as shown below.



Figure 11.9    Multiple locations with the same boundary statistics. The same statistics will be applied to the upstream boundaries in main stem, STtrib1 and STtrib2. To add a line place the cursor in the last line or if no lines are present in the upper left hand corner and use the right tab key on the keyboard.

Once the uncertainties have been decided upon the Standard Deviation Editor is used to define the individual standard deviation items. The standard deviation is given an ID. This ID is subsequently used for selecting which Standard Deviation to apply to the individual boundaries. Further, set the standard deviation method to be used:

– constant, a value is specified in the value/file field
– relative, a percentage is specified in the value/file field
– a time series, a filename and a time series is specified by browsing.

If a relative or a time series type of standard deviation is selected, specify (optionally) an upper and a lower limit to be applied.

10. Finally, the time constant for the temporal development of the errors are set. Only the time constant prior to the time of forecast is used. The time constants describe the "colouring" of the noise. If pure white noise is to be applied at the boundaries, the time constant should be set to zero. Typically, the noise at the boundaries will not be independent from time step to time step (white noise), thus a non-zero time-constant (coloured noise) is recommended. Please refer to section 11.5 for a description of the time constants.

11. Decide what additional output is required. Tick the appropriate boxes for HD and/or AD and supply the confidence intervals percentage. Note that the confidence intervals are found by sorting each of the simulation values in the ensemble at every grid point. The confidence internal of say 90% is found by locating the values in the lowest 5% range of the ensemble and the values in the highest 5% range. Thus, as an absolute minimum the ensemble size must be sufficiently large to represent this fraction. For the 90% case an ensemble size of at least 20 (1/20 = 5%) is required. It is recommended to use a larger ensemble size than the minimum dictated by the confidence interval.

## 11.8 A Step-by-step Guide to Updating Using the Kalman Filter Method

If water level, discharge or concentration measurements are available within the model domain these may be utilized for updating the model. Below follows a step-by-step guide for setting up a model that uses the Kalman filter updating method.

1. Follow the step-by-step guide for setting up an uncertainty run given in section 11.7.

2. Determine the location of the measurements that are to be utilized for the updating. The location of the measurement is inserted through the use of a river name and a chainage. Remember that the measurements do not need to be located at a cross section.

3. Set the variable type of the measurement. If a concentration measurement is to be used, then the advection dispersion component number is also required.

4. Specify the location of the file holding the measured data by browsing for the file. The data must be in the dfs0 format with a calendar axis.

5. The description used for the standard deviation is to be supplied. The standard deviation may be of three types

   – constant, a value is specified in the value/file field
   – relative, a percentage is specified in the value/file field
   – a time series, a filename and an time series is specified by browsing.

   If the relative or the time series option has been selected, bounds can optionally be applied to the standard deviation.

6. Items 3 - 5 are repeated for every measurement. Augmenting the list with a measurement is done by placing the courser in a row in the overview at

the bottom of the measurements menu and then subsequently pressing the right tab key until a new row appears.

7. Additionally, the updating method may be used for forecasting by setting a time of forecast within the simulation period specified in the simulation editor. Remember that for each measurement data must be available from simulation start until time of forecast.

8. Finally, the forecast mode should be selected. Either deterministic (quick, no forecast statistics) or stochastic (slow, but with forecast statistics).

## 11.9 A Step-by-step Guide to Updating Using the Weighting Function Method

Below follows a step-by-step guide for setting up a model that uses the Weighting function updating method.

1. Follow steps 1-5 in the step-by-step guide for setting up an uncertainty run given in section 11.7.

2. Select 'Measurement update' and 'Weighting function' in the General menu.

3. If forecasting is performed, a time of forecast is set. The time of forecast should be within the simulation period specified in the simulation editor.

4. Determine the location of the measurements that are to utilized for the updating. The location of the measurement is inserted through the use of a river name and a chainage. Remember that the measurements do not need to be located at a cross section.

5. Set the variable type of the measurement. If a concentration measurement is to be used, then the advection dispersion component number is also required.

6. Specify the location of the file holding the measured data by browsing for the file. The data must be in the dfs0 format with a calendar axis. The measurement data do not need to cover the entire period up to the time of forecast. Updating is simply performed for each measurement up to the time of the last data point in the dfs0 file.

7. Set the type of weighting function, the amplitude of the function in the measurement point and the lower and upper chainages for which the function should be applied. The amplitude should be set to 1 if a perfect match to the measurement is wanted, whereas an amplitude less than 1

should be chosen to account for uncertainties in the measurements. The type of weighting function and the bounds for which it applies should reflect the correlation between the model error at the measurement location and the errors at nearby grid points. Note that the weighting function can only be applied on the branch where the measurement point is located.

8. If error forecasting should be applied, activate the 'Apply error forecast' tick box and choose the error forecast model from the pull-down menu. The error forecast models are defined in the Equation editor.

9. Items 4 - 8 are repeated for every measurement. Augmenting the list with a measurement is done by placing the courser in a row in the overview at the bottom of the measurements menu and then subsequently pressing the right tab key until a new row appears.

## 11.10  Examples

With the installation of MIKE 11 follows a number of examples illustrating the use of the Data assimilations methods.

The examples are all based on a simple set-up with three branches. The examples are designed to illustrate the main features of the data assimilation. The are 5 examples available all located in the folder:

*'...\examples\m11\DataAssimilation\example1'*

There five examples are described below

### 11.10.1 Uncertainty assessment on hydrodynamic simulation

The example located in the folder '*uncertainty*' illustrates how an uncertainty assessment simulation is set up. Please run the simulation to generate the results.

**Note** that confidence intervals are stored in the additional hydrodynamic result file. Figure 11.10 illustrates the temporal variation of the discharge through the structure located in Trib2 along with a 90% confidence interval band.

Figure 11.10   The discharge through the structure in Trib2. The upper and lower limits of the 90% confidence interval are also illustrated.
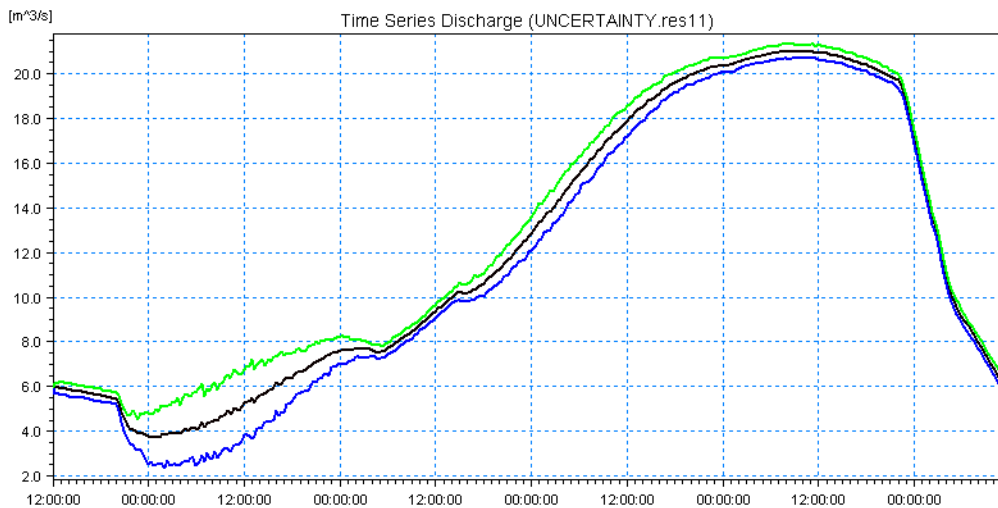
## 11.10.2 Kalman filter updating on hydrodynamic set-up

Based on the set-up from the example described previously an updating model has been implemented. The full set-up is located in the folder '*UpdateKalman*'. The example illustrates how the Kalman filter is capable of recreating a lost peak of a hydrograph. Figure 11.11 illustrates the effectiveness of the updating technique.
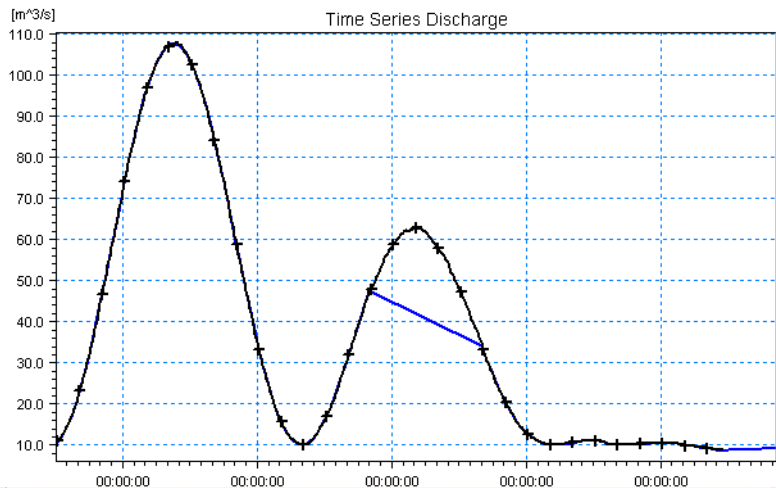
Figure 11.11  Updating using the Kalman filtering technique. The discharge at the upstream end of the main stem is illustrated. The blue curve is the result obtained using MIKE 11 without updating the black curve with crosses is the result obtained using the updating method. The updating is based on a measurement located further downstream in the main stem

## 11.10.3 Uncertainty assessment on advection dispersion simulation

The example located in the folder '*AD_uncertainty*' illustrates how an advection dispersion uncertainty assessment simulation is set up.

Figure 11.12 shows the temporal variation of the concentration at the junction of the main stem and Trib2 along with a 90% confidence interval band.
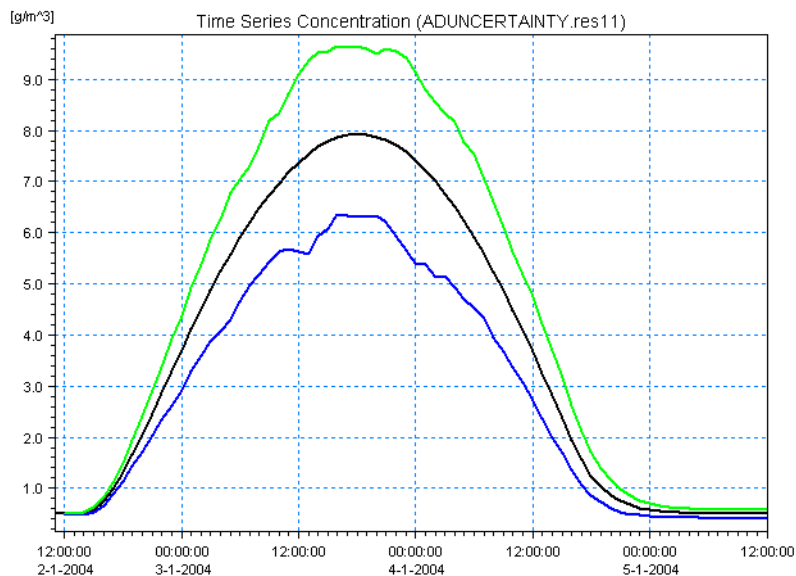
Figure 11.12 The concentration at the junction of the main stem and trib2. The upper and lower limits of the 90% confidence interval are also illustrated

## 11.10.4 Kalman filter updating on advection dispersion set-up

The advection dispersion set-up from the previous example has modified so that the upstream boundary condition consists of a constant concentration of 0.5 g/m$^3$. The present example illustrates how the updating technique also is effective for the transport equation. The set-up is located in the folder '*AD_UpdateKalman*'. Figure 11.13 illustrates the effectiveness of the updating technique for AD-simulations.
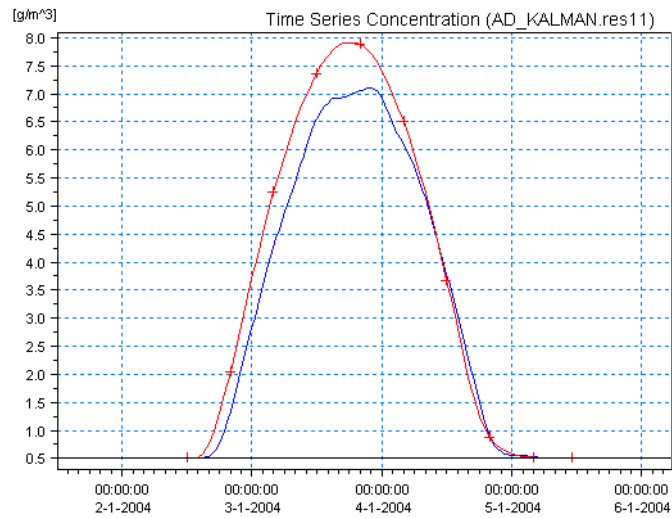
Figure 11.13  The concentration at the junction of the main stem and trib2. The red curve illustrates the original simulation without erroneous boundary condition. The blue curve illustrates the updated results. Finally for comparison the erroneous results (no update) are illustrated as a horizontal line at 0.5 g/m$^3$

# BATCH SIMULATION EDITOR

# 12    Batch Simulation Editor

The Batch Simulation Editor offers a possibility for setting up a batch simulation from the MIKE Zero shell. That is, the Batch Simulation Editor is used to pre-define a number of simulations, where all items included in a simulation (input-files, simulation parameters, output files etc.) can be changed from simulation to simulation and multiple simulations are then performed automatically when starting the Batch simulation.
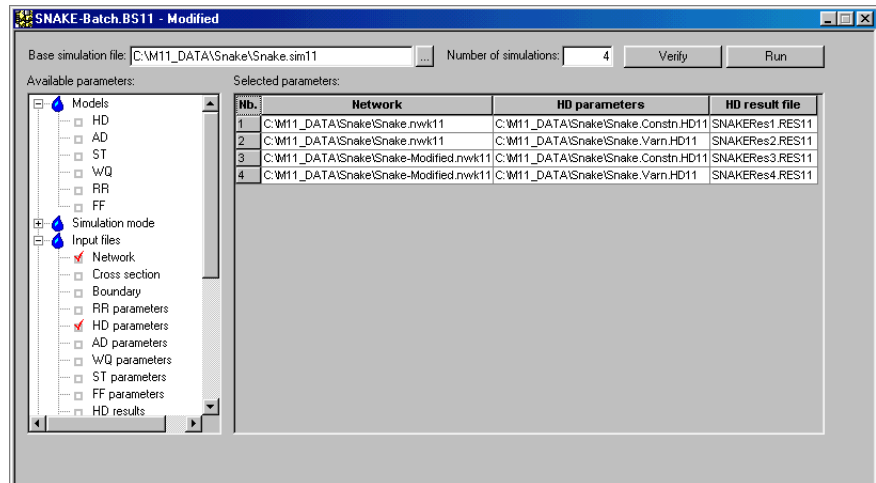


Figure 12.1    The MIKE 11 Batch Simulation Editor.

Figure 12.1 above shows an example on how to use the Batch Simulation Editor. The figure presents a batch simulation setup, where two different network files are combined with two different HD Parameter files. A setup like this could be used to investigate the impact of variations in bed resistance values (Manning numbers) at locations where a hydraulic structure (weir) has been planned. The two different network files will then be identical except one of the files will contain description on the new proposed weir, and the two HD Parameter files will only differ in the local variation of the Manning numbers.

Output from the four different batch simulations has also been defined such, that results from each simulation are saved in different result-files.

The Batch Simulation Editor has been developed in cooperation with CTI Engineering, CO., Ltd., Japan.

## 12.1    Setting up a Batch Simulation

The following steps are necessary to setup the Batch Simulation:

● Pre-define base simulation file

- Define parameters to adjust in batch simulation
- Specify input parameters for each simulation

Each of the steps are described in the following:

## Pre-define base simulation file

The Batch Simulation Editor is designed such, that a Base simulation file (*.sim11) with all relevant information on simulation models and simulation mode, input files, simulation period, time step, initial conditions and output filenames must be defined prior to using the Batch Simulation Editor. Batch simulations are then performed with this Sim11-file as a basis and only if other parameters or filenames have been defined by the user in the Batch Simulation Editor, will the definitions in the Base Sim11-file be changed.

Filename and path to the base Sim11 file must be defined in the 'Base Simulation File' field (Use the '...' button to browse for the Base Sim11 file on your computer).

## Define parameters to adjust in batch simulation

The user must define the number of simulations to be performed in the batch simulation by specifying a number in the 'Number of simulations' field. A number of (empty) rows will be introduced in the 'Selected Parameters' grid after specifying the this number and leaving the edit field using the <TAB> key or clicking with the pointer anywhere in the editor.

Each line in the 'Selected Parameters' grid shall only contain specifications of the parameters or input files which should be different from the definitions in the base simulation file. Parameters which should differ from the base simulation file is selected in the tree-view on the left-part of the Batch Simulation Editor, (see Figure 12.2). Open any tree-view items by clicking the '+' and select the item/parameter, which should be modified in the batch simulation by double-clicking the empty square in front of the specific item. After double-clicking the item, a new column will be introduced in the 'Selected Parameters' grid and a red check-mark is presented in the square in front of the selected item in the tree-view. Within the new column in the Selected Parameters, it is now possible for the user to select different input files or define variations in input parameters for the different simulation performed in the batch simulation.
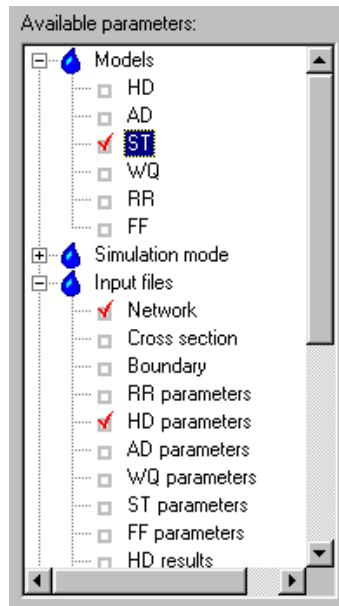
Figure 12.2    Tree view from the Batch Simulation Editor dialog for selecting batch simulation parameters

## Specify input parameters for each simulation

Input parameters for the batch simulation can be different input filenames, different simulation parameters, activating or deactivating simulation models (e.g. activate and/or deactivate AD-model in some simulations) etc.

If e.g. the Network file should be different in some simulations, open the 'Input files' item in the tree-view and double-click the Network square. After this a Network column is presented in the 'Selected Parameters' grid and network-files can now be specified in this column - either manually or by pressing the '...' button to browse for the required file. If e.g. the network file in one simulation should be the same as in the base simulation file - but other parameters are changed - the 'base network file' must be defined in the network field, as it is not allowed to have any blank cells in the 'Selected Parameters' grid.

Additionally, e.g. the AD-model should be deactivated in some simulations, open the 'Models' item in the tree-view and double-click the AD square. In the 'Selected Parameters' grid you will now have the possibility in the AD column to set the value to False (model deactivated) or True (Model activated in simulation).

After all files and parameters for the batch simulation have been specified, it is required to save the data to a Batch Simulation file (*.BS11).

## Verify Batch Simulation definitions

The 'Verify' button can be used to make a test of all batch-setups in the Batch Simulation file. The verification procedure includes a test of all input-files, simulation parameters etc. and therefore, if problems exist in some of the input files or other simulation parameters, the user will be informed about this through the verification procedure.

## Start Batch Simulations

After the verification of the setup has been performed, press the 'Run' button to start the batch simulations.

# 13    Correlation Analysis And Gap Filling Tool

## 13.1    Introduction

This tool can, as it's name indicates, do the following:

1. Make a correlation analysis between 2 or more time series data.

2. Fill gaps (i.e. generate values artificially) in time series data where values are missing.

To start the tool you should open or create a Correction Analysis & Gap Filling file (.CGF) by selecting Open or New in the File menu.

The tool itself consists of one window being divided into a Tree view, a Data view and an Error Message view, see Figure 13.1. When applying the tool you would have to step through all pages by selecting the items in the tree view from top to bottom. The content of the tree view may be less than shown in Figure 13.1.
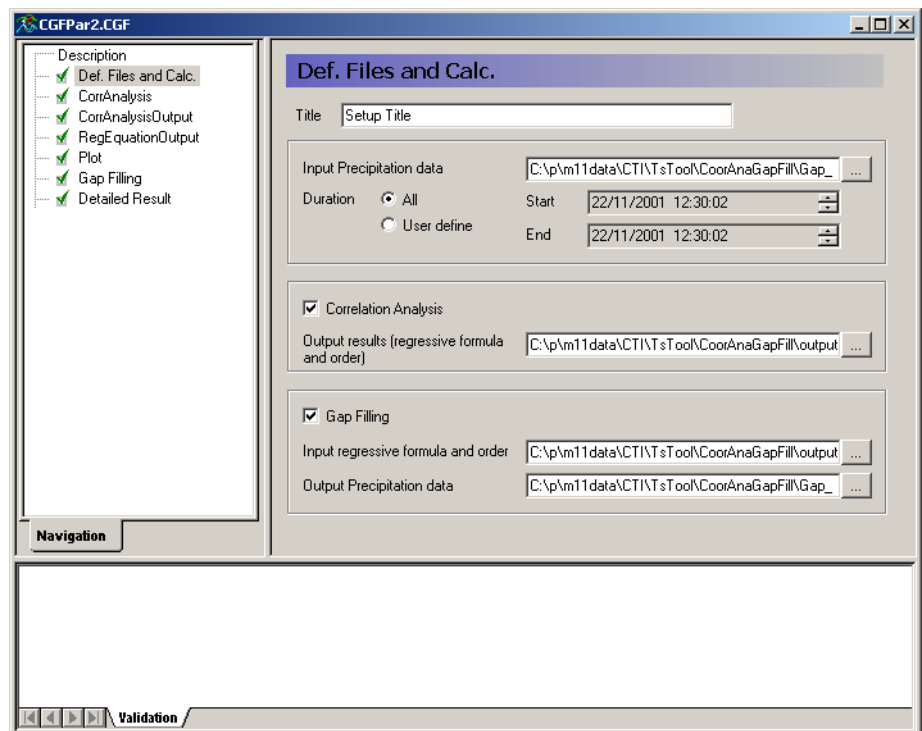


Figure 13.1    Layout of user interface.

## 13.2   Def. Files and Calc. page

When a new .CGF file is created the tree view contains only one item in the tree view, but as selections are made on the first page the tree view expands.

### Input Time Series Data.
The time series data for the correlation analysis and/or gap filling is selected. The tool can be used for any type of data, i.e. not limited to precipitation. The time axis in time series file can be either equidistant or non-equidistant. The selection of duration allow the user to control the time period used for the analysis. The choices are "User Defined", or "All" corresponding to all data in the selected data file. The tool always applies all items in the time series file.

### Correlation Analysis
The choice of including correlation analysis or not has to be made. If selected an output file name has to be specified. The output file is an ASCII/PFS file.

### Gap Filling
The choice of including gap filling or not has to be made. If selected a regressive formula input and an gap filling output file name has to be specified. Most often the user will specify the regressive formula input file as being equal to the output file from the correlation analysis. The output file is a .dfs0 time series file which will contain the result of the gap filling.

Figure 13.2 shows the page when all data selections and choices has been made.



Figure 13.2    Def. Files and Calc. page

## 13.3 CorrAnalysis page

### Regression Formula

On this page the user selects what regression formula to use. Four types of regression formulas all being linear are available:

1. Y=AX+B (Y regression)
2. Y=AX+B (XY regression)
3. Y=AX (Y regression)
4. Y=AX (XY regression)

### Lower Limit

For some data types, for instance rainfall, is may be desirable to exclude values less than a lower limit in the analysis.

### Run

Once the run button is pressed the correlation analysis in carried out, and the result is written to the output file specified on the first page.

The CorrAnalysis page is shown in Figure 13.3.



Figure 13.3    CorrAnalysis page.

## 13.4 CorrAnalysis Output page

The result of the correlation analysis is shown on this page. See Figure 13.4.

The page contains a table with number of rows and columns equal to the number of items in the time series input file. For each item in the file a row is

shown holding the result of the correlation analysis against all items in the time series file. The result of each analysis is shown with two numbers: the correlation coefficient and number of values applied in the analysis. The number of values applied may differ from one analysis to the next as the number of missing values may differ from one item to the next. In the diagonal of the table the correlation coefficient is always 1.000 as this is the result of correlating the item to itself.



Figure 13.4    CoorAnalysisOutput page

## 13.5   RegEquationOutput page

The coefficients in the regression equation are shown in this page. See Figure 13.5.

The page contains a table with number of rows and columns equal to the number of items in the time series input file. For each item in the file a row is shown holding the coefficients in the regression equation corresponding to the other items in the time series file. The two numbers shown are the coefficients A (upper) and B (lower) in the equation Y=AX+B. In the diagonal of the table the A and B is not shown as these always are 1 and 0 respectively.

Figure 13.5    RegEquationOutput page

## 13.6    Plot page

This page allows for plotting the data in one item against data from another. See Figure 13.6.

### Row / Column number / Change Plot
Here you select which items to plot. Once you changed the numbers you should press the "Change plot" button to update the plot.

### Copy
Pressing the button copies the plot to the clipboard as Windows meta file information. This allows for subsequent paste into for instance a word processing tool.

### Print
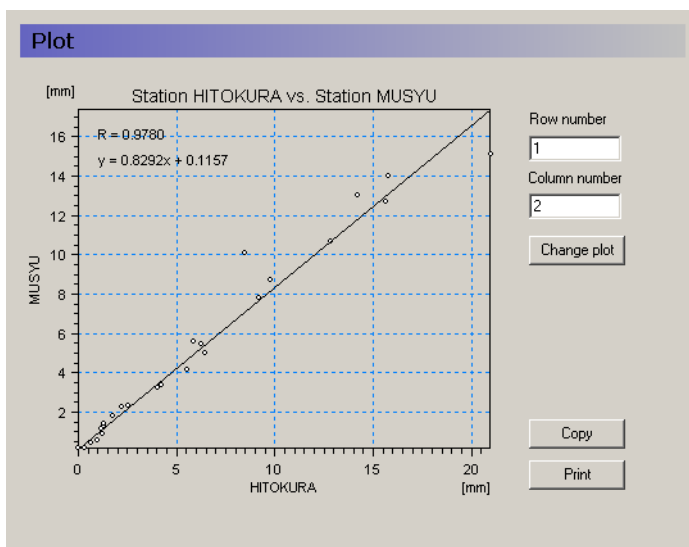Pressing this button will send the plot to a printer.

Figure 13.6    Plot page.

## 13.7    Gap Filling page

In this page parameters for the gap filling are specified and the gap filling process is launched. See Figure 13.7.

The gap filling works such that for each missing value in each item an estimated value will be calculated by using the value at the same time step from another item transformed using the corresponding regressing equation. For each item the other items are ranked in the order of decreasing correlation coefficient. When a missing value occurs a non-missing value is searched for in the other items at the same time step, and the non-missing value in the item with the highest rank (lowest order) is selected.

The parameters for the gap filling are:

### Gap filling boundary value.
Gap filling will only occur if the value applied to generate the missing value through the regression equation is larger then the boundary value.

### Lower limit for sample numbers.
If the number of non-missing values in an item is less than the specified limit this item will not be used for gap filling.

### Lower limit for correlation coefficient.
If the correlation coefficient for an item is less than the specified limit this item will not be used for gap filling.

## Detailed condition

In the table the item specific conditions for the gap filling is given. For each item to be gap filled a row with specifications is available. The first column in the row indicate if the specifications are to be achieved automatically (equal to the output of the correlation analysis) or to be user defined. In the remaining columns the items are shown in order with the following information for each item:

1. Item number in time series input file.

2. Regression equation

3. Number of non-missing values

4. Correlation coefficient.

These parameters can be edited if the check box in the first column is off.

## Run

Pressing this button starts the gap filling process, and the output .dfs0 file is generated. The result is shown also in two pages Details Results and Outlined Results. See Figure 13.8.



Figure 13.7    Gap Filling page

# 13.8    Detailed Result page

The results of the gap fillings are shown in detailed on this page. See Figure 13.8.

The detailed results shows all the time series data after gap filling. Artificial values that has been calculated by the gap filling has an integer number to

the left of the number. This number is the item number used for the gap filling. There may still be gaps in the data as no gap filling is done if all items on a particular time step has missing values.

| No | Date | 1 HITOKURA | | 2 MUSYU | | 3 MUSHU_2 |
|---|---|---|---|---|---|---|
| 4 | 1982-07-31 13:00:00 | 0.000 | | 0.000 | | 0.000 |
| 5 | 1982-07-31 14:00:00 | 0.000 | | 0.000 | | 0.000 |
| 6 | 1982-07-31 15:00:00 | | | | | |
| 7 | 1982-07-31 16:00:00 | 0.000 | | 0.000 | | 0.000 |
| 8 | 1982-07-31 17:00:00 | 0.000 | | 0.000 | | 0.000 |
| 9 | 1982-07-31 18:00:00 | 0.000 | | 0.150 | 2 | 0.150 |
| 10 | 1982-07-31 19:00:00 | 8.450 | | 10.090 | | 10.090 |
| 11 | 1982-07-31 20:00:00 | 0.670 | | 0.460 | | 0.460 |
| 12 | 1982-07-31 21:00:00 | 0.000 | | 0.000 | | 0.000 |
| 13 | 1982-07-31 22:00:00 | 0.180 | | 0.110 | | 0.110 |
| 14 | 1982-07-31 23:00:00 | 0.000 | | 0.000 | | 0.000 |
| 15 | 1982-08-01 00:00:00 | 0.070 | | 0.040 | | 0.040 |
| 16 | 1982-08-01 01:00:00 | 0.000 | | 0.000 | | 0.000 |
| 17 | 1982-08-01 02:00:00 | 0.000 | | 0.000 | | 0.000 |
| 18 | 1982-08-01 03:00:00 | 0.000 | | 0.080 | | 0.080 |
| 19 | 1982-08-01 04:00:00 | 0.950 | | 0.600 | | 0.600 |
| 20 | 1982-08-01 05:00:00 | 1.330 | | 1.310 | | 1.310 |
| 21 | 1982-08-01 06:00:00 | 1.320 | | 1.450 | | 1.450 |
| 22 | 1982-08-01 07:00:00 | 1.190 | | 1.180 | | 1.180 |
| 23 | 1982-08-01 08:00:00 | 2 | 5.256 | 6.200 | | 6.200 |
| 24 | 1982-08-01 09:00:00 | 14.230 | | 13.030 | | 13.030 |
| 25 | 1982-08-01 10:00:00 | 1 | 3.310 | 1 | 2.860 | 1 | 2.880 |
| 26 | 1982-08-01 11:00:00 | 2.210 | | 2.300 | | 2.300 |
| 27 | 1982-08-01 12:00:00 | 1.790 | | 1.820 | | 1.820 |
| 28 | 1982-08-01 13:00:00 | 2.560 | | 2.360 | | 2.360 |
| 29 | 1982-08-01 14:00:00 | 4.230 | | 3.350 | | 3.350 |
| 30 | 1982-08-01 15:00:00 | 5.840 | | 5.600 | | 5.600 |
| 31 | 1982-08-01 16:00:00 | 6.440 | | 5.000 | | 5.000 |
| 32 | 1982-08-01 17:00:00 | 4.060 | | 3.270 | 2 | 3.270 |

Figure 13.8    Detailed Result page.

# 13.9   Outlined Output page

A summary of the gap filling process is shown on the page. See Figure 13.9.

For each item following information is shown (from left to right)

1. Number of missing values before gap filling.

2. Number of missing values after gap filling.

3. For each item (other than the item itself) two numbers are shown:
   – The item number
   – Number of values from this item used in the gap filling.

### Outlined Result

| Station No & Name | Number of missing data | Number of not filled data | Used station No. and filled data number | | | | | |
|---|---|---|---|---|---|---|---|---|
| | | | Order 1 | | Order 2 | | Order 3 | |
| 1<br>HITOKURA | 2 | 1 | 2 | 1 | | | | |
| 2<br>MUSYU | 4 | 1 | 3 | 2 | 1 | 1 | | |
| 3<br>MUSHU_2 | 5 | 1 | 2 | 3 | 1 | 1 | | |

Figure 13.9    Outlined Results page

# FLOW RESISTANCE AND VEGETATION A

# A.1    Flow Resistance and Vegetation

Only a few detailed investigations have been made on establishing relation-ships between flow resistance in a stream filled with vegetation and flow resistance in the same stream without any vegetation. A quantitative evalua-tion of the influence of vegetation on flow resistance has been performed in a few Danish gauging programmes. For each of the programmes it has been possible to identify the influence of the weed on the flow resistance, but it has not been possible to transfer the results to other streams and environments. Therefore, it is evident, that description of the weeds influence on flow resist-ance and hydraulic conditions in general is always a matter of calibrating the modelling system by adjusting values of the bed resistance parameter.

Results and findings from the Danish gauging programmes and investiga-tions on the weeds influence on flow resistance are described in the follow-ing.

## A.1.1  Flow Channels in Halkaer AA

Jensen et. al, /4/ describes experiments performed in a Danish stream named 'Halkær Å' (= Stream of Halkaer). A straight-line stretch of the stream with very dense vegetation was chosen for the experiment, and regulators for control of the discharge into the stretch were introduced. The object of the experiment was to determine *Q-h* relations for different weed densities. *Q-h* relations were established for natural (very dense) weed conditions, and additionally for situation where flow channels of different widths were cut in the weed. Widths of 0.5 m, 1 m and 2.5 m (equals weed-free conditions) were investigated. The vegetation type was Bur Reed (Latin: Sparganium sp.; Dan-ish: Pindsvineknop) with few occurrences of Water Thyme (Latin: Helodea sp.; Danish: Vandpest). The obtained Q-h relations are presented in Fig A.1.1
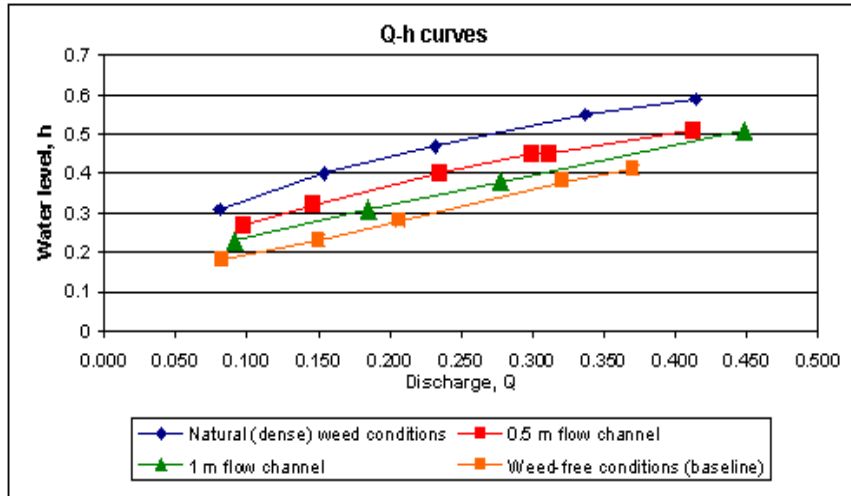
*Fig A.1.1*        *Q-h curves determined for varying flow channel width*

Calculated Manning numbers (Manning's *M*) are presented in Fig A.1.2 as a function of Discharge, *Q*. From this figure, it can be seen, that the flow resistance in a weed-filled stream can be up to 4 times larger compared to weed-free conditions in the same stream.
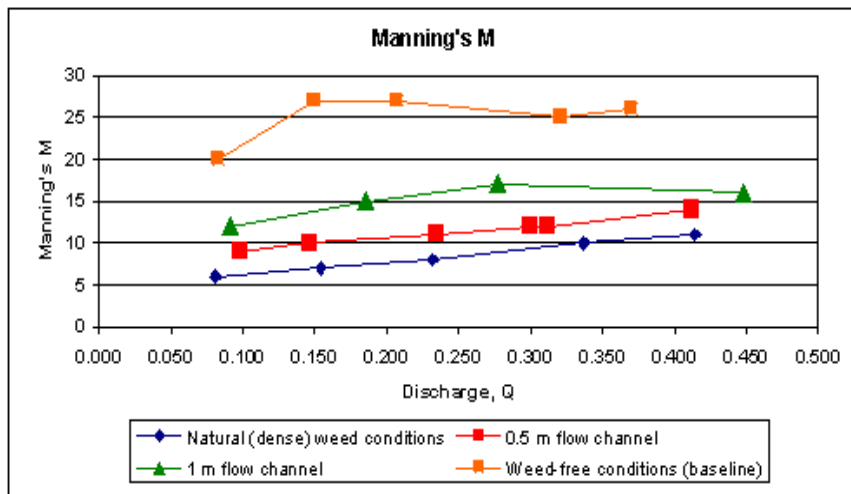


*Fig A.1.2*        *Manning's M calculated as a function of Discharge, Q*

## A.1.2  Laboratory measurements using Bur Reed

Jensen /3/ describes a laboratory experiment using a 15 m long and 0.3 m wide flow channel. A weed-bank of 2 meters in length was prepared using leaves of Bur Reed (Latin: Sparganium emersum Rehman; Danish: enkeltbladet pindsvineknop). The experiment included a series of measurements with varying weed density. Fig A.1.3 shows the results from the measurements. Manning's n is plotted against the product; Velocity, *V*, times the hydraulic radius, *R*, for two different densities of weed (defined by mass of dry material per area) and a complete weed-free situation. From the results it can be seen, that the flow resistance varies with a factor of 4 to 6 from a weed-free channel to a situation with very dense vegetation (325 g dry material/m$^2$).
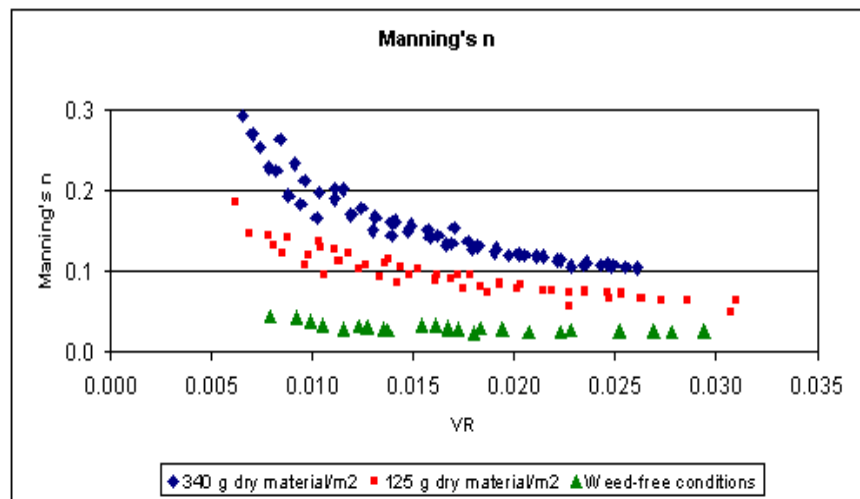


Fig A.1.3          *Manning's n vs VR (VR: Velocity times Hydraulic Radius)*

Jensen /3/ discusses the possible correlation of flow resistance and hydraulic parameters and presents arguments, stating that the variation in flow resistance can be correlated to the product, *VR* for a specific weed density by the following equation:

$$n = a\ln(VR) + b \tag{A.1.1}$$

where, *n* is Manning's *n*, *V* is the average flow velocity, *R* hydraulic radius and *a* and *b* are coefficients determined by regression.

A verification trial of eq. (A.1.1) using measurements from another Danish stream; 'Simested Å', was unsuccessful. Application of eq. (A.1.1) is, however, supported by Bakry /1/ where statistics have been made on 12 cross sections with 'drowned weed', that is, weed which primarily gets its nourishment from the water and therefore is not limited to the area near the stream

banks. In this series of investigations it was found, that in case the weed is limited to the banks only it is suitable to use the following expression:

$$n = aD_\eta^\beta \tag{A.1.2}$$

where *a* and *b* are coefficients as described for equation (A.1.1) and $D_h$ is the hydraulic depth calculated from:

$$D_\eta = \frac{A}{B} \tag{A.1.3}$$

where *A* is the flow area and *B* is the width of the section at water surface.

It should be noted, that eq. (A.1.1) depends significantly on the flow velocity compared to eq. (A.1.2). This reflects the fact, that weed along banks (non-drowned) is less liable to lie down due to high flow velocities than fully drowned weed.

## A.1.3 Experiments in 'Kimmeslev Møllebæk'

Høybye et. al, /2/ describes how *Q-h* curves have been determined in a Danish stream named 'Kimmerslev Møllebæk' for both a winter and a summer situation. These situations are practically identical to periods with no weed in the stream and periods with very dense vegetation present in the stream. In the summer situation the weed is primarily bank vegetation and to a smaller extent bed vegetation. Bottom width of the cross section is approx. 2 m, bank slopes approx. 30 degrees and measurements have been performed - for both situations - for depths between approx. 6 and 50 cm.

Results showed, that Manning's *M* in the winter situation varies from 15 $m^{1/3}$/s at small water depths up to 30 $m^{1/3}$/s for large water depths. Fig A.1.4 shows the calculated Manning numbers as a function of water depth. For comparison expressions of the form (A.1.2) have been fitted to the data.
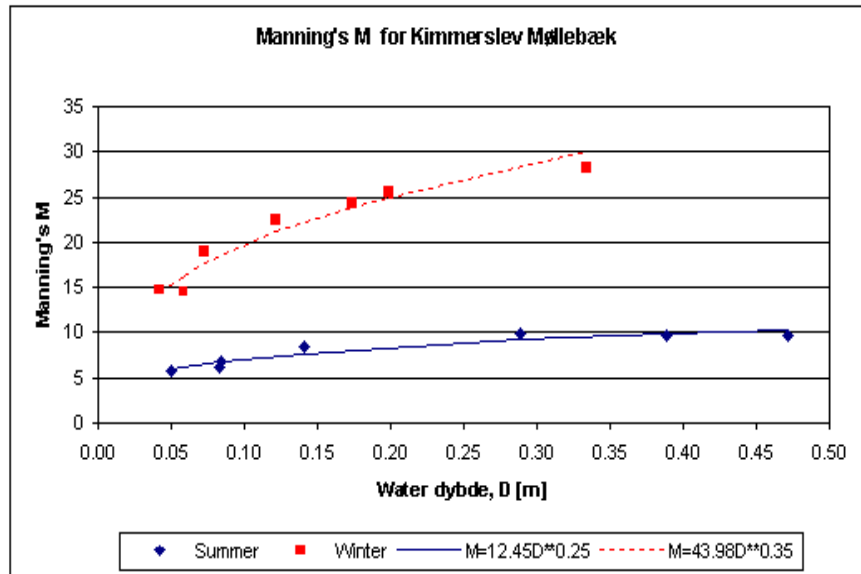
*Fig A.1.4*      *Manning's M for Kimmerslev Møllebæk in summer and winter period. Results calculated with the formulas of the form M = αD$^β$ are also included*

## A.1.4 Experiments in 'Arn Å'

Høybye et al., /2/, describes a gauging programme with the purpose of determining the variation of Manning's *M* in the period from May 1990 till October 1991. In the beginning of the period, Manning's *M* is approx. 10 m$^{1/3}$/s, increasing to approx. 15 m$^{1/3}$/s in August 1990 as a result of weed cutting. Thereafter Manning's *M* increases during winter to a value of approx. 25 m$^{1/3}$/s. From April it is found, that Manning's *M* starts to drop and ends at approx. 10 m$^{1/3}$/s in late summer.

These results - an annual variation in Manning's *M* between approx. 10 m$^{1/3}$/s and 25 m$^{1/3}$/s - are identical to the variations observed in 'Kimmerslev Møllebæk'.

## A.1.5 References

/1/      Bakry, M.F.; T.K.Gates; A.F.Khattab:

"Field Measured Hydraulic Resistance Characteristics in Vegetation Infested Canals". *Journal of Irrigation and Drainage Engineering. Vol 118 No. 2, 1992.*

/2/ Høybye, J. Alex Andersen:

"Eksperimentel Undersøgelse af Friktionsformler for Åbne Vandløb". *Hedeselskabet. Afd. for Hydrometri og Vandressourcer, 1996*

"Experimental investigations of friction formulae for open channels". *Hedeselskabet, dep. for Hydrometry and Water Resources, 1996 (In Danish)*

/3/ Jensen, K.R.:

"Undersøgelse af Vandløbsvegetationens Hydrauliske Indflydelse." *Afgangsprojekt, AUC, 1992*

"Investigation of the influence of stream vegetation on hydraulic conditions" *B.Sc. Thesis from University of Aalborg, Denmark (In Danish)*

/4/ Jensen, S.A.B.; Niels Olsen; Jan Pedersen:

"Strømrender i Grødefyldte Vandløb". *Afgangsprojekt, AUC, 1990*

**"**Flow channels in weed-filled streams". *B.Sc. thesis from University of Aalborg, 1990.*

ADDITIONAL TOOLS B

# B.1 Additional Tools

Apart from the catalogue of features which are accessible from the MIKE Zero interface some additional application tools also come with a MIKE 11 installation. These are:

- pfsmerge: An application which is used for merging two or more pfs files (.nwk11,.bnd11,.ad11, etc.)

- res11read: A tool for converting result files from MIKE 11 (*.res11 files) to text files (ascii). The tool is only available for *.res11 files i.e. the classic MIKE 11 calculation kernel.

## B.1.1 Merging .pfs files

In some instances it may be necessary to merge set-ups. To do so the pfs-merge.exe program may be used. This program merges two or more files in the pfs format into one. The application may be applied to the following types of files:

- Network files (.nwk11). Please note the feature  Number Points Consecutively *(p. 49)* under the network editor.

- Boundary files (.bnd11).

- Rainfall-Runoff files (.rr11).

- Hydrodynamic parameter files (.hd11).

- Advection dispersion files (.ad11).

- Water quality files (.wq11).

- Eutrophication editor (.eu11).

- Sediment transport (.st11).

- Flood forecasting files (.ff11).

The application runs in a dos prompt and has the following syntax:

...\PFSMERGE pfsfile1 ... pfsfileN pfsfiletotal

where

...\ denotes the full path to the application located in the bin director of the installation.

pfsfile1 ... pfsfileN: The list of files to merge.

pfsfiletotal: The name of the combined pfsfile.

**Note** that the above syntax is based on a call from the data directory (the directory where the pfsfiles are located).

## B.1.2  Converting simulation results to text files (Res11Read)

The application named 'Res11Read' is designed for extracting information from one or more MIKE 11 result files and convert results or additional information on comparison etc. to an ASCII text file. Thus, the tool may be used as a conversion tool for subsequent post-processing of MIKE 11 result files stored in the binary res11-file format.

The Res11Read application must be launched from a command line in a Dos Prompt window. Res11Read is located in the MIKE Zero installation subfolder 'bin' with the name Res11Read.exe.

To execute the program, open a Dos Prompt (on Windows XP the Dos Prompt can normally be found under; Start-> Programs -> Accessories -> Command Prompt).

The syntax for running the application is:

> ...\Res11Read.exe Option(s) Res11FileName1 ... Res11FileNameN
> OutputFileName

where:

> '...\' denotes the full path to the application located in the bin directory of the MIKE Zero installation.

> 'Option(s)' are user-defined options instructing the Res11Read utility on which operation to perform (see below for list and explanation of options available)

> 'Res11FileName1 ... Res11FileNameN' is the list of *.res11 files to convert.

> 'OutputFileName' is the name of the ASCII output file.

### Options available for the Res11Read utility
The different Options available for usage with the application is listed and described briefly below:

- '-xy'
  Exports X- and Y-coordinates as well as levels for all calculation points (h- and Q-points) in the model.
  (example: 'Res11Read -xy xx.res11 out.txt')

- **'-xyh'**
  Exports X- and Y-coordinates and levels for all the h-points in the model.
  (example: 'Res11Read -xyh xx.res11 out.txt')

- **'-xyq'**
  Exports X- and Y-coordinates and levels for all the Q-points in the model.
  (example: 'Res11Read -xyq xx.res11 out.txt')

- **'-xyxsec'**
  Exports X- and Y-coordinates and levels for all h-points defined by cross sections in the model.
  (example: 'Res11Read -xyxsec xx.res11 out.txt')

- **'-raw'**
  Exports Raw data for all cross sections used in the simulation.
  (example: 'Res11Read -raw xx.res11 out.txt')

- **'-sim'**
  Content of the .sim11 file used for the simulation is saved as static data in the Res11-file. This can be extracted using this option (useful for extracting details on simulation settings).
  (example: 'Res11Read -sim xx.res11 out.txt')

- **'-minX'**
  Exports the absolute Minimum values obtained in all grid points through-out the simulation for item no X.
  The item-number can be found by using the option '-Items' initially. (For a standard HD-Result file Water Level is Item number 1 and Discharge is Item-number 2).
  (example: 'Res11Read -min2 xx.res11 out.txt')

- **'-maxX'**
  Exports the absolute Maximum values obtained in all grid points through-out the simulation for item no X
  (example: 'Res11Read -max1 xx.res11 out.txt')

- **'-xsecids'**
  Exsports the Cross section ID's from cross sections active in the simula-tion.
  (example: 'Res11Read -xsecids xx.res11 out.txt')

- **'-usermarks'**
  Exports User defined marks from the result file.
  (example: 'Res11Read -usermarks xx.res11 out.txt')

- **'-items'**
  Exports a List of all dynamic items present in the result file.
  (example: 'Res11Read -items xx.res11 out.txt')

- **'-allres'**
  Exports all simulation results (all time steps) for all calculation points in the model.
  Note: This option can occasionally result in a large ascii output file!
  (example: 'Res11Read -allres xx.res11 out.txt')

- **'-someresFILE'**
  Simulation results from selected calculation points only are written to the output file. The selected points to be exported is defined in a configuration file with a user-defined filename. In this example the file is named; 'FILE' (see format of FILE below).
  (example: 'Res11Read -SomeresInp.txt xx.res11 out.txt'
  where 'inp.txt' is the input configuration filename)

- **'-compareFILE'**
  This option allows a comparison of two Res11-files from the same model, and the option will issue an error if differences in result-values between simulation points in the compared results are larger than a user-defined threshold value.
  The FILE is the configuration file for this option and contains information on which threshold value shall be active for which result items. (see format of FILE below).
  (example: 'Res11Read -CompareInpFile.txt xx.res11 yy.res11 out.txt'
  where 'InpFile.txt is the configuration filename)

- **'-silent'**
  Writing to prompt is cancelled. Used in conjunction with one or more of the other options.
  (example: 'Res11Read -allres -silent xx.res11 out.txt')

- **'-MessageCompare'**
  Writes additional message on Return code in output file on result of comparison (0 or 2 for the Compare results option).
  (example: 'Res11Read -CompareInp.txt -MessageCompare xx.res11 yy.res11 out.txt'
  where 'Inp.txt' is the configuration file for file comparison)

- **'-DHIASCII'**
  Option for suppressing header information in the Output file in order to convert e.g. results/Time series into a DHI standard ascii format.
  Should be used in conjunction with one or more of the above.
  (example: 'Res11Read -max1 -dhiascii xx.res11 out.txt')

- **'-FloodWatch'**
  Special option for converting results from a MIKE 11 simulation into a specific format used by the FLOOD Watch real-time data handling system from DHI.

- '-MakeDfs0'
  Option to automatically create a time series containing results from specific calculation points.
  Note: This option can only be used in conjunction with the -SomeresFILE option
  (example: 'Res11Read -SomeResFILE -MakeDfs0 xx.res11 out.txt')

- '-SeperatorSTRING'
  Option for defining the seperator string between output-results.
  Note: This option is only applicable in conjunction with the Flood Watch option
  (example: 'Res11Read -Floodwatch -Separator, xx.res11 out.txt')

- '-RRResults'
  Extraction of Results from Rainfall Runoff simulation specifically dedicated toward a FLOOD Watch system. Hence, this option is only applicable with the FloodWatch option.
  (example: 'Res11Read -Floodwatch -RRResults rr.res11 out.txt')

- '-ArcView'
  Exports results into an ArcView table import format. Used in conjunction with other options. Typically RRResults as a sub-option to Floodwatch.
  (example: 'Res11Read -Floodwatch -RRResults -ArcView rr.res11 out.txt')

Some of the above listed options includes a 'FILE' in the calling conversion. FILE is here a name of a configuration file used by the Res11Read application to detect details on what results shall be converted.

When FILE is used in connection with the option 'someresFILE', the format of the configuration file is as presented below:

```
ItemNumber   Chainage Rivername
    1          100      MAIN
    1          200      MAIN
```

*Fig B.1.1        Format of FILE used in connection with the 'someresFILE' option*

**Note:** Column Separators in the configuration file (FILE) as presented above must be blank space characters. It is not allowed to use tab-characters as separators.

ItemNumber can be extracted from the results-files by running Res11Read with the '-Items' options. This will export all items present in the result-file and their respective numbers.

Note that if results are extracted from a Rainfall Runoff simulation results file, the format of the FILE is slightly different.
Instead of Chainage and Rivername you must instead specify the Catchment-Area and Catchment-Name as parameters as presented below:

```
ItemNumber    Catchment_Area   Catchment_Name
    12            2300.000        CATCHMENT_A
    10            2300.000        CATCHMENT_A
```

*Fig B.1.2        Format of FILE when using the 'SomeRes' option and extracting results from a Rainfall-Runoff result-file*

**Note** that the file has a header line at the start.

An additional option is available to redirect individual selections to additional text files.

This can be done using the ">" character at the end of a line in FILE:

```
ItemNumber Chainage Rivername
    1        100     MAIN
    1        200     MAIN>main_chain200_output.txt
    1        300     MAIN>main_chain300_output.txt
    1        400     MAIN
    1        500     MAIN>main_chain500_output.txt
```

*Fig B.1.3        Format of FILE used for the 'someresFILE' option with alternate output file option*

Using the option 'CompareFILE' the format of the FILE is presented below:

```
ItemNumber  Limit      Rivername
    1        1e-6      Main 10000
    2        1e-6      Main 10000
```

*Fig B.1.4        Format of FILE using the 'CompareFILE' option*

The 'Limit' parameter defined in the FILE is the threshold for the comparison between files, such that comparison errors will not be issued if differences are smaller than the define Limit-value.

## B.1.3 Updating rating curves through the command prompt

The updating of the Q/H-relations for broadcrested weirs, culverts and link channels may be done using a separate executable. To execute the program, open a command prompt (on Windows XP the Dos Prompt can normally be found under; Start-> Programs -> Accessories -> Command Prompt). The syntax for running the application is:

...\M11QHCalcLauncher.exe simulationfilename.sim11

where

...\ denotes the full path to the application located in the bin directory of the MIKE Zero installation.

simulationfilename.sim11 is the simulation filename. If the calculation is successful then the user is prompted to acknowledge this. If any errors occur then the structure type along with the locations are reported in the file errorQHcalc.log which is located at the same location as the simulation file.

INDEX