MIKE 11

A modelling system for Rivers and Channels

User Guide

DHI Software 2000

Software development by : xxxxx

Written by: xxxx

E:\LRi2World\M11Manuals\UserManual\Cover.fm 11 September 2001 3:24 pm

Please Note

Copyright

Limited Liability

Printing History

September 2001
Edition 2001

MIKE 11

S I M U L A T I O N   E D I T O R

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 Mike11 editors. The editing of cross sections is a typical example of this link, where the graphical view of the network editor is used to select cross sections from the cross section editor. The linkage requires a file name to be specified for each of the required editors. The file names are input on the Input Property Page of the simulation editor. An alternative is to select a file from the File Menu which will recall the appropriate editor. The edit menu can then be used to edit the objects

1.1 Models

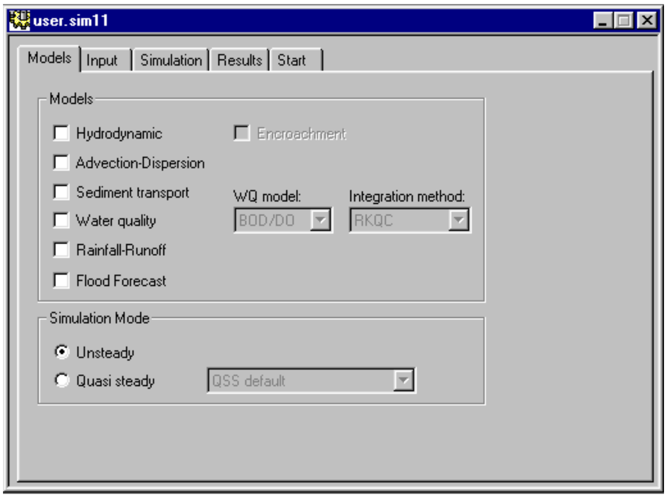


Figure 1.1 The Models tab. Note that the Simulation Mode Box may differ if a quasi two dimensional steady state solver with vegetation is not installed.

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 module names are used:

HD Hydrodynamic

AD Advection-Dispersion

ST Sediment Transport

WQ Water Quality

RR Rainfall-Runoff

FF Flood Forecast

When selecting a hydrodynamic model an additional tick box entitled Encroachment becomes active. When selecting the latter all other tick-boxes become inactive since the encroachment module 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.

In conjunction with selection of a water quality model two selection boxes become active:

- Selection of WQ model:
  - BOD/DO: Biological Oxygen Demand/Dissolved Oxygen.
  - EU: Eutrophication.
  - HM: Heavy Metal.
- Integration method.
  - RKQC: Fifth order Runge-Kutta with Quality Control.
  - RK4: Fourth order Runge-Kutta.
  - EULER: Euler or linear solution.

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 1.1. 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

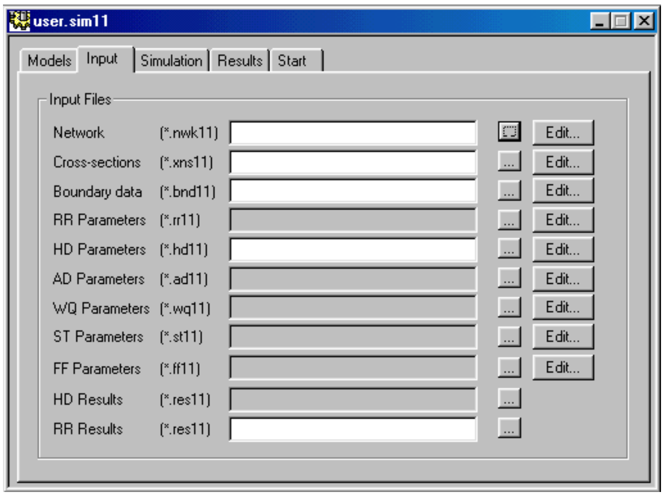
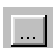


Figure 1.2 The Input tab.

Based on the model selection from the Models Property Page the user is required to specify a range of input file names.



This button opens a file selection box.

This button opens the relevant editor.

Note that the files required are indicated by the active fields. Two exceptions are the fields for the result files. A hydrodynamic result file is required if

- a stand alone Advection-Dispersion/Sediment transport simulation is to be carried out

or

- if lateral sources from a previous MIKE SHE/MIKE 11 coupled model run are to be included in a hydrodynamic simulation.

A Rainfall Runoff result file is only required if the hydrodynamic and rainfall model are to run uncoupled.

1.3 Simulation

The simulation property page contains details of simulation time and initial conditions for each of the chosen types of models.

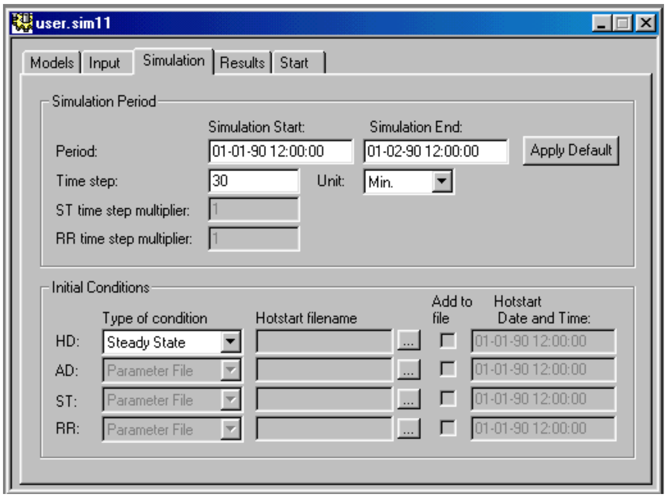


Figure 1.3 The Simulation tab.

1.3.1 Simulation Period

Period

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

Time Step

The simulation time step.

ST Time Step Multiplier

The ST module may not operate using the same time step as the HD model. The ST Time Step Multiplier specifies the ST time step as a multiple of the HD time step.

RR Time Step Multiplier

The RR module may not operate using the same time step as the HD model. The RR Time Step Multiplier specifies the RR time step as a multiple 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. 230)* Property Page of the Hydrodynamic Editor *(p. 223)* 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 the existing result file from which the initial conditions should be loaded.

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

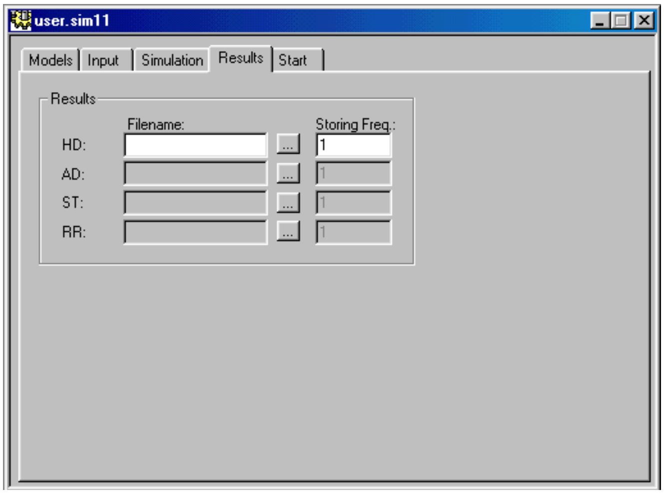


Figure 1.4 The Results tab.

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 Freq.

To limit the size of the result files the user can specify a save step interval. The storing frequency is the number of time step intervals between each saving of the results.

1.5 Start

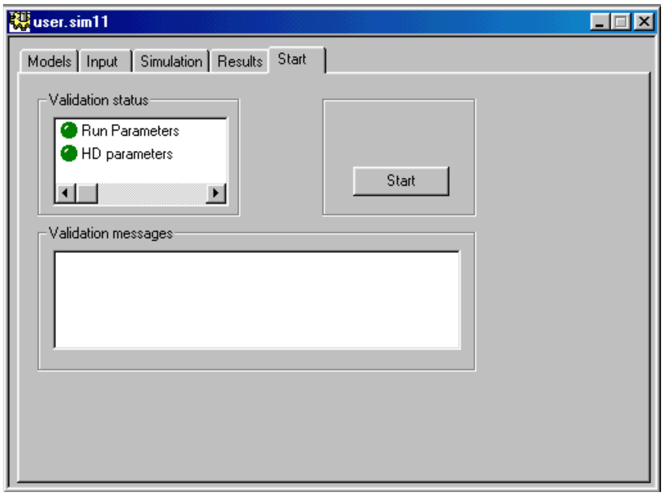


Figure 1.5 The Start tab.

If all specified input files exist, the “Start” button can be pressed and the simulation will commence. The simulation will take place as a separate process (MIKE11.EXE) and the progress of the simulation will be reported in a separate window.

Any error or warning message from the simulation will be saved in a file with the same name as the simulation file and a .log extension. If any errors or warnings are encountered during simulation the user is given the choice of viewing these at the end of the simulation.

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

R I V E R   N E T W O R K   E D I T O R

2 RIVER NETWORK EDITOR

The River Network Editor gives an overview of the current setup and provides a common link to the various MIKE 11 editors. The network editor has two main functions:

1 River network input and editing.

This includes:

– Digitising river networks and branch connections.

– Definition of hydraulic structures (weirs, culverts etc.).

– Definition of catchment inflow points (for rainfall run-off model).

2 Overview of all model information in the current simulation.

The editor provides an overview display in a graphical window. Settings for the graphical view are found in the Settings menu.

The current simulation setup is defined using the Simulation Editor *(p. 15)*.

NOTE: Cross sections are edited using the River Cross Section Editor *(p. 135)*, which is accessible from the River Network Editor.

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), 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. Additional graphical views 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) these include insert, edit and delete functions. Typically the Pop-Up Menu is used for editing of cross section geometry, parameters, hydraulic structures and data held in other MIKE 11 editors. Note that to access information from another editor other than the Network Editor, an editor file name must be specified using the Simulation File Editor.

Example of insertion of a Catchment link using the Pop-Up menu is shown in Figure 2.1

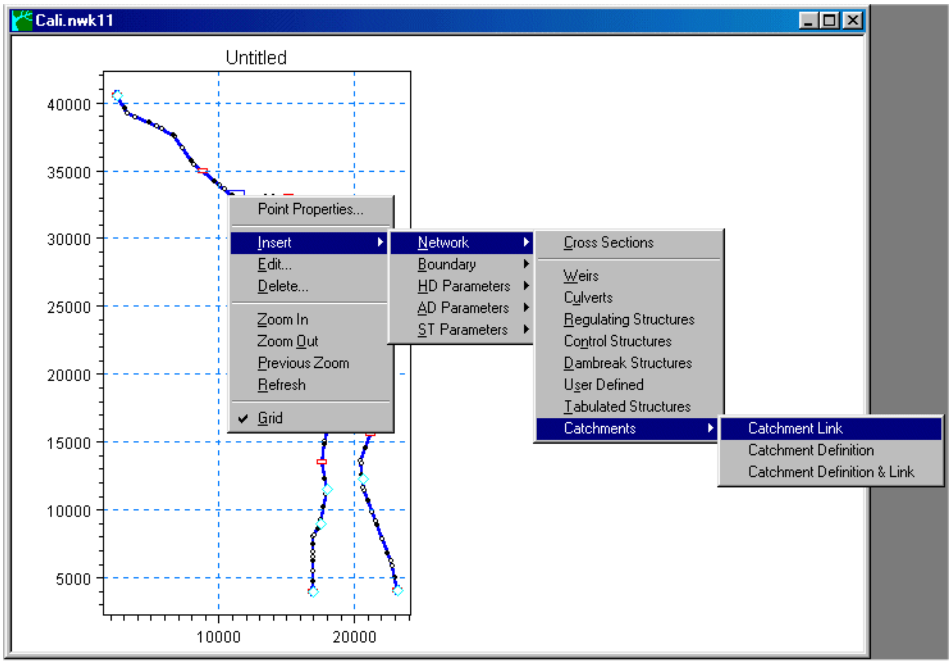


Figure 2.1 Illustration of right mouse pop up menu from where all data editors can be accessed.

2.1.1 File

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 the following format:

x-coordinate y-coordinate Branch_Name Chainage

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 2.2. 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 2.2 The format used for importing alignment line data.

2.1.2 View

Tabular view

Used when the tabular view of the network file must be shown.

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.

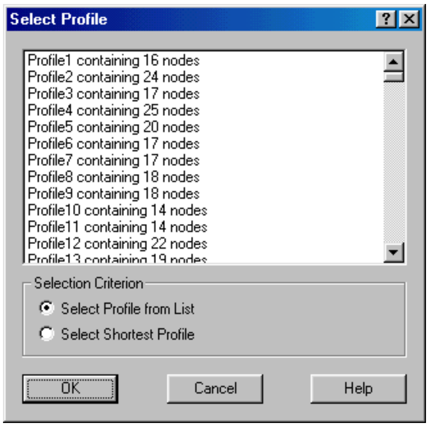


Figure 2.3 Menu used to select between several possible longitudinal profiles.

Network

Here the presentations of the different network objects can quickly be turned on or off. For a more detailed layout of the graphical view see Graphics *(p. 31)*.

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. 31)*.

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. 31)*.

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. 31)*.

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. 31)*.

Draw Grid

The drawing of the grid can be switched on and off by using this option.

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.

2.1.3 Network

Resize area

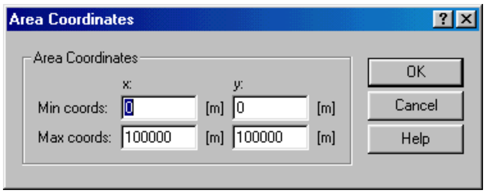


Figure 2.4 Menu for resizing the area of the graphical view.

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

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. The method used to connect the branches can be selected in Network data *(p. 33)*.

Disconnect All Branches

Choosing this options will remove all branch connections.

Auto Boundary (..) Free Branch Ends

This feature will create boundaries in the boundary file for all free branch ends. It will be done for the HD module, the AD module or the ST module depending on the selections in Network data *(p. 33)*.

Auto Update Chainages

When this option is selected the chainages of the points will be updated automatically.

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

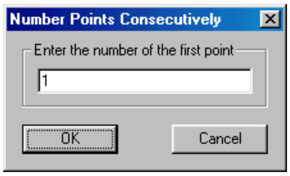


Figure 2.5 The menu in which the number of the first point can be entered.

When joining two network files (see B.1.1 Merging .pfs files *(p. 381)*) 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 in one of the network files.

2.1.4 Layers

Import

It is possible to import background maps into the graphical view of the network file. The following file types can be used: bmp, gif87a and emf.

2.1.5 Settings

Network

The network settings dialog contains the following property pages:

- Graphics
- Mouse
- Network data
- Select and edit

Graphics

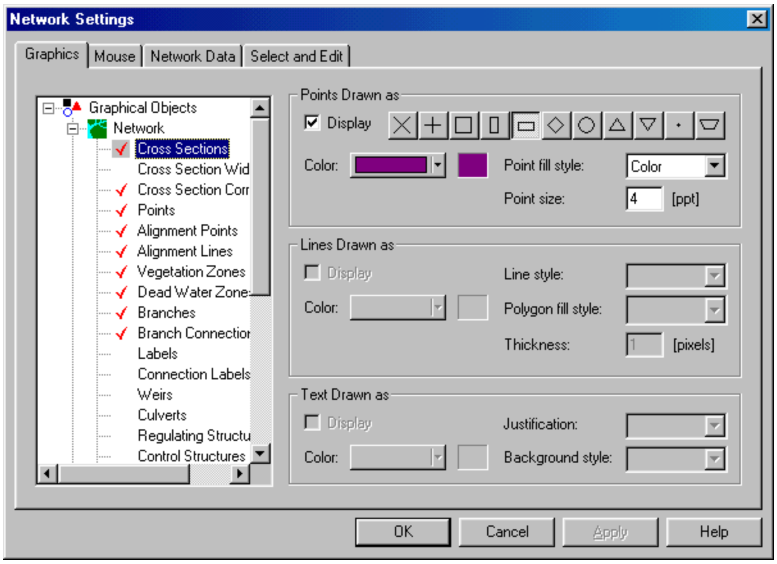


Figure 2.6 The Graphics property page.

This property page controls the layout of the graphics.

On the left hand side, the dialog shows the items organized in a tree structure. Each graphical item has branches for points, lines, labels etc. By selecting a branch it's settings can be changed in the right hand side of the dialog.

It is also possible to control if items are displayed or not by using the right mouse button on a branch. This can be done on different levels in the tree.

Mouse

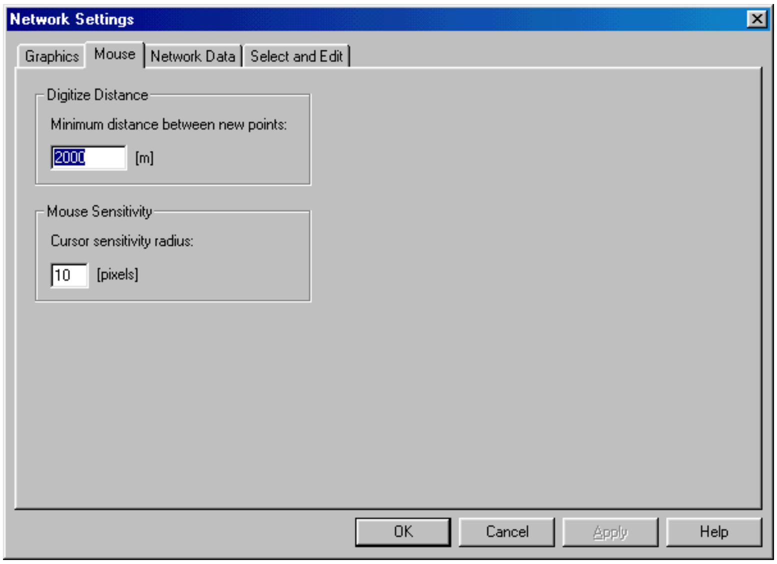


Figure 2.7 The Mouse property page.

This property page sets the properties for the mouse. This minimum distance for which a new point is generated when digitizing is set by using the 'Digitize Distance' field. The radius in pixels for which the mouse detects points can be set in the 'Mouse Sensitivity' field.

Network data

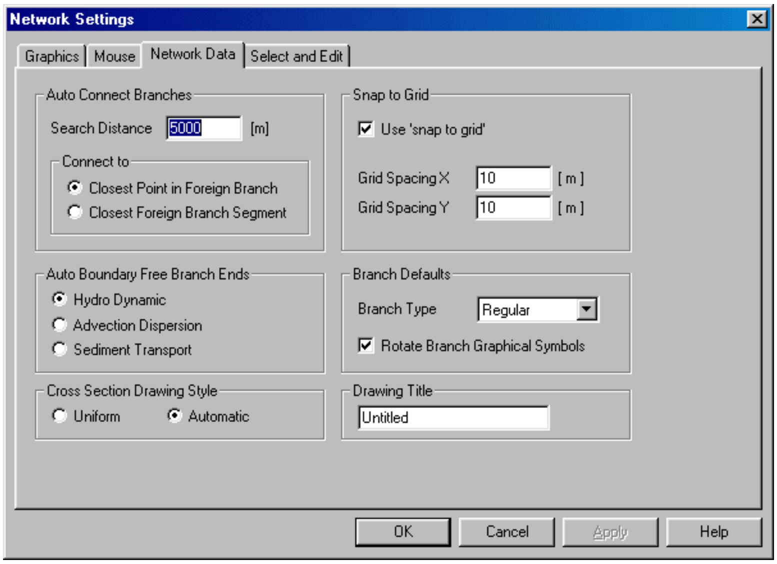


Figure 2.8 The Network Data property page.

– 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 facility Auto Boundary Free Branch Ends can generate boundary conditions for the HD, AD or ST models. The desired models are selected here.

– 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 default type of branch is set here. The user can chose between Regular, Link Channel and Routing.

The Rotate Branch Graphical Symbols checkbox enables the rotating of graphical symbols such as triangles, rectangles etc. on the plan plot. Without the activation of this checkbox symbols are always oriented 'north-south' but if the feature is enables, the symbols will be oriented towards the direction of the river branches, see Figure 2.9

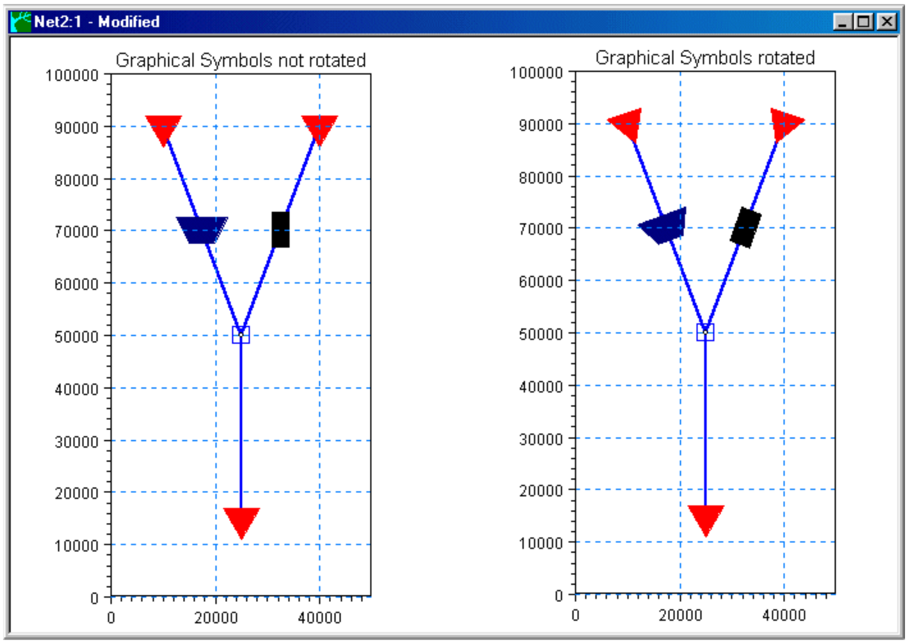


Figure 2.9 Illustration of 'Rotate Symbols' feature in MIKE 11 Network Editor

– Cross section chainage correction

This switch may be used if chainage corrections should be drawn.

Select and edit

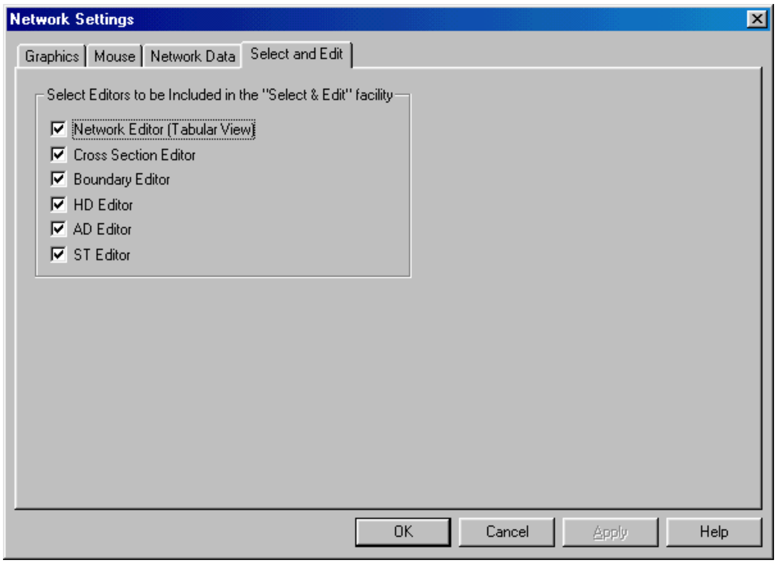


Figure 2.10 The Select and Edit property page.

Here the user specifies which editors are to be included when using the Select & Edit tool.

Font

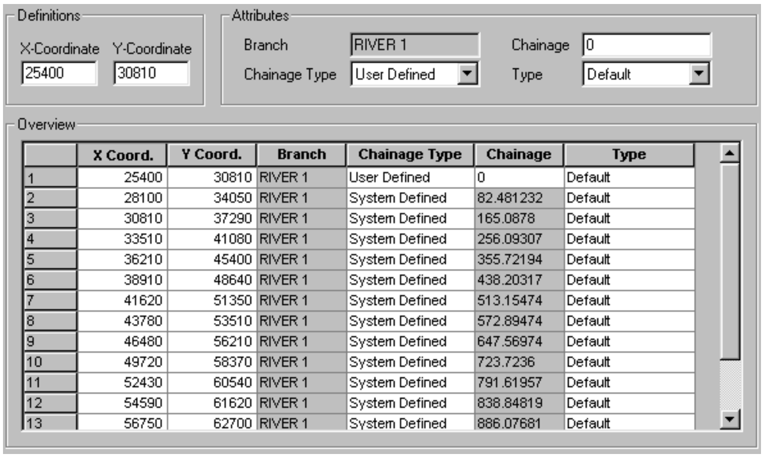
Here it is possible to select the font used in the graphical view.

2.2 Tabular view: Network

The tabular view gives an overview of branches, structures, rainfall catchments etc.

2.2.1 Points

The position of the points in the network may edited here. The dialog is shown in Figure 2.11.



	X Coord.	Y Coord.	Branch	Chainage Type	Chainage	Type
1	25400	30810	RIVER 1	User Defined	0	Default
2	28100	34050	RIVER 1	System Defined	82.481232	Default
3	30810	37290	RIVER 1	System Defined	165.0878	Default
4	33510	41080	RIVER 1	System Defined	256.09307	Default
5	36210	45400	RIVER 1	System Defined	355.72194	Default
6	38910	48640	RIVER 1	System Defined	438.20317	Default
7	41620	51350	RIVER 1	System Defined	513.15474	Default
8	43780	53510	RIVER 1	System Defined	572.89474	Default
9	46480	56210	RIVER 1	System Defined	647.56974	Default
10	49720	58370	RIVER 1	System Defined	723.7236	Default
11	52430	60540	RIVER 1	System Defined	791.61957	Default
12	54590	61620	RIVER 1	System Defined	838.84819	Default
13	56750	62700	RIVER 1	System Defined	886.07681	Default

Figure 2.11 The points property page.

Definitions

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

Attributes

Different attributes are available for editing.

Chainage type

The chainage may either be chosen as user defined or system defined.

Chainage

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

Type

The type of the point may be set here. Three types are available:

1. Default: The point is neither an *h*- or a *Q*-point.
2. Forced *h*-point: The point is used as an *h*-point.
3. Forced *Q*-point: The point is used as an *Q*-point.

Branch

This displays the river branch to which the present point belongs and is only for verification purposes.

Overview

An overview of the points is given in this box.

2.2.2 Branches

Branches and points can be inserted or deleted from existing branches using the Graphical Editing Toolbar.

Alternatively the branch dialog may be used (see Figure 2.12).

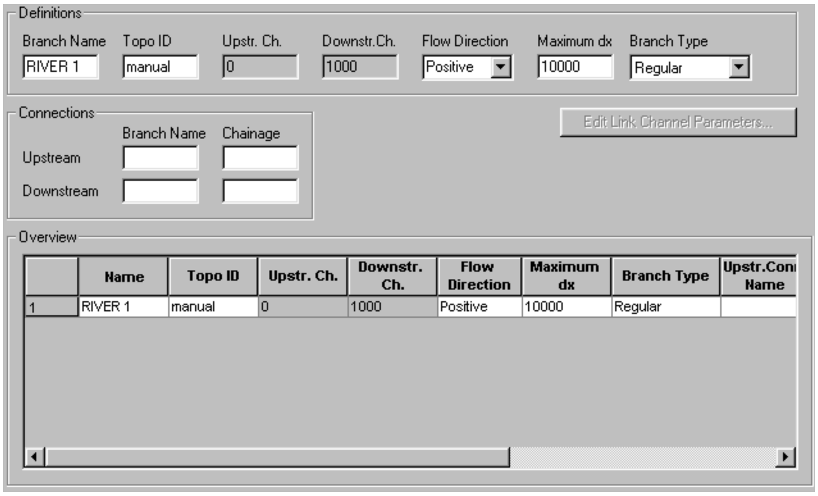


Figure 2.12 The branch property page.

Definitions

Branch name

Name of the branch.

Topo ID

Topo ID.

Upstr. Ch.

The chainage of the first point in the branch.

Downstr. Ch.

The chainage of the last point in the branch.

Flow Direction

If specified as positive, 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.

Maximum dx

Maximum distance between to adjacent h-points. See Tabular View: Grid Points *(p. 124)*

Branch Type

- Regular: 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 Edit Link Channel Parameters button.
- Routing: No cross sections are required. Only the flow is calculated, no water levels. See section 2.4 Tabular view: Routing *(p. 108)*.
- 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: If stratified flow is to be included in the simulation. The branches for which this vertical resolution is required are to be specified as stratified.

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.

Edit Link Channel Parameters

This option is available when the branch type has been set to link channel.

Purpose

The link channel is a short branch used to connect a flood plain to the main river 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 weir branch and will contain only three computation points.

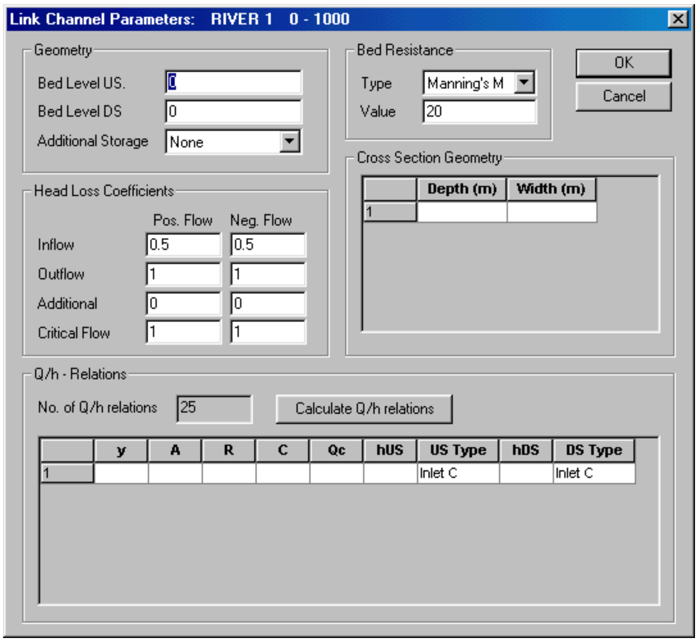


Figure 2.13 Link channel property page.

The link channel dialog (see Figure 2.13) is used for specifying all parameters appropriate for the link channel e.g. geometry, head los coefficients etc.

Geometry

The longitudinal 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 combo box defines if 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 on a cross section page.

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 a link channel. Both the depth and the width must be increasing.

Q/h - relations

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 are changed the *Q/h* relations must be re-calculated.

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 informartion 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 2.14 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.

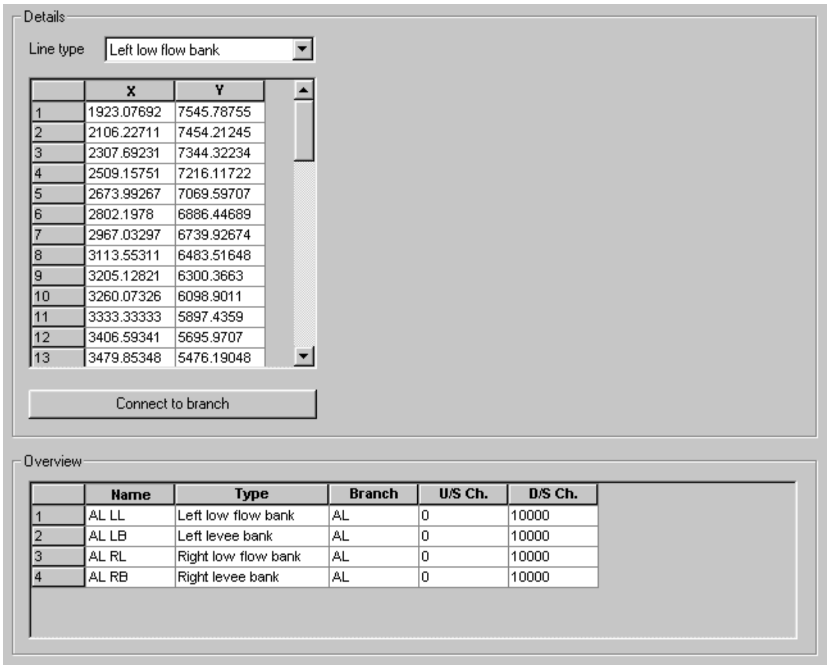


Figure 2.14   The alignment lines property page.

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 pairs 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.

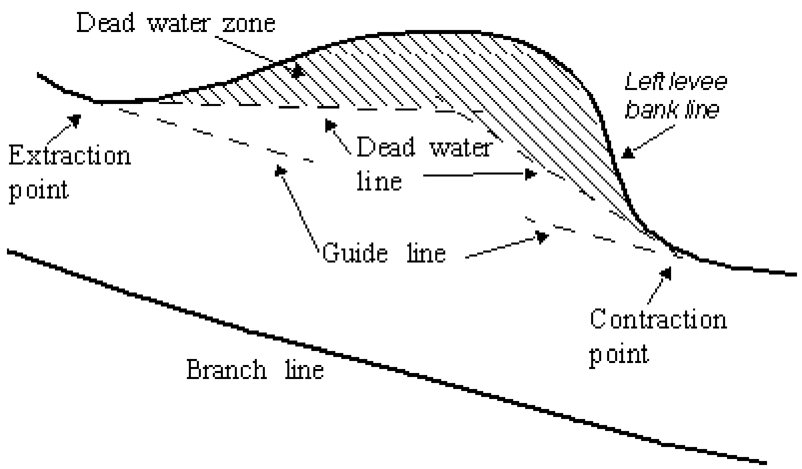


Figure 2.15 Definition of dead water zone along bank.

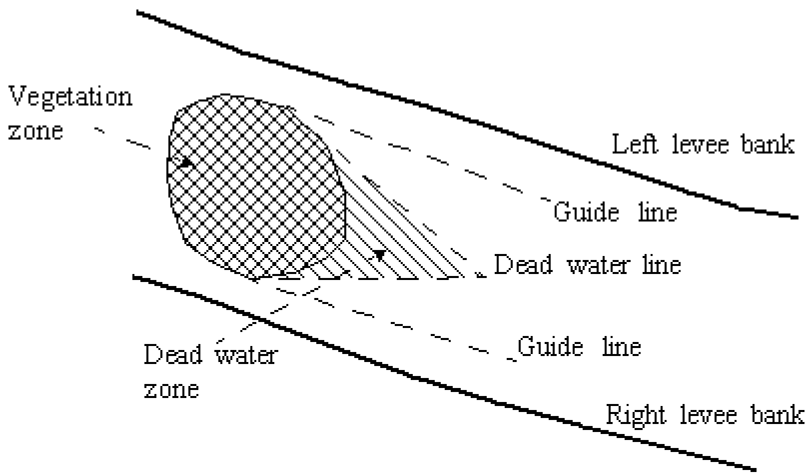


Figure 2.16 Definition of dead water zone behind vegetation zone.

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 2.7.2 Tool Bar for Alignment Lines *(p. 129)*). 2) Editing the numbers in the tabular view. 3) Using the File menu to import the coordinates from a text file.

Figure 2.17 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.

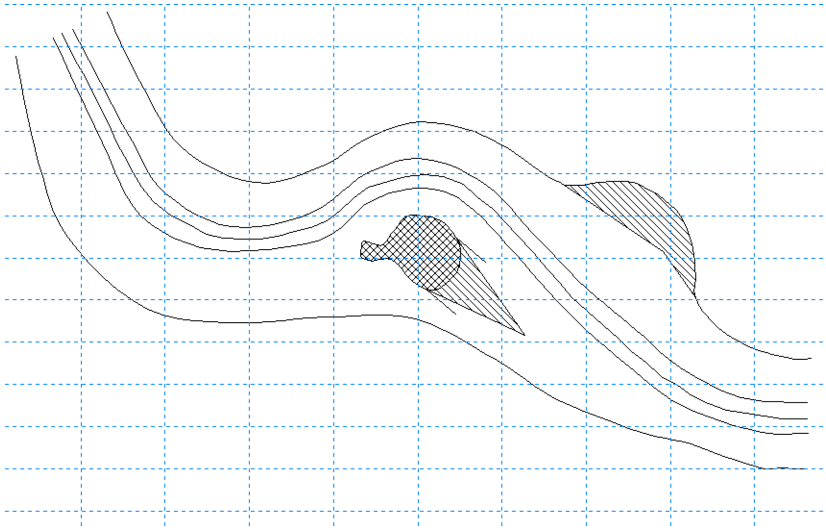


Figure 2.17 Example of a river network with alignment.

2.2.4 Junctions

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

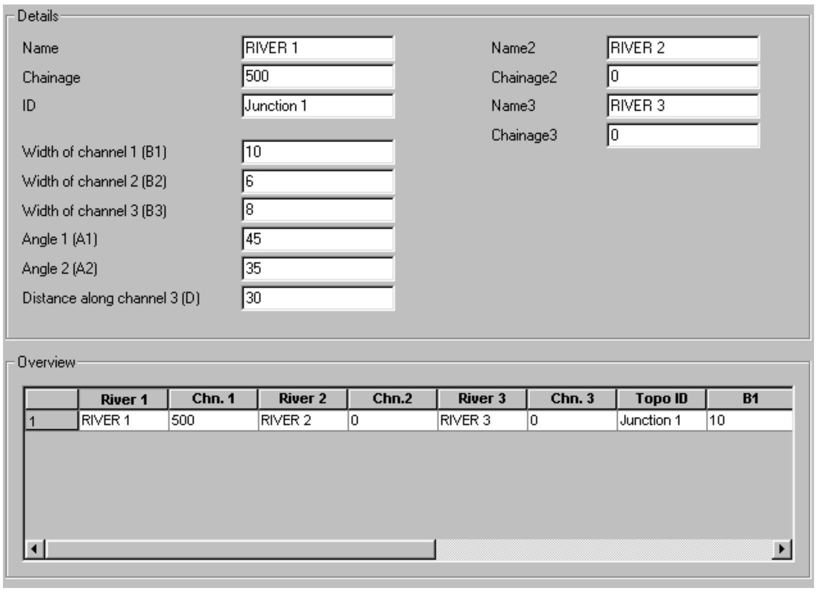


Figure 2.18 The Junctions dialog.

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

A number of structures such as weirs, culverts, bridges etc. may be implemented in a simulation. Structures are situated at *Q*-points. The flow over most of the structures is modelled using the energy equation so that local head losses can be incorporated. The effect of the bed-friction is not taken into account thus it is recommended that the *h*-points up- and downstream are situated close to the structure.

2.3.1 Weirs

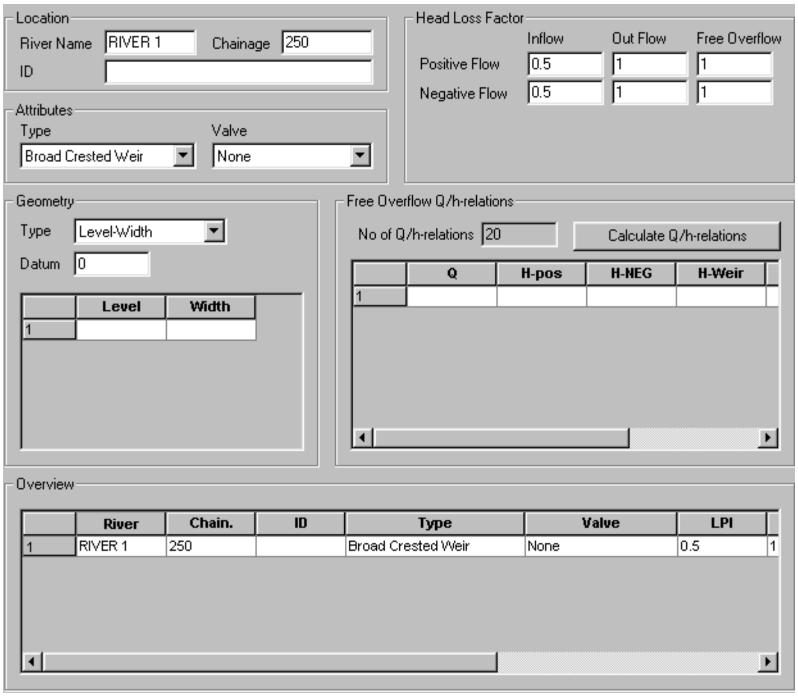


Figure 2.19 The weir property page.

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. It is used to identify the weir if there are multiple structures at the same location. It is recommended always to give the weir an ID.

Attributes

- **Type**:

  Broad Crested Weir: The calculation of *Q/h* relations assumes critical flow at the crest.

  Special Weir: The *Q/h* relationship table must be specified.

  Weir Formula 1: A standard weir expression is applied. See the Reference Manual.

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

– **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 broad crested weir and special weir).

Geometry

Only broad crested weir and special weir.

Type:

– **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. 37)*.

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

Level/Width: Weir shape defined as levels and corresponding flow widths. Values in the levels column must be increasing.

Weir formula Parameters (only weir formula 1)

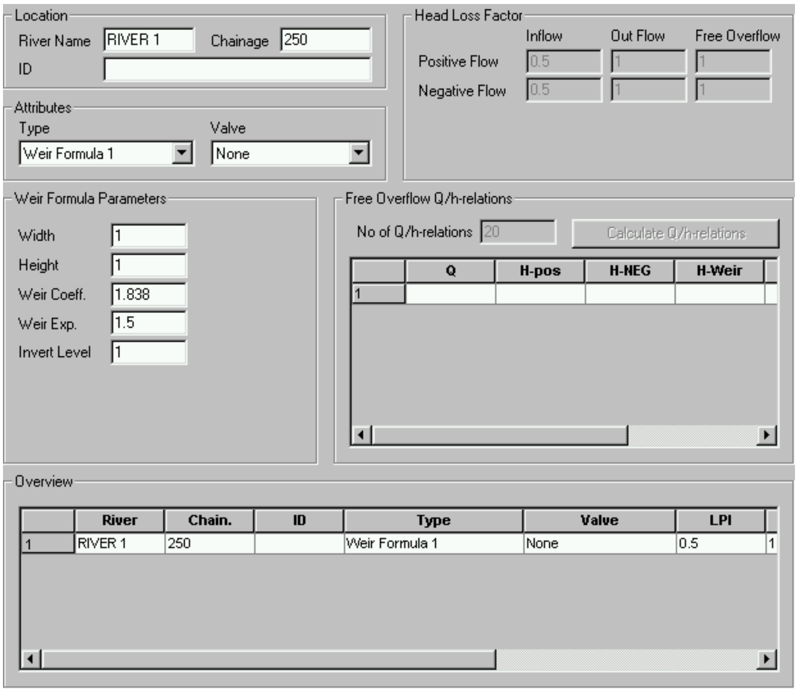


Figure 2.20 The weir property page, formula 1.

Width: Width of the flow.

Height: Weir height. See Figure 2.22

Weir Coeff.: Multiplication coefficient in the weir formula.

Weir Exp.: Exponential coefficient in the weir formula.

Invert Level: Bottom datum level. See Figure 2.22

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

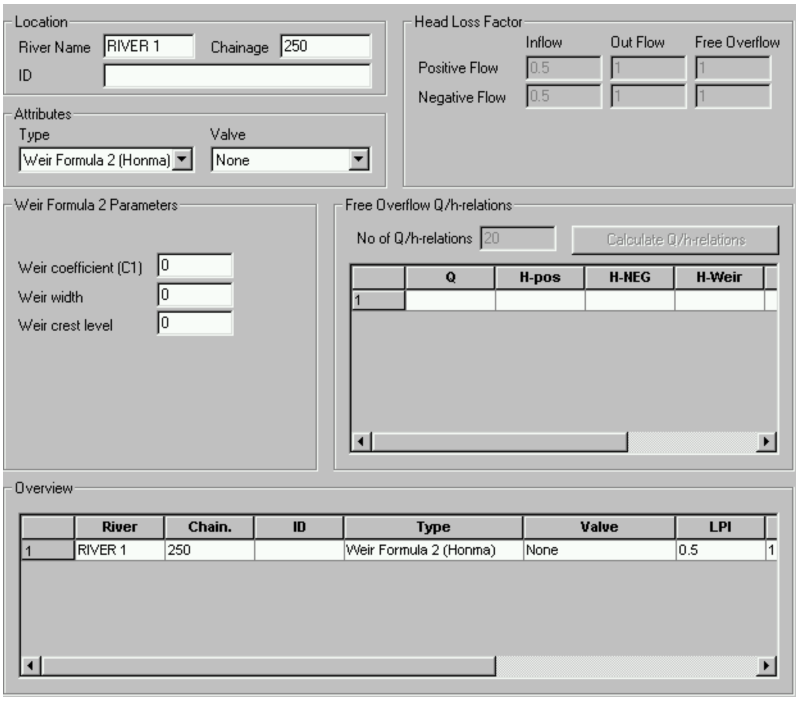


Figure 2.21 The weir property page, formula 2.

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

Weir width: Width of the flow.

Weir crest level: Weir level. See Figure 2.22

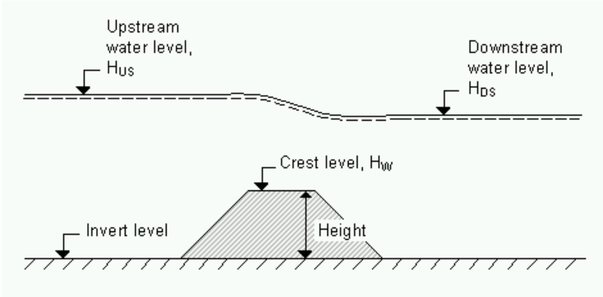


Figure 2.22 Definition sketch for Weir formula.

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. 38)*) 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.
- **Special weir:** Unlike a broad crested weir, the user must specify *Q/h* relations corresponding to free overflow conditions. These must be specified for both positive and negative flows.

Note that *Q/h* relations must be recalculated if any changes are made to the weir or if the cross sections up- or downstream have been altered. Further note that since a weir in MIKE 11 is defined as a structure causing a contraction loss and subsequently an expansion loss some constrains 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!

2.3.2 Culverts

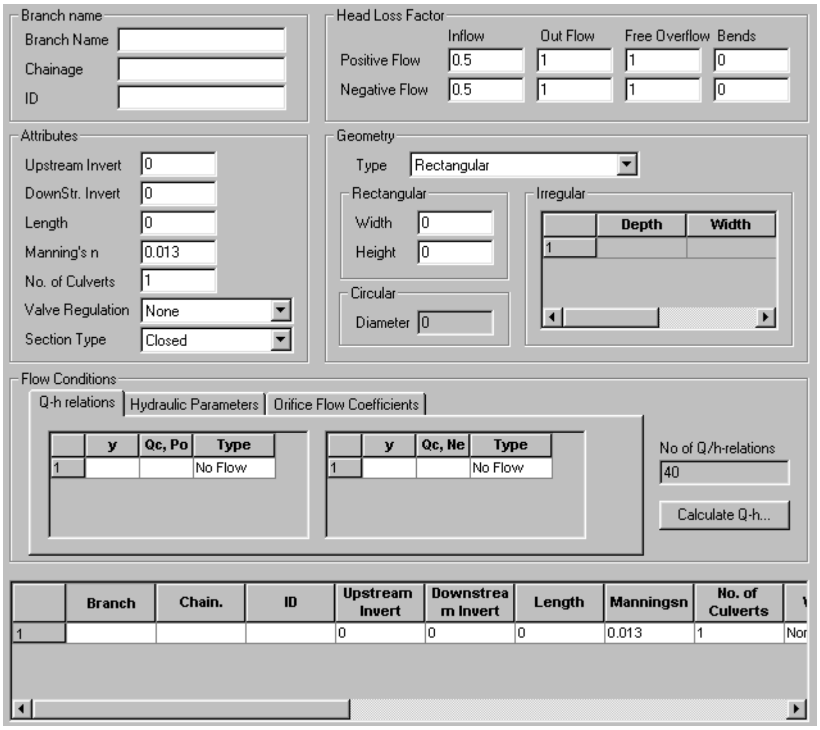


Figure 2.23 Culvert editor page.

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.

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.

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.

Head Loss Factors

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

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.

– **Irregular Depth-Width Table**: The geometry is specified using a depth/width table. Values in the width 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. 37)*.

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. 38)*) 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. 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 upstream of the culvert for all water levels. Similarly the cross sectional area at the outflow end must be less than the cross sectional immediately downstream of the culvert.

2.3.3 Bridges

Eight types of bridges may be implemented:

1 FHWA WSPRO bridge method.

2 USBPR bridge method.

3 Fully submerged bridge.

4 Arch Bridge (Biery and Delleur).

5 Arch Bridge (Hydraulic Research (HR)).

6 Bridge piers (D'Aubuisson's formula).

7 Bridge piers (Nagler).

8 Bridge piers (Yarnell).

It is possible to combine the free surface bridge flow with submerged and overflow solutions - except for Fully submerged bridge and Bridge piers (D'Aubuisson's formula).

Also note that the use of the two bridge types Fully submerged bridge and Bridge piers (D'Aubuisson's formula) requires the installation of a separate module.

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 **trouble shooting**, consult the reference manual 'Trouble shooting for bridge structures'.

Submergence methods:

1 FHWA WSPRO.

2 MIKE11 Culvert.

Overflow methods:

1 FHWA WSPRO.

2 MIKE11 weir.

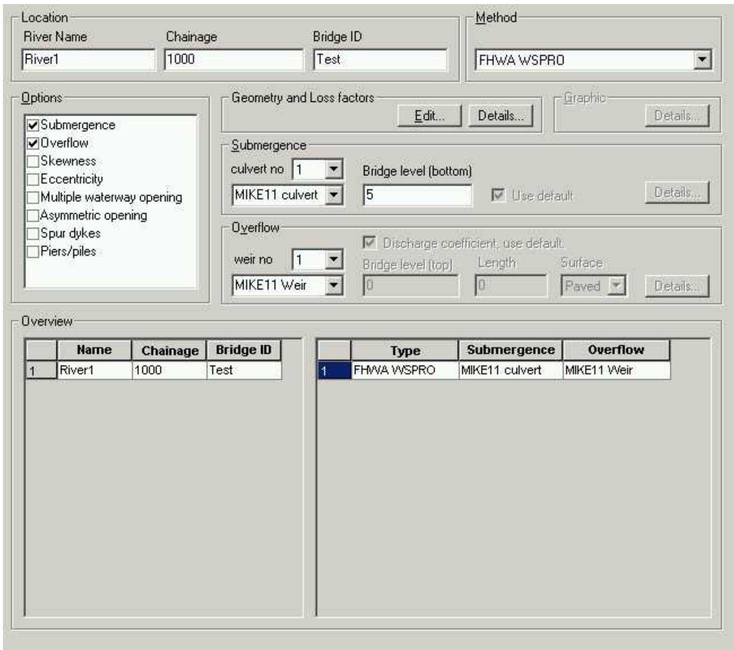


Figure 2.24 Bridge structure property page.

- **Name**: The river name.
- **Chainage**: The location on the river (not in a point where a cross section is specified).
- **Bridge ID**: String identification of the bridge.
- **Method**: Type of bridge.
- **Options**: Checkboxes for options available for the chosen method.
- **Geometry and Loss factors**: Edit button for entering geometric and loss factor options. Detail button for entering loss coefficient tables.
- **Graphic**: Not available in present release.
- **Submergence**: (Pressure flow) Available if Submergence checkbox is marked (See Options). The user must select method, FHWA WSPRO or MIKE 11 culvert. For details on FHWA WSPRO see Submergence, FHWA WSPRO.
- Culvert no: When choosing MIKE 11 culvert details of the culvert structure are in the culvert menu (See section 2.3.2). Culvert no is chosen as the number marked in the overview box in the culvert menu.

– Bridge level (bottom): Vertical level for the bottom of the girders.

– **Overflow**: Available if Overflow and Submergence checkbox is marked (See options). Selection of FHWA WSPRO or MIKE 11 weir method. For details on FHWA WSPRO see Overflow, FHWA WSPRO.

– Weir no: When choosing MIKE 11 weir details of the weir structure are in the weir menu (See section 2.3.1). Weir no is chosen as the number marked in the overview box in the weir menu.

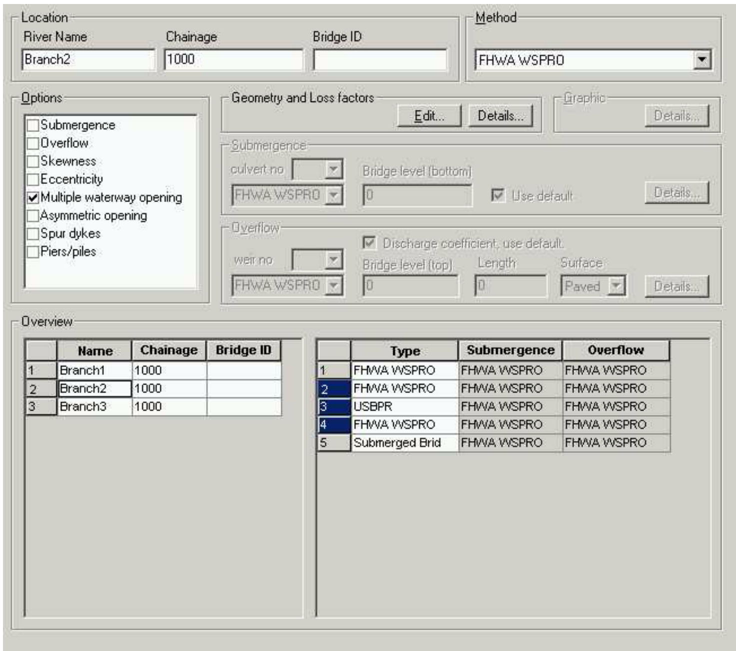


Figure 2.25 Overview, Branch2 has three bridge openings - 2, 3 and 4. Marked in the right part of the overview window.

– **Overview**: Left part show River name, Chaninage 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 2.25). In order to ad additional openings, mark a row in the right part of the overview window and press insert on the keyboard.

Working with Loss factor tables:

Loss / adjustment factor tables are viewed by pressing the Details buttons. 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 more lines.

FHWA WSPRO & USPBR Bridge

The FHWA WSPRO and the USPBR methods describe free surface flow through a bridge opening. The methods use the up and down stream cross sections inserted in the cross section editor. It is recommended that the distance between the bridge and the cross sections are one opening width (see Figure 2.26).

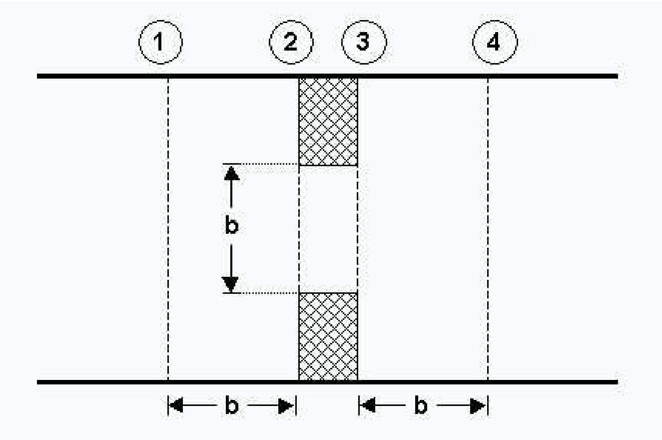


Figure 2.26 Location of up- and downstream cross section. 1: Upstream river cross section. Defined in the cross section editor. 2: Upstream bridge cross section. Defined in the network editor, bridge geometry. 3: Downstream bridge cross section. Defined in the network editor, bridge geometry. 4: Downstream river cross section. Defined in the cross section editor.

Available options for FHWA WSPRO Bridge:

– Submergence

- Overflow
- Skewness, Used when the embankments is 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 is 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:

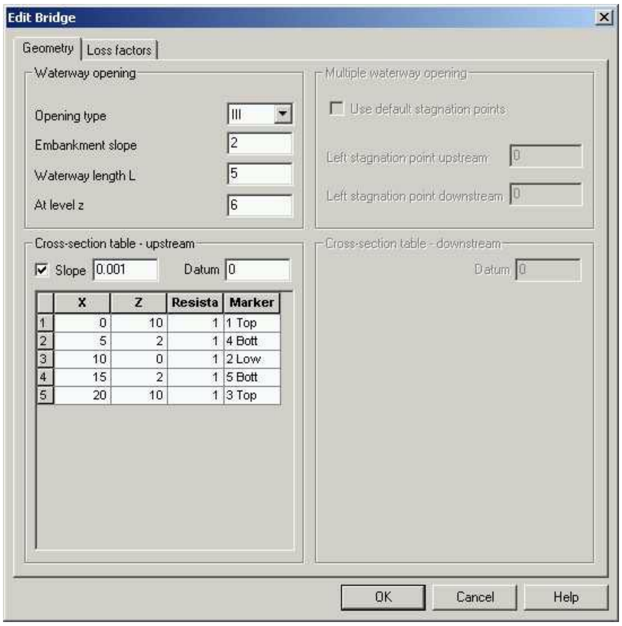


Figure 2.27 Geometry property page.

– **Opening type**: (see definition sketch Figure 2.28 - Figure 2.31). 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.

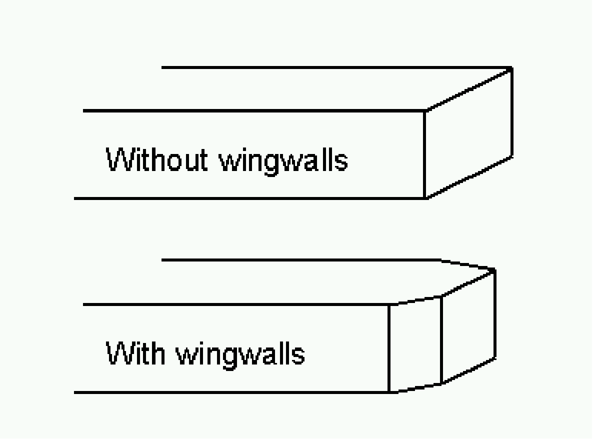


Figure 2.28 Definition sketch of type I opening, vertical embankments and vertical abutments, with or without wingwalls (after Matthai).

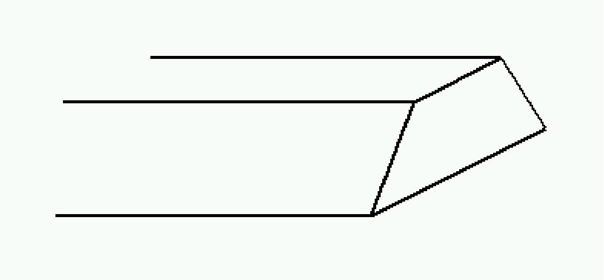


Figure 2.29 Definition sketch of type II opening, sloping embankments without wingwalls (after Matthai).

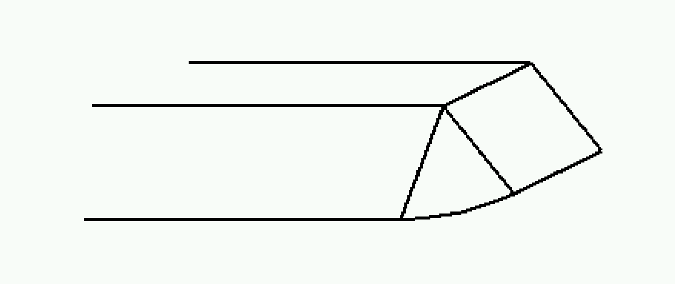


Figure 2.30 Definition sketch of type III opening, sloping embankments and sloping abutments (spillthrough) (after Matthai).

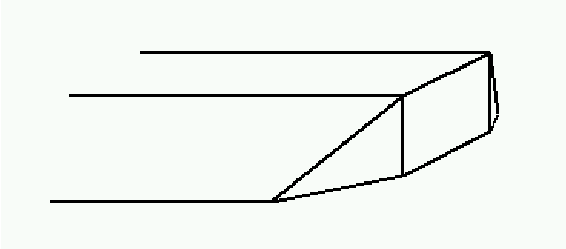


Figure 2.31 Definition sketch of type IV opening, sloping embankments and vertical abutments with wingwalls (after Matthai).

Geometry, Cross-section table:

- **Slope**: If the slope check box is marked the only the upstream bridge cross section must be defined. The downstream cross section is generated be copying the upstream cross section and adding 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 2.26).
- **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 2.26) are placed correct in respect to the x-values.
- **Z**: Vertical level of the cross section point.
- **Resistance**: Additional resistance in the cross section point. 1 is resistance corresponding to the manning number.
- **Marker**: Define the abutments (See Figure 2.32).

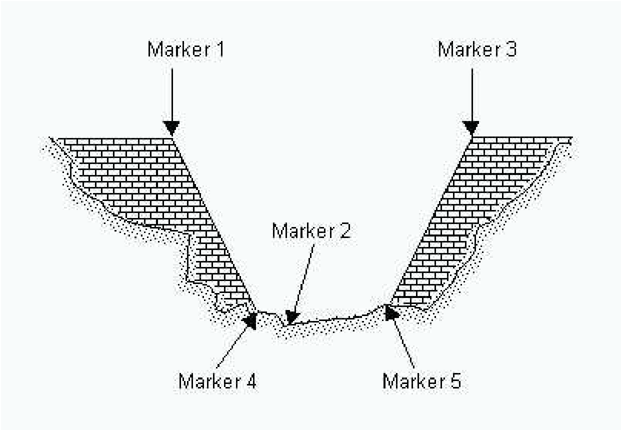


Figure 2.32 Definition of the bridge cross section markers.

Geometry, Multiple waterway opening:

Geometry and loss factors are defined for each opening when working with multiple waterway openings (see Figure 2.25). 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 2.33)

– **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 2.26. 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 2.26. 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.

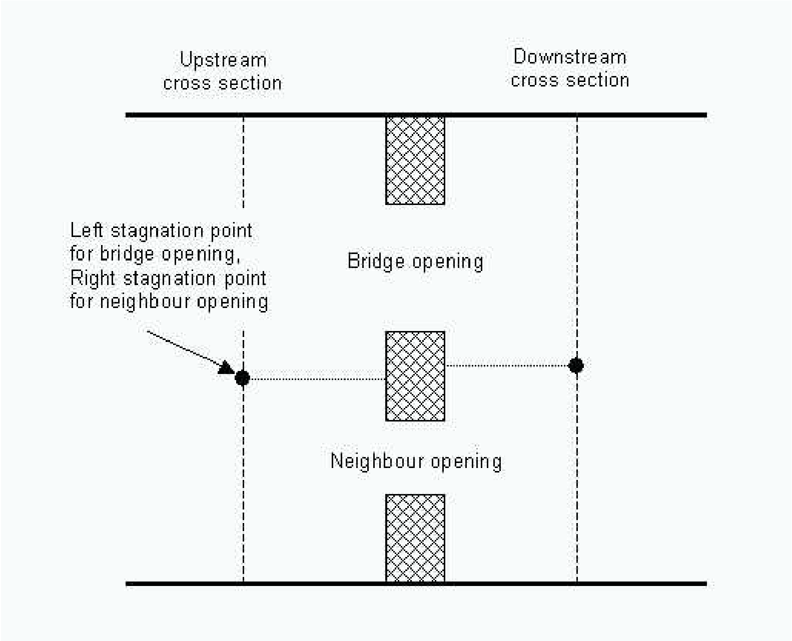


Figure 2.33 Multiple openings and stagnation points.

Loss factor:

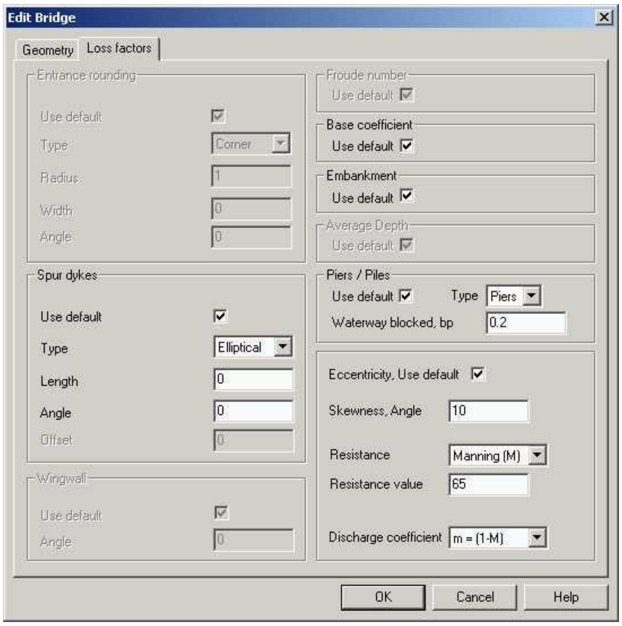
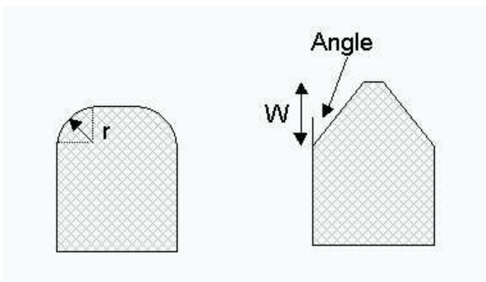


Figure 2.34 Loss factor property page

– **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, a default loss factor table will be generated. For Straight spur dykes the user must enter length and offset from the bridge opening. For Elliptical spur dykes the user must enter length and angle.

- **Wingwall**: Loss factor for FHWA WSPRO, opening type IV. When, use default, 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, 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, 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, a default loss factor table will be generated.
- **Average Depth**: Loss factor for FHWA WSPRO, opening type II. When, use default, a default loss factor table will be generated.
- **Velocity distribution coefficient**: Loss factor for the USBPR method. When, use default, a default loss factor table will be generated.
- **Piers / Piles**: Loss factor when 'piers / piles' is marked in options. When, use default, a default loss factor table will be generated. Choose Type (piers or piles) and enter the proportion of 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, 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 M or n as the unit for resistance.
- **Resistance value**: The value for resistance on the bridge structure between markers 1 and 4 and between 5 and 3 (see Figure 2.32). 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.

Loss factor tables for FHWA WSPRO:

Table 2.1

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_θ	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 as axis 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.

Loss factor tables for USBPR:

Table 2.2

Table		**Function of:**	
Base coefficient	k_b^*	M or m	
Velocity distribution coefficient	α_2	M or m	α_1
Eccentricity	e	M or m	Δk_e^*
Skewness	Δk_ϕ^*	M or m	
Piers	Δk_p^*	M or m	

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

Where:

M Bridge opening ratio.

m Channel contraction ratio.

e Degree of eccentricity.

α_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 since the bridge formula assumes that the bridge is fully submerged

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

Figure 2.35 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.

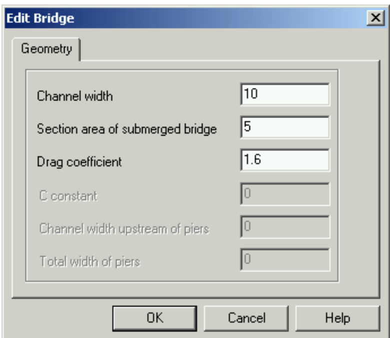


Figure 2.35 Submerged bridge geometry property page.

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 Mike11.ini file by setting the variable:

BRIDGE_FROUDE_CRITERIA.

Arch Bridge (Biery and Delleur) & (Hydraulic Research (HR))

The Arch Bridge methods describes free surface flow trough an arch bridge opening.

Available options for Arch Bridge:

– Submergence

– Overflow

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

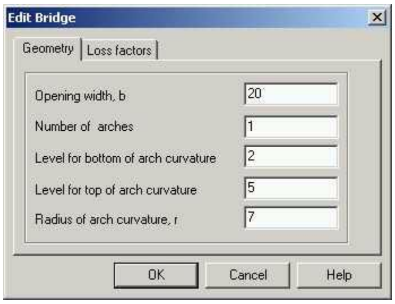


Figure 2.36 Arch Bridge Geometry property page.

Figure 2.37 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**.
- **Coefficient of discharge, Use default**: When, use default, a default loss factor table will be generated.
- **Opening/Contraction ratio**: Choose channel contraction ration m or bridge opening ratio M as parameter in the loss factor tables.

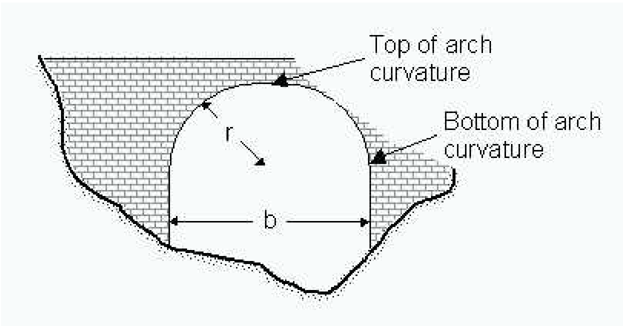


Figure 2.38 Hydraulic variables for Arch Bridge.

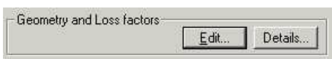
Loss factor tables for arch bridges:

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 2.39 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 constant 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.

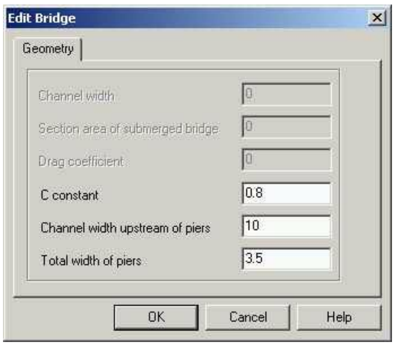


Figure 2.39 D’Aubuisson Bridge piers geometry property page

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.

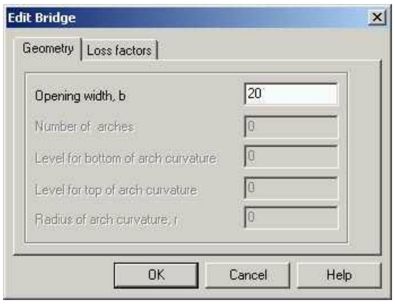


Figure 2.40 Nagler and Yarnell Bridge piers geometry property page.

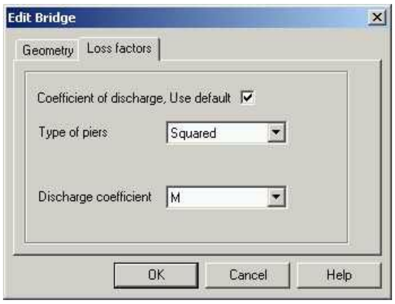


Figure 2.41 Nagler and Yarnell Bridge piers Loss factor property page.

– **Opening width, b**: The total opening width between the piers.
– **Coefficient of discharge, Use default**: When, use default, 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 θ and β 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.

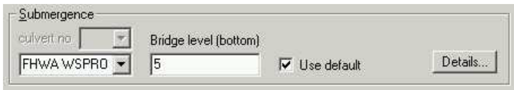


Figure 2.42 Submergence, FHWA WSPRO, property page.

- **Bridge level (bottom)**: Vertical level of the bottom of the girders.
- **Use default**: When use default 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 orificed flow is set to ON in the MIKE11.ini file. orificed flow is in general not recommended.

Overflow, FHWA WSPRO

The method describes weir flow 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 2.43 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 a default loss factor table will be generated.
- **Surface**: When Use default 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 .

2.3.4 Regulating

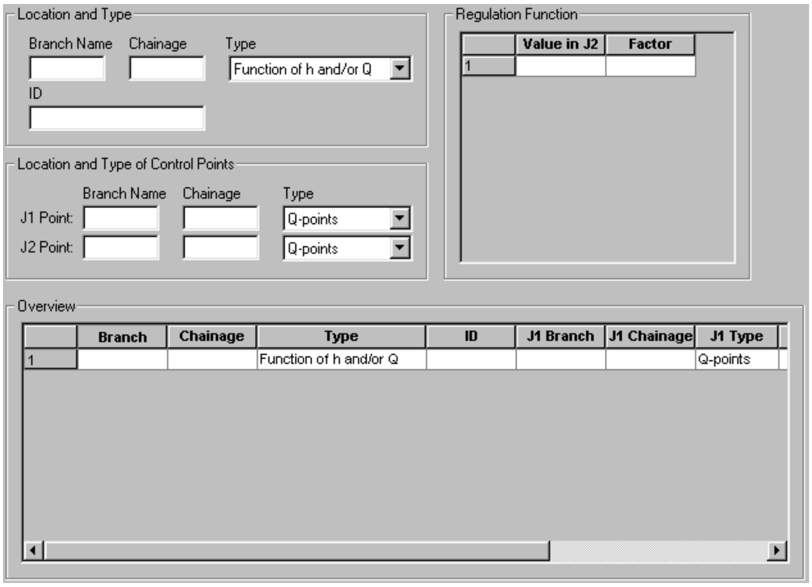


Figure 2.44 Regulating structure property page.

This structure type is 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 2.44) whose 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 weir is located.

Type:

– **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. 159)* using the Hydrodynamic *(p. 161)* tab.

– **Function of *h* and/or *Q*:** The discharge through the structure is defined as a function of h or Q at two locations (J1 and J2) 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.

2.3.5 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.

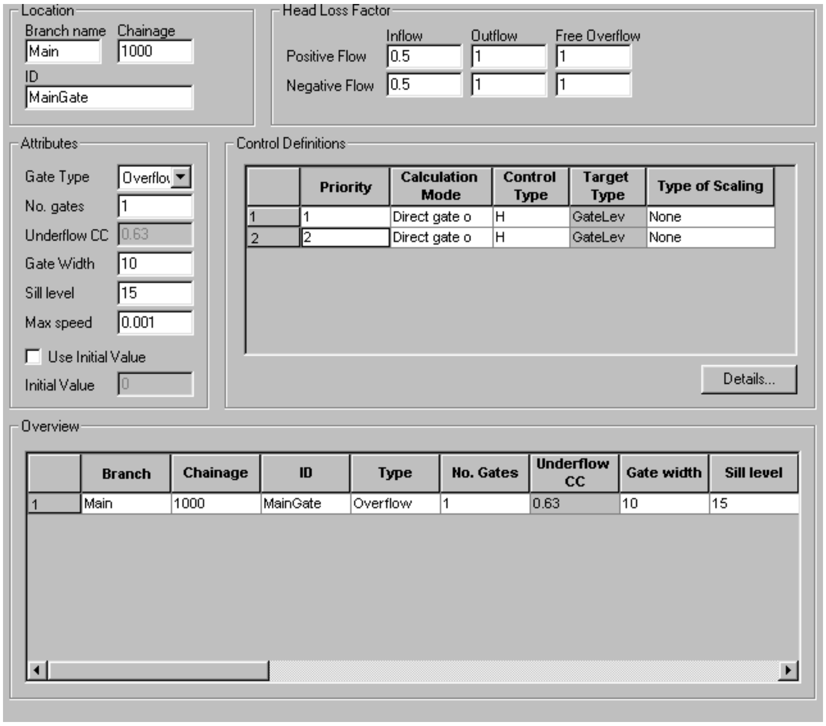


Figure 2.45 The control structure property page.

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.

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. 80)*.

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 in gate level. This is probably not realistic, further it can create instabilities in the computation.

Use initial value

If an initial value is requested this check box must be checked.

Initial Value

If the ‘Use initial value’ checkbox is checked the initial value must be written in this field.

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 2.46.

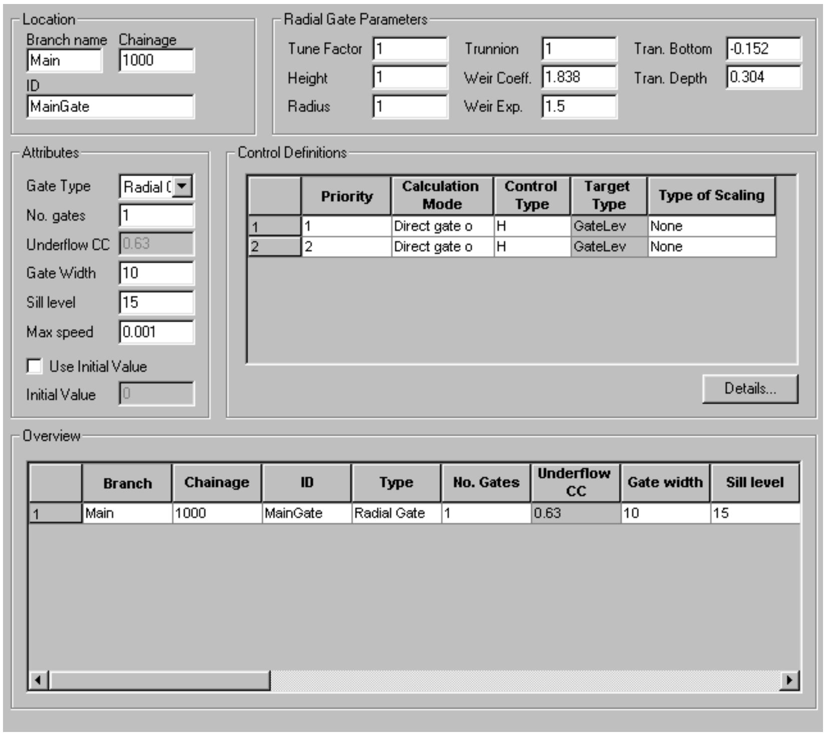


Figure 2.46 The control structure property page when a radial gate has been selected.

In Mike11 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.

Tune Factor

Discharge calibration factor. This factor is used only on the part of the discharge that flows below the gate.

Height

Height above sill of the overflow crest of the gate when the gate is closed, see Figure 2.47.

Radius

Radius of gate, see Figure 2.47.

Trunnion

Height above sill of centre of gate circle, see Figure 2.47.

Weir Coeff.

Coefficient used when calculating the flow above the gate.

Weir Exp.

Coefficient used when calculating the flow above the gate.

Tran. Bottom

Parameter used to define the transition zone between free flow and submerged flow. Corresponds to $y_{Tran,Bottom}$ defined in Hydraulic Aspects - Radial Gates in the reference manual.

Tran. Depth

Parameter used to define the transition zone between free flow and submerged flow. Corresponds to $y_{Tran,Depth}$ defined in Hydraulic Aspects - Radial Gates in the reference manual.

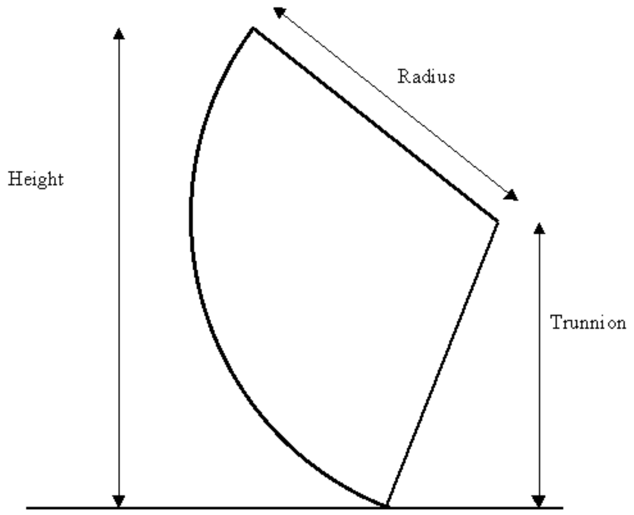


Figure 2.47 Definition of a radial gate.

Control definitions

The way the gate level is calculated is determined by 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.

From above it is realized that it takes two things to define a control strategy: The conditions that must be fulfilled for the strategy to be executed and 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). As 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 Mike11 there are different Calculation Modes. 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 Mike11 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 Mike11 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 more details must be defined. We start with a description of the main parameters. As seen in Figure 2.45 or Figure 2.46 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. 82)* it is possible to make Mike11 choose between an arbitrary number of control strategies. These control strategies are organized 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

- **Direct Gate Operation**: 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:

$$u^n = \alpha_1 K\left\{1 + \frac{T_s}{T_i} + \frac{T_d}{T_s}\right\}\{y^n_{ref} - y^n\} \tag{2.1}$$

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

where u^n is the gate level (or discharge in case of a discharge structure) at the n'th 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 n'th time step, y^n the actual value of the target point at the n'th time step, α_1, α_2, α_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 (2.1).

The variables K, T_d, T_i, α_1, α_2 and α_3 are entered by the user, see Iteration / PID *(p. 90)*. The rest is calculated by Mike11.

– **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. 82)* 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. 90)*.

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.

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'.

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. 86)* 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.

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 Mike11. There are four property pages: ‘Control Strategy’, ‘Control- and Target point’, ‘Iteration/PID’ and ‘Logical Operands’.

Control Strategy

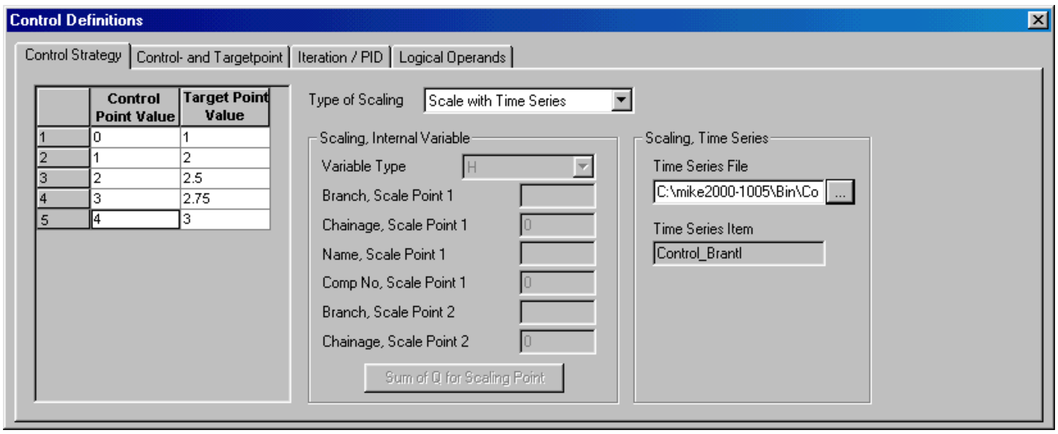


Figure 2.48 The Control Strategy property page.

Here the relationship between the value of the Control Point and the value of the Target Point are entered. This is done in the table on the left side of the property page.

Also the information about 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. 85)*. 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.

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. 95)*.

Control and Target Point

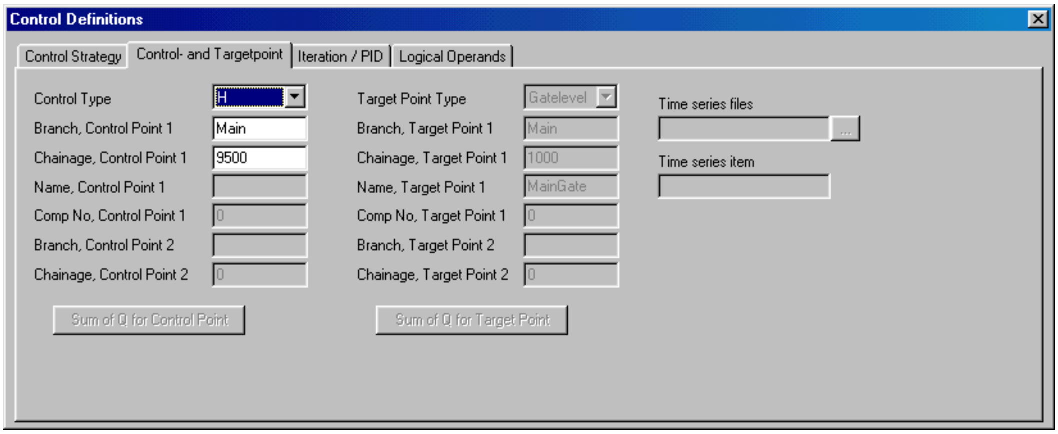


Figure 2.49   The Control- and Target point property page.

Control Type: Here the type of Control Point is chosen. This field is linked to the Control Type field described in Control Type *(p. 84)*.

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. 95)*.

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. 85)*.

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. 95)*.

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.

Iteration / PID

PID-Section: Here the necessary data is entered if the calculation mode is chosen as PID-operation.

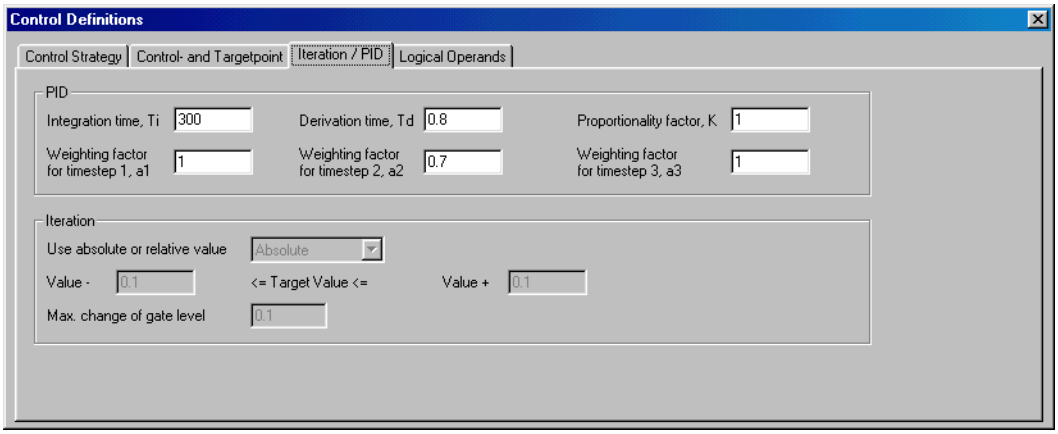


Figure 2.50 The Iteration / PID property page when calculation mode is chosen as PID-Operation.

Integration Time, Ti: Corresponds to T_i in eq. (2.1).

Derivation Time, Td: Corresponds to T_d in eq. (2.1).

Proportionality Factor, K: Corresponds to K in eq. (2.1).

Weighting factor for time step 1, a1: Corresponds to α_1 in eq.(2.1).

Weighting factor for time step 2, a2: Corresponds to α_2 in eq. (2.1).

Weighting factor for time step 3, a3: Corresponds to α_3 in eq. (2.1).

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. Mike11 use a criteria that can be expressed like:

$$TP_{Required} - Limit_{Low} \leq TP_{Act} \leq TP_{Required} + Limit_{High} \tag{2.2}$$

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.

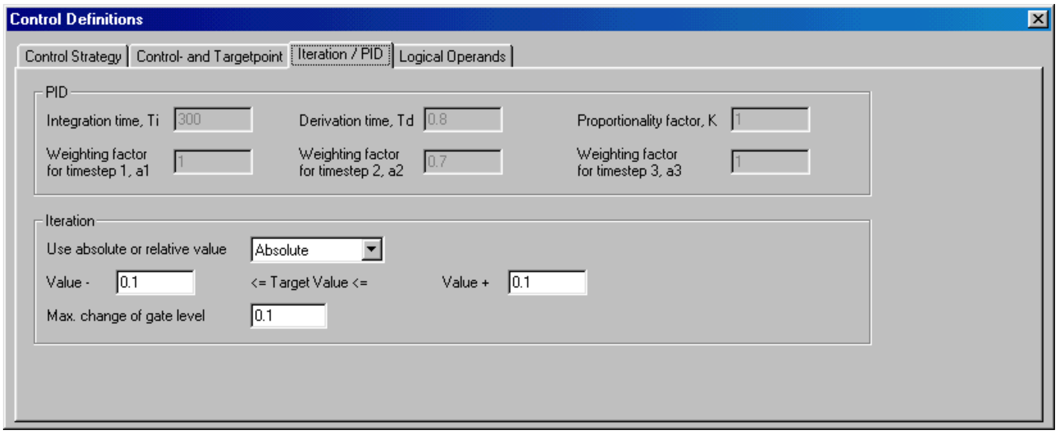


Figure 2.51 The Iteration / PID property page when calculation mode is chosen as Iterative Solution.

Value -: This field corresponds to $Limit_{Low}$ in eq. (2.2).

Value +: This field corresponds to $Limit_{High}$ in eq. (2.2).

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.

Logical Operands

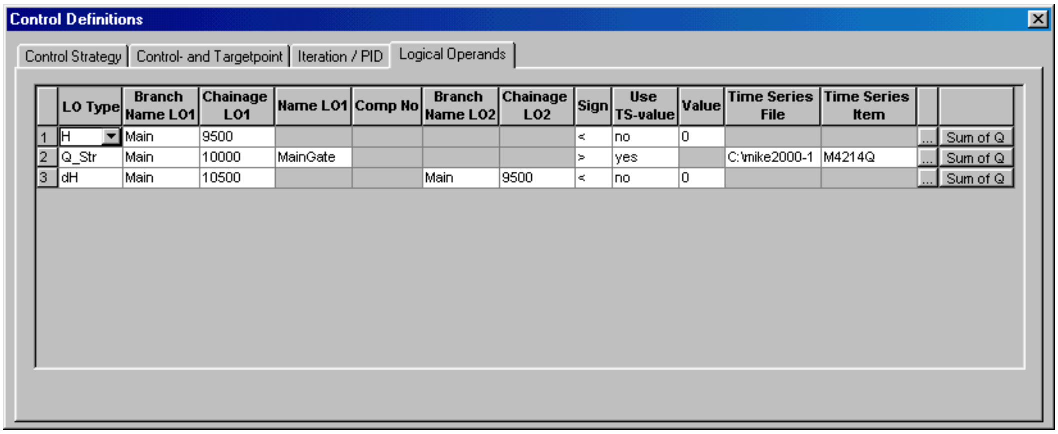


Figure 2.52 The Logical Operand property page.

As stated in Control definitions *(p. 82)* 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 Mike11 called ‘Logical Operands’.The logical operands are entered in the Logical Operand property page, see Figure 2.52. 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. 95)*.
- **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 Mike11 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.

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

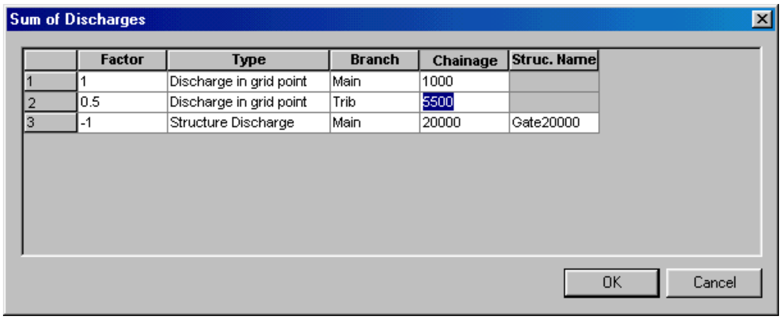


Figure 2.53 Input page to Sum of Discharges.

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:

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

n is the number of discharges to sum, fac_i the factor to be multiplied with the i'th discharge, Q_i.

Factor: This corresponds to fac_i in eq. (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.

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.

2.3.6 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 2.54.

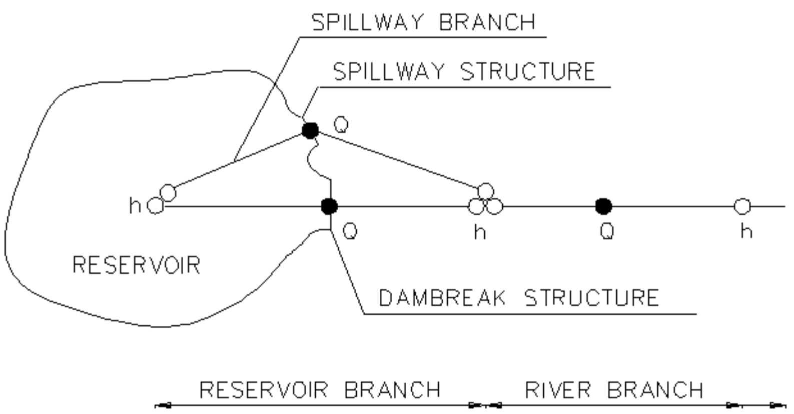


Figure 2.54 Typical setup for dambreak simulation

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 ΔX step used between the adjoining h-points is of no consequence. The maximum ΔX 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 2.54.

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

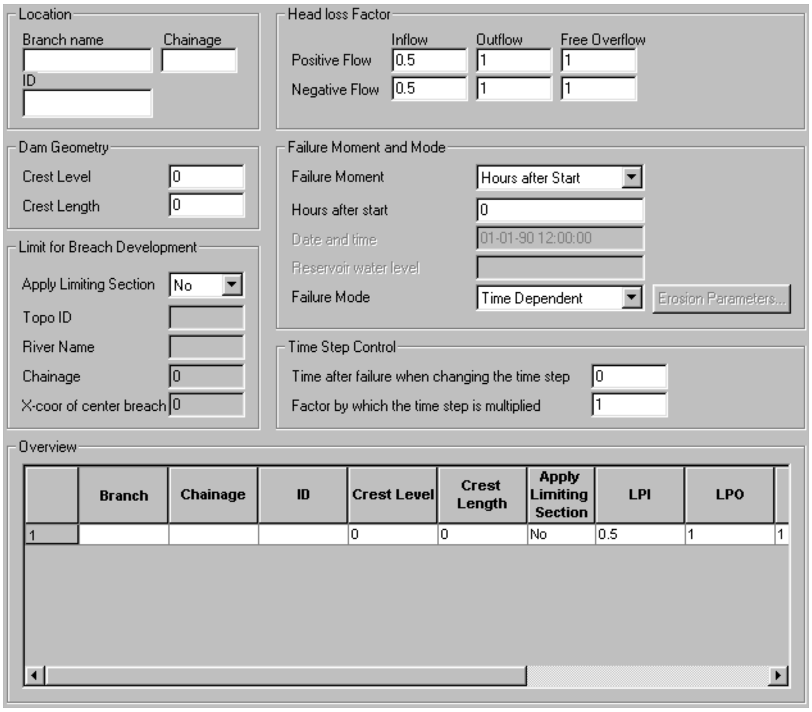


Figure 2.55 The Dambreak structure property page.

This Dambreak structure property page is used for inserting dambreak structures in a given network. The property page (see Figure 2.55) 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.

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

– **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. 135)*.

– **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.

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.

This specification takes place through the Hydrodynamic *(p. 161)* property page in the Boundary Editor *(p. 159)*.

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.

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) 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.

2.3.7 Dambreak Erosion

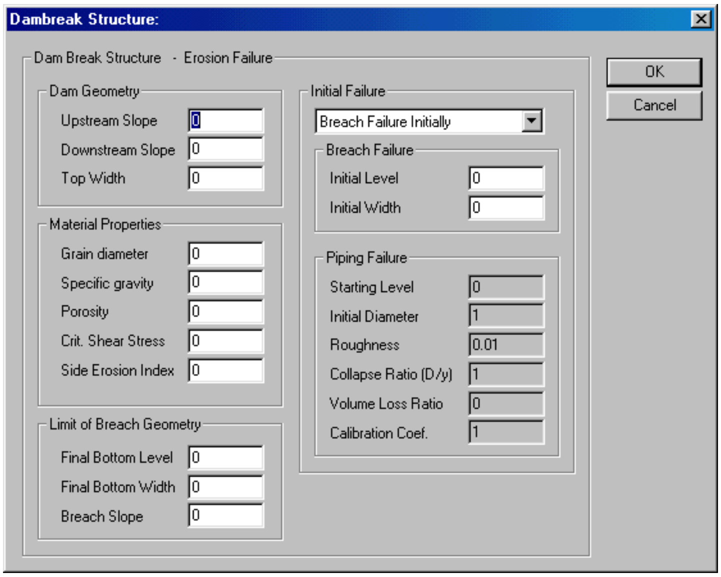


Figure 2.56 The Erosion property dialog.

This dialog (Figure 2.56) is accessed from the Dambreak Str. *(p. 96)* property page in the Tabular view: Structures *(p. 46)* by pressing the

. 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:

- as a breach starting at the top of the dam

or

- 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.

2.3.8 User Defined Structures

User defined structures are yet to be implemented in the calculation kernel of MIKE 11.

2.3.9 Tabulated Structure

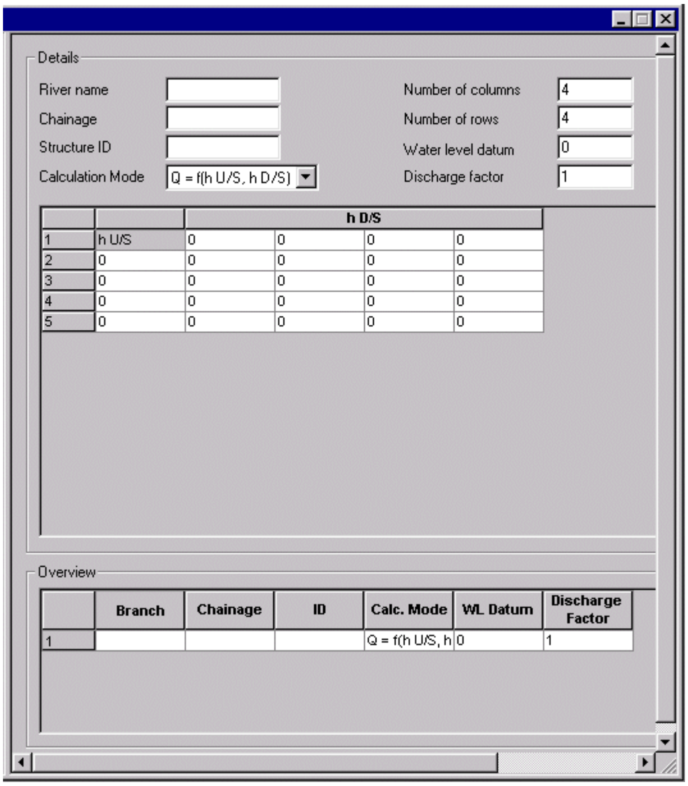


Figure 2.57 The Tabulated Structure property dialog.

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 2.57) 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.

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 right direction and the downstream water level must increase in the downward 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.

2.3.10 Energy Loss

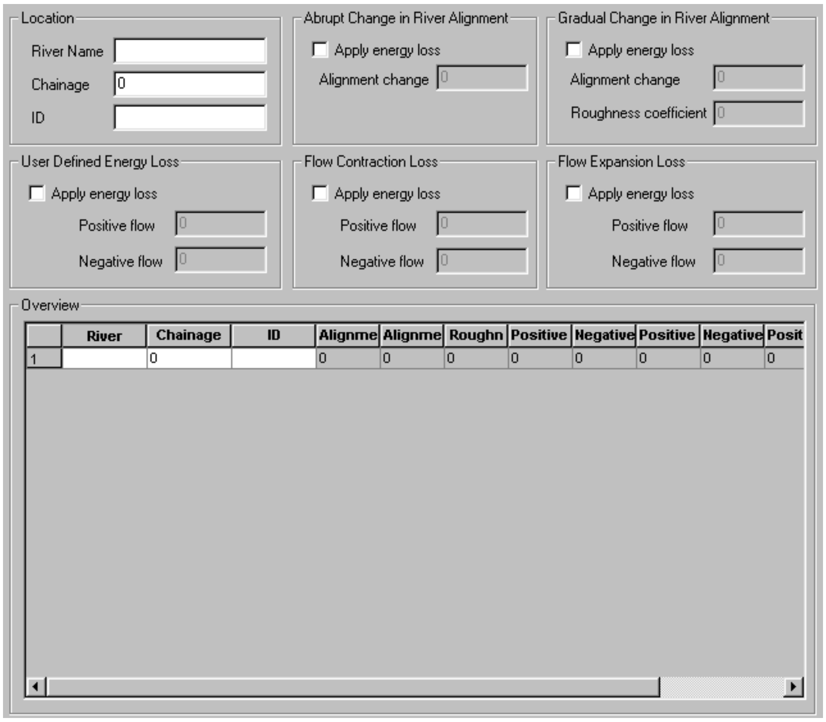


Figure 2.58 Energy Loss property page dialog.

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:

$$\Delta H = \zeta \frac{Q^2}{2gA^2} \tag{2.4}$$

in which ΔH is the energy loss, g is the acceleration of gravity, Q is the discharge and A is the cross-sectional wetted area. The quantity, ζ, 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 solving the St. Venant equations. This requires cross section information, however, if such is not available routing may be an alternative. There are no water levels calculated in routing branches, and what routing does is transforming 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. Typically 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 set to “Routing”. See section 2.2.2 Branches *(p. 37)*.

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.

A routing component is any of the data types described in the following sections.

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.

2.4.1 Channel routing

Currently (version 2000) only non-linear storage function is available for channel routing. The dialog for specifying the parameters for channel routing is shown in Figure 2.59.

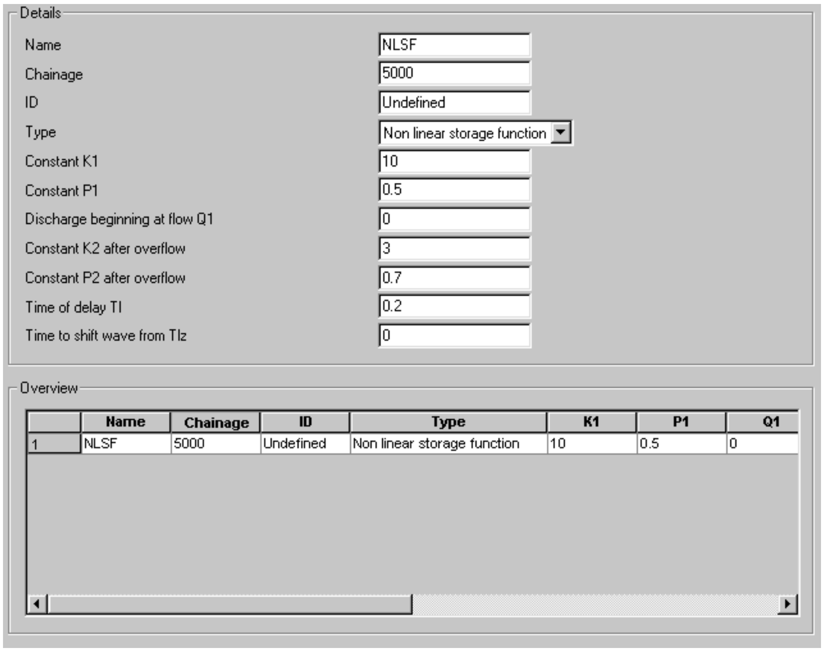


Figure 2.59 Dialog for channel routing.

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, Tl, Tlz: Parameters for the calculation. 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 2.60.

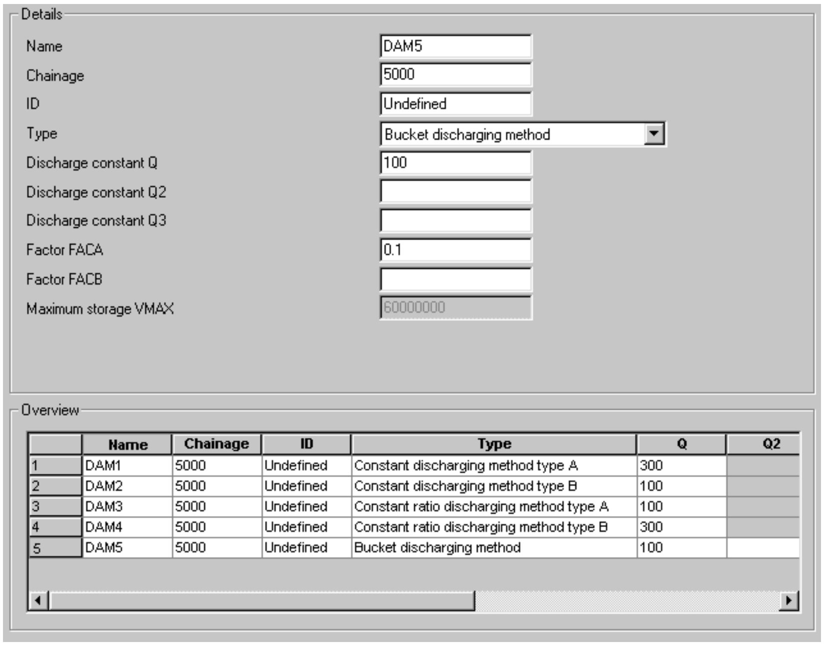


Figure 2.60 Dialog for flood control Q and Q-rate.

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 2.61.

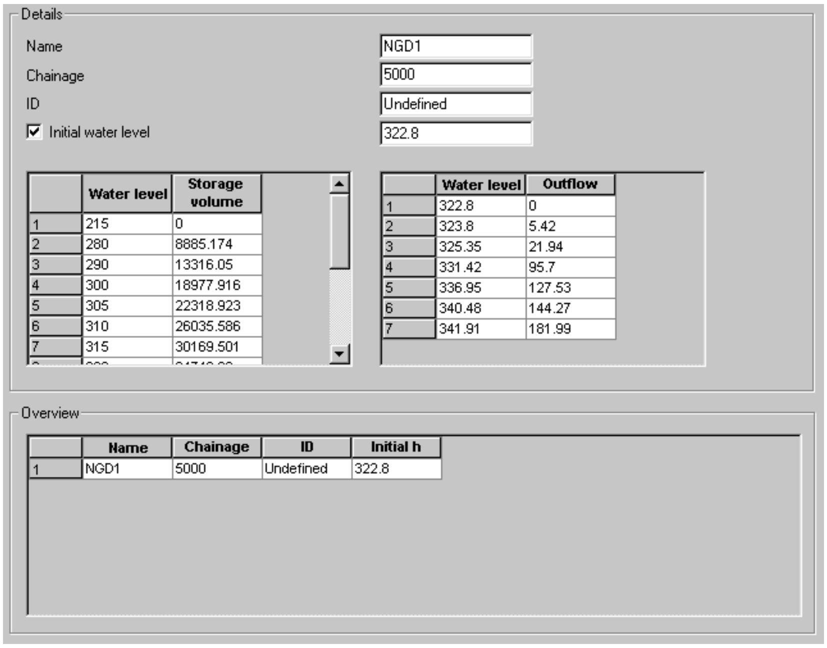


Figure 2.61 Dialog for flood control H-Q / H-V curve.

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 2.62.

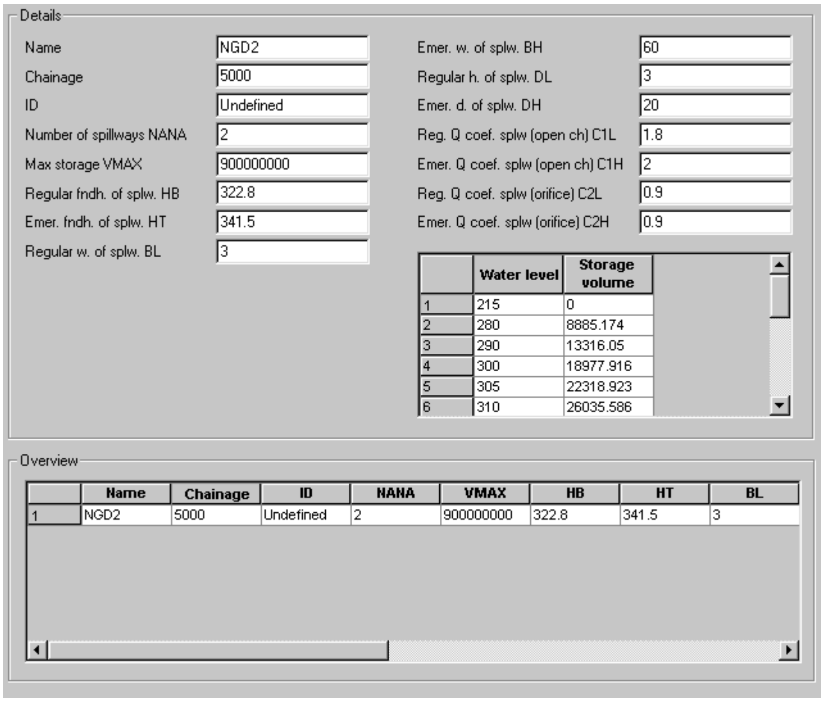


Figure 2.62 Dialog for flood control by orifice.

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 2.63.

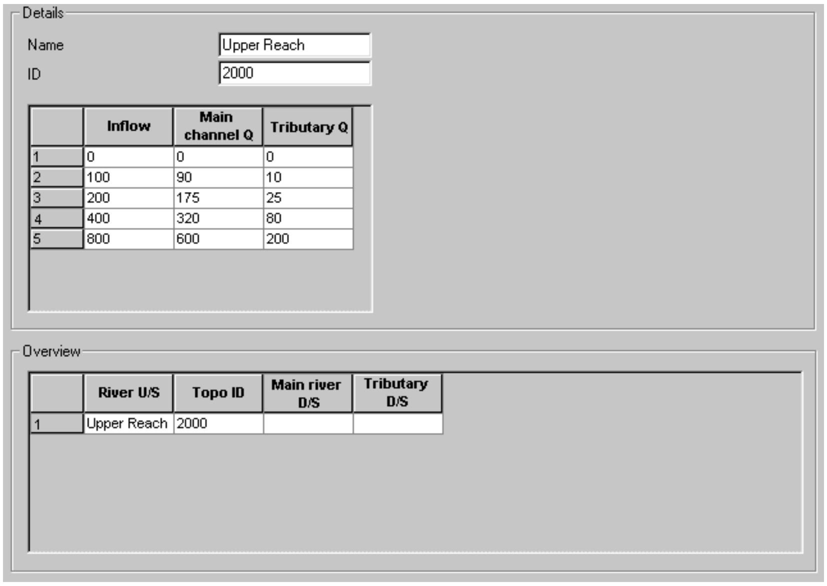


Figure 2.63 Dialog for diversion of flow.

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 2.64 shows how this is done when a branch splits into three branches.

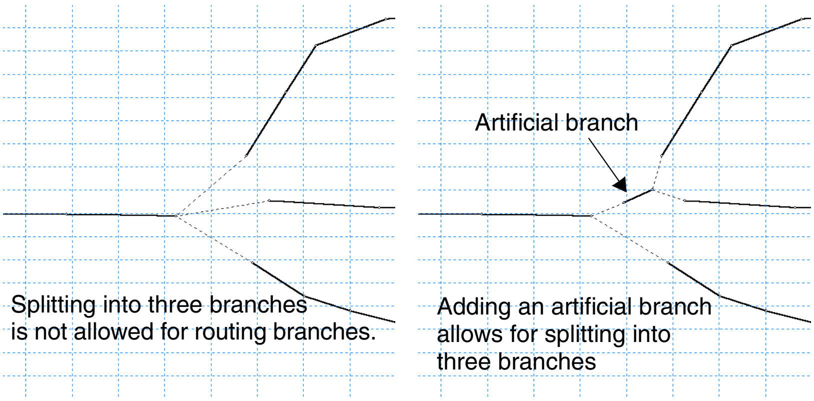


Figure 2.64 Splitting routing branch into three branches.

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.

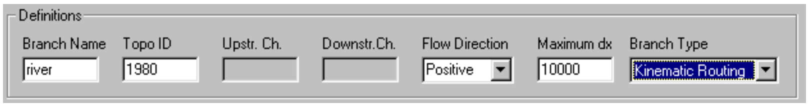


Figure 2.65 Definition of Kinematic Routing branches.

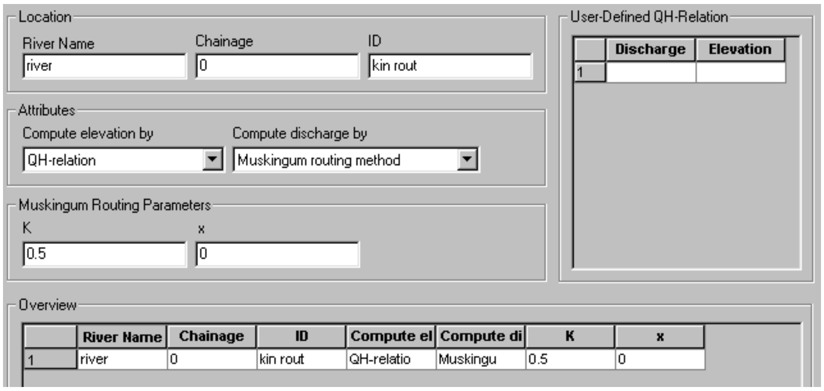


Figure 2.66 Definition of Kinematic Routing elements.

The dialog used to define a Kinematic Routing branch is shown in Figure 2.65, while the dialog used to define Kinematic Routing elements is shown in Figure 2.66.

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 his 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 DHI's groundwater model MIKE SHE.

2.5.1 MIKE SHE links

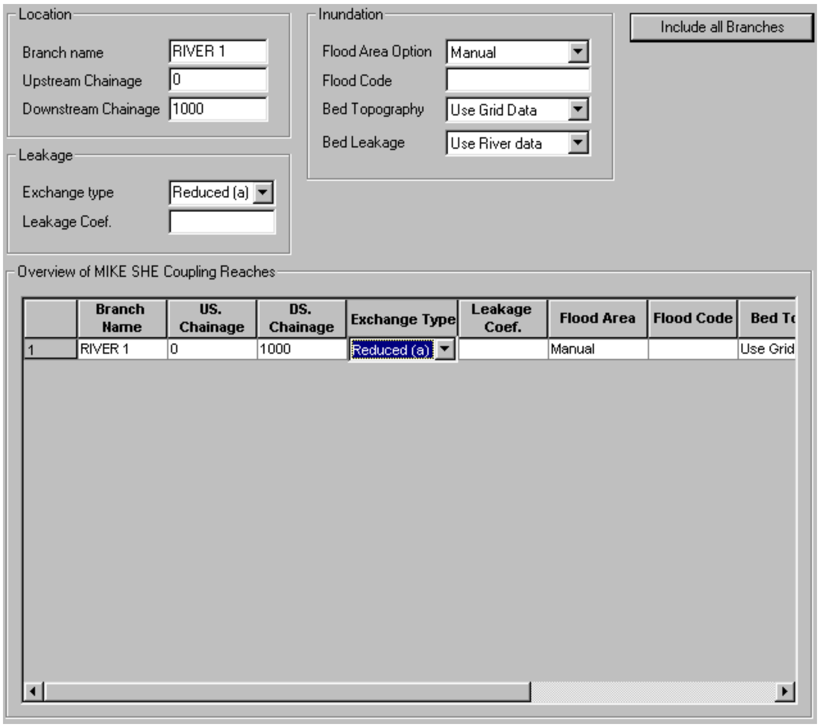


Figure 2.67 MIKE SHE links dialog.

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 coupling 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

A,b or c should be chosen and refers to the 3 different river aquifer exchange types (described in the technical documentation of the MIKE SHE User Manual) of exchange between surface water and aquifer. 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 is either b or c.

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

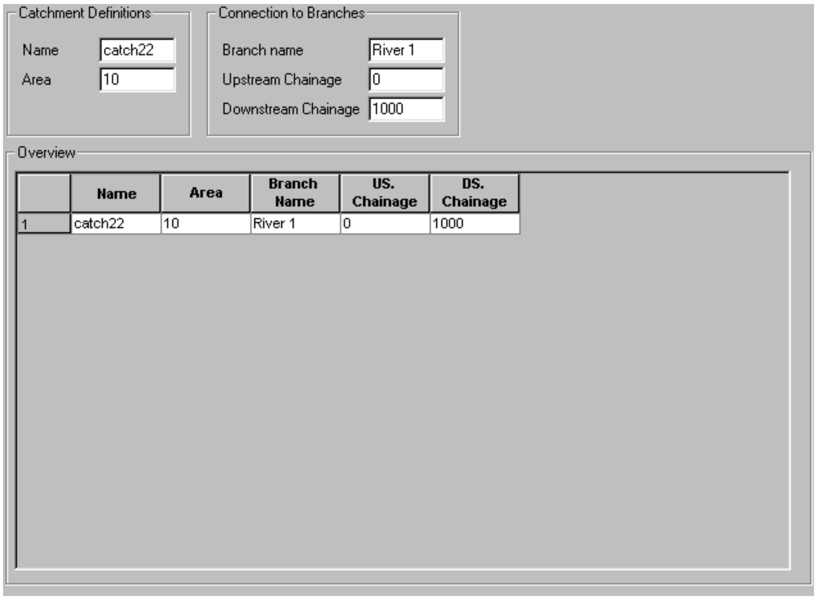


Figure 2.68 Rainfall-runoff links dialog.

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 in the river network.

Catchment Definitions

Name: Name of input catchment.

Area: Catchment area.

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 catchments.

2.6 Tabular View: Grid Points

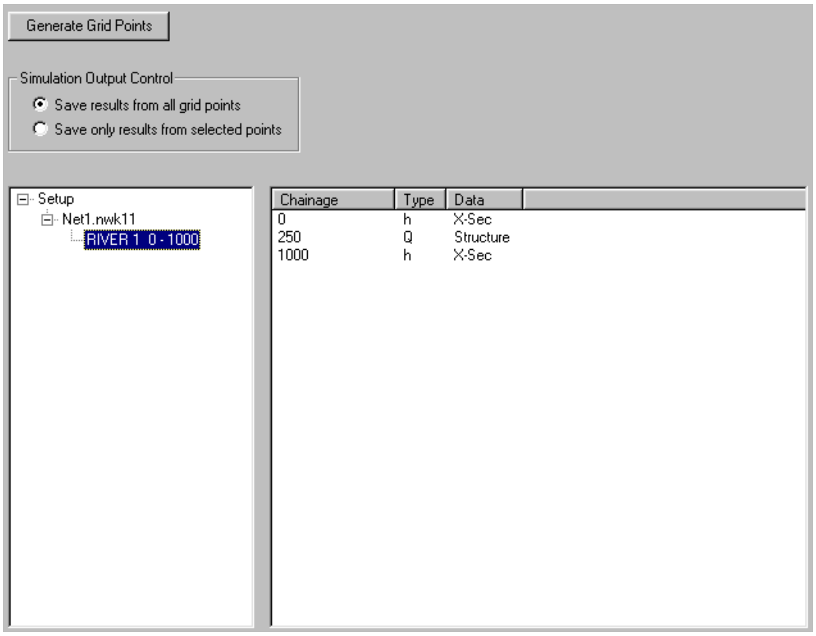


Figure 2.69 Grid points dialog.

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 the 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 2.3

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:

Table 2.3

Level in tree view (right hand part)	**Content of the list view (left hand part)**	
		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). The "-" symbol in a Q-point row indicates this is a standard Q-point where the momentum equation is solved. The word "X-sec" in an h point row indicates that a cross section exists at this location. The word "Structure" in a Q point indicates that a structure is located at this location.

2.7 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. 40)* for more details about alignment lines)

2.7.1 Tool Bar for River Network

The tool bar for graphical editing of the river network is shown in Figure 2.70. In the following the functionality of each of the icons in the tool bar is explained.

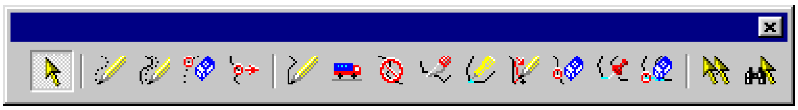
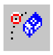


Figure 2.70 Tool bar for editing river network

Select object. This icon activates the selection mode which is also 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.

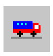
Add points and define branch. This tool creates points and branches in a single operation. Point and click at successive locations along a desired path. Points can also be added by pressing the left mouse button and holding it down while moving. Double click on the last point to end the branch.



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



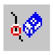
Auto route branch. This tool automatically determines a river branch route from a set of free points. To use this tool you select the first point and drag to the last point on the branch. The editor automatically determines a path through intermediate free points by always searching for the closest point.

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.

Care should be taken when connecting four or more branches. In such cases all branches connections should be made to a single junction point as shown in Figure 2.71.

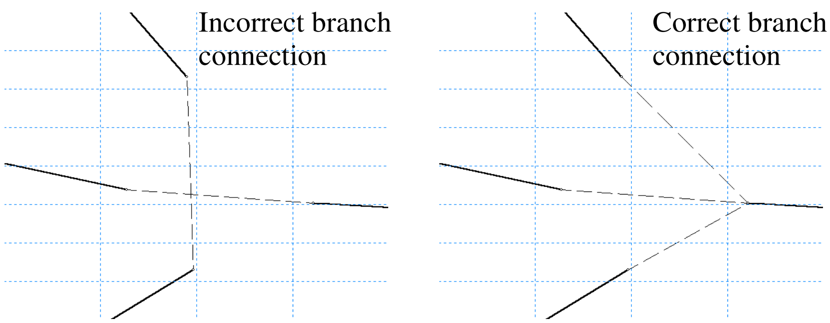


Figure 2.71 Connection of four or more branches.

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.

2.7.2 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.

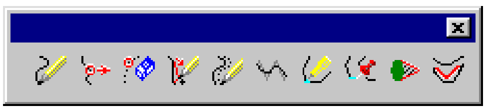
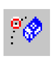


Figure 2.72 Tool bar for editing alignment lines.

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 into 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 2.73 shows an alignment line before and after splining. Five points have been added between all existing points

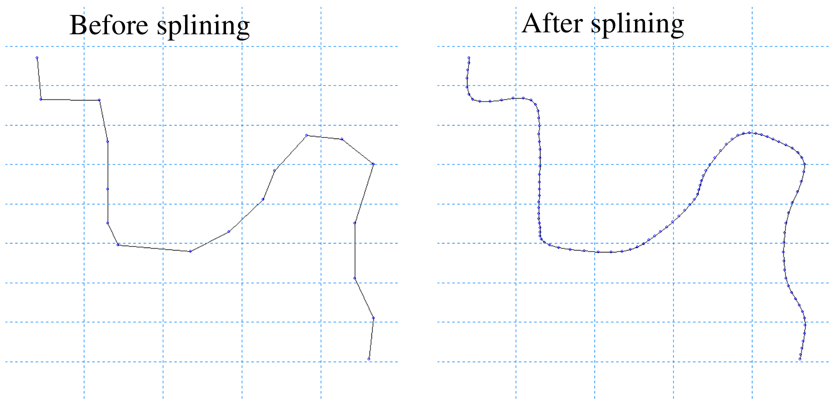
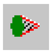


Figure 2.73 Alignment line before and after splining.

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 2.16). 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 2.15). 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.

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 (positive direction downstream).

Refer to one of the following sections for more Information:

3.1 Raw data View *(p. 135)*

3.2 Processed data view *(p. 147)*

3.3 Importing cross sections using File Import *(p. 150)*.

3.4 Exporting cross sections using File Export *(p. 155)*

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.

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).

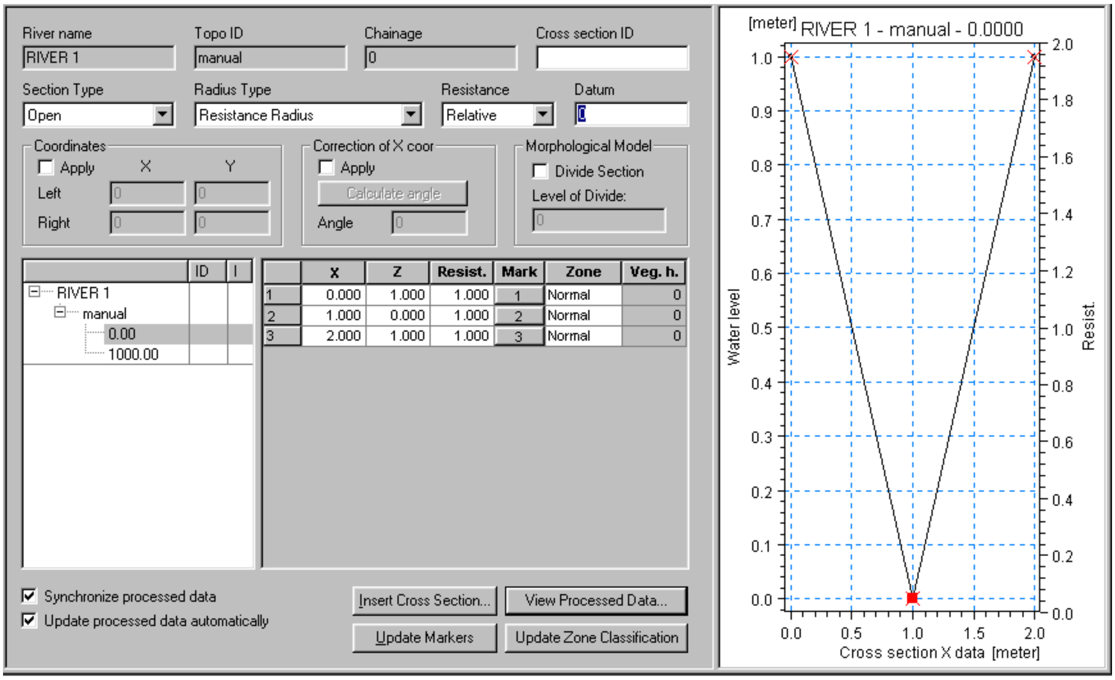


Figure 3.1 *The raw data view.*

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 *Dialog boxes*

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 Context sensitive pop-up menus).

Cross section ID

An identification tag 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 possibilities are listed:

– Open section: The typical setting for river cross sections.

– Closed irregular: Closed sections with arbitrary shape.

– Closed circular: Circular shape where only the diameter need to be given.

– Closed rectangular: Width and height is required.

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.

Resistance

Only used in conjunction with the quasi two dimensional steady flow with vegetation module.

The setting for the resistance column in the tabular view (see Figure 3.4).

Datum

A datum may be entered here which is added to all vertical coordinates in the tabular view.

Coordinates

Plane coordinates may be entered here for the left/right end of the cross section. If non-zero values are entered the values are used in the graphical view of the network to display the cross section width.

Correction of X-coor

This 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 center line of the river.

The correction angle can be automatically calculated by activating the 'Calculate angle' button.

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

$$x_{cor} = x\cos\theta \tag{3.1}$$

where θ is illustrated below

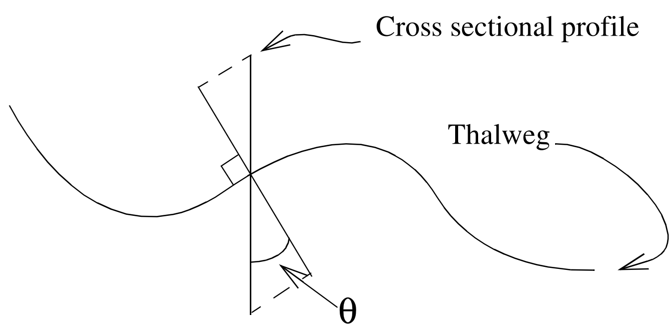


Figure 3.2 Definition sketch of the correction angle.

Morphological Model

A level of division can be entered. This level of division is used for activating the flood plain description as specified in section 6.3 Flood Plain Resistance *(p. 229)*.

Additional tick boxes

At the bottom left corner of the editor two tick boxes are present.

Synchronize processed data

By ticking this box the processed and raw data 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.

Additional buttons

Insert cross section

Pressing this button activates a pop up dialog as shown below.

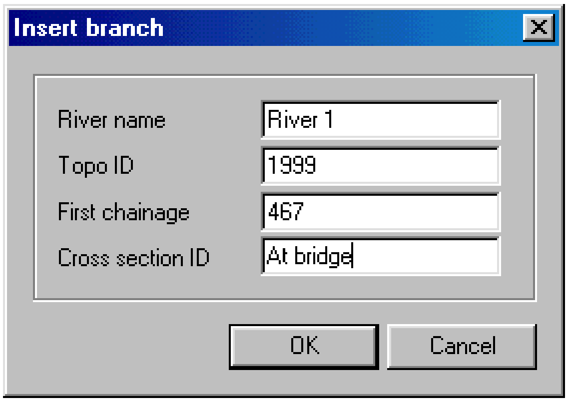


Figure 3.3 The Insert branch dialog.

In this dialog the appropriate information is entered and OK is pressed.

View processed data

This button activates the processed data view.

Update markers

This button updates markers 1,2 and 3 as the extremes of the cross section. Note this facility overwrites user settings of these three markers unless the appropriate boxes under Settings –> Cross section.... –> Update Markers have been unticked.

Update zone classification

Only used in conjunction with the quasi two dimensional steady flow with vegetation module.

Used for updating the zone classifications in the cross section.

3.1.2 Tabular view

The tabular view is only appropriate 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.

Resist.

This column is used for setting relative resistance.

In conjunction with the quasi two dimensional steady flow with vegetation module it is used for setting local values of Manning's *M* or *n*. Depending on the setting in the resistance combo box (see Figure 3.4).

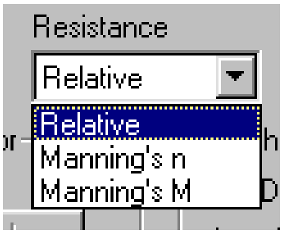


Figure 3.4 Resistance combo box. Only visible if a Quasi two dimensional steady state solver with vegetation has been installed.

Mark

The column is used for setting the markers 1 to 9 plus possible user defined marks. Clicking an element in the column opens a marker dialog as shown below.

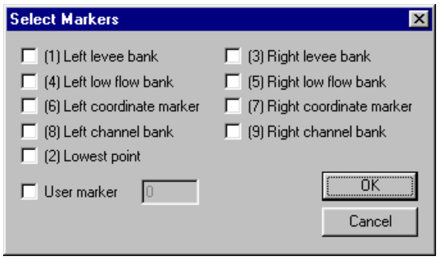


Figure 3.5 Select marker dialog box.

A number of markers may be set in this dialog:

- Left/right levee bank: Defines the extend of the cross section used for the calculations.
- Left/right low flow bank: Defines the extent of the low flow channel. Only used in conjunction with the quasi two dimensional steady flow with vegetation module.
- Left/right coordinate markers: Defines the points in the cross section corresponding to the coordinates used for determining the correction angle.
- Left/right channel bank markers: Defines the points in the cross section corresponding to extend of the main channel. The markers have an effect on the calculation of the processed data. For details consult the reference manual.
- Lowest point: The lowest point of the river may be set using this marker. The marker is used for post processing only and thus does not affect the calculations.

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.

Context sensitive pop-up menus

Selecting a river branch or cross section with the right mouse button will open **context sensitive pop-up menus**. The following editing facilities are available:

Copy Cross Section

The Copy Cross Section dialog is activated from the pop-up menu in the tree display of the raw data view. A cross section chainage must be selected before activating the pop-up menu.

A dialog requests a Topo-ID, branch name and chainage before copying the cross section.

Rename Cross Section

The rename cross section dialog is activated from the pop-up menu in the tree display of the raw data view. A cross section must be selected before activating the pop-up menu. A new cross section chainage must then be entered.

Insert Branch

The Insert Branch dialog is activated from the Insert Cross Section button on the raw data view or by selecting Insert on the tree view pop-up menu.

A dialog requests the River Name, Topo-ID and Chainage of a cross section to be inserted on the selected river branch.

Copy Branch

The copy branch dialog is activated from the pop-up menu in the tree display of the raw data view. The Topo-ID of the river branch must be selected before activating the pop-up menu.

A dialog requests a river branch name and Topo-ID before copying the cross sections.

Rename Branch

The Rename Branch dialog is activated from the pop-up menu in the tree display of the raw data view. A river branch must be selected before activating the pop-up menu. A new river branch name must then be entered.

Combine Branch

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.

A typical example occurs when combining survey (SUR) cross sections with digital elevation model (DEM) sections. A DEM is used to extract sections from broad flood plains while survey is used to obtain river sections. The combine feature will produce a composite section which can be saved under a new Topo-ID.

- Topo-ID of DEM profiles:

  Topo-ID of the DEM or first sections.

- Topo-ID of SUR profiles:

  Topo-ID of the SUR or second sections.

- Topo-ID of combined profiles:

  Topo-ID of the combined sections. Section will only be created at locations with corresponding chainage.

- Maximal difference:

  The Maximal difference is the tolerance limit within which sections are considered to correspond.

- Synchronize to:

  Specifies the method for combining sections. Currently only one option is available:

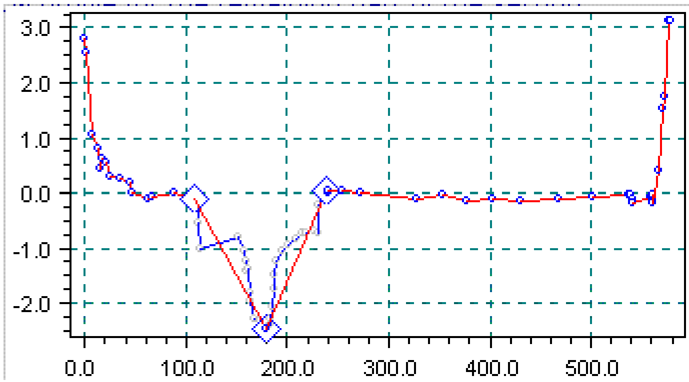


Figure 3.6 Center (mark 2) The plot shows a DEM section (indicated by the blue/dark dots) and a SUR section (indicated by the grey dots). The combined section will include the DEM section from x=0 to the left marker (the diamond) of the SUR profile, then the SUR profile to the right marker (third diamond from the left) of the SUR profile and finally the DEM profile for the remaining part of the section.

Insert Interpolated

This feature is only available when selecting a cross section. The cross section editor gives the user the possibility of inserting interpolated cross sections in a given set-up. When selecting this feature a dialog appears.

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 cross sections and also 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.

3.1.3 The Cross section pull down menu

When the cross section editor is active the cross section pull down menu may be activated. This menu has two 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. To change any of the below please tick the respective ‘change’ tick box.

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 division can be set here.

Raw data - Resistance Type

The global resistance type may be changed using this facility.

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.

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.
- Update markers.
- Recompute all.

Please note that 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.

3.1.4 Graphical Settings

Graphics settings

The graphics settings consists of a tree structure view of all possible settings for the graphical elements of the cross section editor. The desired elements are ticked and the properties are set using the right hand side of the dialog box.

Drawing Settings

A number of settings are available:

- **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 is drawn in black. 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.)

3.1.5 Miscellaneous settings

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

Dialog used for setting the default settings of the resistance column in the tabular view of the cross section editor.

3.1.6 Update Markers settings

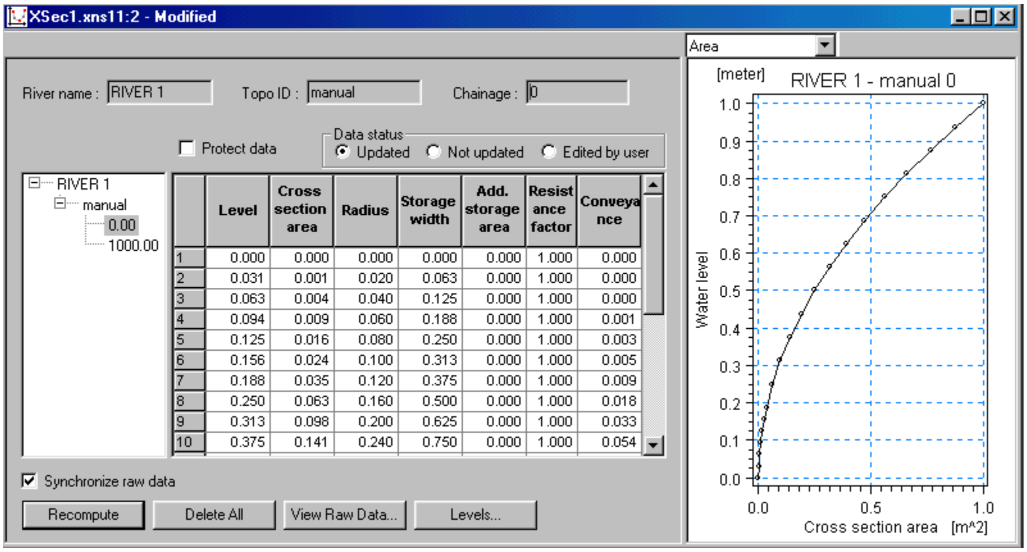
This dialog is used for setting the markers which should be automatically updated when activating the update markers button.

3.2 Processed data view

Selecting the *View Processed Data* button on the Raw data View *(p. 135)* activates the processed data view.

Note that 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 provides all processed data and a graphical view on the right hand side displays the processed data graphically (see Figure 3.7).



	Level	Cross section area	Radius	Storage width	Add. storage area	Resistance factor	Conveyance
1	0.000	0.000	0.000	0.000	0.000	1.000	0.000
2	0.031	0.001	0.020	0.063	0.000	1.000	0.000
3	0.063	0.004	0.040	0.125	0.000	1.000	0.000
4	0.094	0.009	0.060	0.188	0.000	1.000	0.001
5	0.125	0.016	0.080	0.250	0.000	1.000	0.003
6	0.156	0.024	0.100	0.313	0.000	1.000	0.005
7	0.188	0.035	0.120	0.375	0.000	1.000	0.009
8	0.250	0.063	0.160	0.500	0.000	1.000	0.018
9	0.313	0.098	0.200	0.625	0.000	1.000	0.033
10	0.375	0.141	0.240	0.750	0.000	1.000	0.054

Figure 3.7 The processed data view.

3.2.1 Tabular View

The processed data is calculated from the raw data and contains the following parameters:

Level

The level in the cross section. The calculation levels can be manually set using the Levels Dialog activated by the *Levels* button.

Cross section area

Cross sectional flow area.

Radius

A resistance or hydraulic radius depending on the selected type. (Selection is made in the raw data view.)

Storage width

Top/Storage width of the cross section.

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.

Resistance factor

This factor can be used to apply a variable resistance for incremental levels of flow height in the section cross section.The roughness parameter in the Bed Resistance *(p. 232)* Property Page of the Hydrodynamic Editor *(p. 223)* is multiplied by the resistance factor.

Conveyance

Conveyance is not used in the simulation but is displayed for the purposes of checking that the conveyance relationship is monotonously increasing with increasing water level.

Note that this may not be the case for closed sections or in some instances of sudden width increase in the section geometry when using the hydraulic radius option.

Processed Data Levels

The levels dialog controls the number and method of processed data levels selection.

Level Selection Method

There are three methods by which the levels can be selected:

1. **Automatic:** The levels are selected automatically. If the resistance radius is applied the levels are selected according to variations in section flow width. If hydraulic radius is applied the levels are selected according to variation in the section conveyance.
2. **Equidistant:** The levels are selected with equidistant level difference.
3. **User defined:** The levels can be fully or partially selected. 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. (The default is the lowest point in the section).

Maximum Level

The maximum calculation level.

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 it is possible to read cross-section data (raw or processed) from a text file into MIKE 11.

The read facilities can be used to read cross-sectional data stored in an external data base format. The cross-sections are then read in 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:

```
Branch Name
TOPO_ID
               Chainage
COORDINATES
    0
FLOW DIRECTION
    0
DATUM
      0.00
RADIUS TYPE
    0
DIVIDE X-Section
0
SECTION ID
     Bridge
INTERPOLATED
    0
ANGLE
    0.00
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 3.8 File format of ASCII file used for importing data into MIKE11.

In Figure 3.8 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 centerpoint 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.

X x-coordinate

Z z-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
<#32>	marker 6
<#64>	marker 7
<#128>	marker 8
<#256>	marker 9

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.

Markers 4-7 are only of concern for the quasi two dimensional steady state with vegetation module.

Each type of information must start 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

  "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

  "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

  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
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)      B(1)          Afl(1)    rf(1)
     L(2)          A(2)        R(2)      B(2)          Afl(2)    rf(2)
      |             |           |         |             |          |
      |             |           |         |             |          |
     L(M)          A(M)        R(M)      B(M)          Afl(M)    rf(M)
*******************************
```

Figure 3.9 Format used for importing processed data.

The first three lines are 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)

- 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 levels) lines with the hydraulic parameters can be specified. 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.

3.4 Exporting cross sections using File Export

Via File –> Export it is possible to write cross-section data (raw or processed) from the MIKE 11 data base to a text file.

There are three possibilities:

1 Export All...: Both raw and Processed data is exported to a 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.

BOUNDARY EDITOR

4 BOUNDARY EDITOR

The boundary editor consists of four tabs:

- Hydrodynamic *(p. 161)*
- Advection dispersion *(p. 163)*
- Sediment transport *(p. 166)*
- Rainfall-Runoff *(p. 168)*

For each of these four tabs there exists some common features described below.

4.0.1 Item Selection

Selection of items are simplified with the following functions:

- Edit *(p. 159)*
- Browse *(p. 159)*
- Items *(p. 160)*
- Specifying Time series File and Item *(p. 160)*

A boundary condition is defined by assigning a time series item to a boundary point. The selection of the required items is simplified using the following available functions:

Edit

The 

button opens the time series file listed in the Time Series File field by activating the Time Series Editor. The time series can be modified using edit, add and delete functions.

The Edit button is only active if a file-name of an existing "dfs0" file is listed in the Time Series File field.

Browse

The 

button opens the file selection dialog for selection of a time series file (*.dfs0). This function is used when defining a boundary condition.

The selected time series file (*.dfs0) can be located in any directory on the disk. Use the File selection dialog to locate the "dfs0" file and select the required file.

Items

The 

button activates the DFS Time Series Item Selector.

From the time series item selector a specific item can be selected and linked to a boundary definition.

Specifying Time series File and Item

Time series items are saved in a time series file (*.dfs0). A boundary condition can be defined by selecting a time series file using the Browse button or by manually typing the full name and path of the required dfs0-file (including extension) in the Time Series File field. The required items stored in the dfs0 file can then be linked to the model boundary points.

A dfs0-file contains one or more time series items of different types (e.g. water level, Discharge etc.). The Items button activates the DFS Item Selector, which lists the available items for the specified boundary type (e.g. Water Level, Discharge etc.). An item can be selected from the list and will be displayed in the Items Tree view.

Time series items can be modified using the Time Series Editor, activated by the Edit button on the Property Page.

4.1 Hydrodynamic

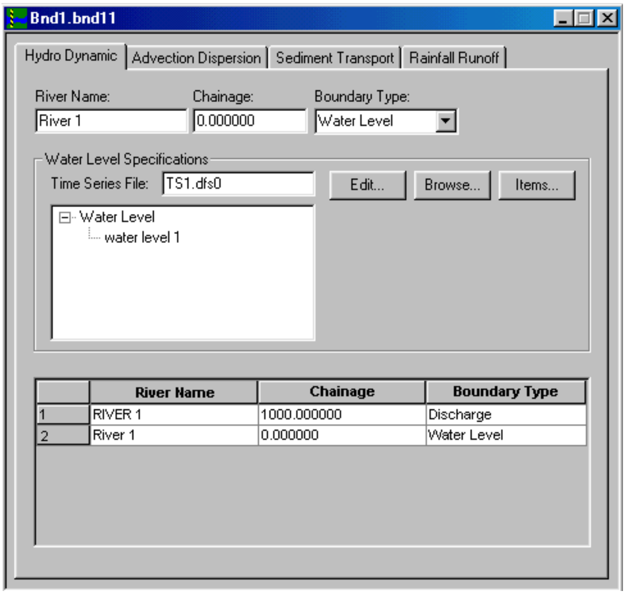


Figure 4.1 The Hydro Dynamic Property Page.

Boundary conditions for hydrodynamic (HD) simulation are defined using the Hydro Dynamic Property Page (Figure 4.1).

To define a boundary condition a time series item must be assigned to a boundary point.

4.1.1 Boundary types, Hydrodynamic model:

Water level

Must be applied as either an upstream or downstream boundary condition.

Discharge

Can be applied as either an upstream or downstream boundary condition. A discharge boundary type can also be applied as side tributary flow (lateral inflow) or as a time dependent regulating function/structure where $Q_{struc} = Q(t)$.

Q/h Relation

Q/h Relations can only be applied at downstream boundaries.

Q/h boundaries are applied at downstream boundary conditions when it is not reasonable to apply a water level boundary with constant or time varying water level. Such a situation may exist where the model does not extend to a sea or lake, and where the water level at the boundary, therefore, is not independent of the upstream conditions.

As a rule *Q/h* boundaries should only be used at boundary points where outflow from the model area takes place.

Note: MIKE 11 will not extrapolate the *Q/h* relation past the highest specified water level value. If this situation arises during a computation, the program will halt with an error message.

Wind Field

Wind fields can be applied globally or as branch specific factors. The global wind field is applied by specifying GLOBAL in the river name field. Branch specific wind fields are introduced by specifying the river name of a branch. If a branch specific wind field is not specified a global wind field is assumed. Wind fields are applicable to whole branches and it is not possible to specify a chainage value for specific reaches. Inclusion of wind shear stress calculations in the computation is specified in the Hydrodynamic Editor *(p. 223)* under Wind *(p. 231)*. The user can reduce the effect of the wind shear stress by applying a topographical wind factor in certain reaches using the Hydrodynamic Editor *(p. 223)*. When applying a wind field as a boundary condition both the wind velocity and wind direction must be specified. The direction of the wind is degrees in clock wise direction from (model) north. (See Figure 4.2).

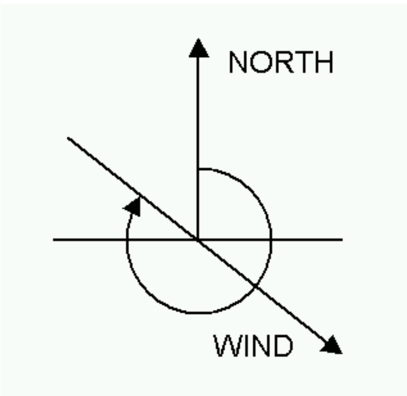


Figure 4.2 Definition of wind direction.

Dambreak

Time dependent failure modes for a dambreak computation must be specified using a time series file. The file defines the shape and development of the breach which includes, dam breach level, breach width and breach slope.

Resistance Factor

A time variant resistance factor can be specified using a time series roughness factor. The roughness parameter (Manning or Chezy) values specified in the HD Parameter File will be multiplied by the time variant factors. The factor applies only to branches specified in the river name field i.e. the factor is applied to the whole branch thus a longitudinal variation is not possible. If the river name is specified as GLOBAL, the multiplication factors are applied to all branches without a specific definition - a branch specific factor always overrides a global definition. Any number of resistance factors can be applied. Only global or branch specific resistance factor series are applicable and it is not necessary to enter a chainage.

Heat Balance

Three items are needed for specifying a heat balance boundary:

- Air temperature
- Relative humidity
- Sunshine

Heat balance is also taken into account in normal Mike 11 branches.

Dam

In case one would like to include a dam in a heat balance set-up the boundary type dam should be chosen. Here one needs to specify a discharge through the dam and a gate level, width and height.

4.2 Advection dispersion

Boundary conditions for advection-dispersion (AD) simulations are defined using the Advection-Dispersion Property Page (Figure 4.3).

To define a boundary condition a time series item must be assigned to a boundary point.

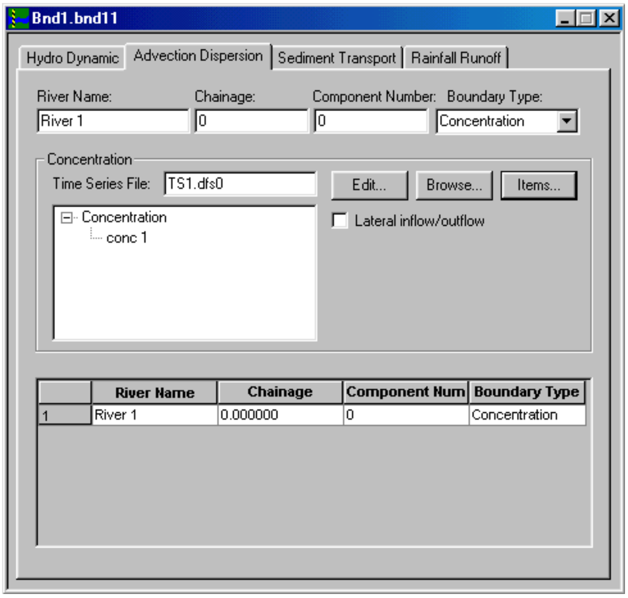


Figure 4.3   The Advection Dispersion property page.

4.2.1   Component number

The AD and WQ models are capable of modelling the advection dispersion and water quality processes of multiple substances/components. Each boundary definition must be assigned a component number that corresponds to the number of components defined in the Components *(p. 270)* Property Page from the Advection-Dispersion Editor *(p. 261)*.

A boundary time series for component ‘0’ will be applied to all components not otherwise specified. This allows multiple concentrations to be specified using a single time series (e.g. at an open boundary or closed boundary).

With the exception of Temperature, it is possible to apply time series using a unit specification (e.g. mg/l) different from the global unit specified in the Advection-Dispersion Editor *(p. 261)* (e.g. g/l). The time series units will be adjusted automatically.

4.2.2   Lateral inflow/outflow

A lateral inflow introduces mass to the model simulation; consequently the concentration time series must include discharge values for the mass to be calculated. When entering a concentration time series for lateral inflows the ‘Lateral Inflow/outflow’ must be enabled and both concentration and discharge items must be selected from the Items Selector.

The user has the option to include (or neglect) tributary inflows in an HD simulation. If lateral inflows are included the discharge series applied for the HD simulation should match the discharge series applied to the concentration time series.

4.2.3 Boundary Types, Advection Dispersion/Water Quality models

Concentration

Concentration time series may be used as external boundary conditions or as lateral source points. If an external boundary condition is applied in the AD-simulation, the discharge from the HD simulation will be used compute the mass transport in or out of the system. Units are selected using the 'Unit' column.

Bacteria Concentration

Concentration of E Coli bacteria. Unit in Counts x1E6 / 100 ml

Temperature

Temperature is used if the WQ module is activated in conjunction with an AD simulation.

WQ Temperature function

Temperature forcing function used when the Eutrophication module is activated in conjunction with an AD simulation.

WQ Sun Radiation function

Incident light radiation forcing function used if the Eutrophication module is activated in conjunction with an AD simulation.

4.3 Sediment transport

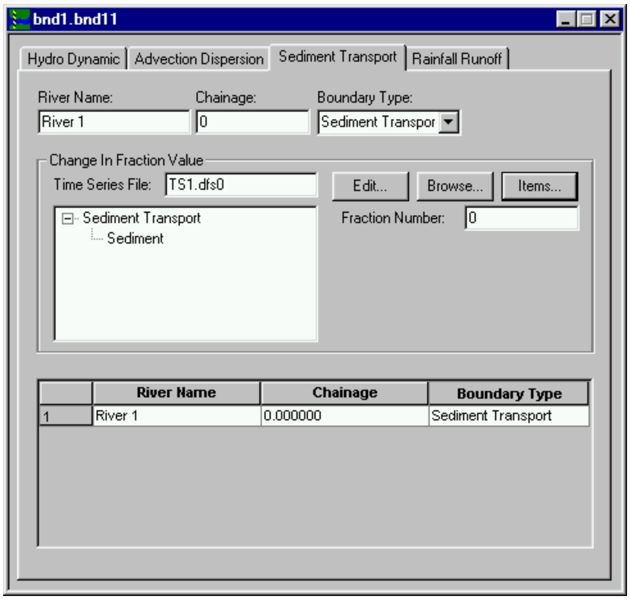


Figure 4.4 The Sediment Transport property page.

The Sediment Transport property page (Figure 4.4) contains all boundary conditions required for Sediment Transport (ST) simulations.

4.3.1 Fraction Number

Boundary conditions for each fraction (particle size) specified in the Sediment Transport Editor *(p. 333)* (Data for graded ST *(p. 342)* property page) must be defined when using the graded sediment transport model. Fraction numbers must be specified for the boundary types options listed as; ‘Sediment transport’, ‘Fraction value’ and ‘Change in Fraction value’.

4.3.2 Boundary Types, Sediment Transport model

Sediment Transport

A Sediment transport boundary condition can be applied as an external boundary or as a lateral sediment inflow. Sediment transport boundary can be applied with both positive and negative values imposing either sediment addition or abstraction (e.g. dredging).

Bottom Level

Bottom level can be applied as an external boundary condition only. It is used in applications of the sediment transport module including morphological updating of the river bed level.

Change in Bottom Level

Change in bottom level can be applied the same way as for the bottom level type described above. Change in bottom level is specified in either depth/s or depth/day.

Fraction Value

Fraction of each particle size in the graded sediment module.Fraction values can be applied both as external and lateral sediment inflows and are specified in percentage distribution. The sum of all fractions at each boundary must equal 100% over the entire simulation period.

Change in Fraction Value

Change in fraction value can be applied the same way as for the fraction value type above. The value is expressed as change of percentage distribution/day.

Sediment supply

The Sediment boundary condition is an alternative to defining a sediment transport boundary with measured sediment transport values. Choosing this option, no time series file and item shall be defined and the sediment transport at the boundary locations will be calculated automatically based on the HD-conditions at the location at the actual time.The Sediment supply boundary condition could be used if e.g. measurements are not available for the sediment transport at the boundary or if it is simply desired to let the model determine the sediment transport at the inflow boundary point purely based on the HD-conditions at the given time.

4.4 Rainfall-Runoff

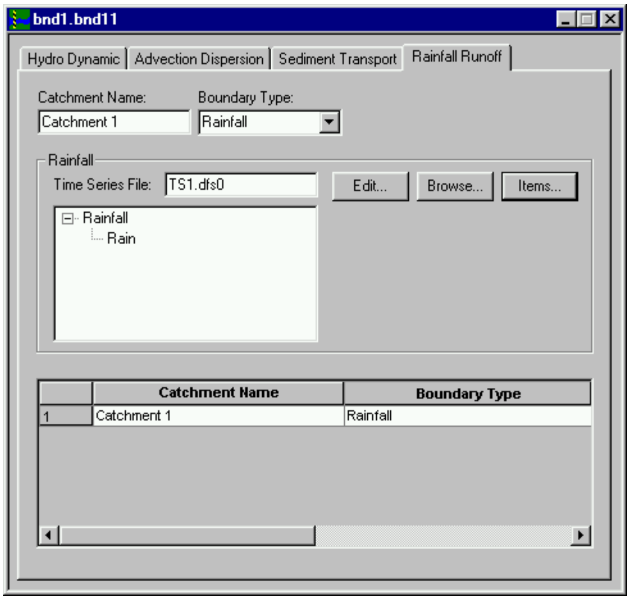


Figure 4.5 The Rainfall-Runoff property page.

The Rainfall-Runoff property page (Figure 4.5) contains all boundary conditions and setup parameters required for rainfall-runoff (RR) simulations.

Note that all parameters of the rainfall-runoff model are edited using the Rainfall-Runoff Editor *(p. 173)* and all boundary conditions and setup parameters required for rainfall-runoff (RR) simulations are stored in the file “default.bnd11”. Thus the Rainfall-runoff property page is included for verification purposes only.

4.4.1 Boundary Types, Rainfall-Runoff model

Rainfall

Rainfall appears as accumulated precipitation from the time of the preceding rainfall value.

Evaporation

Evaporation appears as accumulated evaporation from the time of the preceding rainfall value.

Temperature

The temperature is used when the snow routine is activated.

Irrigation

Irrigation for a specific catchment. Only used in connection with the Irrigation module in the RR-model.

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 timeseries**. 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 builds a file containing all the specified data with extension .RR11. Once the catchments have been defined and 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 noticed that:

- The time series specified in the Time series page are collected in a MIKE 11 boundary file for use in MIKE11. The saving of the RR-editor file will automatically generate this **“default.bnd11**" file, which is used as boundary file on the input page in the simulation editor. The “default.bnd11” will normally automatically appear in the boundary data field after having selected the RR input file. The boundary data field will appear gray as shown on Figure 5.1.
- Time step: 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 on Figure 5.2.

- Simulated catchment results can be linked with the River Network. Catchment runoff/discharges and be inputted as lateral inflows and summed 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 ready to be used for model calibration and result presentation. These are described in Section 5.8

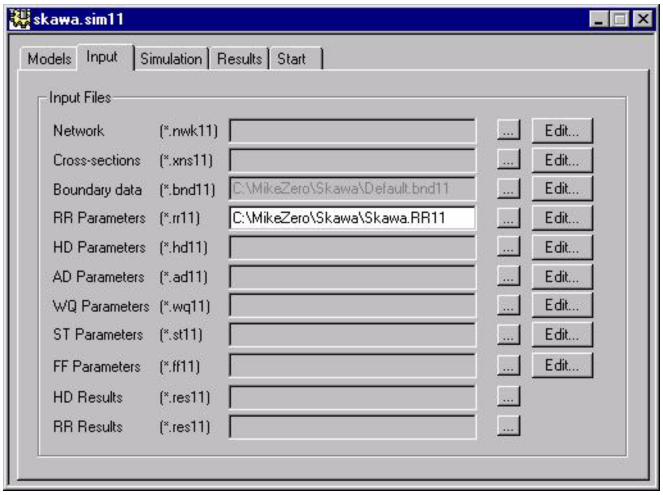


Figure 5.1 Input page to the rainfall-runoff simulation in the Simulation Editor

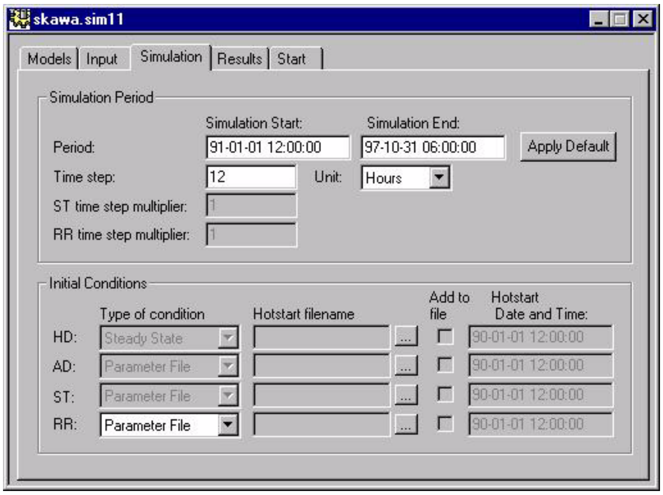


Figure 5.2 Simulation page to the rainfall-runoff simulation in the Simulation Editor. In this example aTimestep=12 hours.

Editing using the clipboard

Overviews in the Editor shown in 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 clipboard. 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).

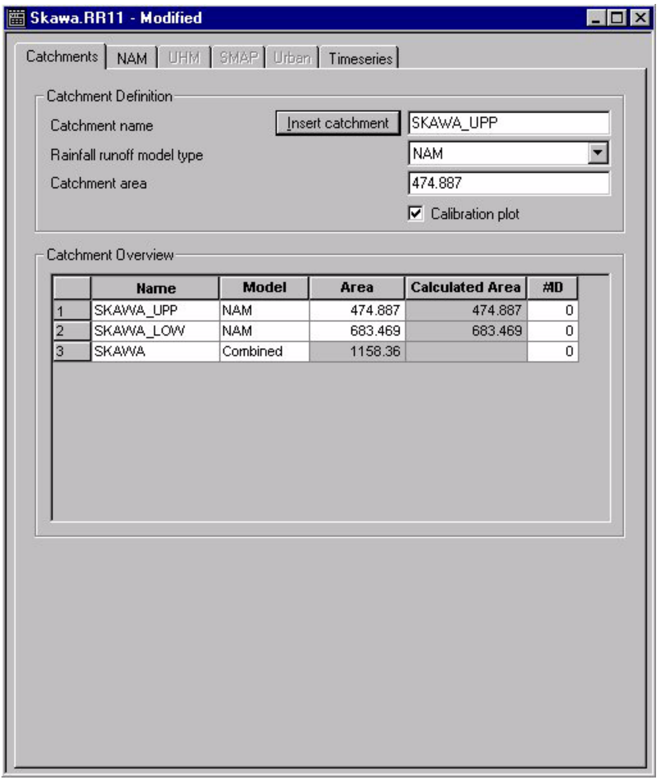


Figure 5.3 The Catchment page. Additional catchments are prepared via the Insert Catchment dialog. The Example includes 2 sub-catchments and a combined catchment which includes the 2 sub-catchments

Inserting Catchments

New catchments are defined via the Insert Catchment dialog (see Figure 5.4). The insert catchment dialog is automatically activated for the first catchment, when creating a new RR-parameter. A new RR parameter File is created from the MIKEZero File dialog. Additional 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.

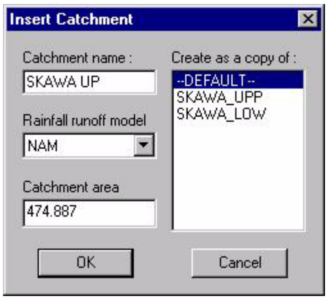


Figure 5.4 Insert Catchment Dialog.

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). Following models can be selected:

1 **NAM**: A lumped, conceptual rainfall-runoff model, simulating the overland-, inter- flow, and base-flow components of catchment runoffs as a function of the moisture contents in four storages. NAM includes a number of optional extensions, including an advanced snow-melt rou-

tine 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 composed and is saved in the subdirectory RRCalibration with the file name: Catchment-name.plc. The time series in these plots are also available in DFS0 format in the subdirectory RRcalibration with the file name: Catchmentname.dfs0.
Figure 5.28 shows an example on 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 5.7. 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 on 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 5.9: A step-by-step procedure for using 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 below.

5.2.1 Surface-rootzone

Parameters used in the surface and the root zone are described below (see Figure 5.5).

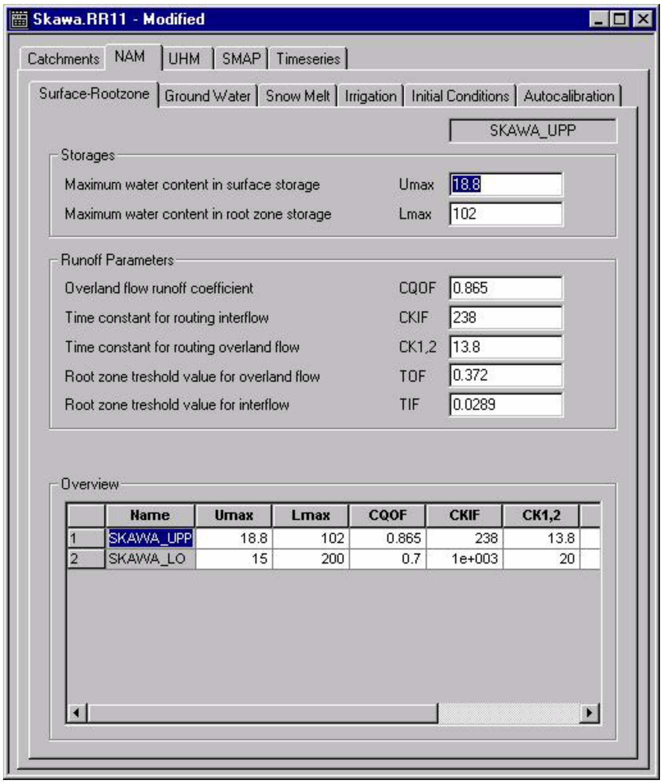


Figure 5.5 NAM - Surface Rootzone.

Maximum water content in surface storage (Umax).

Represents the cumulative total water content of the interception storage (on vegetation), surface depression storage and storage in the uppermost 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, 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 70% 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 70% of Lmax and the maximum value allowed is 0.99.

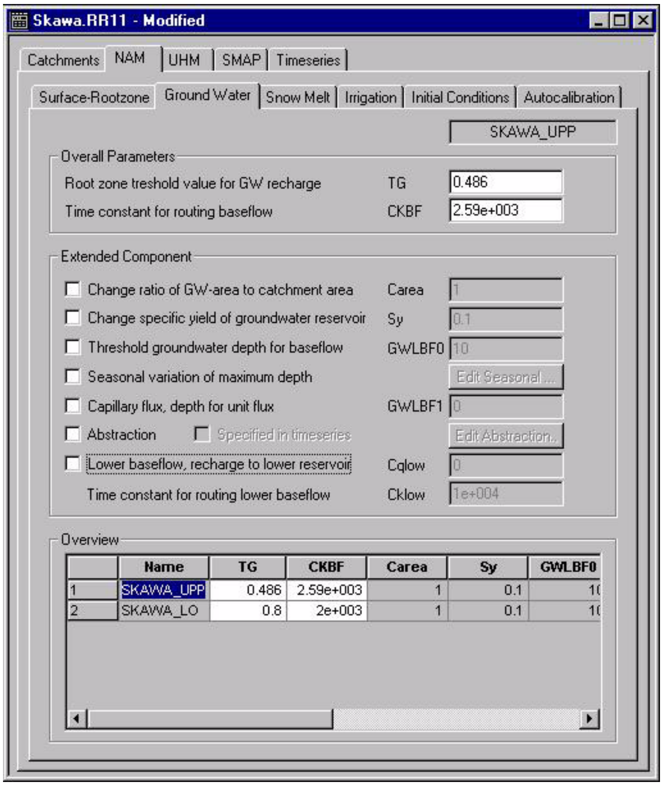


Figure 5.6 NAM - Ground Water.

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, 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 Timeseries page.

The Snow Melt parameters are described below (see Figure 5.7).

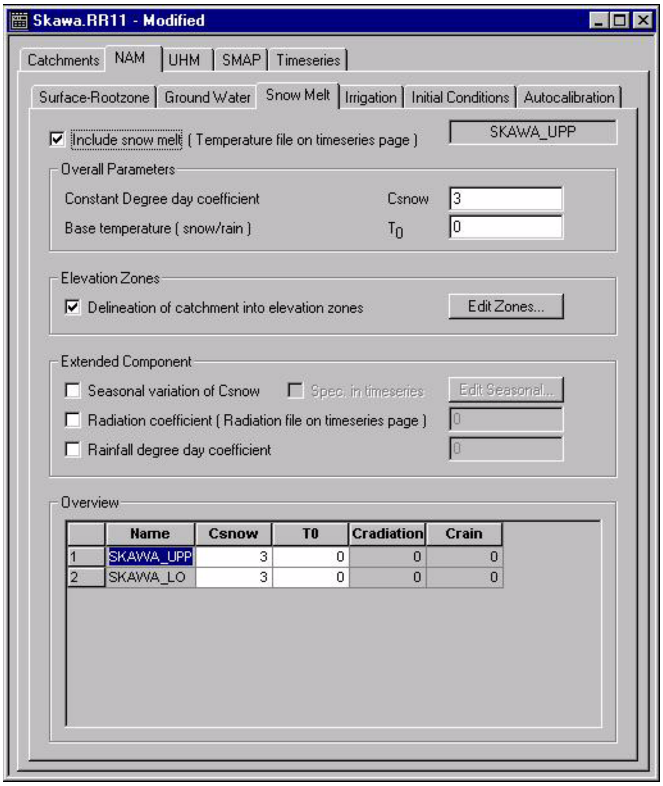


Figure 5.7 NAM - Snow Melt.

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 timeseries 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 timeseries 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 timeseries 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.

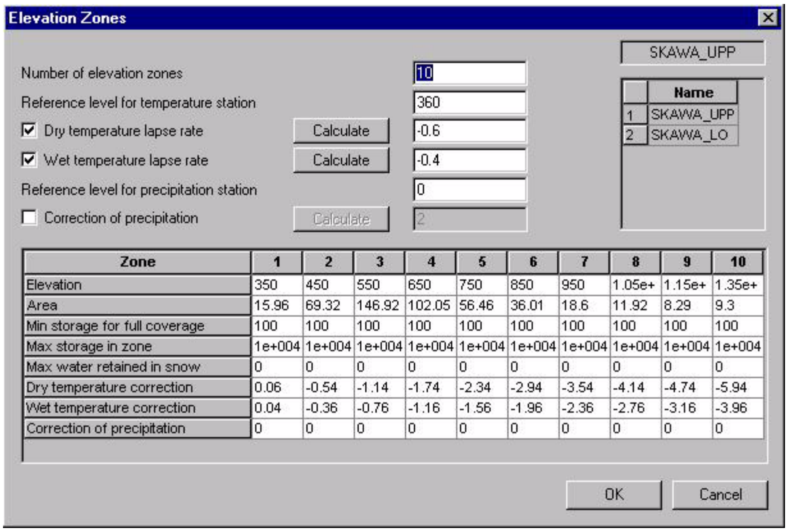


Figure 5.8 NAM - Snow Melt, Elevation Zones.

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 defined to accumulate the runoff.

A time series of applied irrigation must be specified as a rainfall series on the timeseries page.

The Irrigation parameters are described below (see Figure 5.9).

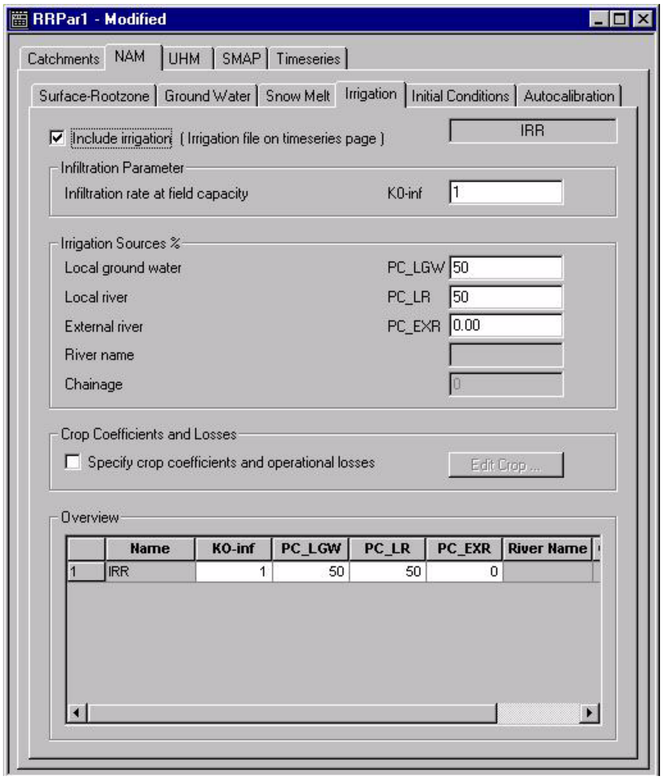


Figure 5.9 NAM - Irrigation.

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 substitutes the standard NAM infiltration calculation, and the overland flow coefficient CQOF and the threshold value TOF are consequently not required, 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).

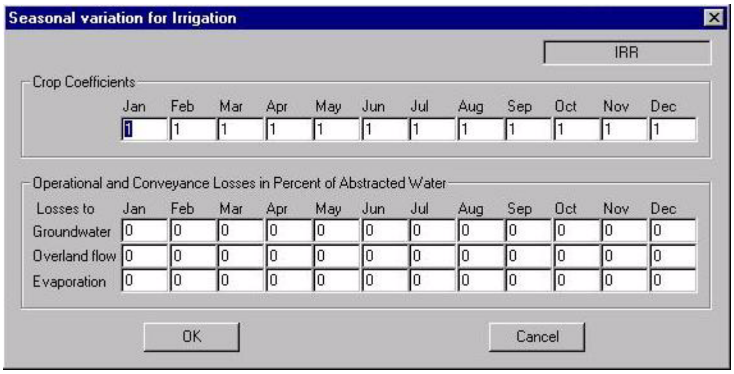


Figure 5.10 Seasonal variation of crop coefficients and losses.

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.

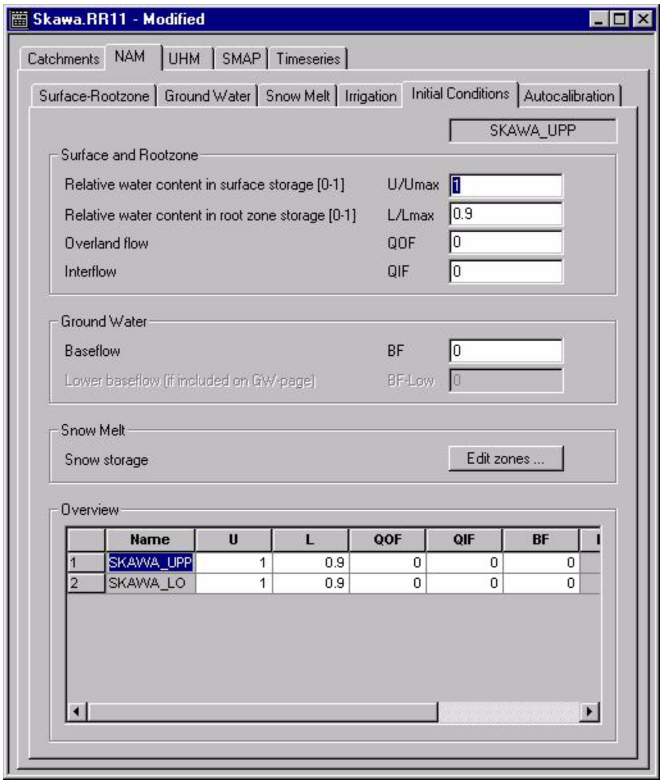


Figure 5.11 NAM - Initial Conditions.

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).

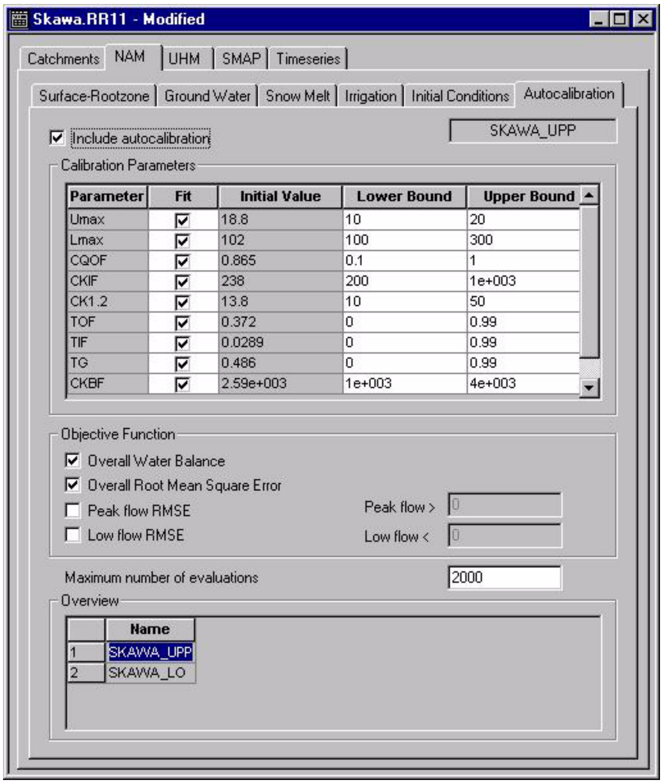


Figure 5.12 NAM - Autocalibration.

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 autocalibration 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).

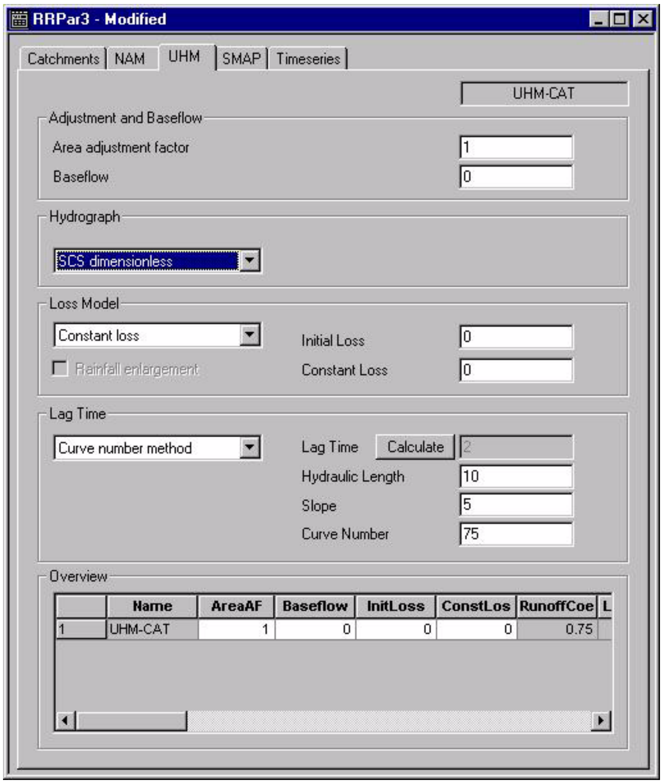


Figure 5.14 UHM Parameters

Areal adjustment and Baseflow

An **areal 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.

Four other methods for describing the hydrograph will be made available in subsequent versions of UHM. These are

– Storage Function

– Quasi Linear Storage Function

– Nakayasu

– Rational method

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 will be made available in subsequent versions of UHM. Theses are

– 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 subtropical 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):

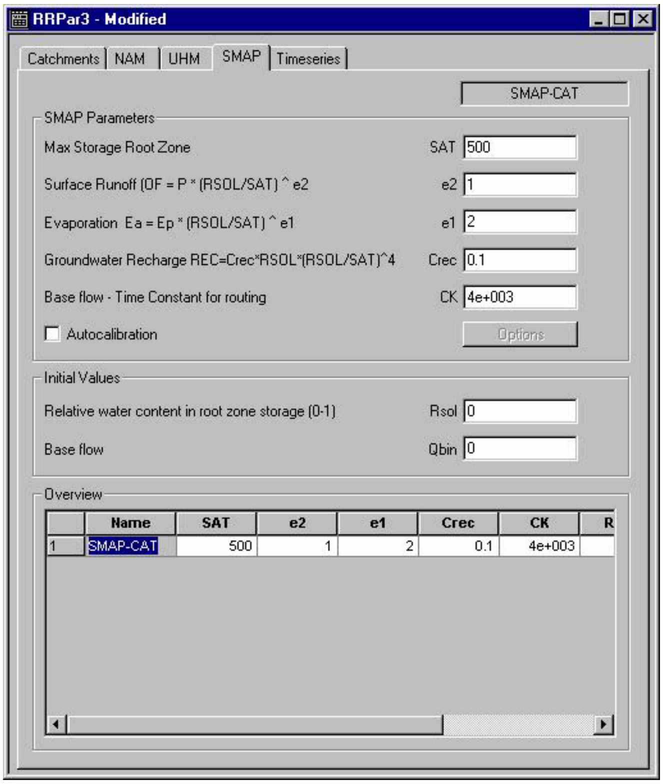


Figure 5.15 SMAP Parameters

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 yet 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, however, that the discharge output in the main result file is in m3/s and represent an instantaneous value at by 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 as two different runoff models:

Model A) Time/area Method

Model B) Non-linear Reservoir (kinematic wave) Method

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)

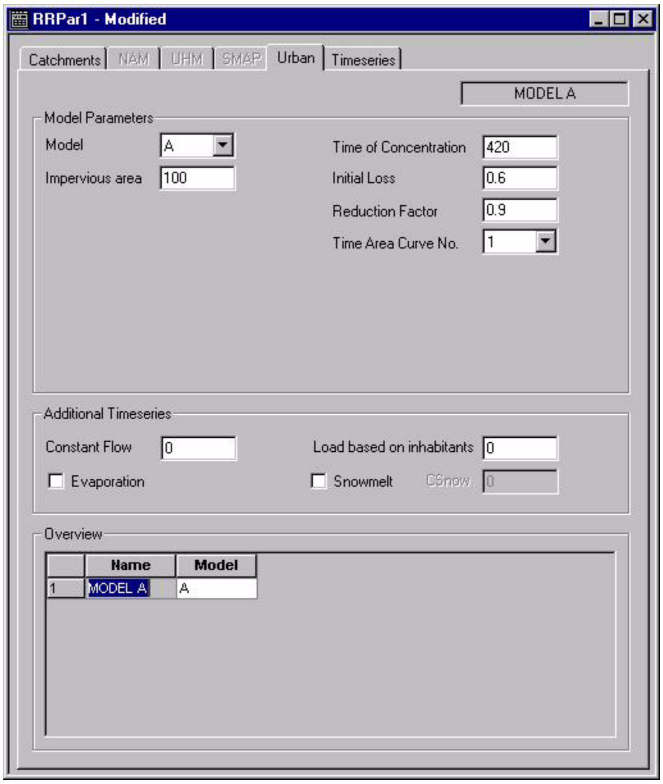


Figure 5.16 Urban Page. Model A, Time/area Method.

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. evapotranspiration, 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

3) convergent catchment

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, taking the gravitational and friction forces only. The runoff amount is controlled by the various hydrological losses and the size of the actually 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 a base for the kinematic wave computation (Manning equation).

The Parameters for Model B are described below (see Figure 5.17)

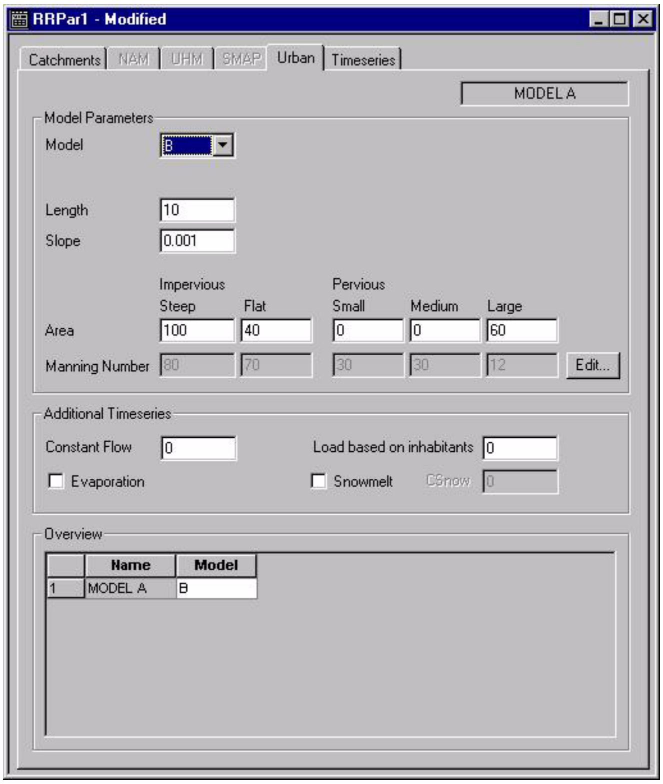


Figure 5.17 Urban Page. Model B, Non-linear Reservoir (kinematic wave) Method

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 for the runoff computation according to Manning.

Area

The area distribution percentages divide the catchment area into 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%.

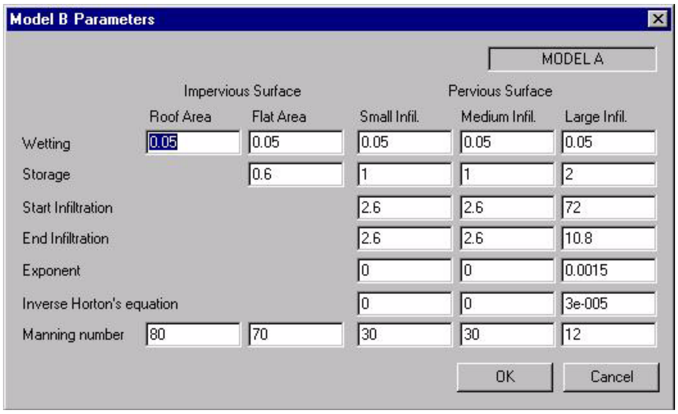


Figure 5.18 Model B, Hydrological Parameters for individual sub-catchments.

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.

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". Determines the dynamics of the infiltration capacity rate reduction over time during rainfall. The actual infiltration capacity is made dependent of 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, i.e. in a drying period.

Manning's number

Describes roughness of the catchment surface, used in hydraulic routing of the runoff (Manning's formula).

5.5.4 Additional Time series

Additional runoff

Additional runoff Evaporation check box - controls if the evapo-transpiration process will be included in the runoff computations can be specified as a constant flow or specified as **load based on inhabitants** (PE). An additional time series for load (qload) is specified on 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

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

Snow melt check box - controls if snow melt is 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 Time Series

The Time series page serves two purposes: Input of time series and calculation of weighted time series (see Figure 5.19)

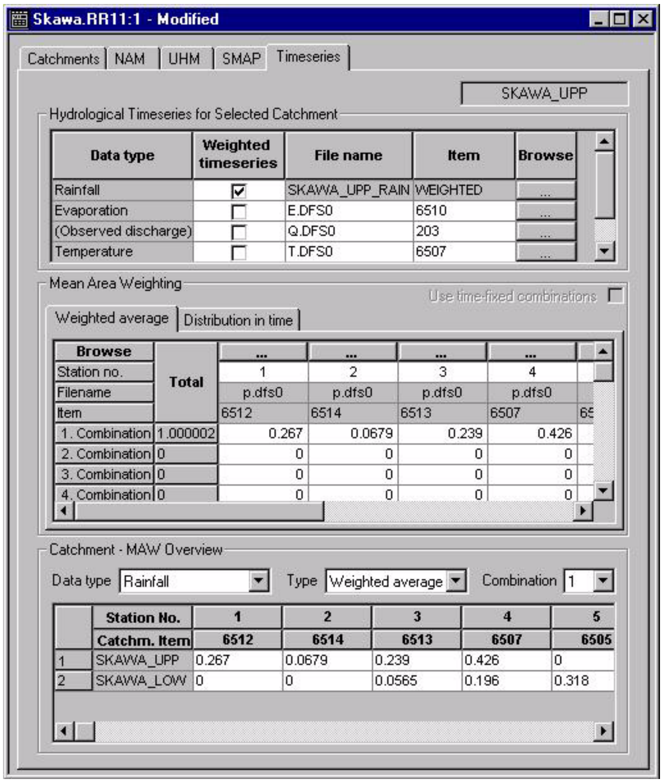


Figure 5.19 Time series Page.

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 saving of the RR-editor file will automatically generate the file **"default.bnd11**" which is used as boundary file on the input page in the simulation editor. 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 should be the rainfall volume accumulated since the previous value.

Evaporation

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

Temperature

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

Irrigation

An input time series is required to provide information on the amount of irrigation water applied, if the irrigation module is included in a NAM simulation

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.

Radiation

A time series of incoming solar radiation can be used as input to the extended snow melt routine.

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.

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 5.8.

Calculation of Weighted time series

This calculation usually needs only 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 periods of the input files specified in “default.bnd11” are known in the simulations editor.

After having calculated the weighted time series once the calculation can be disconnected when 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.

5.7 Basin View

The Basin View provides an 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 as default not activated when a Rainfall Runoff file is opened or created. It is often not required to activate the Basin View for preparation of the RR-file.

5.7.1 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.

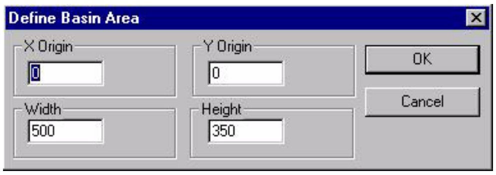


Figure 5.20 Define Basin Area Dialog.

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 same time as the Basin View. This initializes the Rainfall Runoff Editor. The default catchment can afterwards be deleted from the catchment page in the Tabular View, such as the catchments in the Basin View and on the Tabular View are the same.

5.7.2 Importing Layers

The layer management tool is used to import a graphical image used 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.

5.7.3 Basin Work Area

The Basin Work Area dialog selected from the top menu bar contains following facilities (see Figure 5.21).

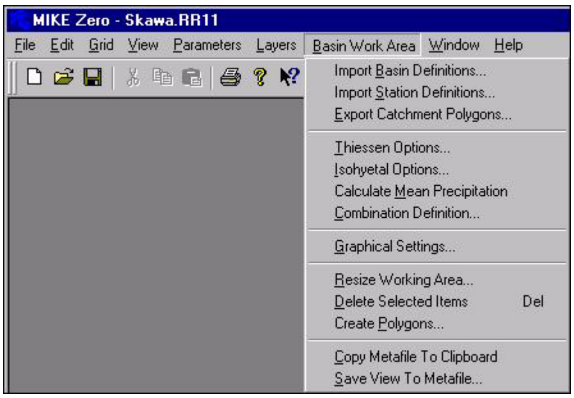


Figure 5.21 Basin Work Area.

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: Number of catchment boundary sections and pairs of (x,y)-coordinates.

Line 2 - Line4 : (x,y) -coordinates for first catchment boundary section.

Line 5: Number of catchment boundary sections and 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 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 number, (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 from the “Thiessen Option”-dialog. Select number 1 for the first combination and press OK (see Figure 5.22). Thiessen weights have now been prepared on the Time series page (see Figure 5.19, Time series page in the Rainfall Runoff Editor).

Figure 5.22 The Thiessen Option dialog.

Apply the weight “-1.00” (for stations with missing data) on the timeseries page before calculating of other combinations.

Showing Thiessen polygons for a catchment on the Basin View:

1 Press the Thiessen icon ( ) on the Basin View toolbar.

2 Right click on the basin view.

3 Select 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 Timeseries page in the Rainfall Runoff Editor. It should therefore be noticed that the Isohyetal Option can not be used to prepare weights and time series of mean area rainfall used as input to the rainfall-runoff calculation. Select the Isohyetal Options to activate the Isohyetal Option dialog (see Figure 5.23). 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 ( ).

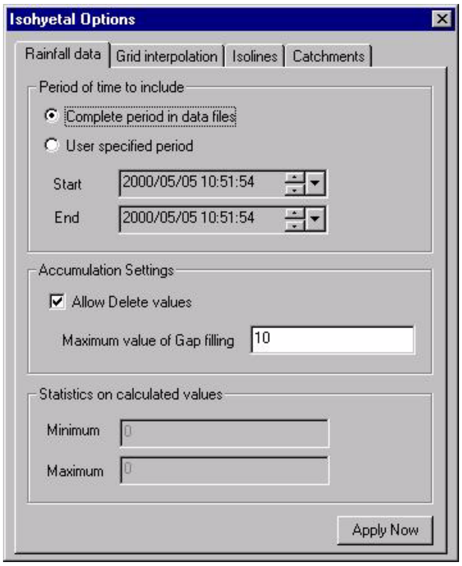


Figure 5.23 Isohyetal Options dialog

Calculate Mean Precipitation

After having prepared the Thiessen weights (see Figure 5.19, 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 5.24).

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.

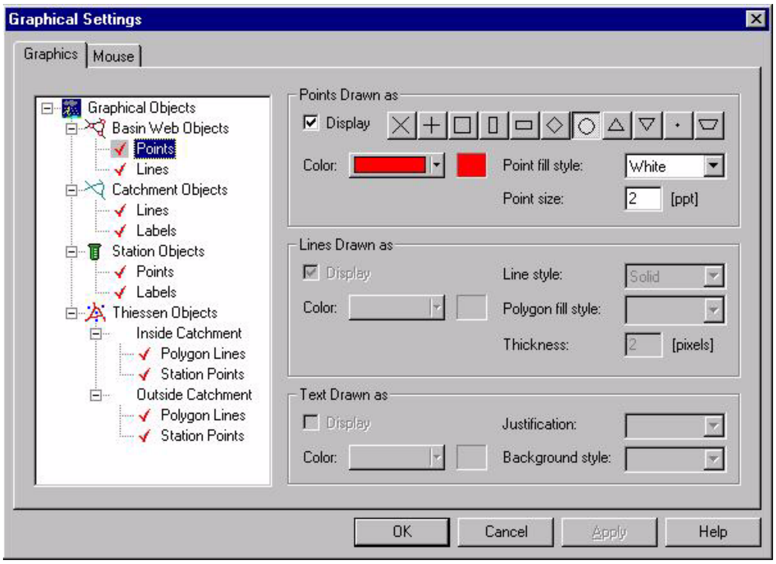


Figure 5.24 Graphical Settings.

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.

5.7.4 *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.

5.7.5 *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 5.25).

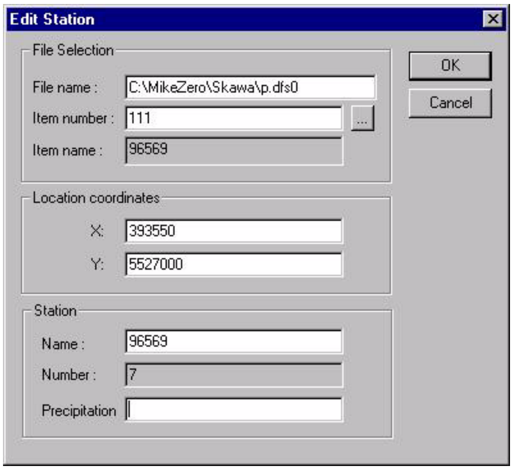


Figure 5.25 Edit Station Dialog.

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”.

5.7.6 *Preparing Thiessen weights*

Thiessen weights are prepared from the menu bar (Basin View | Thiessen Options).

Figure 5.26 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.

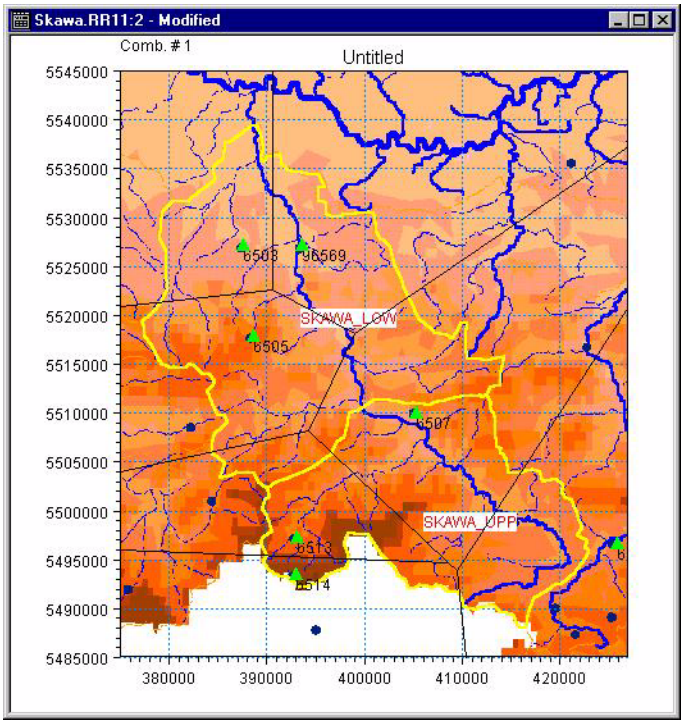


Figure 5.26 Basin View with catchment boundaries, rainfall stations and Thiessen polygons.

5.8 Result Presentation

Results

MIKE11 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. The format of the result file should be selected before running the simulation. Three facilities are available to plot and analyze the **results** of a rainfall-runoff simulation:

1. MikeView. To apply MikeView 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. MikeZero 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. MikeZero Plot Composer. The MIKEZero 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 Timeseries 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 5.27 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 5.27 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 off on the catchment page. The calibration can be loaded from the Plot composed and is saved in the subdirectory RRCalibration with the file name: Catchment-name.plc. The time series in these plots are also available in DFS0 format in the subdirectory RRcalibration with the file name: Catchment-name.dfs0. The plot shows following results (see Figure 5.28):

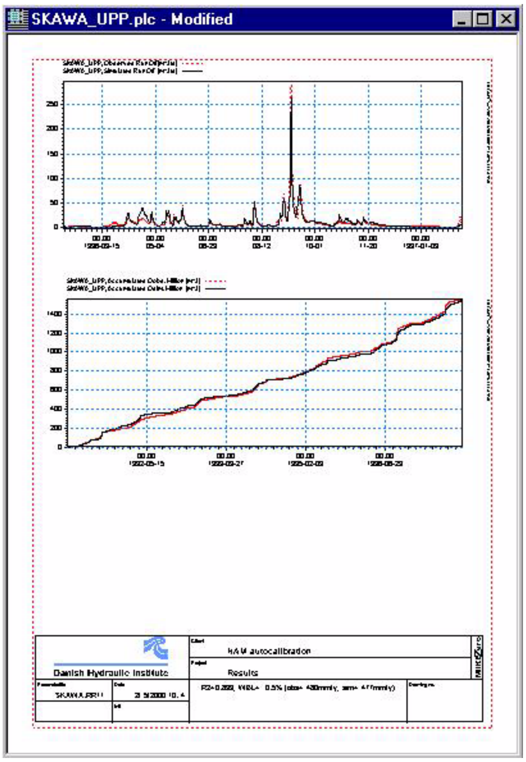


Figure 5.28 Example on a Calibration Plot

- 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 noticed that the calibration plot requires the results saved for each simulation timestep (See Simulation editor, Results Page).

A combined catchment has no input timeseries and is therefore not represented on the Timeseries page. The observed discharge for a combined catchment is therefore included as the observed discharge for the previous catchment on the Timeseries Page.

5.9 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 step were performed:

1 **Opening of a new MIKE11 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 of the Basin View** (select View | Basin View).

3 **Import of a background images** (select Layers | Layers management). The imported image was prepared and geo-referenced from an ArcView application.

4 **Digitising of catchment boundaries**. The catchment was subdivided into two sub-catchments defining the Upper and Lower part of the Catchment (see Section 5.7.4).

5 **Creation of polygon catchments** (see Section 5.7.4), 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 on the two catchment were modified to SKAWA_UPP and SKAWA_LOW and the default catchment was deleted from the Catchment Overview.

6 **Setup of a combined catchment**. A Combined catchment was defined as the sum of the two sub-catchments (see Figure 5.3).

7 **Inserting of the rainfall station**s. Stations included in the calculation of catchment rainfall were included in the Basin View (see Section 5.7.5).

8 **Preparation of Thiessen Weights** (see Section 5.7.3, Thiessen Options). The calculated Thiessen Weight Polygons are shown on Figure 5.26, which also shows the two sub-catchments. The weights

which were automatically transferred to the Time series Page are shown on Figure 5.19.

9 **Calculation of Mean Precipitation**. The Weighted timeseries were calculated based on the weights prepared as described in the previous step.

10 **Setup of other input time series**. Input Time series for Evaporation, Temperature and Observed Discharge were included on the Time series Page (Figure 5.19).

11 **Setup of 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 included in an ArcView application. 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).

13 **Setup of 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 **Start of simulation editor**. After having saved the Rainfall Runoff Parameters, a MIKE11 simulation editor was opened. The Input page includes the RR Parameter file and the default boundary file (see Figure 5.1). The Simulation period was prepared from the Apply default button and a time step on 12 hours were found appropriate for the simulation (see Figure 5.2).

15 **Estimation of 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 on 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 on 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 **Presentation of Results**. Results from the simulation were finally compared in tables and on plots. Figure 5.27 shows example on summarised output from the upper catchment, while Figure 5.28 shows the calibration plot for the upper catchments

HYDRODYNAMIC EDITOR

6 HYDRODYNAMIC 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 editor has a number of tabs which are listed below and described in the following:

- Quasi Steady *(p. 223)*
- Add. Output *(p. 225)*
- Flood Plain Resistance *(p. 229)*
- Initial *(p. 230)*
- Wind *(p. 231)*
- Bed Resistance *(p. 232)*
- Bed Resistance Toolbox *(p. 235)*
- Wave approx *(p. 237)*
- Default values *(p. 238)*
- User Def. Marks *(p. 240)*
- Encroachment *(p. 242)*
- Mixing Coefficients *(p. 247)*
- W. L. Incr.- Curve *(p. 249)*
- W. L. Incr.- Sand Bars *(p. 251)*
- Heat Balance *(p. 253)*
- Stratification *(p. 255)*

6.1 Quasi Steady

Various parameters required for the quasi steady simulation to be carried out are set here.

6.1.1 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.

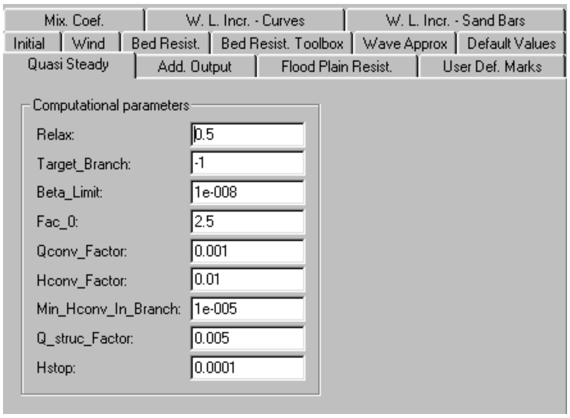


Figure 6.1 The Quasi Steady property page.

Target_Branch

Computed water levels/discharges are shown on the screen at each iteration for branch number equal to ‘Target Branch’. No computations are shown if ‘Target Branch’ is negative.

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.

6.2 Add. Output

A number of simulated parameters can be selected for storage in an additional output result file (with the file name extension 'RES11'). 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 MIKE11 result files.

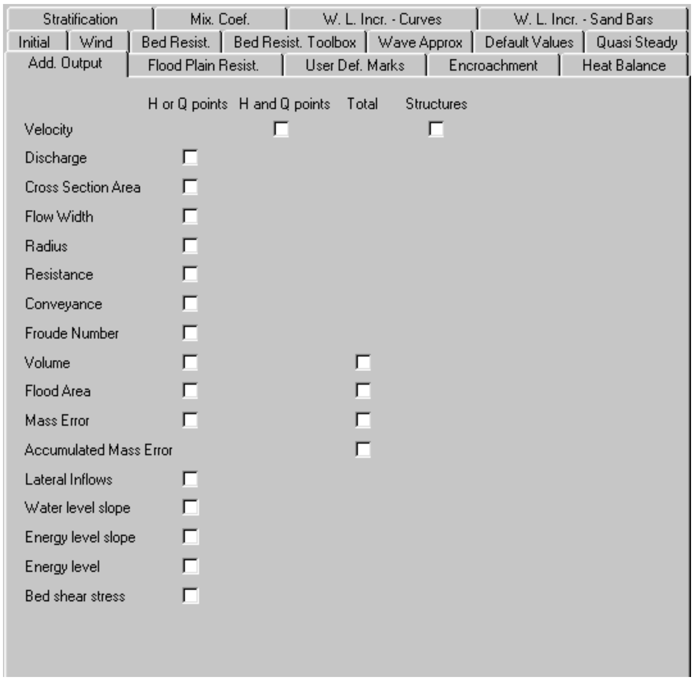


Figure 6.2 The additional output property page.

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:

$$F = \frac{Q}{A\sqrt{g\frac{A}{b_s}}} \tag{6.1}$$

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 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.

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 at water level points given as

$$\tau = \rho g R \frac{\mathrm{d}E}{\mathrm{d}x} \tag{6.2}$$

where E is the energy level and x is the longitudinal coordinate along the river.

6.2.1 Additional output for QSS with vegetation

Please note that when utilizing the quasi two dimensional steady state with vegetation module, additional data can be obtained by setting the following switch in the "mike11.ini" file:

CREATE_QSSVEG_VELOCITY_FILE=ON

With this setting 8 .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 ρ.
- 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.

6.3 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 6.3): 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.

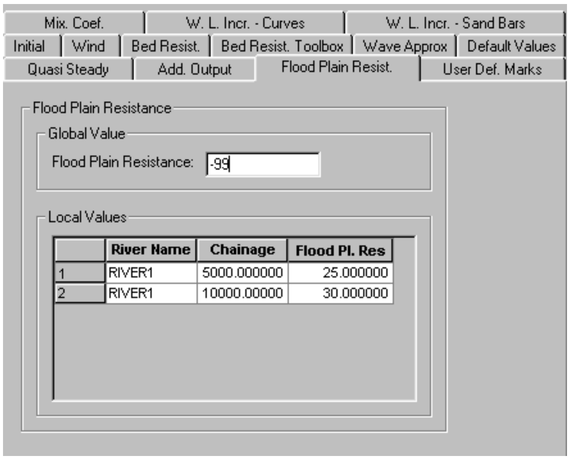


Figure 6.3 The Flood Plain Resistance property page.

6.4 Initial

Initial conditions for the hydrodynamic model are specified on this page. 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. Local values will override the global specification.

Example (Figure 6.4): The global water level and discharge have been specified as 5.00 and 1.400 respectively. Local values have been specified in the branch "RIVER 1". The local initial water levels vary from 5.70 to 5.00 with a linear relationship between chainage 0 and 3000. The discharge also varies between 1.000 and 1.400 with a linear relationship over the 3000 branch length.

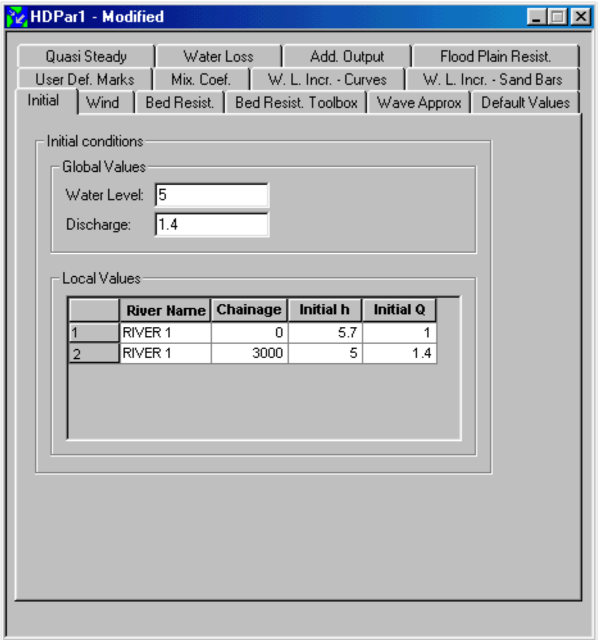


Figure 6.4 Initial value tab

6.5 Wind

Wind fields can be applied to the entire model network using the wind property page of the HD editor. The property page contains an "on/off" switch a global wind factor and a table of local wind factors. A wind field is applied globally to the model using a hydrodynamic boundary file (.bnd11) and can be scaled by using the global and local factors section.

Example (Figure 6.5): The global wind factor is set to 0.70. It varies linearly from 0.70 to 0.30 in the branch named "RIVER 1" from chainage 0 to 5000.

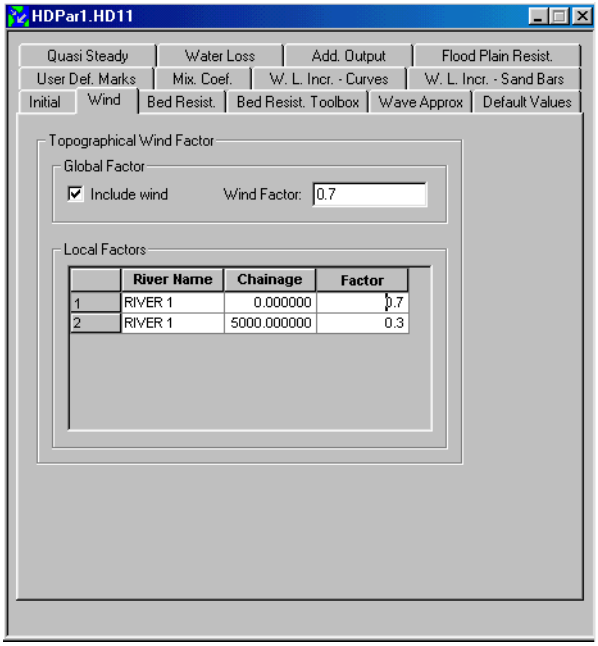


Figure 6.5 Wind tab.

6.6 Bed Resistance

Two approaches may be applied for the bed resistance. Either a uniform or a triple zone approach can be specified.

6.6.1 Uniform approach

The bed resistance is defined by a type and a corresponding global value. Local values are entered in tabular form at the bottom of the editor.

There are three resistance type options:

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.

The resistance number is specified in the parameter 'Resistance Number'. This number is multiplied by the water level depending 'Resistance factor' which is specified for the cross sections in the cross section editor (.xns11 files) to give a resulting bed resistance.

Example (Figure 6.6): 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.

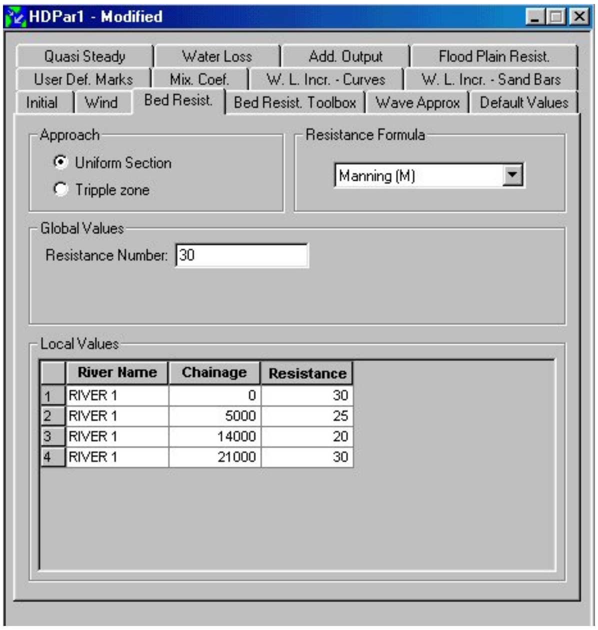


Figure 6.6 Uniform approach for implementation of the bed resistance.

6.6.2 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 6.7

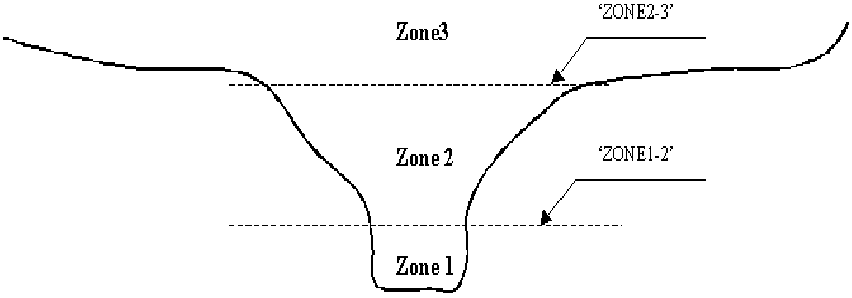


Figure 6.7 Triple Zone division of cross section

Zone separator lines must be defined in the User Defined Markers page (see description in Activation of Bed resistance Triple Zone Approach *(p. 241)*).

Global and local values of bed resistance for each zone can be specified as described for the Uniform approach.

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.

6.6.3 Vegetation and bed resistance

Only few detailed investigations have been made to establish relationships between flow resistance and vegetation growth. A quantitative evaluation of the vegetation's influence on the flow resistance has been performed in a few danish gauging-programmes. These are referred to in A.1 Flow Resistance and Vegetation *(p. 373)*.

6.7 Bed Resistance Toolbox

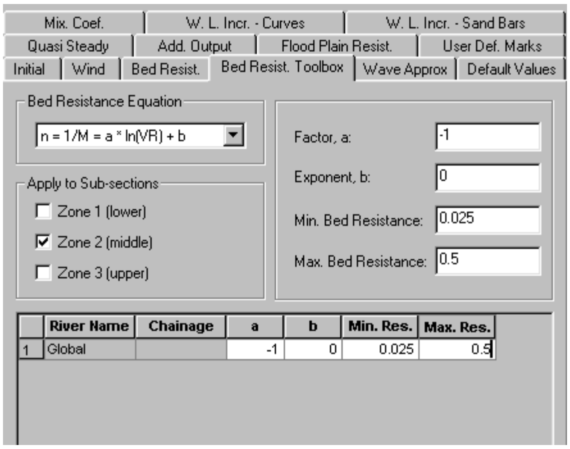


Figure 6.8 The Bed Resistance Toolbox property page.

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.

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). The velocity and the radius is calculated in metric units i.e. m/s and m.

- n = 1/M = a*D^b

  The bed resistance is calculated as a function of the Water depth. The water depth used is evaluated in meters and is the mean depth across the whole cross section.

- n = 1/M = a*V^b

  The bed resistance is calculated as a function of the velocity. The velocity is the mean cross sectional velocity evaluated in m/s.

- 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.

6.8 Wave approx

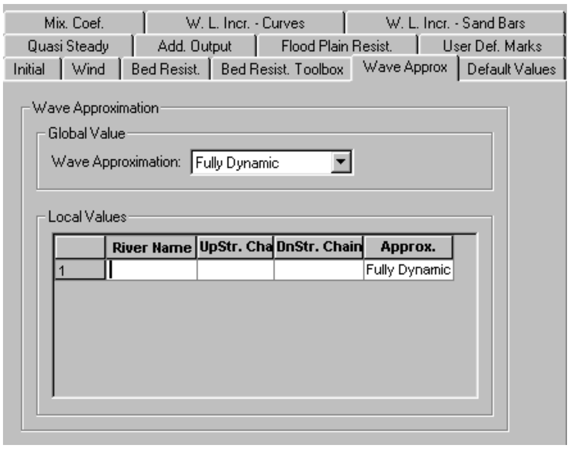


Figure 6.9 The Wave Approximation property page.

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).

6.8.1 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 allows simulations to be performed at longer time steps than the ‘fully dynamic’ description.

6.8.2 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.

6.8.3 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.

6.9 Default values

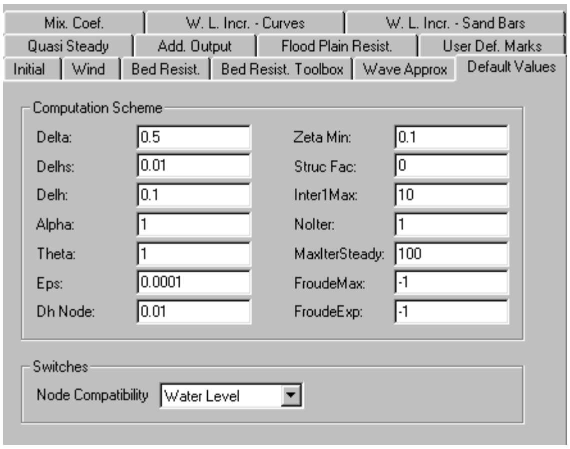


Figure 6.10 The Default Values property page.

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. The following brief descriptions are provided (see also section Coefficients, HD default parameters in the Reference Manual).

6.9.1 Computation Scheme

Delta

The time-centring of the gravity term in the momentum equation.

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

The maximum number of iterations permitted at each time step to obtain a solution at a structure.

Number of Iter

The number of iterations at each time step, generally 0, 1 or 2.

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.

6.9.2 Switches

Node Compatibility

This switch should be set to water level since the energy compatibility has not yet been implemented.

6.10 User Def. Marks

The User Defined Markers page offers a possibility for the user to define special 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. Markers could be the location of an important hydraulic structure, a gauging station 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 column 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 interpolation 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 un-checked.

6.10.1 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.

Please 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 6.11shows 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.

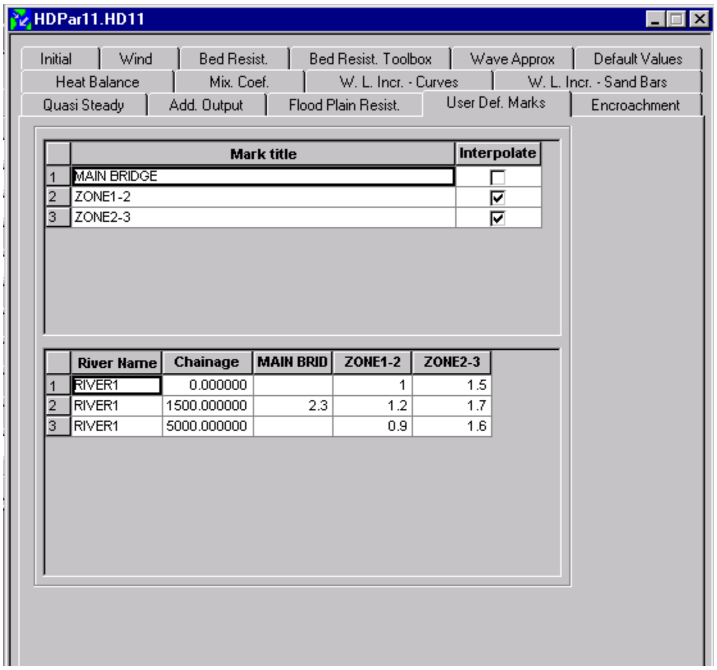


Figure 6.11 Example of defining User Defined Marks

6.11 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 *(p. 246)*. 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.

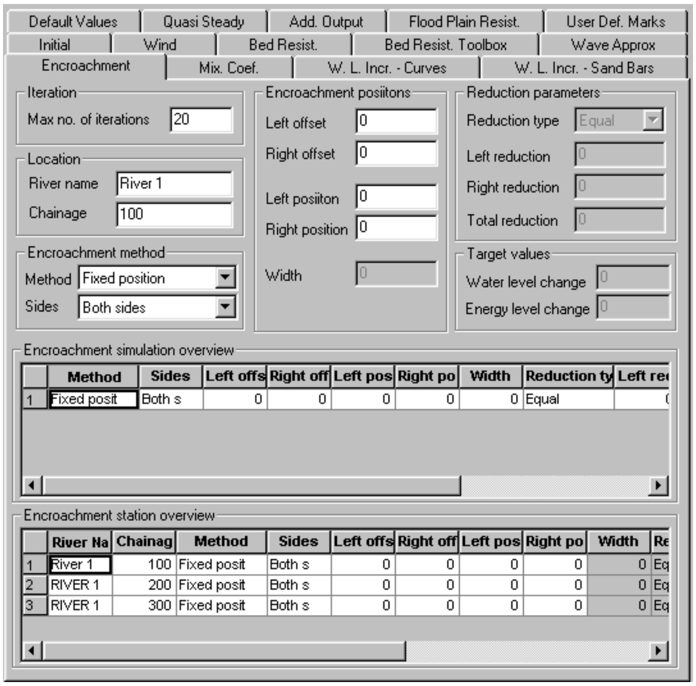


Figure 6.12 The encroachment property page.

6.11.1 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.

6.11.2 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.

6.11.3 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.

6.11.4 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 8 and 9.

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.

6.11.5 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 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.

6.11.6 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.

6.11.7 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.

6.11.8 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).

6.11.9 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.

6.12 Mixing Coefficients

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.

Local values are shown at the bottom in table form.

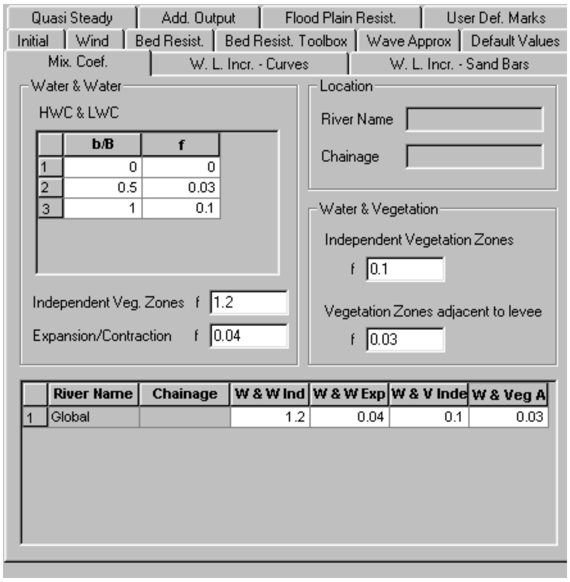


Figure 6.13 The Mixing Coefficients property page.

6.12.1 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.

6.12.2 Location

The river name and location (chainage) is displayed here.

6.12.3 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.

6.13 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 6.14 with all the different features all of which are described below.

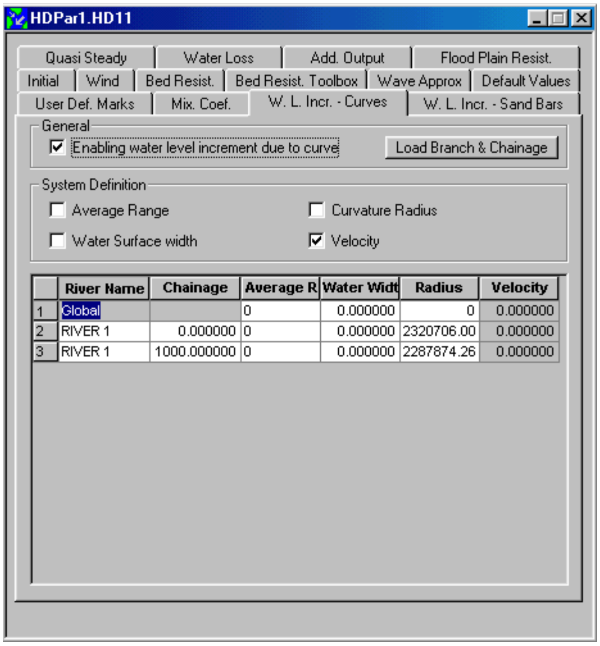


Figure 6.14 Water level increment due to curves.

6.13.1 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.

6.13.2 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.

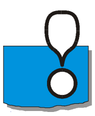
Note! If either 2 or 3 is ticked the velocity is also automatically ticked.

6.13.3 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.

6.14 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 6.15 with all the different features all of which are described below.

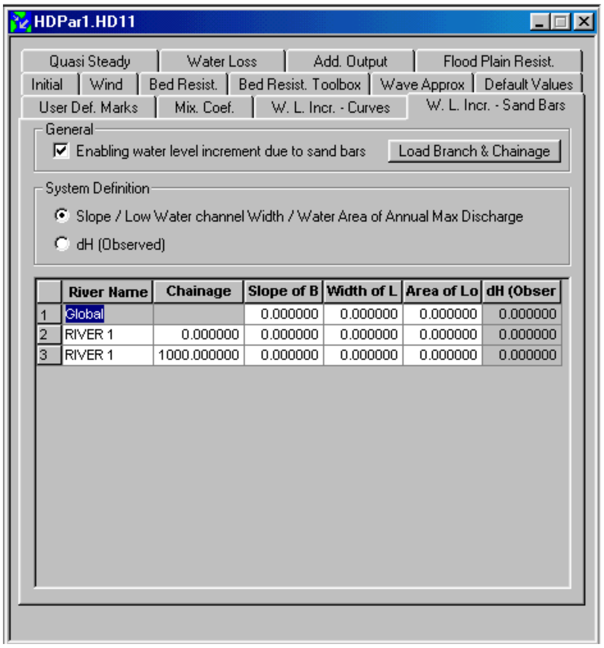


Figure 6.15 Water level increment due to sand bars.

6.14.1 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.

6.14.2 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.

6.14.3 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.

6.15 Heat Balance

The property page used for setting up heat exchange simulations is illustrated in Figure 6.16.

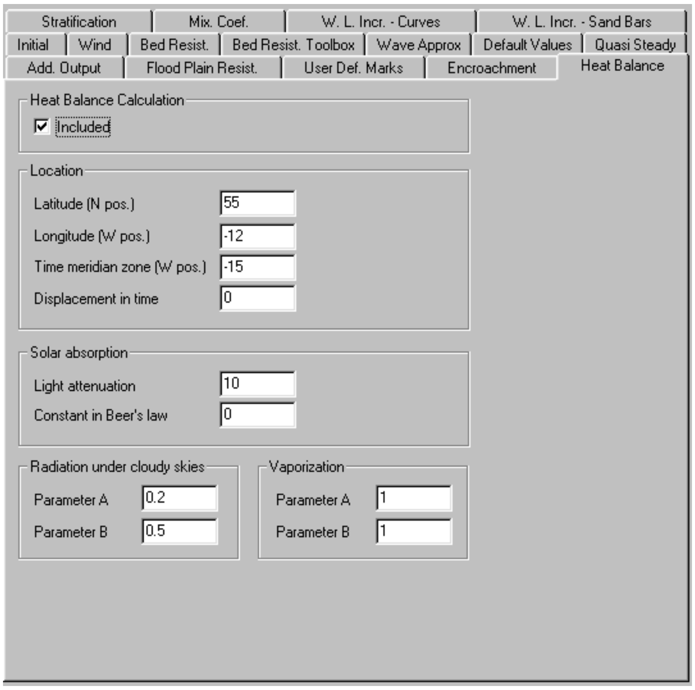


Figure 6.16 The heat balance property page.

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:

$$E_{fac} = \frac{I_{sun}}{I_{tot}}(1-\beta)\exp(-a(D-z)) \tag{6.3}$$

where I_{sun} is the solar radiation, β is constant in Beer's law, (D-z) is distance from surface and a is light absorption.

Radiation Parameter *A*

Daily radiation under cloudy skies is determined by:

$$\frac{H}{H_0} = A + B\frac{n}{N_d} \tag{6.4}$$

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:

$$q_e = LC_e(A + BW_2)(Q_W - Q_a) \tag{6.5}$$

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.

6.16 Stratification

The property page used for setting up stratified flow simulations is illustrated in Figure 6.17.

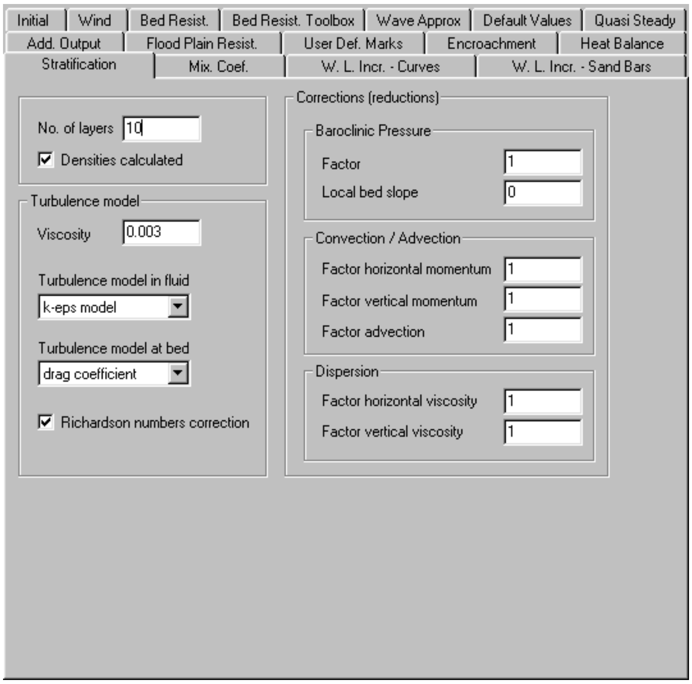


Figure 6.17 The stratification property page.

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-ε turbulence models. It is recommended to use the k-ε 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.

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 module
- Cohesive sediment transport module
- Advanced cohesive sediment transport module

A brief description of each of these modules is provided below.

7.0.1 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. 262)* 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 very steep fronts.

7.0.2 Water Quality module (WQ)

The water quality (WQ) module deals with the basic aspects of river water quality in areas influenced by human activities: e.g. oxygen depletion and ammonia levels as a result of organic/nutrient loadings. The WQ-module is coupled to the AD module, which means that the WQ module deals with the chemical/biological transforming processes of compounds in the river and the AD module is used to simulate the simultaneous transport process. The WQ module solves a system of coupled differential equations describing the physical, chemical and biological interactions in the river. The relevant water quality components must be defined in the AD editor.

7.0.3 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.

7.0.4 Advanced Cohesive Sediment Transport module (A CST)

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.

7.0.5 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:

$$\frac{\partial AC}{\partial t} + \frac{\partial QC}{\partial x} - \frac{\partial}{\partial x}\left(AD\frac{\partial C}{\partial x}\right) = -AKC + C_2 q \qquad (7.1)$$

where

C : concentration

D : dispersion coefficient

A : cross-sectional area

K : linear decay coefficient

C_2 : source/sink 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/sink term is considered to mix instantaneously over the cross-section.
- The substance is conservative or subject to a first order reaction (linear decay)
- Fick's diffusion law applies, i.e. the dispersive transport is proportional to the concentration gradient.

To operate the AD-module a number of dialogs are available all of which are described in the following.

7.1 Sediment layers

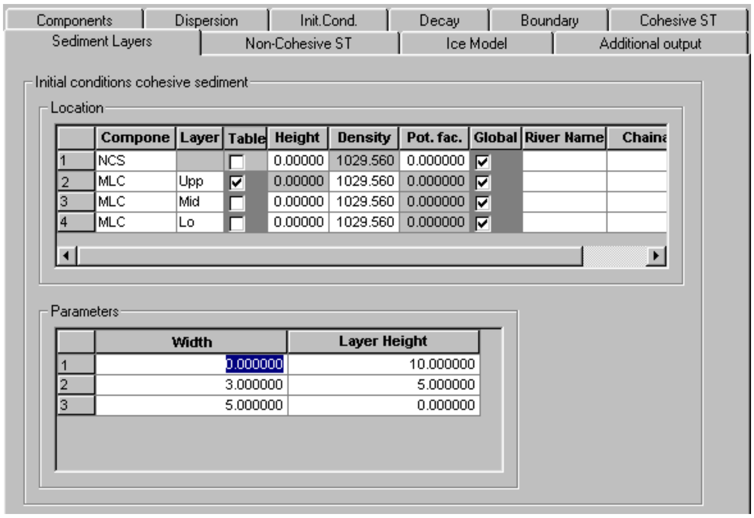


Figure 7.1 The Sediment Layers property page.

Initial conditions for the sediment layers are defined on the Sediment Layers page. Selection pop down menus are available for the component types 'Single cohesive', 'Multi cohesive' or 'Non cohesive'.

Location

Component

Three types can be selected: Single Layer Cohesive, Multi Layer Cohesive and Non-Cohesive.

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. 265)* a more detailed initial condition can be specified using a width-Height table, see Parameters *(p. 265)*.

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. 266)* property page.

Density

The density of the layer.

Pot. fac.

Initial amount of BOD attached to the sediment. Only applicable for a Single Layer component.

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 cross section.

7.1.1 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:

Potency factor

Initial amount of BOD attached to the sediment (kg BOD / kg sediment).

7.2 Non-cohesive ST

Figure 7.2 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.

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.

7.3 Ice model

This property page contains parameter information for the MIKE 11 ice module. The following parameters must be specified:

- Active ice model
- Constant cross section area

- Latitude
- Latent heat
- Specific heat of water
- Density of water
- Heat flux
- Ice density
- Air temperature
- Wind speed
- Cloudness
- Visibility
- Cloud density
- Precipitation
- Ice thickness
- Ice cover
- Ice quality

The latter three parameters can also be given local values in the grid control.

7.4 Additional output

The additional output page contains check boxes which can be used to store internal model parameters in result files with the extension (.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.

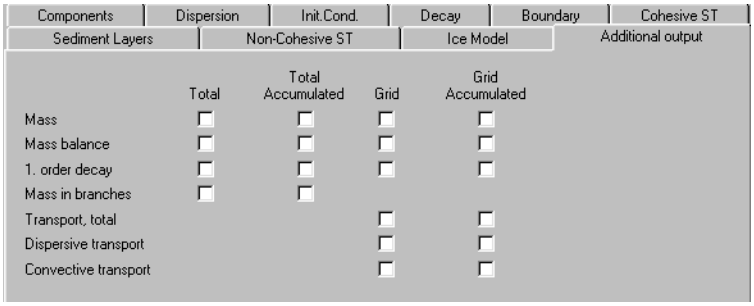


Figure 7.3 The additional output property page.

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.

7.5 Components

Component names and numbers must be specified in this dialog.

The components can be user defined or selected using the pre-defined component sets provided with the water quality module. Each component is modelled using a defined concentration ‘unit’ and ‘type’.

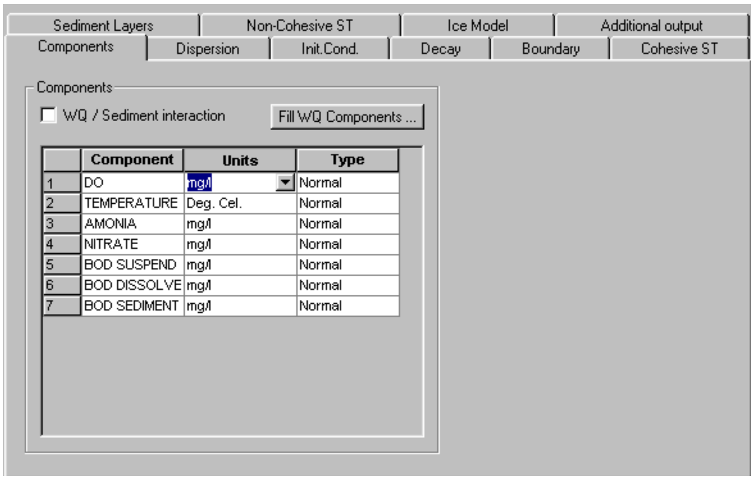


Figure 7.4 The component property page.

WQ / Sediment interaction

If this check box is checked Mike11 will include the exchange of BOD between the water and the sediment. Both cohesive sediment and non-cohesive sediment will be included. All together four components will be added to the component list:

- **COHE**: Cohesive sediment. Type must be ‘Single Layer Cohesive’.
- **COHE BOD**: BOD attached to cohesive sediment. Type must be ‘Normal’.
- **NON_COHE**: Non-cohesive sediment. Type must be ‘Non-Cohesive’.
- **NON-COHE BOD**: BOD attached to non-cohesive sediment. Type must be ‘Normal’.

Fill WQ components

By selecting the Fill WQ components button a number of pre-defined component sets for the water quality modules can be accessed.

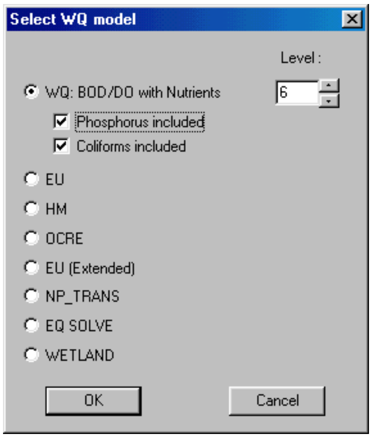


Figure 7.5 Selecting different WQ model components.

A short description of the WQ model types is listed below.

- **BOD/DO**: Components used for the standard water quality (WQ) model. Up to 6 levels can be chosen using the levels option. Coliformal bacteria and phosphorus components can also be included.
- **EU**: Components used for the eutrophication module.
- **EU extended**: Components used for the extended eutrophication (EU) module.
- **HM**: Components used for the heavy metal (HM) module.
- **OCRE**: Components used for the iron-oxidation(OCRE) module.
- **NP_TRANS**: Components used for the nutrient transport module.
- **EQ SOLVE**: Components used for the equation solver module.

Component

Here all components for AD and/or WQ simulations are defined.

Units

Here the unit of the component is specified.

- **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 Celcius.
- **Counts x 1E6/100 ml**: Bacterial counts.

Type

- **Normal**: A component used for AD and/or WQ simulations.
- **Single layer cohesive**: A component used only in the single layer cohesive sediment transport model.
- **Multi cohesive**: A component used in the multi layer cohesive sediment transport model.
- **Non-cohesive**: Used only if WQ/Sediment interaction is chosen, see WQ / Sediment interaction *(p. 270)*. Note that this non-cohesive sediment model can not be used for morphological simulations. It is only used to simulate the exchange between the water and the sediment of BOD attached to the sediment.

7.6 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).

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.

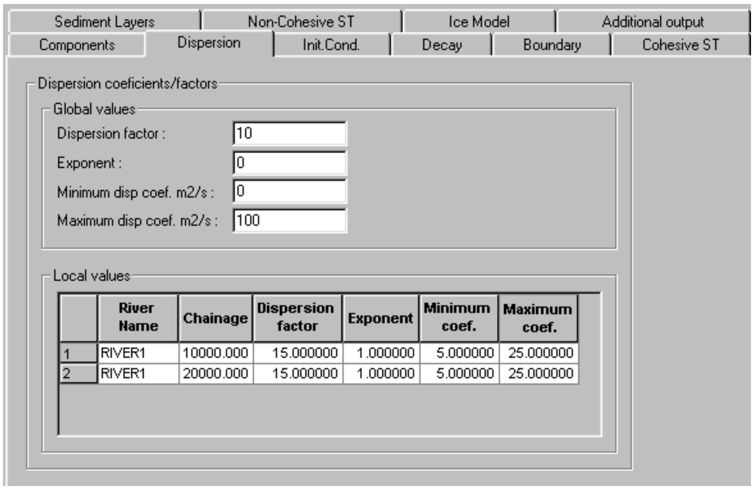


Figure 7.6 The dispersion property page.

Global values

The dispersion can be defined for the whole setup at once by entering data in the Global Values section.

Dispersion factor

Here the dispersion factor is entered. This corresponds to a in (7.2).

Exponent

Here the dispersion exponent b from (7.2) is entered.

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 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 can be entered here.

Local Values

Mike11 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 7.6 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). In the reach between chainages 10000 m and 20000 m the dispersion coefficient is dependent on the velocity ($D = 15V$, $5 < D < 25$)

7.7 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.

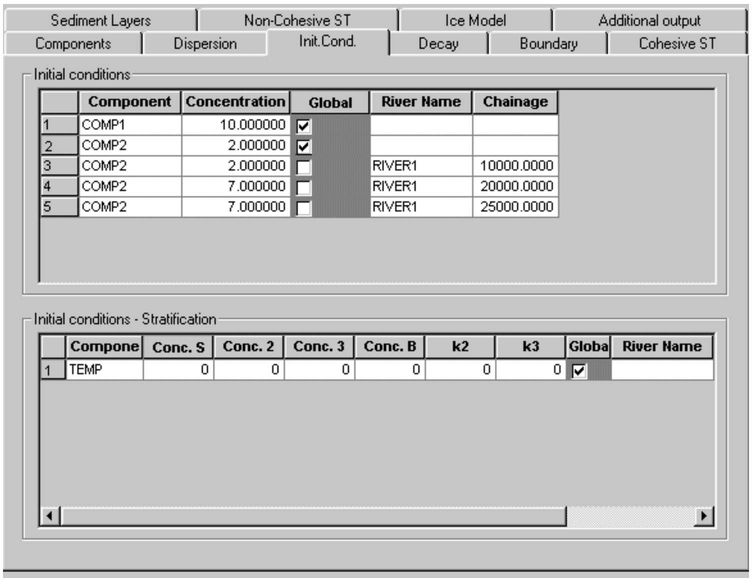


Figure 7.7 The initial conditions property page.

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, see Components *(p. 270)*.

Concentration

Here the value of the initial condition is entered.

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 7.7 two components are simulated, COMP1 and COMP2. The initial concentration of COMP1 is set to 10.00 for the entire river network. The initial concentration of COMP2 is set globally with a value of 2.00. However, the initial concentration of COMP2 varies linearly between 2.00 and 7.00 in the branch ‘RIVER 1’ from chainage 10000 to 20000. From chainage 20000 to 25000 the initial concentration of COMP2 is 7.00.

Initial conditions - stratification table

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.

7.8 Decay

This page contains information for non-conservative components. These components are assumed to decay according to a first-order expression:

$$\frac{dC}{dt} = KC \tag{7.3}$$

Where K is a decay constant. C is the concentration. Both global and local values of the decay constant K can be specified. **NOTE** If the components selected are used for a water quality simulation (WQ) then decay constants should not be specified.

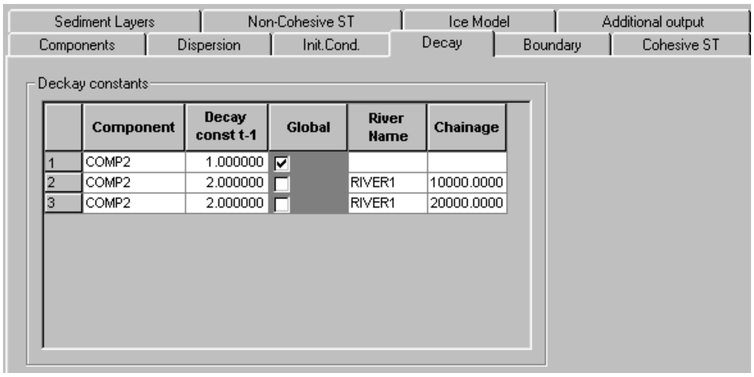


Figure 7.8 The Decay property page.

Component

Here the component in question is selected. It is possible to choose between the components defined in the Components property page, see Components *(p. 270)*.

Decay const

Here the value of the decay constant are entered.

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 7.8 the component COMP2 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.

7.9 Boundary

At all external model boundaries a concentration condition must be specified for all components. An external boundary can be defined as one of three types:

- Open concentration boundary
- Open transport boundary
- Closed boundary

7.9.1 Which boundary description to use?

Open concentration boundary

Open concentration boundary conditions should be applied at locations where outflow (of water and component mass) from the model area occurs. Open concentration boundaries in the AD model correspond to open/water level or Q-h boundaries in the HD model. At each open boundary, a time series of the concentration must be specified in the boundary editor (.bnd11 files).

Time varying and/or constant concentration inflow boundaries are developed in a similar way to that of discharges and water levels in the hydrodynamic model. When an outflow boundary becomes an inflow boundary (during flow reversal at the boundary), the boundary condition concentration is adjusted according to:

$$C = C_{bf} + (C_{out} - C_{bf})e^{-t_{mix}k_{mix}} \tag{7.4}$$

where

C_{bf} the boundary concentration (specified in the input)

C_{out} the concentration at the boundary immediately before the flow direction changed

K_{mix} a time scale specified in the input, unit: hours^{-1}

t_{mix} the time since the flow direction changed

When flow occurs out of the model, the concentrations at the boundary point are computed within the AD model. For flow into the model (e.g. at flow reversal in tidal applications), the specified boundary concentrations are used. (These inflows are assumed to be unaffected by the previous model outflows).

The parameter, K-mix, is used to ensure a smooth transition between calculated and specified boundary concentrations in the case of a flow reversal.

Open transport boundary

Open transport boundary conditions should be used at boundaries 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. It is important to note that the computed concentration at the boundary point can therefore differ from the concentration specified in the boundary file. The open boundary outflow condition is defined as,

$$\frac{\partial^2 C}{\partial x^2} = 0 \tag{7.5}$$

This condition is applied for both Open concentration and Open transport boundaries.

Closed boundaries

Closed boundaries occur where no mass is transported in or out of the model area. A closed boundary condition should only be specified in this menu if a similar boundary condition has been specified in the hydrodynamic computation (i.e. $Q = 0$) Closed boundary points do not need to require concentration time series. This boundary condition is characterized by a zero discharge and is defined as:

$$\frac{\partial C}{\partial x} = 0 \tag{7.6}$$

(i.e., there is no transport through the boundary.)

7.9.2 Entering the data

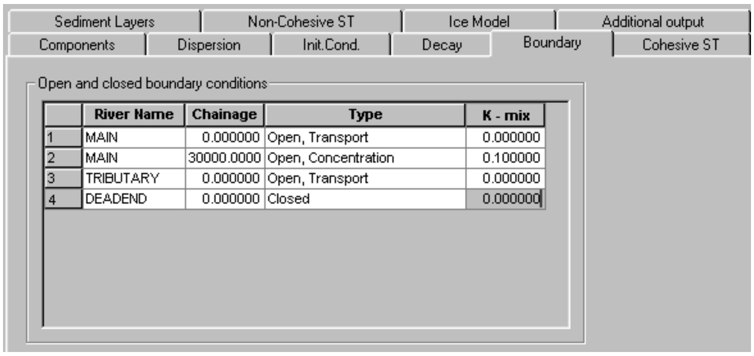


Figure 7.9 The Boundary property page.

Open and closed boundary conditions

River Name

Name of the river for which boundary type is specified.

Chainage

Chainage in river for which boundary type is specified.

Type

- Closed, see description at Closed boundaries *(p. 279)*.
- Open, Concentration, see description at Open concentration boundary *(p. 278)*.
- Open, Transport, see description at Open transport boundary *(p. 279)*.

K-mix

Here K_{mix} from (7.4) is entered.

7.10 Cohesive ST

Data used for the cohesive sediment transport models 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.

7.10.1 Single Layer Cohesive Model

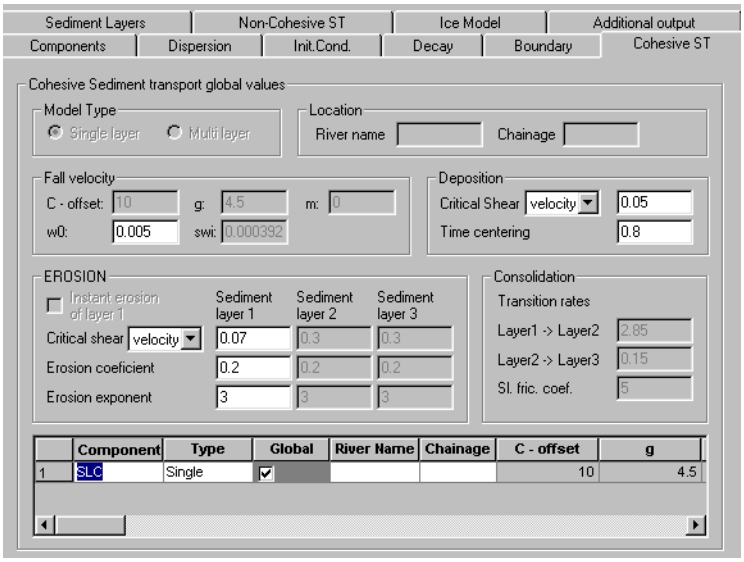


Figure 7.10 The Cohesive sediment property page when a single layer model is selected.

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.

7.10.2 Multi Layer Cohesive Model

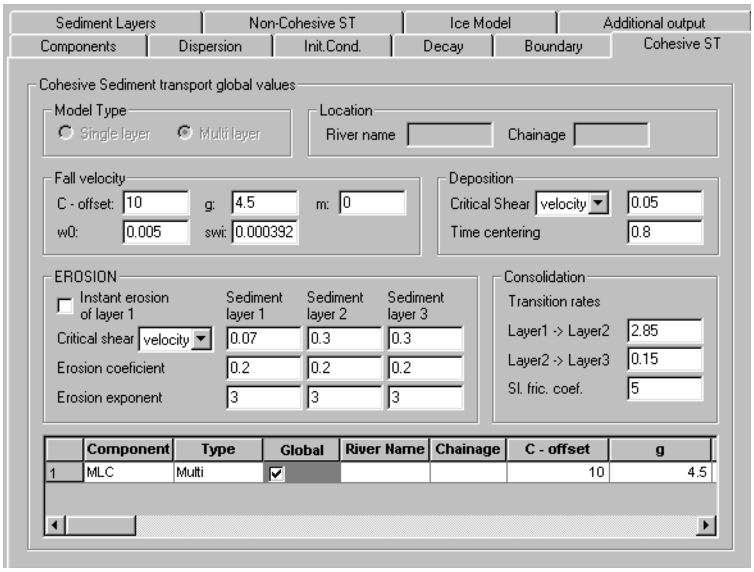


Figure 7.11 The cohesive sediment property page when a multi layer model is selected.

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 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.

7.10.3 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:

$$S = \frac{WC}{h^*}\left(1 - \frac{\tau}{\tau_{cd}}\right), \ \tau \le \tau_{cd} \tag{7.7}$$

where,

S is the source term in the advection dispersion equation

C is the concentration of the suspended sediment (kg/m^3)

w is the mean settling velocity of suspended particles (m/s)

h^* is the average depth through which the particles settle

τ is the critical shear stress for deposition (N/m^2)

τ_{cd} is the bed shear stress (N/m^2)

The bed shear stress can be given by the Manning formula (as an example):

$$\tau = \rho g \frac{V^2}{M^2 h^{1/3}} \tag{7.8}$$

where,

ρ fluid density (kg/m^3)

g acceleration of gravity (m/s^2)

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:

$$S = \frac{WC}{h^*}\left(1 - \left(\frac{V}{V_{cd}}\right)^2\right),\ V \le V_{cd} \tag{7.9}$$

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.

$$S = \frac{M^*}{h}\left(1 - \frac{\tau}{\tau_{ce}}\right)^n,\ \tau \le \tau_{ce} \tag{7.10}$$

where

S source term in the advection dispersion equation

τ bed shear stress (N/m^2)

τ_{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:

$$S = \frac{M^*}{h}\left(1 - \left(\frac{V}{V_{ce}}\right)^2\right)^n, \; V \geq V_{ce} \tag{7.11}$$

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:

$$S_d = W_s\left(1 - \frac{\tau_b}{\tau_{c,\,d}}\right)c, \; \tau < \tau_{c,\,d} \tag{7.12}$$

where

S_d rate of deposition (kg/m^2/s)

τ critical shear stress for deposition (N/m^2)

c suspended sediment concentration (kg/m^3)

All deposited material is added to sub-layer 1.

The model concentration c is weighted in time according to the following expression:

$$c = (1-\theta)c_j^n + \theta c_j^{n+1} \tag{7.13}$$

where:

j spatial index

n time index

θ 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:

$$S_E = E_o(\tau_b - \tau_{c,e})^n,\ \tau_b > \tau_{c,e} \tag{7.14}$$

where,

S_E rate of erosion (kg/m2/s)

E_o erosion coefficient (kg/s/N)

$\tau_{c,e}$ critical shear stress for erosion n: erosion 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.

WATER QUALITY EDITOR

8 WATER QUALITY EDITOR

8.1 Level for Water Quality Modelling

It is possible to choose among six model levels corresponding to different sets of state variables for the water quality and/or different descriptions of the transformation of the state variables in the river. The higher the model level, the more execution time the water quality module will require.

The whole idea of having a number of model levels with varying complexity is to have a model which would apply to very simple problems (in terms of variables involved), and also to much more complicated situations where nutrient transport and fractionation of the organic matter (dissolved, suspended and deposited) are included. Using such approach the model will be quick to use and would be able to solve small though still important problems. It can also be used in the most complex situations where the next step would be to apply ecological type models such as a eutrophication model.

8.2 Model Level

Model level 1: BOD and DO

A simple oxygen balance model, only including immediate oxygen demand from degradation of BOD and reaeration.

Model level 2: BOD, with bed/sediment exchange and DO

As model level 1, except that here resuspension and sedimentation are included in the calculation of the BOD balance, and a sediment oxygen demand is included in the dissolved oxygen balance.

Model level 3: BOD, DO, and nitrification

As for model level 1 with the addition of the ammonia / nitrate balances, and the oxygen consumption from the nitrification process. No denitrification is assumed.

Model level 4: BOD, with bed/sediment exchange, DO, Nitrification and Denitrification

Includes all processes from model levels 2 and 3: resuspension and sedimentation are included in the calculation of the BOD balance, and the ammonia / nitrate balances, plus the oxygen consumption from the sediment oxygen demand and the nitrification process are included. Moreover, denitrification is included.

Model level 5: BOD and DO, including delayed oxygen demand

BOD at this model level is split into three different fractions: dissolved in the water phase, suspended in the water phase, and settled at the river bed. Degradation of the settled BOD fraction at the river bed gives rise to the delayed oxygen demand. This level does not include the nitrogen components ammonia and nitrate.

Model level 6: All processes

Dissolved BOD, suspended BOD, BOD at the river bed, oxygen, ammonia and nitrate. BOD is described as for level 5, and nitrogen components are described as for level 4.

Coliforms (see 8.12) and phosphorus (see 8.13 - 8.16) are optional at all model levels. Wetlands processes can be included at level Nos. 3, 4 and 6 (see 8.22).

Each data entry menu, has been divided into two parts into which values can be typed.

1 Fields with global values

2 Fields with local values

Global values apply everywhere in the river system, where no local values have been specified.

Local values apply to specific locations in the river system. During computation model values will be interpolated between the locally defined values. Outside the locally specified areas the global parameter values will be used.

8.3 Arrhenius

Arrhenius gives the temperature dependency of a process rate by multiplying with the factor

$$\Theta^{(T - T_0)} \tag{8.1}$$

In the WQ model all temperature dependencies are described in this way, and the reference temperature, T_0, is 20 °C. If Θ is set to be 1.07, the process rate doubles when temperature increases by 10 °C. This is generally a reasonable approximation for chemical processes. Biological processes, however, can show more variability.

8.4 Degradation

This property page offers the possibility to add and edit degradation related data.

There are three parameters specifying the degradation of BOD. In the first field, a global value for the first order decay rate for dissolved BOD at 20°C, K_1, is shown. The physical unit is 1/day.

In the second field a global value of the Arrhenius temperature coefficient for the decay rate is shown, it is dimensionless.

In the last field the half-saturation oxygen concentration in the Michaelis-Menten expression describing the influence of oxygen in the BOD decay, K_s, is shown, in the unit of g O_2/m^3. The BOD decay decreases at low O_2 concentrations due to the depression of bacterial BOD degradation under anaerobic conditions.

The decay of BOD is calculated as

$$\mathrm{Degradation} = K_1^{\mathrm{BOD}} \cdot \Theta^{(t-20)} \cdot \frac{\mathrm{DO}^2}{K_s + \mathrm{DO}^2} \tag{8.2}$$

where DO is the concentration of dissolved oxygen.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.5 Degradation at the bed (levels 5 and 6)

This property page offers possibility to add and edit bed degradation related data.

There are four parameters for degradation of organic matter at the river bed.

In the first field, a global value for the first order decay rate for sediment BOD at 20°C, K_1, is shown. The physical unit is 1/day.

In the second field a global value of the Arrhenius temperature coefficient for the decay rate is shown, it is dimensionless.

In the third field the background sediment oxygen demand at 20°C is specified. The unit is g O_2/m^2/day. The baseline sediment oxygen demand is the basic demand of oxygen originating from the river bed due to natural sources of organic matter, that is, not as a result of the pollution sources studied by the modelling. The oxygen demand due to settling of BOD from the pollution sources is taken into account by the BOD decay at the river bed, for which the rate constant is specified by the first parameter in this menu.

In the last field the Arrhenius temperature coefficient for the oxygen demand at the river bed is specified.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.6 Bed/sediment (levels 2 and 4)

This property page offers possibility to add and edit bed / sediment related data in connection with modelling bed / sediment exchange.

There are five coefficients for Bed/Sediment on this model level.

In the first field the background sediment oxygen demand at 20°C is specified. The unit is g O_2/m^2/day. The sediment oxygen demand is the basic demand of oxygen originating from the river bed due to natural sources of organic matter, that is, not as a result of the pollution sources studied by the modelling.

In the second field the Arrhenius temperature coefficient for the oxygen demand at the river bed is specified.

In the third field the resuspension of sedimented organic matter is specified as g BOD/m^2/day.

In the fourth field the settling velocity is specified for suspended organic matter in m/day.

In the fifth field the critical flow velocity, where net resuspension/deposition is zero, is specified. The critical flow velocity is given in m/sec. When the flow velocity is below this value, sedimentation is assumed, and the parameter specified in the previous field is used. When the flow velocity exceeds this value, resuspension is assumed, and the parameter specified in the second field is used.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.7 Bed / Sediment (Model Levels 5 and 6)

This property page offers possibility to add and edit sediment / bed related data in connection with modelling delayed oxygen demand.

There are five coefficients for Bed/Sediment on this model level.

In the first field a 1st order adsorption constant must be specified for the adsorption of dissolved organic matter from the water on the river bed. The unit is m/day.

In the second field the resuspension of sedimented organic matter is specified as g BOD/m²/day.

In the third field the settling velocity is specified for suspended organic matter in m/day.

In the fourth field the critical flow velocity, where net resuspension/deposition is zero, is specified. The critical flow velocity is given in m/sec. When the flow velocity is below this value, sedimentation is assumed, and the parameter specified in the previous field is used. When the flow velocity exceeds this value, resuspension is assumed, and the parameter specified in the second field is used.

In the last field the critical concentration of organic matter at the river bed is specified as g BOD/m². In case of concentrations below this value there will be no resuspension of organic matter from the bed irrespective of the flow velocity (this is effectively the same as setting the parameter in field No. 2 to zero).

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.8 Nitrogen Contents (Model Levels 3 and 4)

This property page offers possibility to add and edit nitrogen contents related data.

The title Nitrogen contents covers the nitrogen release from BOD decay and the uptake of ammonia by bacteria and plants. The parameters for these processes are necessary, in order to describe the nitrogen transport and transformation in the river. The menus for the immediate oxygen demand levels (3 and 4) and the level including both immediate and delayed oxygen demand (6) are different due to the differences in the description of the BOD, e.g. modelling three BOD fractions when delayed oxygen demand is included.

Three parameters for nitrogen contents are required in the case of modelling only immediate oxygen demand.

In the first field a value for the release of ammonia-nitrogen is given for the degradation of organic matter, in the unit g NH4-N/gBOD.

In the second field the uptake by the plants of ammonia-nitrogen relative to the net photosynthesis (= photosynthesis - respiration) at the maximum rate of photosynthesis is specified. The unit is g NH4-N uptaken / g O_2 released.

In the last field the uptake of ammonia-nitrogen by bacteria must be specified relative to their uptake of oxygen. The unit is g NH4-N uptaken/ g O_2 used.

In summary:

First field
Global value for the release of ammonia at BOD decay (g NH4-N/g O_2)

Second field
Global value for the uptake of ammonia in plants proportional to the net photosynthesis (g NH4-N/g O_2)

Third field
Global value for the ammonia uptake in bacteria proportional to their degradation of BOD (g NH4-N/g O_2)

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.9 Nitrogen Contents (Model Level 6)

This property page offers possibility to add and edit nitrogen contents related data.

The title Nitrogen contents covers the nitrogen release from BOD decay and the uptake of ammonia by bacterial and plants. The parameters for these processes are necessary, in order to describe the nitrogen transport and transformation in the river. The menus for the immediate oxygen demand levels (3 and 4) and the level including both immediate and delayed oxygen demand (6) are different due to the differences in the description of the BOD, e.g. the fractionation when delayed oxygen demand is included.

Five parameters for nitrogen contents are required in the case of modelling both immediate and delayed oxygen demand.

In the first three fields values for the release of ammonia-nitrogen are given for the degradation of dissolved organic matter, suspended organic matter and sedimented organic matter respectively. All three values have the unit g NH4-N/gBOD.

In the fourth field the uptake by the plants of ammonia-nitrogen relative to the net photosynthesis (= photosynthesis - respiration) at the maximum rate of photosynthesis is specified. The unit is g NH4-N uptaken/g O_2 released.

In the last field the uptake of ammonia-nitrogen by bacteria must be specified relative to their uptake of oxygen. The unit is g NH4-N uptaken/g O_2 used.

In summary:

First field
Global value for the release of ammonia at BOD dissolved decay (g NH4-N/g O_2)

Second field
Global value for the release of ammonia at BOD suspended decay (g NH4-N/g O_2)

Third field
Global value for the release of ammonia at BOD bed decay (g NH4-N/g O_2)

Fourth field
Global value for the uptake of ammonia in plants proportional to the net photosynthesis (g NH4-N/g O_2)

Fifth field
Global value for the ammonia uptake in bacteria proportional to their degradation of BOD (g NH4-N/g O_2)

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.10 Nitrification

This property page offers possibility to add and edit nitrification related data.

There are four parameters related to nitrification.

Select either n = 1 for an ordinary 1st order reaction, or n = 0.5 for a ½-order reaction (biofilm resisted transport).

In the first field the rate constant for the nitrification at 20°C is stated. If n = 1 has been selected as reaction order, the rate constant must be specified as 1/day. If n = 0.5 has been selected as reaction order, the unit is $(mg/l)^{1/2}$/day.

In the second field the Arrhenius temperature coefficient for the nitrification rate must be specified.

In the last field, oxygen demand by nitrification is given in the unit of g O_2 /g NH4+-N.

The half-saturation constant for oxygen, specified under BOD degradation, is also applied for nitrification.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.11 Denitrification

This property page offers possibility to add and edit denitrification related data.

The denitrification is a process which takes place under anaerobic conditions in the river, e.g. in biofilm at surfaces of stones, gravel and plant leaves. By this process nitrate is transformed into free nitrogen, which eventually escapes to the atmosphere due to its low water solubility. The denitrification can be an important process for nitrogen removal in the river. In order to have a proper nitrogen balance, this process has to be included whenever it is known to occur.

Three parameters are required to model denitrification.

Select either n = 1 for an ordinary 1st order reaction, or n = 0.5 for a ½-order reaction, (biofilm resisted transport).

In the first field the rate constant for the nitrification at 20°C is stated. If n = 1 has been selected as reaction order, the rate constant must be specified as 1/day. If n = 0.5 has been selected as reaction order, the unit is $(mg/l)^{1/2}$/day.

In the second field the Arrhenius temperature coefficient for the denitrification rate must be specified.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

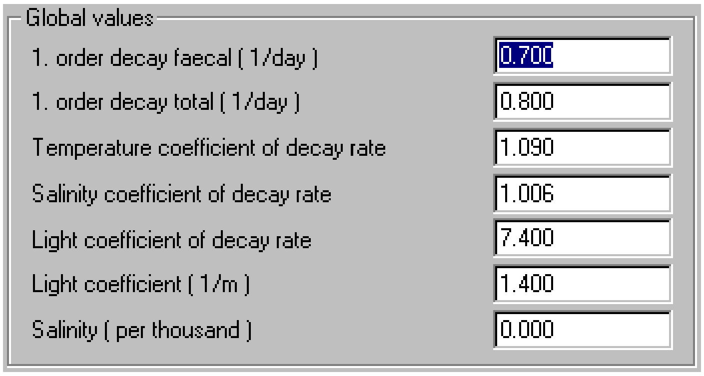
8.12 Coliforms

This property page offers possibility to add and edit coliform related data.

The decay of coliforms is dependent on the light intensity in the water column, the temperature and the salinity.

$$\text{Coli decay} = K \cdot C_{coli} \cdot \Theta_T^{(T-20)} \cdot \Theta_S^{SAL} \cdot \Theta_I^{I} \qquad (8.3)$$

Global values for the first order decay rates, *K*, for faecal and total coliforms, respectively, are specified in the first two fields, at 20 ºC, total darkness and zero salinity. The default decay coefficients have been found from experiments with water polluted with coli bacteria where corrections for temperature, salinity and light have been made.



The Arrhenius temperature coefficient, T, is specified in the third field.

The salinity coefficient, S, is specified in the fourth field.

The light coefficient, I, is specified in the fifth field.

The light extinction coefficient, η, is specified in the sixth field. The light intensity, I, is the average light intensity calculated as:

$$I = I_0 \frac{(1 - \exp(-\eta Z))}{\eta Z} \tag{8.4}$$

where I_0 is the surface light intensity, and Z the water depth.

The salinity, SAL, is specified in the last field.

8.13 Phosphorus Content (Model Levels 1 to 4)

This property page offers the ability to add and edit data related to phosphorus modelling.

There are two parameters to be specified on this page, describing the content of phosphorus in organic matter (BOD) originating from pollution sources, and in plants.

In the first field, the phosphorus content in BOD must be specified, as g P / g O_2.

In the second field, uptake of phosphorus by plants per g O_2 produced (nettoproduction = production – respiration) is specified.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.14 Phosphorus Content (Model Levels 5 and 6)

This property page offers the ability to add and edit data related to phosphorus modelling.

There are four parameters to be specified on this page, describing the content of phosphorus in organic matter (BOD) originating from pollution sources, and in plants.

In the first three fields, the phosphorus content must be specified, as g P / g O_2 for dissolved, suspended and bottom BOD, respectively.

In the last field, uptake of phosphorus by plants per g O_2 produced (net production = production – respiration) is specified.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.15 Phosphorus Processes in the Water Phase

This property page offers the ability to add and edit data related to phosphorus modelling.

There are four parameters to specify on this page, dealing with the degradation and formation of particulate phosphorus suspended in the water phase.

In the first field a first order decay rate is specified, at the reference temperature 20 °C.

In the second field the corresponding Arrhenius temperature coefficient is specified.

In the third a first order rate for the formation of particulate phosphorus from orthophosphate is specified (first order with respect to orthophosphate). This rate is also given at the reference temperature 20 °C.

In the last field the corresponding Arrhenius temperature coefficient is specified.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.16 P. exchange with the bed

This property page offers the ability to add and edit data related to phosphorus modelling.

There are three parameters to specify on this page, dealing with phoshorus exchange between the river bed and the water phase.

In the first field, the resuspension rate is specified.

In the second field, the sinking velocity for particulate phosphorus is specified.

In the last field the critical flow velocity, where resuspension = deposition, is specified. If the flow velocity (calculated by the HD-module) is below the critical velocity, sedimentation is assumed to occur, with the sinking velocity specified in the second field. If the flow velocity exceeds the critical velocity, resuspension is assumed to occur, with the rate specified in the first field.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.17 Temperature

This property page offers possibility to add and edit temperature related data.

The temperature will be computed as the result of the difference between solar energy input (only during light hours) and the energy loss due to emitted heat radiation (during night and day).

There are four parameters for temperature.

1 In the upper field on the menu the latitude (degrees) of the location of the river is given.

2 In the second field a global value of the maximum heat radiation of the river is specified. The unit is $kJ/m^2/hour$.

3 In the third field a global value of the displacement of the maximum temperature of the stream from 12 noon is specified. If the river temperature reaches its maximum after 12 noon, the displacement of time will be positive. Conversely, the displacement of time will be negative if the maximum temperature is reached before 12 noon. The displacement of time is stated in hours.

4 In the last field a global value of the emitted heat radiation from the river is entered using the unit of kJ/m²/day.

The global values will be used by the WQ-module throughout the river system. Global values can be substituted for specific locations.

Note: Temperature is not modelled when the Mike 12 thermocline/halocline hydrodynamic model is used as basis.

The global values will be used by the WQ module throughout the river system. Local values can be given for specific locations.

8.18 Oxygen processes

This property page offers possibility to add and edit oxygen processes related data.

The factors affecting the oxygen concentration are photosynthetic production and respiration, reaeration (exchange with the atmosphere), BOD decay and nitrification. The two latter processes are specified on separate menus. The oxygen menus include the photosynthetic processes and the reaeration.

There are six parameters for oxygen processes.

1 In the upper field the number (1 through 6) of the selected expression for the calculation of the reaeration constant at 20°C is shown. The Thyssen expression (1) is recommended for application to small streams, O'Connor-Dubbins (2) to ordinary rivers, and the Churchill-expression (3) to rivers with high flow velocities. The equations 4 through 6 can be specified by the user, by pressing the button

See later for a description of how to apply different expressions for reaeration at different locations of the river setup.

2 In the second field the Arrhenius temperature coefficient for the reaeration constant is specified.

3 In the third field, respiration of plants and animals at 20°C is given. The unit can be specified to be either g O_2/m^2/day or g O_2/m^3/day.

4 In the fourth field the Arrhenius temperature coefficient for respiration of plants and animals is entered.

5 In the fifth field, maximum oxygen production by photosynthesis is given, in the same unit as specified for respiration.

6 In the last field the displacement of the time of the maximum oxygen production of the river from 12 noon is stated. If the river has its oxygen maximum after 12 noon, the displacement of time will be positive. Conversely, the displacement of time will be negative, if the maximum oxygen concentration is reached before 12 noon. The displacement of time is stated in hours.

A first estimate of the oxygen parameters: production, respiration and reaeration constants can be carried out from measured diurnal variations of the oxygen concentrations. This can be done with measurements from only one station, but this approach requires a number of vague assumptions concerning the conditions of the river (uniform topography and uniform oxygen fluctuations for the entire river). Measurements from two stations, however, do not involve assumptions concerning the physical conditions of the river and is therefore recommended. The method is based on a simplified oxygen balance which implies that the river reach must be unaffected by pollution sources, e.g. BOD decay, ammonia oxidation and sediment oxygen demand. The measurement stations thus have to be positioned in a non polluted part of the river. The simple balance reads:

$$\frac{\mathrm{d}C}{\mathrm{d}t} = K_2 \cdot (C_m - C) - R + P(t) \tag{8.5}$$

where

C oxygen concentration (mg/l)

C_m oxygen concentration at saturation (mg/l)

R respiration (g O_2/m^3/day or g O_2/m^2/day)

$P(t)$ photosynthetic production (g O_2/m^3/day or g O_2/m^2/day)

K_2 reaeration constant (day^{-1})

t time (day)

The oxygen production at night is nil, which means the respiration and reaeration can be estimated from the night measurements. The reaeration constant is usually not constant hence it varies with the physical conditions of the river and is calculated in the model by an expression including this (either a standard equation or a user defined). Also the production and respiration can very well change along the river. However, the outlined estimation method can be a great help as the first attempt to estimate respiration and production. They will, however, very often have to be tuned during the calibration. The unit for respiration/production can be specified per m^2 (benthic production) or per m^3 (pelagic production)

The reaeration coefficient can be calculated either according to some standard expressions applicable for different types of rivers or streams or from user defined expressions. The major factors affecting the reaeration constant are the current velocity, the river slope, the water depth and the temperature. The temperature is included by an Arrhenius temperature function as mentioned above.

For more information about reaeration, see Reaeration *(p. 307)*.

Defining the local values must follow a number of specific rules. First of all, there is no interpolation between the selected expressions at different chainages. This means that if you want an expression to be valid at a stretch of the river you have to specify the chainage and the requested expression number at the beginning and end of the stretch. An example of this is shown below. The global value is applied everywhere else. The expressions 1 through 4 can be used for river stretches.

Secondly, if you only specify an expression to be used at one point (chainage) then it will only be applied at this specific location. This is actually the way to use the expressions 5 and 6. These can only be applied for one chainage, which as an example could be a weir at which the reaeration process is very different from the river conditions. In the example shown below, the global expression is valid everywhere except for the chainages 1.000 and 2.000 and 5.000 - 6.000, for which expressions Nos. 2 and 4 are used, respectively. Chainages 3.000 and 4.000 are defined as structures. At these two points expressions 5 and 6 will be applied, respectively.

The figure below shows the resulting combination of expressions in a schematic way.

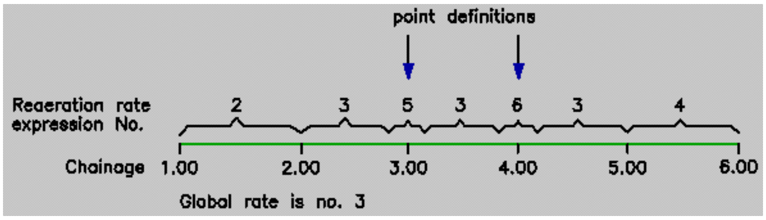


Figure 8.1 Combination of reaeration expressions

The third general rule for specification of local expressions is that only one expression can be specified for a chainage. This means that if a point definition is made in a stretch which is already defined by a local expression this stretch must be split. When expression Nos. 5 and 6 are to be used at points of the river where a local expression is applied, then this river stretch has to be split into sections at each side of the point definition. The order of the definitions is arbitrary.

8.19 Degradation in the water phase

This property page offers possibility to add and edit water phase degradation related data.

There are five parameters specifying the degradation of BOD in the water.

In the first and the third field, a global value for the first order decay rate, K_1, for dissolved and suspended BOD, respectively, at 20°C is shown. The physical unit is 1/day.

In the second and fourth field, a global value of the Arrhenius temperature coefficient for the decay rate, Θ, for dissolved and suspended BOD, respectively, is shown. It is dimensionless.

In the last field the half-saturation oxygen concentration in the Michaelis-Menten expression describing the influence of oxygen in the BOD decay, K_S, is shown, in the unit of g O_2/m^3. The BOD decay decreases at low O_2 concentrations due to the depression of bacterial BOD degradation under anaerobic conditions.

The decay of BOD is calculated as

$$\text{Degradation} = K_1^{\text{BOD}} \cdot \Theta^{T-20} \cdot \frac{\text{DO}^2}{K_s + \text{DO}^2} \tag{8.6}$$

where DO is the concentration of dissolved oxygen. This equation applies for dissolved, as well as suspended, BOD. Different values for K_1 are, however, used.

8.20 Reaeration

The reaeration coefficient can be calculated either according to some standard expressions applicable for different types of rivers or streams or from user defined expressions. The major factors affecting the reaeration constant are the current velocity, the river slope, the water depth and the temperature. The temperature is included by an Arrhenius temperature function. In this menu an expression for the reaeration constant K_2 at 20°C is chosen.

This dialog offers possibility to modify the user's own expressions for reaeration, and to view parameters applied in the three built-in expressions.

All expressions have the same mathematical formulation, only the parameters differ.

$$K_2 = a \cdot u^b \cdot h^c \cdot I^d \tag{8.7}$$

K_2 reaeration constant at 20°C (g O^2/m^2/day)

u flow velocity (m/s)

h water depth (m)

I river slope

The user can choose between six options for the expression for the reaeration constant, see below, and in the Oxygen processes *(p. 303)* property page.

All six expressions are based on empirical relationships between the reaeration constant and flow velocity, water depth and river slope. The three first expressions are standard expressions, whereas the fourth, fifth and sixth can be specified by the user.

1 Thyssen expression

$$K_2 = 27185 \cdot u^{0,931} \cdot h^{-0,692} \cdot I^{1,09} \tag{8.8}$$

The Thyssen expression is recommended for calculations in small streams.

2 O'Conner Dubbins expression

$$K_2 = 3,9 \cdot u^{0,5} \cdot h^{-(1,5)} \tag{8.9}$$

The O'Conner Dubbins expression is recommended for ordinary rivers.

3 Churchill expression

$$K_2 = 5,233 \cdot u \cdot h^{-1,67} \tag{8.10}$$

The Churchill expression is recommended for rivers with high flow velocities.

Custom expressions

4

$$K_2 = a_3 \cdot u^{b_3} \cdot h^{c_3} \cdot I^{d_3} \tag{8.11}$$

5

$$K_2 = a_2 \cdot u^{b_2} \cdot h^{c_2} \cdot I^{d_2} \tag{8.12}$$

6

$$K_2 = a_1 \cdot u^{b_1} \cdot h^{c_1} \cdot I^{d_1} \tag{8.13}$$

The user can specify coefficients for three different reaeration expressions. No. 4 can be applied globally, Nos. 5 and 6 can only be applied at point locations. This can be done in structures, where the water is strongly aerated.

If (4-6) Own expressions are chosen, coefficients must be specified for the expressions. The fourth expression can be applied for a river system similar to the three standard equations. These expressions (e.g. 1-4) can be specified locally as well as globally. The expressions five and six are applicable only for a chainage not for a river stretch. They are intended to be used at weirs, falls etc. where the reaeration process has to be described different from the river.

First of all a global expression has to be specified. This is either done by editing the upper field of the global part of the oxygen processes property page, e.g. by typing the number of the expression (1 through 4, remember to define No. 4 if that is chosen) or by selecting the appropriate expression in the present dialog.

The coefficients of the Own expressions can be specified. The own values are:

a coefficient of the reaeration expression (proportionality factor)

b exponent for the flow velocity of the water

c exponent for the water depth

d exponent for the river slope

8.21 Non point pollution interface

Please read the lower part of the dialog explaning the different parameters used.

8.22 WETLAND

8.22.1 Introduction

The retention and removal of nutrients (N and P) in areas such as riparian wetlands, flood plains, reed swamps and engineered wetlands, have long been considered as an energy-efficient treatment system.

The agricultural and industrial utilisation of the natural wet zone between the terrestrial and the aquatic system has reduced these important buffer zones in the last 30-100 years. Nowadays, artificial or constructed wetlands are used as primary or secondary treatment of waste water in many countries. Non-point sources of nutrients are becoming more and more the bottleneck in controlling the nutrient run-off to rivers and lakes, especially in the Western-like countries. The use of wetlands as nutrient traps is one available method for reducing the nutrients in waste water or water transported with rivers.

The WETLAND model has been designed with the purpose to have an instrument capable of simulating the significant nutrient removal processes, their interrelationship, and their effects on the river water quality.

8.22.2 Integration with WQ

The WETLAND developed by DHI is an integral part of the Water Quality (WQ) model. This means that river branches with adjacent flood plains, and selected branches that constitute a wetland area, can be modelled explicitly with WETLAND - in parallel with WQ modelling for conventional river branches.

The user can specify branches that shall be modelled with WETLAND (substituting WQ) and the threshold water level for which the WETLAND processes shall be activated.

This means that is possible within the same simulation to simulate WQ in the open river channel and WETLAND in the adjacent flood plain areas.

The integration works in the following way: In each time step, the branches that are defined as Wetlands will be activated. If the actual water level is below the specified wetland flooding level, there will be no impact on the river water quality from wetland processes. Should the actual water level exceed the wetland flooding level, the WETLAND calculations will influence the river water quality proportional with the water volume that is in contact with the flood plains. The WQ processes will also be calculated and affect the river water quality with the water volume representing the open channel part in the cross-section, see Figure 8.2

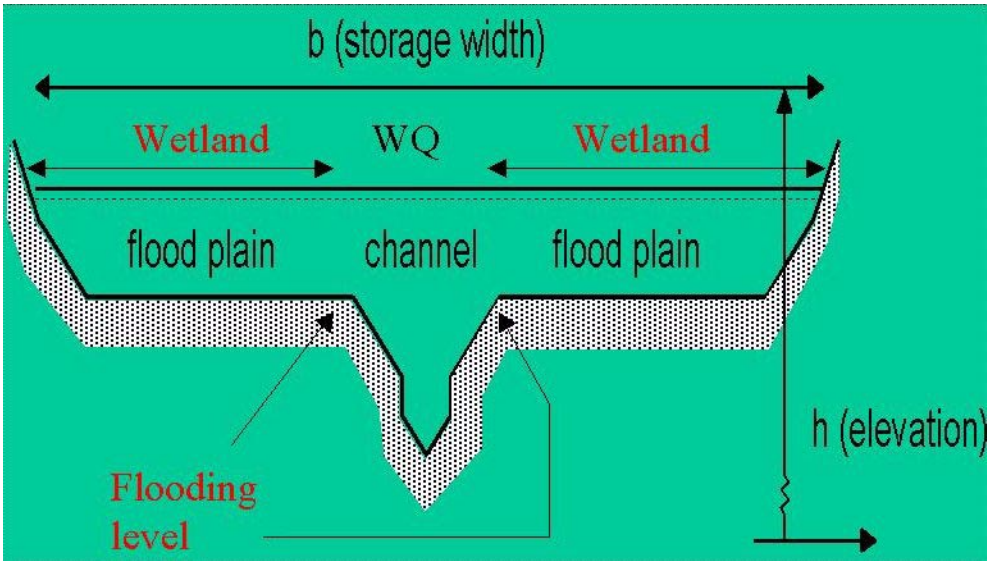


Figure 8.2 WETLAND integration with WQ

8.22.3 The Model

The WETLAND model embraces 4 components in the water phase (NH_4-N, NO_3-N, PO_4-P and Particulate-P) and 8 attached to the sediment/peat (plant N/P biomass, labile N/P organic matter, stabile N/P organic matter, immobile N and adsorbed N).

In the WETLAND model, which is a relatively simple model, both aerobic (assumed to occur in a constant oxic microzone) and anaerobic processes (in the saturated zone) are simulated, e.g. nitrification is modelled in the aerobic zone while denitrification takes place in the anaerobic zone. Uptake and mineralization of nutrients together with immobilisation/adsorption processes and sedimentation/resuspension of particulate matter are also modelled. The modelled processes are illustrated in fig. 8.3

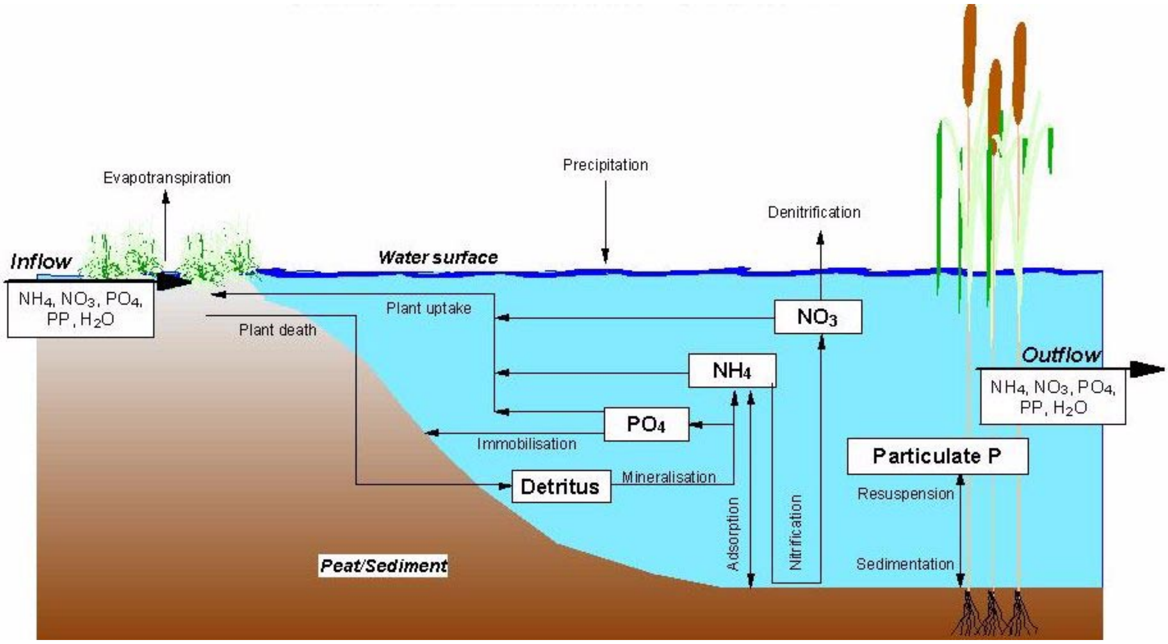


Figure 8.3 The WETLAND model

The biological/physical system described in the WETLAND model consists of coupled processes where changes in one component could influence other components depending on the biological/chemical reactions. The model describes 4 components or variables in the water phase, all which are coupled to the transport/spread simulation from the AD module:

AMMONIA	NH_4-N concentration in surface water
NITRATE	NO_3-N concentration in surface water
PHOSP. DISS.	PO_4-P concentration in surface water
PHOSP. PART.	Particulate-P concentration in surface water

These four components are identical with the components for nutrients applied in WQ model level Nos. 3, 4 and 6. This also implies that WETLAND only can be activated at model levels Nos. 3, 4 or 6. Phosphorus processes are as in the WQ model optional.

In addition to the four water phase components 8 more components in the sediment/peat are modelled which do **not** need AD specifications as they not are transported in the water phase (stationary variables). Initial conditions, however, must be specified (see 8.24). The stationary variables are:

PLANT-N	The plant biomass per area expressed in nitrogen
LABILE ORG-N	The concentration of fast degradable (labile) organic matter per volume sediment/peat expressed in nitrogen
STABILE ORG-N	The concentration of slow degradable (stabile) organic matter per volume sediment/peat expressed in nitrogen
ADSORBED-N	The concentration of adsorbed ammonia in peat (per volume sediment/peat)
IMMOBILE-N	The concentration of microbial (immobile) nitrogen in peat (per volume sediment/peat)
PLANT-P	The plant biomass per area expressed in phosphorous
LABILE ORG-P	The concentration of fast degradable (labile) organic matter per volume sediment/peat expressed in phosphorous
STABILE ORG-P	The concentration of slow degradable (stabile) organic matter per volume sediment/peat expressed in phosphorous

In the '<file>WQAdd.res11' (additional output) file you will find the simulated value of these eight stationary variables together with miscellaneous aggregated values.

8.23 Wetland General

This page offers the possibility to include wetland processes for selected branches, and to define some general parameters:

Sediment thickness
An empirical parameter for the relative influence of sediment/peat processes (adsorption and immobilisation of NH_4-N) on the water quality. The sediment/peat processes calculated per volume sediment/peat will be corrected for the specified sediment thickness. The parameter can be interpreted as the mean depth for the horizontal surface water flow through the peat.

Depth when completely flooded
A parameter for the highest point of the wetland area relative to the surface water level. If the value is zero then the surface water always covers the wetland area 100% regardless of the actual water level, whereas a value greater than zero means that the coverage will be less than 100% **if**

the actual water level is greater than the value of the parameter. The actual coverage will be calculated during the simulation. The coverage determines the impact of the biological processes on N and P concentration, e.g. 50% coverage imply 50% impact etc. In the model, hilly wetland areas will thus have a low influence on the water quality compared to a more flat area, if the same amount of water is flowing through the wetland.

Temperature coefficient
The value of 'teta' in the Arrhenius temperature function (see 8.3). The same function is used for all temperature dependant processes in the model.

Activating WETLAND processes, see **Integration with WQ**

8.24 Wetland Nitrogen

This page offers the possibility to specify parameters for wetland nitrogen processes, all applied globally:

Maximum nitrification rate
Max. Production rate per m^2 at 20°C

Half-saturation concentration, nitrification
K_m in the Michaelis-Menten expression

Maximum denitrification rate
Max. Respiration rate per m^2 at 20°C

Half-saturation concentration, denitrification
K_m in the Michaelis-Menten expression

Mineralisation rate, labile pool
1st order decay of fast degradable (labile) organic matter in sediment/peat at 20°C

Mineralisation rate, stabile pool
1st order decay of slow degradable (stabile) organic matter in sediment/peat at 20°C

Immobilisation rate
The immobilisation rate per m^3 (microbial uptake) of NH_4-N in the sediment/peat at 20°C

Half-saturation concentration, immobilisation of NH4-N
K_m in the Michaelis-Menten expression

Microbial mortality rate
1st order mortality (dead rate) of microbial biomass at 20°C

N-adsorption capacity
Adsorption capacity (mass per volume sediment/peat) of NH_4-N in the sediment/peat. The adsorption is reversible.

Half-saturation concentration, adsorption of NH4-N
K_m in the Langmuir adsorption expression

N-Adsorption rate
1st order rate, N-adsorption at 20°C

The yearly N Plant production
The aggregated yearly uptake of N in plants, mass per area. The seasonal distribution follows a latitude-dependant sun radiation formula

Ratio of dead plant material to labile
The ratio of fast degradable (labile) organic nitrogen in plant material.

8.25 Wetland Phosphorus

This page offers the possibility to specify parameters for wetland phosphorous processes, all applied globally:

Half-saturation concentration, adsorption of PO4-P
K_m in the Michaelis-Menten expression

P-adsorption rate
The (irreversible) P-adsorption rate per area to the sediment/peat

Mineralisation rate, labile pool
1st order decay of fast degradable (labile) organic matter in sediment/peat at 20°C

Mineralisation rate, stabile pool
1st order decay of slow degradable (stabile) organic matter in sediment/peat at 20°C

The yearly P Plant production
The aggregated yearly uptake of P in plants, mass per area. The seasonal distribution follows a latitude-dependant sun radiation formula.

Ratio of dead plant material to labile
The ratio of fast degradable (labile) organic phosphorous in plant material

Sedimentation rate, max
1^{st} order rate, max. sedimentation, for particulate-P. The maximum rate is applied when the plant biomass reaches its optimum

Sedimentation rate, min
1^{st} order rate, min. sedimentation, for particulate-P. The minimum rate is applied when the plant biomass reaches its lowest value

Resuspension rate
Rate of resuspension, Particulate-P (mass per area per time)

Critical flow velocity
Value for the critical flow for resuspension/sedimentation of Particulate-P, i.e. velocity above this value implies that resuspension takes place. For velocities below the critical value, Particulate-P will undergo sedimentation.

8.26 Wetland Components

This page offers the possibility to specify initial conditions (global values only) for the eight stationary variables:

Organic-N in peat, e.g. kg/m^3
The concentration of total-N (labile and stabile) in uppermost part of the sediment/peat, i.e. part of the sediment core which affects the quality of the surface water

Ratio of stable organic N
The initial ratio of slow degradable (stable) organic nitrogen

Organic-P in peat, e.g. kg/m^3
The concentration of total-P (labile and stabile) in uppermost part of the sediment/peat, i.e. part of the sediment core which affects the quality of the surface water

Ratio of stable organic P
The initial ratio of slow degradable (stable) organic phosphorus

Adsorbed-N in peat, e.g. kg/m^3
Initial adsorbed NH_4-N in the sediment/peat

Plant-N, e.g. g N/m^2
Initial plant N biomass

Plant-P, e.g. g P/m^2
Initial plant P biomass

Immobile-N in peat, e.g. kg/m^3
Initial immobilised N in the sediment/peat.

EUTROPHICATION EDITOR

9 EUTROPHICATION EDITOR

The former VKI has been developing the Eutrophication model EU since the early seventies. This advanced tool has been used in a long range of contexts for calculating the consequences of human impacts on the environment.

The model describes the growth of phytoplankton, zooplankton, benthic vegetation and oxygen conditions as a consequence of BOD loading, available nutrients and factors such as incident light intensity, water temperature and hydraulic conditions.

The biological/chemical system described in the EU model consist of a network of coupled processes where changes in one component could influence all the other variables depending on the biological reaction. The state variables and processes in the biological system are illustrated as follows:

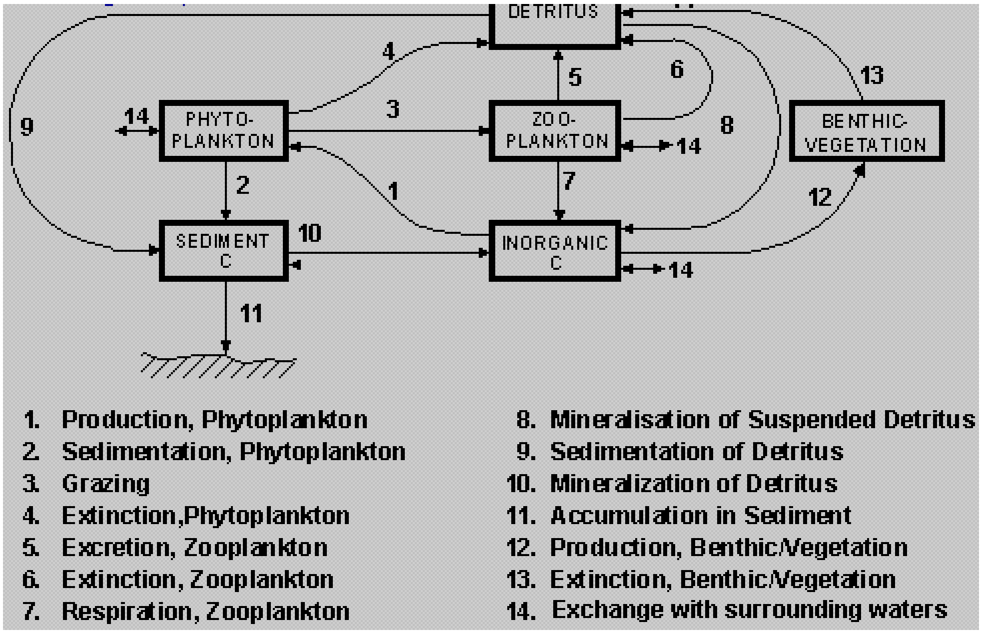


Figure 9.1 State variables and processes in the EU model exemplified by the cycling of Carbon (C).

See also EU Processes *(p. 322)* and Required Data for the EU Model *(p. 324)*.

9.1 Background

9.1.1 EU Processes

In the model the cycling of carbon, nitrogen and phosphorus from inorganic to organic form, and back to inorganic, as well as coupling to dissolved oxygen are described.

The 12 state variables or components in the model are:

PC - phytoplankton carbon

PN - phytoplankton nitrogen

PP - phytoplankton phosphorus

CH - chlorophyll concentration

ZC - zooplankton carbon

DC - detritus carbon

DN - detritus nitrogen

DP - detritus phosphorus

IN - inorganic nitrogen

IP - inorganic phosphorus

DO - dissolved oxygen

BC - benthic vegetation

The content of nitrogen and phosphorus in zooplankton is further included, assuming that the carbon/nitrogen and carbon/phosphorus are constant.

The sediment acts as a source or a sink of N and P depending of many factors, such as temperature, primary production and water flow. The flux of N and P from the sediment is assumed to be proportional to sedimentation of detritus and algae N and P.

The physical, chemical and biological processes for carbon (C), nitrogen (N) and phosphorus (P) described in the EU model are:

Table 1:

	C	N	P
Phytoplankton			
Production of phytoplankton:	PRPC		
Uptake of nutrients:	UNPN	UPPP	
Death of phytoplankton:	DEPC	DEPN	DEPP
Sedimentation of phytoplankton:	SEPC	SEPN	SEPP
Grazing by zooplankton:	GRPC	GRPN	GRPP
Zooplankton			
Production of zooplankton:	PRZC	PRZN	PRZP
Death of zooplankton:	DEZC	DEZN	DEZP
Respiration of zooplankton:	REZC	REZN	REZP
Excretion of org. matter:	EKZC	EKZN	EPZP
Detritus			
Mineralization of detritus:	REDC	REDN	REDP
Sedimentation of detritus:	SEDC	SEDN	SEDP
Sediment			
Mineralization of sediment:	RESC	RESN	RESP
Oxygen			
Oxygen production by phytoplankton:	ODPC		
Oxygen consumption by zooplankton:	ODZC		
Oxygen consumption by detritus:	ODDC		
Oxygen consumption in sediments:	ODSC		

Table 1:

Reaeration:	REAR		
Benthic vegetation			
Production:	PRBC		
Uptake of nutrients:		UNBN	UPBP
Sloughing of vegetation:	SLBC	SLBN	SLBP

These basic processes describe the transfer and reactions on carbon, nitrogen, phosphorus and dissolved oxygen in the modelled area driven by hydrodynamics and external functions. The external functions or EU Forcing Functions include incidence light radiation, temperature and loads of organic matter and nutrients. See also the introduction and the section Required Data for the EU Model *(p. 324)*.

9.1.2 Required Data for the EU Model

The EU model set-up for a lake, reservoir or coastal area consists of a hydraulic sub-model and the eutrophication sub-model. Data on water depth, surface and bottom areas are automatically transferred from the hydraulic results to the EU model during a simulation.

The annual variations in EU Forcing Functions *(p. 325)*, pollution loads or boundary concentrations are, however, required as specific input data to the EU model. These data are:

- Surface light intensity
- Water temperature
- Organic (detritus) loading of C, N and P
- Inorganic loading of N and P
- Concentrations of the pelagic components (phytoplankton, zooplankton, detritus, nutrients and oxygen) at the model boundaries

The recommended calibration data for the EU model are:

- Inorganic nitrogen
- Inorganic phosphorus
- Chlorophyll-a
- Dissolved oxygen
- Secchi depth, or light profiles

As a minimum these measurements are necessary for the model calibration. Measurements should be taken monthly and for a 6-12 months period. In (summer) periods where the primary production is high, a more frequent sampling is recommended. Additionally, it is also recommended to measure:

- Primary production
- Total organic carbon
- Total nitrogen
- Total phosphorus
- Zooplankton biomass
- Detritus quantities
- Macroalgae biomass

The water sampling should be taken at different depths (e.g. in surface layer and 1 m above the bed) at a representative number of stations covering the model entire area.

9.1.3 EU Forcing Functions

The Required Data for the EU Model *(p. 324)* include water temperature and light (sun) radiation at the water surface in order to simulate the biological processes. These data are called forcing functions as the model is "forced" to use them, and should be specified as TimeSeries.

When extracting the two EU Forcing Functions (TimeSeries) to your EU model set-up, the River Name "GLOBAL" must be specified in the MIKE 11 Boundary file. Values for Chainage and Component Number are not required.

9.1.4 Guidelines for Selection of Time Step

The EU model uses two time steps: One for the AD- and one for the EU-computation.

The AD time step is specified as for any other MIKE 11 simulation whereas the EU time step is specified under General model parameters *(p. 330)* (the parameter WQDT).

Regarding WQDT the user should note the following:

- The EU time step (in hours) usually is higher than the AD time step (in minutes) as eutrophication processes only are dynamic on a day-to-day basis, and thus in order to save computing time, the EU-computation should be less frequent than the AD computation. WQDT should, however, be small enough to resolve steep gradients in the state variables caused by movements of the water bodies and pulses in inputs from the boundaries and tributaries
- The AD time step is chosen as for any other AD simulation
- The AD and EU time steps should not be selected independently of each other. The EU time step MUST be a multiple of the AD time step. Otherwise, the results will be erroneous.

9.1.5 EU Results

The result from a model simulation, i.e. the concentration distributions of the 12 state variables will stored in the .res11 file as for any other AD simulation. Additionally, a number of EU Processes *(p. 322)* and some derived variables are stored in a EUAdd.res11 file:

- Total-nitrogen: PN+DN+ZN+IN (g/m^3)
- Total-phosphorus: PP+DP+ZP+IP (g/m^3)
- PRPC: phytoplankton primary production (g/m^2)
- PRBC: net benthic production (g/m^2)
- PRPC+PRBC: total net primary production (g/m^2)
- Secchi depth (m)
- Nitrogen balance for the sediment: SEDN+SEPN-RESN (g/m^2)
- Phosphorus balance for the sediment: SEDP+SEPP-RESP (g/m^2)

NOTE that the storing frequency must be a multiple of the Eutrophication (EU) time step. Only results calculated at time steps where the eutrophication model has been activated are valid. This is illustrated in the sketch below:

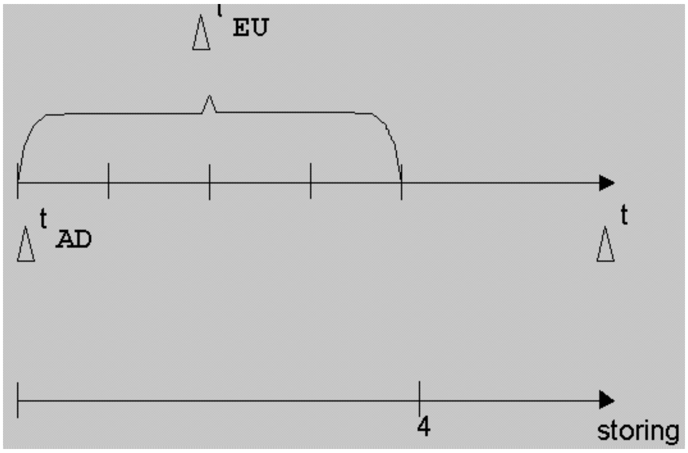


Figure 9.2 Storing frequency

Δt_{EU} = 3 hrs = 180 min

Δt_{AD} = 180, 90, 60, 30, 15, 5 min. and others

The state variables (i.e. .res11 file) and the process rates (i.e. EUAdd.res11 file) can be stored every time the eutrophication module is activated (in the above example every 3 hrs, 6 hrs, 9 hrs etc.).

9.2 EU Property Pages

Data editing forms consist of three entries:

1 Basic calibration parameters, i.e. the model coefficients that can be changed during the calibration phase

2 Ecosystem specific parameters, i.e. the coefficients related to the modelled ecosystem. These coefficients are changed occasionally

3 Physiological/chemical parameters, i.e. parameters which should only be changed if specific knowledge about the modelling area can justify this

The menus are grouped into 8 categories:

- Light extinction *(p. 328)*
- Oxygen *(p. 328)*

- Phytoplankton *(p. 328)*
- Sediment *(p. 329)*
- Zooplankton *(p. 329)*
- Benthic vegetation *(p. 329)*
- Detrius *(p. 329)*
- General model parameters *(p. 330)*

The property pages for each of the above are described below.

9.2.1 Light extinction

This property page offers possibility to edit parameters related to light extinction - the penetration of light through the water column.

All parameters are specified as meter^{-1}.

For general information on the EU Parameters, see EU Property Pages *(p. 327)*.

For specific information on EU parameters, please inspect the EU Technical Reference Manual.

9.2.2 Oxygen

This property page offers possibility to edit the oxygen related parameters.

All rates are specified as the maximum rate, day^{-1} at 20 degree Celsius.

For general information on the EU Parameters, see EU Property Pages *(p. 327)*.

For specific information on EU parameters, please inspect the EU Technical Reference Manual.

9.2.3 Phytoplankton

This property page offers possibility to edit the phytoplankton related parameters.

All rates are specified as the maximum rate, unit is day^{-1} at 20 degree Celsius, except for Maximum Growth rate for Diatoms, which is day^{-1} at 5 degree Celsius.

For general information on the EU Parameters, see EU Property Pages *(p. 327)*.

For specific information on EU parameters, please inspect the EU Technical Reference Manual.

9.2.4 Sediment

This property page offers possibility to edit the sediment related parameters.

All rates are specified as the maximum rate, day^{-1} at 20 degree Celsius.

For general information on the EU Parameters, see EU Property Pages *(p. 327)*.

For specific information on EU parameters, please inspect the EU Technical Reference Manual.

9.2.5 Zooplankton

This property page offers possibility to edit the zooplankton related parameters.

All rates are specified as the maximum rate, day^{-1} at 20 degree Celsius.

For general information on the EU Parameters, see EU Property Pages *(p. 327)*.

For specific information on EU parameters, please inspect the EU Technical Reference Manual.

9.2.6 Benthic vegetation

This property page offers possibility to edit the parameters related to the benthic vegetation.

All rates are specified as the maximum rate, day^{-1} at 20 degree Celsius.

For general information on the EU Parameters, see EU Property Pages *(p. 327)*.

For specific information on EU parameters, please inspect the EU Technical Reference Manual.

9.2.7 Detrius

This property page offers the possibility to edit the detritus related parameters.

All rates are specified as the maximum rate, day^{-1} at 20 degree Celsius.

For general information on the EU Parameters, see EU Property Pages *(p. 327)*.

For specific information on EU parameters, please inspect the EU Technical Reference Manual.

9.2.8 General model parameters

This property page offers possibility to edit the General Model Parameters.

Latitude for the model area is specified. For Southern Hemisphere, the latitude is specify as a negative value.

WQDT is the EU time step in hours. See Guidelines for Selection of Time Step *(p. 325)*.

SAL, is specified as the average salinity for the model area.

SEDIMENT TRANSPORT EDITOR

10 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. 335)*
- Transport model *(p. 336)*
- Calibration factors *(p. 341)*
- Data for graded ST *(p. 342)*
- Preset distribution of sediment in nodes *(p. 343)*
- Passive branches *(p. 344)*
- Initial dune dimensions *(p. 344)*
- Non-Scouring Bed Level *(p. 345)*

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.

10.0.1 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 un-checked (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.

10.0.2 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)

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.

10.1 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.

Figure 10.1 shows an example where the sediment grain diameter is globally set to 1 mm. This value will be used in the entire river network except for the reaches ‘RIVER1’ between 1000 and 2500, where the local grain diameter varies linearly between 1.2 and 1.5. and between 2500 and 4400 where the grain diameter varies linearly between 1.5 and 1.1. At the same chainages, the standard deviation varies linearly between 1.2, 1.2 and 1.0.

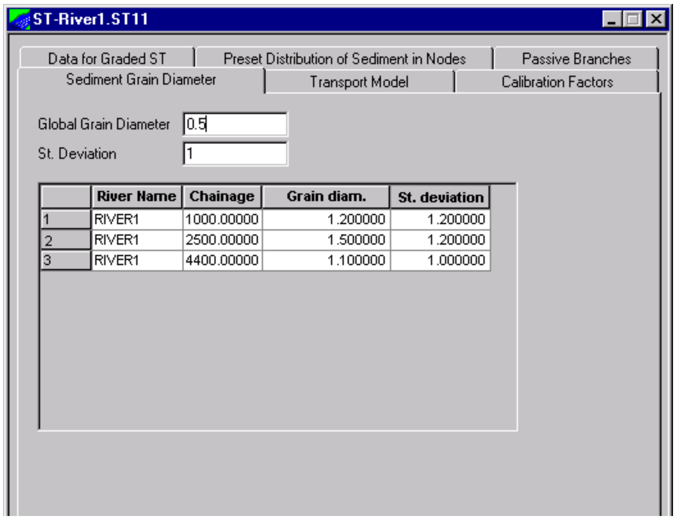


Figure 10.1 Example of implementation of local grain diameter.

10.2 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 checked by the user to set the model type (Total load or bed load/suspended load model), select the appropriate transport model(s) and adjust the transport parameters if required.

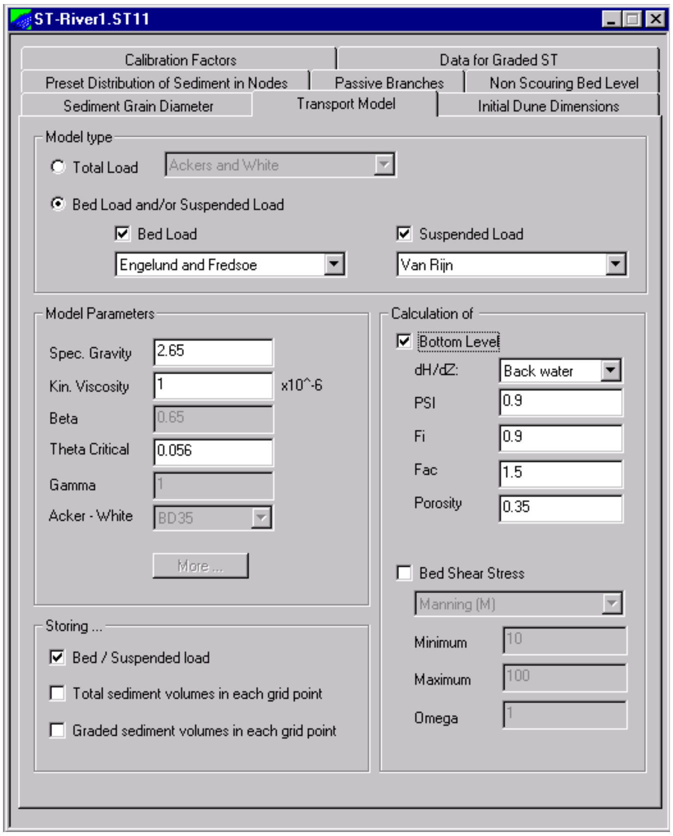


Figure 10.2 Example of implementation of transport model parameters.

Figure 10.2 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.

10.2.1 *Model Parameters*

The transport model parameters can be divided into three sub-groups:

Parameters used by the actual transport models

Spec. Gravity

Specific gravity of the sediment.

Kin. Viscosity

Kinematic viscosity of water.

Please note, that - using SI-Units - the Kinematic Viscosity must be specified as 'value · 10^{-6}' m^2/s. 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.

Theta Critical

Critical Shields' parameter.

Gamma

Calibration parameter for suspended load.

Acker-White

Switch used in the Ackers-White model indicating whether the applied grain size represents d_{35} or d_{65}.

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.

– Graded sediment volumes in each grid point

  Storing of sediment volume of each fraction in each grid point.

Parameters used if a morphological computation is included

Calculation of Bottom Level

A check box is provided to include or exclude bed level updating during the simulation.

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.

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 calculated throughout the ST simulation (Manning's M, Manning's n or Chezy).

Omega

Calibration parameter for the resistance number (ResistanceST = OMEGA + ResistanceHD).

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. Thus, the Chezy or Manning number specified in the cross-section data base may differ from the value(s) applied in the hydrodynamic computations.

10.2.2 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 10.7.

Smart-Jaeggi model

When selecting the Smart-Jaeggi transport model, the model parameters must be edited as for all other transport models. Additionally, coefficients and exponents used in the Smart-Jaeggi formulation can be edited. 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 as shown in Figure 10.3.

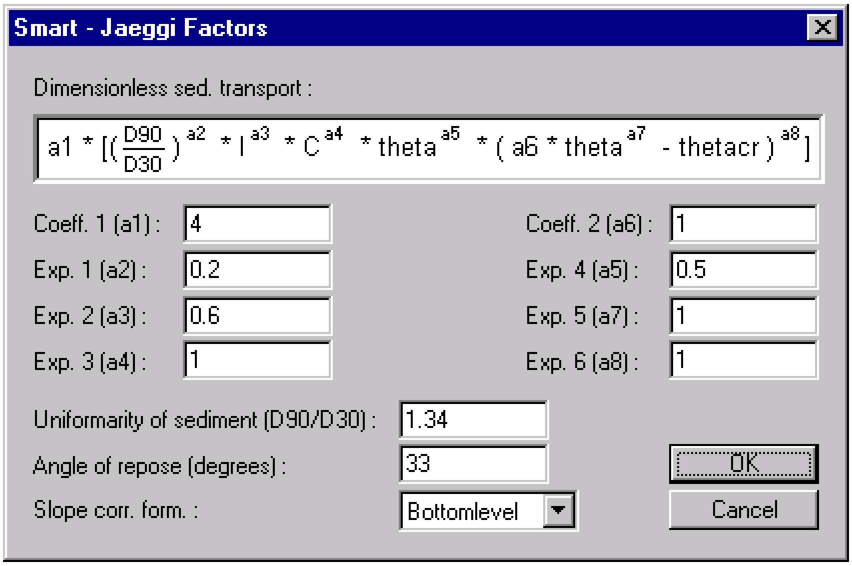


Figure 10.3 Additional dialog for defining Smart and Jaeggi model factors.

The Smart - Jaeggi Factors dialog is activated by pressing the 

button, which can be activated as soon as the transport model selected is ‘Smart and Jaeggi’.

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.

10.2.3 Bottom level update methods

Special options for updating the bottom level exists. The default method is to assume that the whole cross section is moved undistorted up in the case of deposition and down in the case of erosion. Alternatively, an ACSII file named ‘Bedlevel.txt’ can be placed in the data directory (together with the ST11-file) with specification of another update method. The first line in the ascii file is not read by MIKE 11. The second line should contain the Identification Number and the bottom level update method:

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

If the file ‘Bedlevel.txt’ does not exist, the default method (no 4) is applied. If the file exists, the user is prompted to confirm whether the settings in the file should be used before the simulation starts.

A more detailed description on the calculation of bottom levels is given in the NST Reference Manual.

10.3 Calibration factors

The factors ‘Factor 1’ and ‘Factor 2’ can be applied to the calculated transport rates as correction factors.

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, whereas for other models distinguishing between bed load and suspended load, ‘Factor 1’ is used as a multiplication factor for Bed load transport and ‘Factor 2’ as a multiplication factor for suspended load transport. Calibration factors can be specified globally and locally as shown in Figure 10.4, 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.

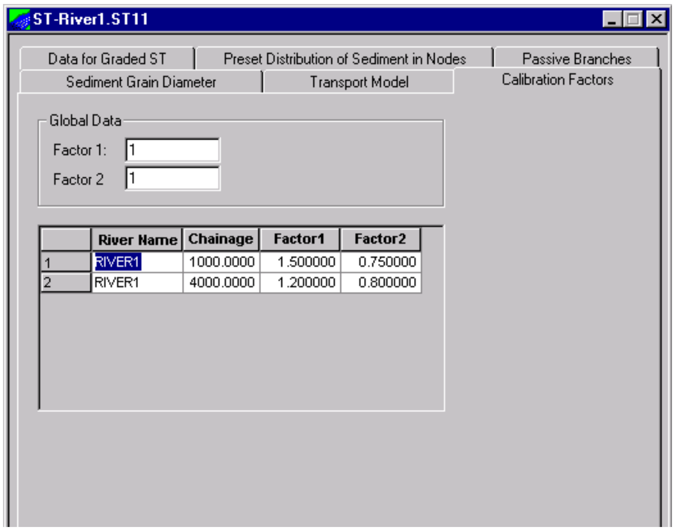


Figure 10.4 Calibration factors dialog

10.4 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 overlying an inactive, passive layer. Each layer is divided into an equal number of fractions (or classes) specified by the user. A mean grain size (mm) 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’ check box.

The percentage contribution and transport rate of each fraction can be stored in the result file by setting a check mark in the ‘Save fraction values’ and ‘Save sed. transport each fraction’ check boxes. If the result file is to be used as a hot start file, the 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 10.5.

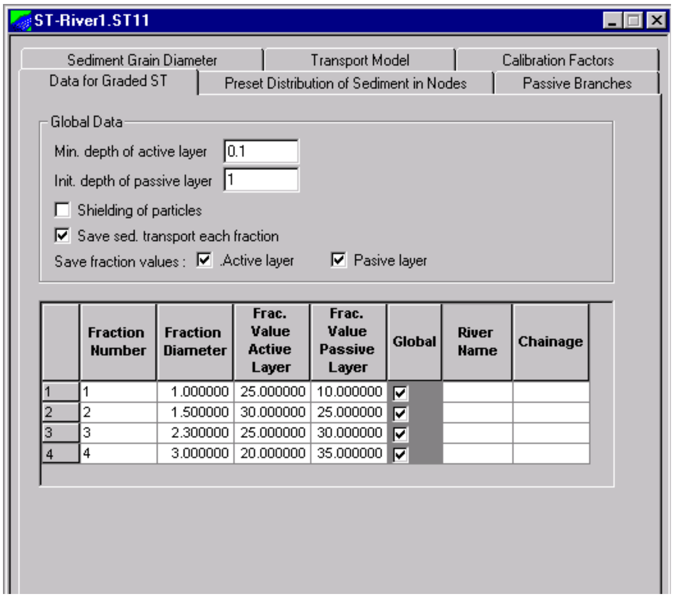


Figure 10.5 Example of specifying Graded ST data (4 fractions)

10.5 Preset distribution of sediment in nodes

The default distribution at a node is carried out according to the ratio of flow discharges. An alternative distribution can be specified on this property page by providing the coefficients and the exponents (K and n values) in the following relationship:

$$Qt_m^{n+1} = \frac{K_m Q_m^{n_m}}{\sum\limits_{\text{downstream branches}} K_i Q_i^{n_i}} \sum_{\text{upstream branches}} Qt_j^{n+1} \tag{10.1}$$

Where

Qt_m^{n+1} sediment transport rate in branch m

The coefficients and exponents are given for each branch, specified by its upstream and downstream chainage, linked to the node. The property page also enables the addition and editing of a preset distribution of sediment in nodes related data.

10.6 Passive branches

Branches in which sediment transport should not be calculated are specified by river name and upstream and downstream chainage as shown in Figure 10.6. Sediment can be transported into a passive branch, but no sediment can be transported out of the branch.

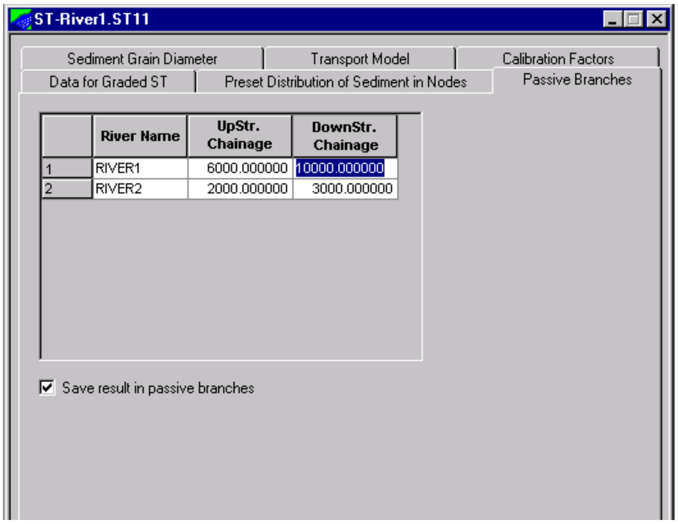


Figure 10.6 Passive branches property page.

10.7 Initial dune dimensions

When selecting the Engelund-Fredsøe 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 10.7 shows an example where the global dune height has been set to 0.25, and the global dune length has been set to 12.50. These values will be used in the entire river network, except in the reach ‘RIVER1’, between chainage 5.000 and 10.000, where the dune height varies linearly between 0.25 and 0.40.

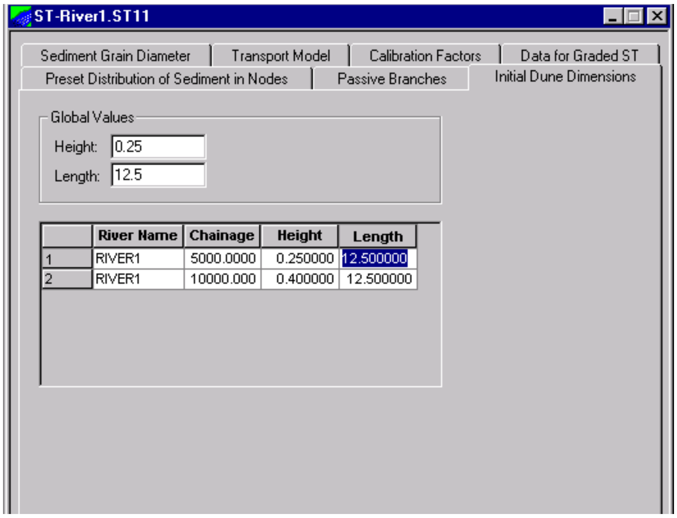


Figure 10.7 Example of an implementation of local initial dune dimensions.

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.

10.8 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 defined the thickness of active layer as half the dune height, but now the value can be user-defined. 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.

(Important to notice, that this item must be defined as a level - and not a height!) 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.

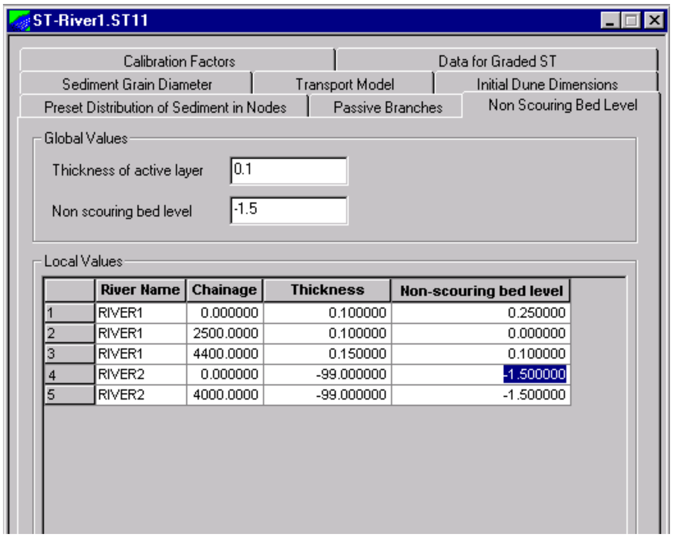


Figure 10.8 Non scouring Bed Level property page

Figure 10.8 shown an example where the global values of Thickness of active layer is defined to 0.1 and Non scouring bed level is set to -1.5. 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.

Please notice, 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.

F L O O D   F O R E C A S T I N G   E D I T O R

11 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.

11.1 Basic definitions

11.1.1 Simulation Period and Time of Forecast

The Time of Forecast (ToF) is defined in relation to the Hindcast and the Forecast Period in Figure 11.1.The Hindcast Period defines the simulation period up to ToF and is specified in the simulation file or calculated by the system; see Chapter 11.1.2, Simulation Mode. The length of the Forecast Period is always specified in the Forecast Menu, see section 11.2.1

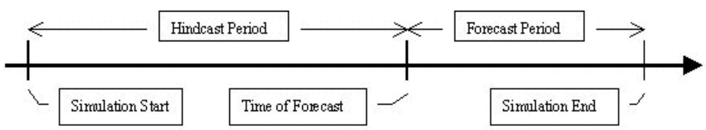


Figure 11.1 Definition of ToF

11.1.2 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 11.2. 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.

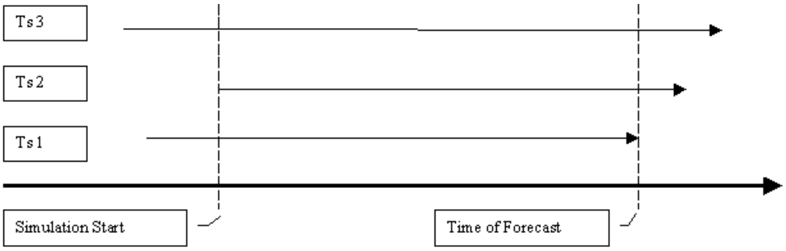


Figure 11.2 Definition of Hindcast Period and ToF

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 11.3 the hindcast period starts on the 4 January 1999 at 12:00 and last up to 7 January 1999 at 12:00.

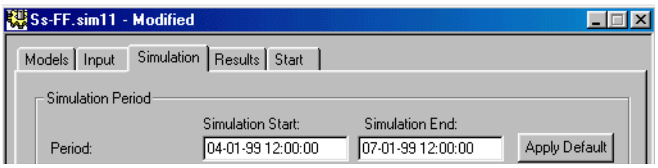


Figure 11.3 Definition of Hindcast period in historical mode

The forecast period is defined in the Forecast Menu.

11.2 Forecast

The main forecast parameters are specified in the Forecast Menu, Figure 11.4.

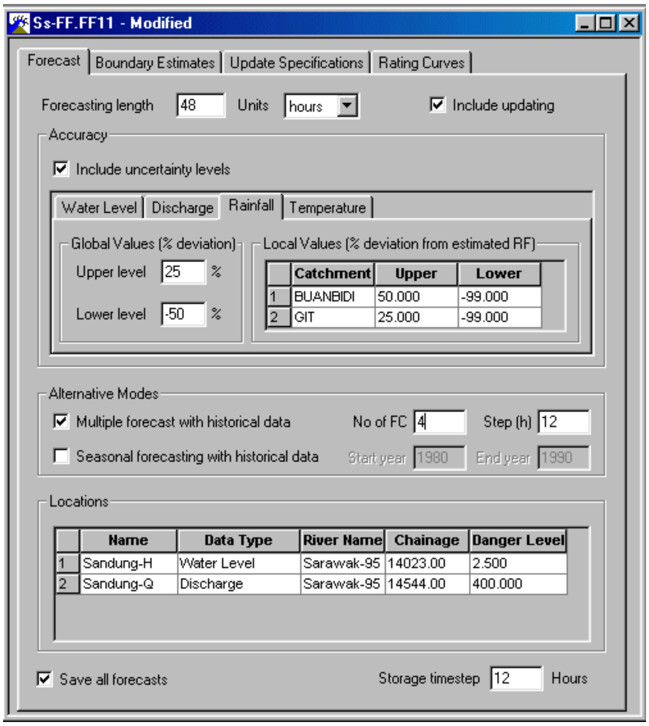


Figure 11.4 Basic Forecast Definitions

11.2.1 Forecast length

The Forecast length is equal to the Forecast Period (Figure 11.4). The length of the Forecast Period can be specified in **hours** or in **days**

11.2.2 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 11.4.

11.2.3 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 11.3.4.

11.2.4 Alternative Modes

Multiple forecast with historical data

To execute simulations in Historical Mode tick on the Multiple forecast check box, see Figure 11.4 or below. Additional information about simulating in Historical Mode can be found in Section 11.1.2.

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.

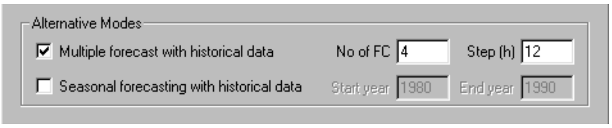


Figure 11.5 Selection of Historical Mode

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 11.6).

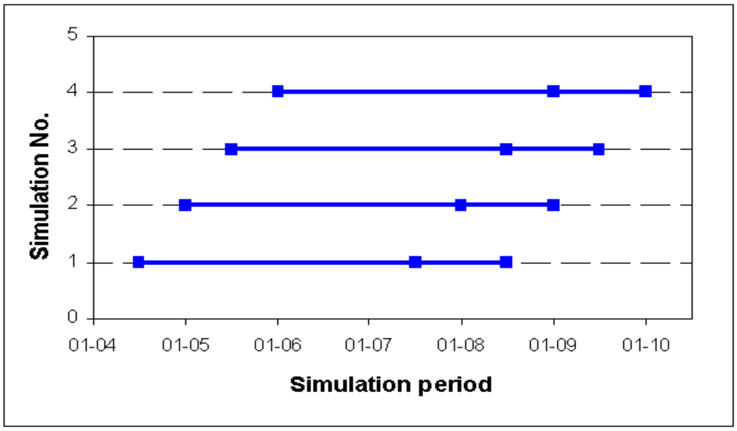


Figure 11.6 Multiple simulations in Historical Mode

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 11.1.2, 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

11.2.5 Location of forecast stations

Forecast points are specified as shown in Figure 11.7 below

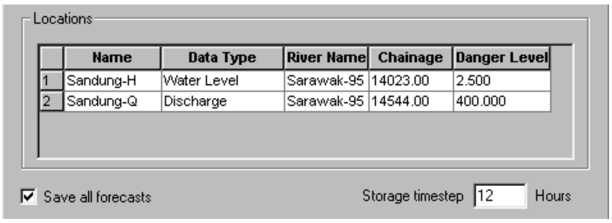


Figure 11.7 Location of Forecast Points

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 11.8.

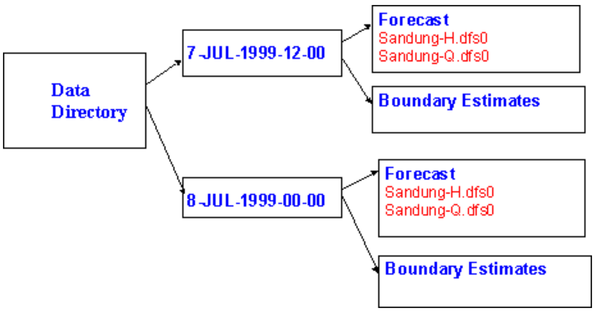


Figure 11.8 Forecast data directory structure

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 11.8. 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 timestep

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.

11.3 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 11.3.3.

Figure 11.9 shows the Boundary Estimates menu.

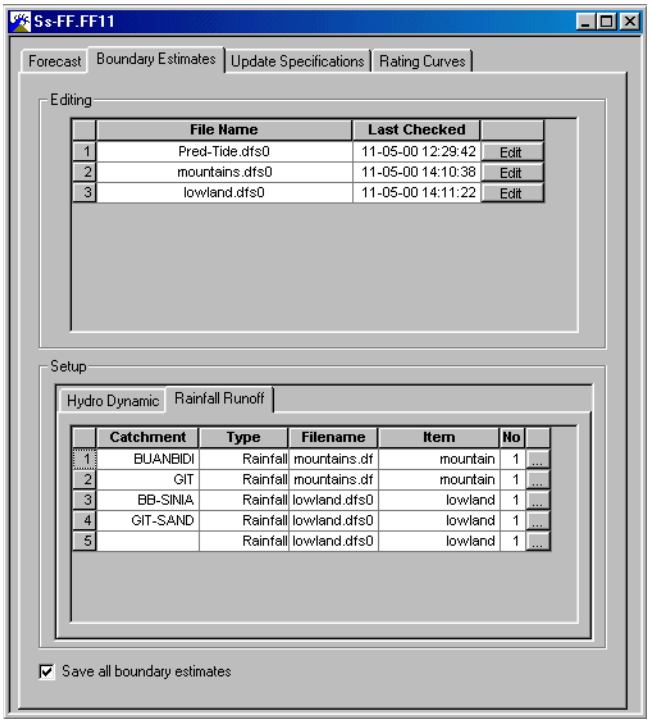


Figure 11.9 Boundary Estimates

11.3.1 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.

11.3.2 Editing

All files included in the setup menu will be listed in the 'Editing' menu as seen in Figure 11.9 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.

11.3.3 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 11.1 and their effect illustrated in Figure 11.10 through Figure 11.14.

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 11.1

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 11.10
The time series covers at least the whole forecast period.	No manipulation is required.	Figure 11.11
Estimated time series starts at ToF but does not cover the whole forecast period	Time series is extrapolated applying the last found value	Figure 11.12

Table 11.1

Case	Estimation method	Illustration
Estimated time series starts after ToF.	Time series is interpolated using hindcast data at ToF and the first entered estimated value	Figure 11.13
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 11.14

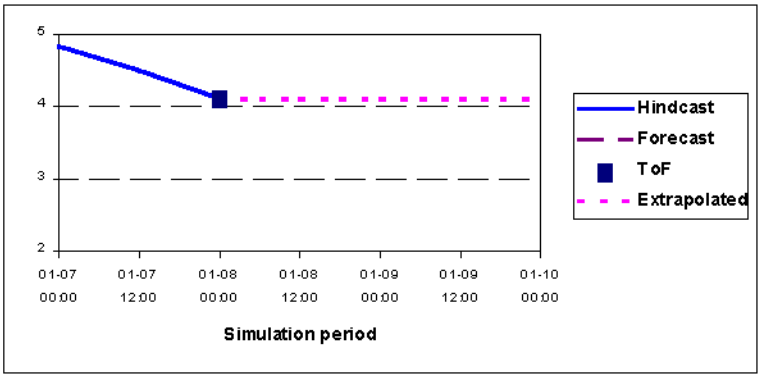


Figure 11.10   Extrapolation from value at ToF

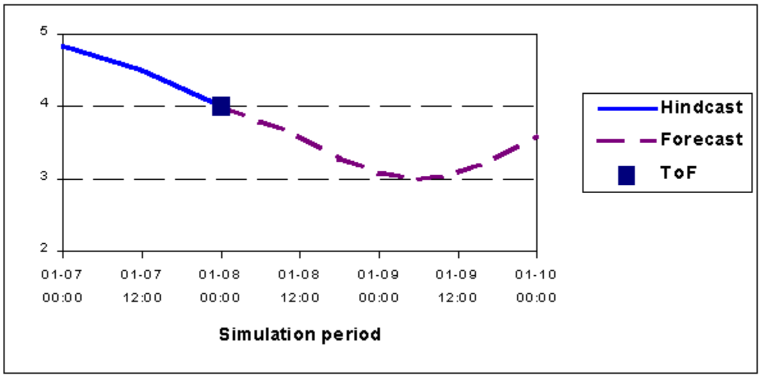


Figure 11.11   Estimated boundary conditions as specified.

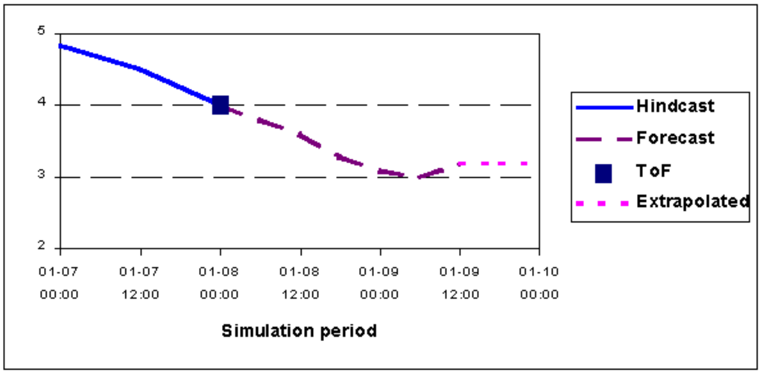


Figure 11.12 Extrapolation of Estimated boundary conditions

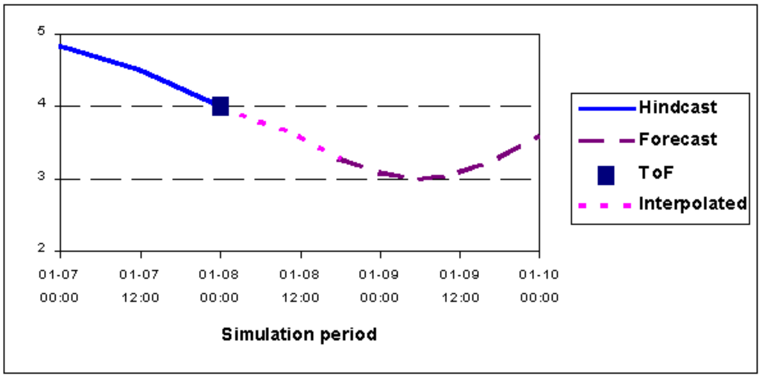


Figure 11.13 Interpolation of Estimated boundary condition

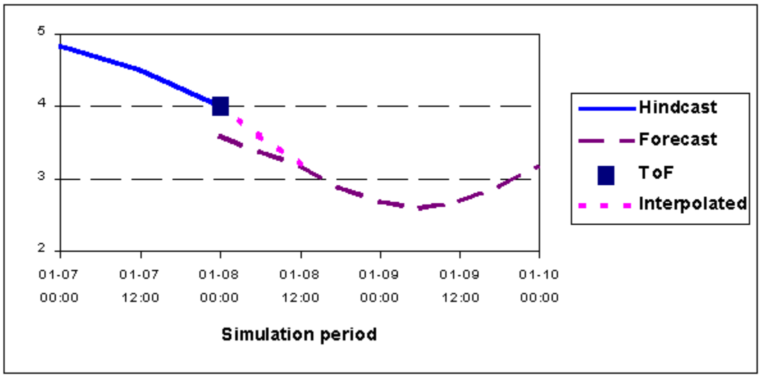


Figure 11.14 Discontinuity at ToF

11.3.4 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 11.2.5 and Figure 11.15 below.

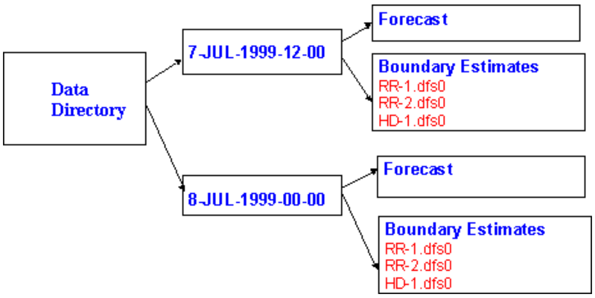


Figure 11.15 Estimated boundary directory structure

11.4 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 11.16 shows the Update Specification menu.

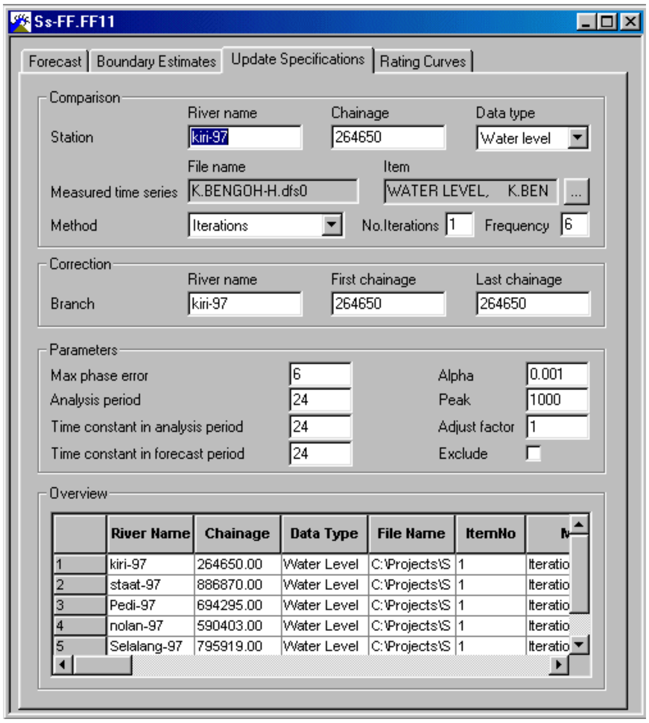


Figure 11.16 Update Specification

11.4.1 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 $q^{n+½}$ 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.

11.4.2 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

11.4.3 Parameters

Table 11.2

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

Table 11.2

Parameter	Main effect	Typical value
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

11.5 Rating curves

Not implemented.

B A T C H   S I M U L A T I O N   E D I T O R

12 BATCH SIMULATION EDITOR

The Batch Simulation Editor offers a possibility for setting up a batch simulation from the MIKEZero 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 will be performed automatically when starting the Batch simulation.

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:

- Predefine 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:

Predefine base simulation file

The Batch Simulation Editor is designed such, that a Base simulation file must be defined with all relevant information concerning Models and simulation mode, input-files, simulation period, time step, Initial conditions and output-file names. Batch simulations will then be performed with this Sim11-file as a basis and only if other parameters or file names have been defined by the user in the Batch Simulation Editor, will the definitions in the Base Sim11-file be modified.

File name 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. According to the number defined in this field a number of (empty) rows will be introduced in the 'Selected Parameters' grid, see example in Figure 12.2, where a number of 4 simulations has been chosen.

Each line in the 'Selected Parameters' grid must only contain specifications of the parameters or input files which should be different from 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 12.1. Open the tree view items by clicking the '+' and select the item/parameter which should be modified in the batch simulation by double-clicking in 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 which makes it possible for the user to select different input files or define variations in input parameters.

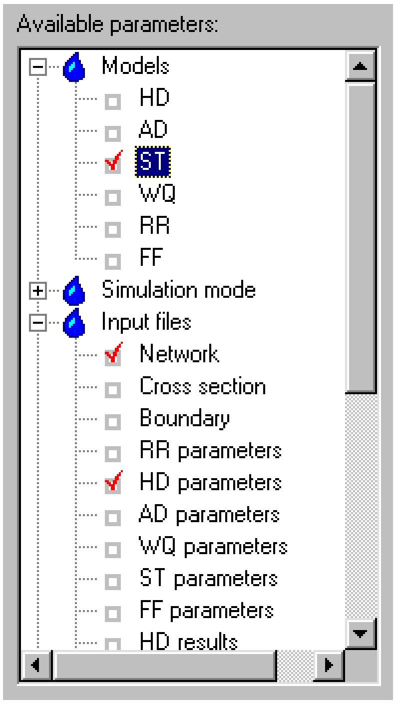


Figure 12.1 Tree view from the Batch Simulation Editor dialog for selecting batch simulation parameters

Specify input parameters for each simulation

Input parameters for the batch simulation can be different input file names, 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).

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.

After the verification of the setup has been performed, press the 'Run' button to start the batch simulations.

Figure 12.2 shows an example of 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 from the one file 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.

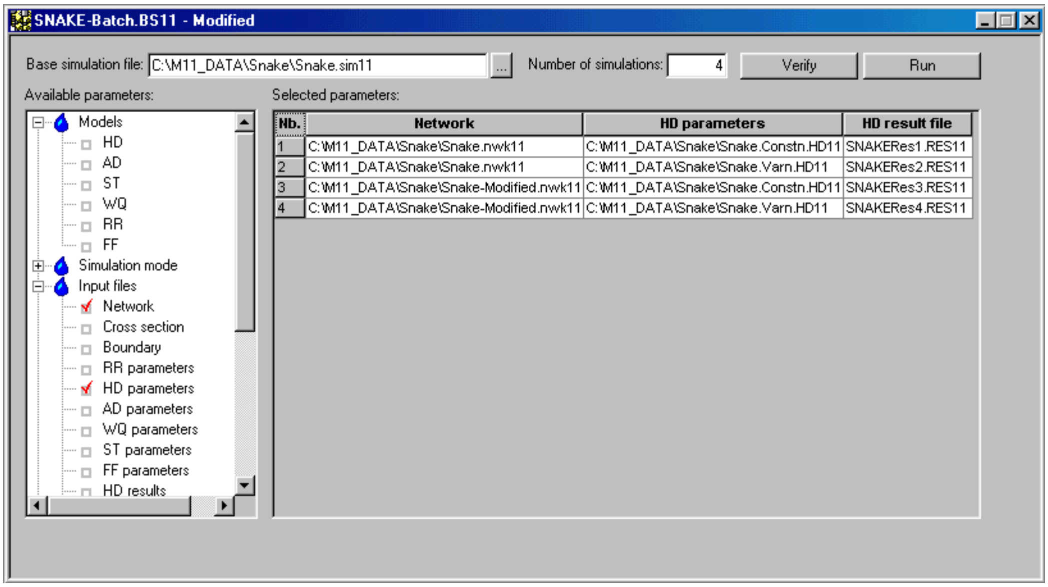


Figure 12.2 Example of Batch Simulation setup.

APP. A: FLOW RESISTANCE AND VEGETATION

A.1 FLOW RESISTANCE AND VEGETATION

Only a few detailed investigations have been made on establishing relationships between flow resistance in a stream filled with vegetation and flow resistance in the same stream without any vegetation. A quantitative evaluation of the influence of vegetation on the 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 resistance 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 investigations on the weeds influence on flow resistance are described in the following.

A.1.1 Flow Channels in Halkær Å

Jensen et. al, /4/ describes experiments performed in a danish stream named ‘Halkær Å’. A straight-line stretch of the stream with very dense vegetation was chosen for the experiment, and regulators for control of the inflowing discharge to 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.; danish: Pindsvineknop) with few occurrences of Water Thyme (latin: Helodea sp.; danish: Vandpest). The obtained Q-h relations are presented in Fig A.1

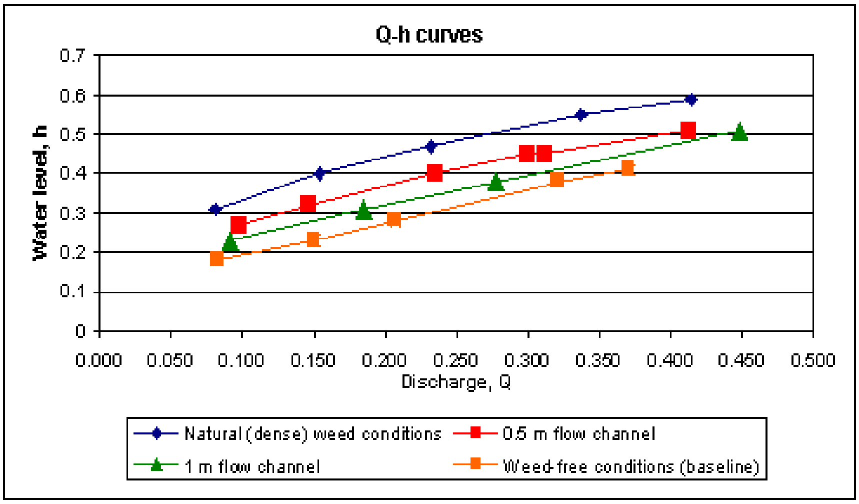


Fig A.1 Q-h curves determined for varying flow channel width

Calculated Manning numbers (Manning's *M*) are presented in Fig A.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.

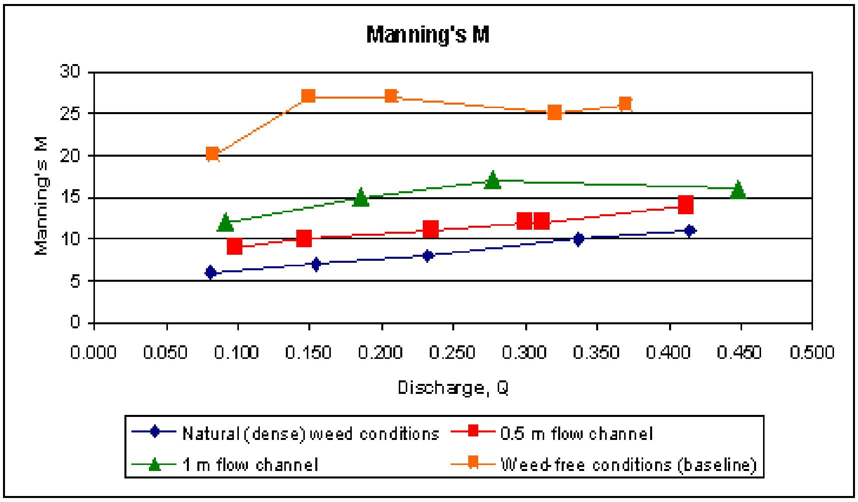


Fig A.2 Mannings M calculated as a function of Discharge, Q

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.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).

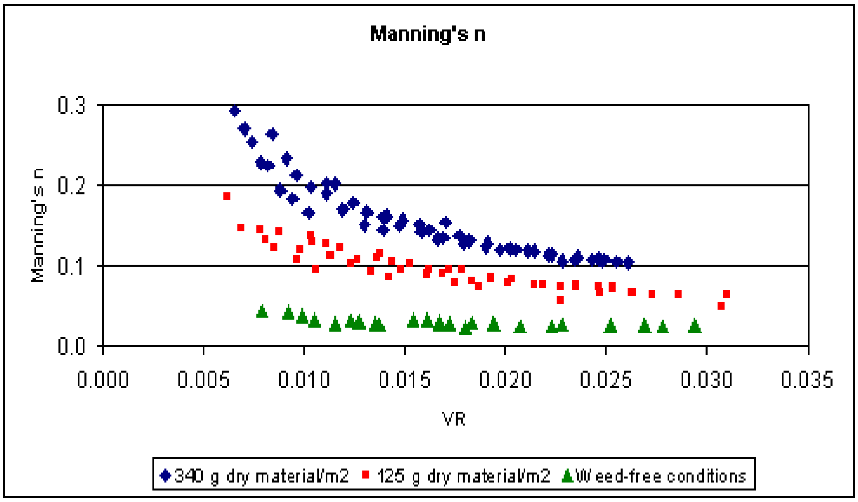


Fig A.3 Manning's n vs VR (VR: Velocity times Hydraulic Radius)

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:

$$n = a\ln(VR) + b \qquad \text{(A.1.1)}$$

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_{η} 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.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.

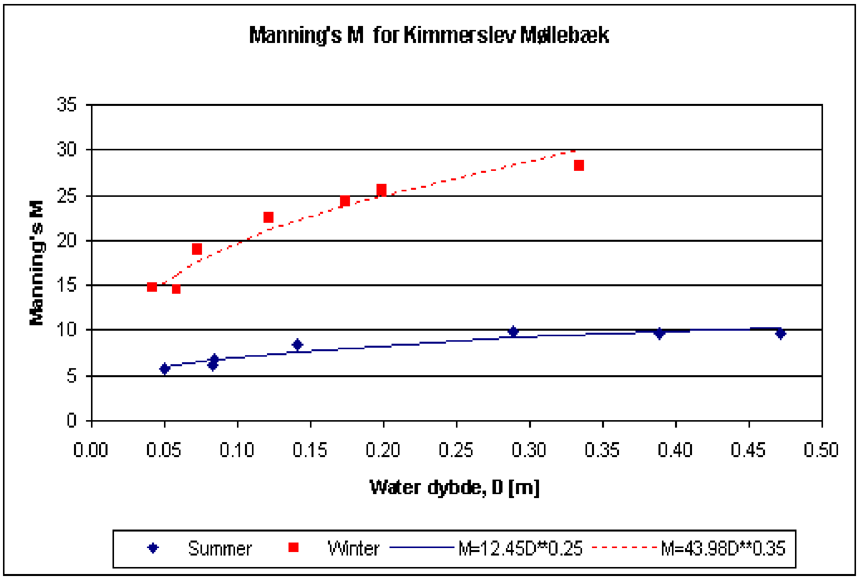


Fig A.4 *Manning’s M for Kimmerslev Møllebæk in summer and winter period. Results calculated with the formulas of the form M =* αD^{β} *are also included.*

A.1.4 Experiments in ‘ArnÅ’

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 frictionformulaes for open channels”. *Hedeselskabet, dep. for Hydrometry and Waterressources, 1996 (In Danish)*

/3/ Jensen, K.R.:

“Undersøgelse af Vandløbsvegetationens Hydrauliske Indflydelse.” *Afgangsprojekt, AUC, 1992*

“Investigagion of the influence of streamvegetations 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.*

APP. B: ADDITIONAL TOOLS

B.1 ADDITIONAL TOOLS

Apart from the catalogue of features which are accessible from the MIKEZero 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.)
- m11conv: This tool is used for converting set-ups from v. 3.2 or earlier to the MIKEZero format.
- res11read: A tool for converting result files from mike11 (.res11 files) to text files (ascii).

B.1.1 Merging .pfs files

In some instances it may be necessary to merge set-ups. To do so the pfsmerge.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. 30)* 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 set-ups from v. 3.2 and prior

m11conv is an application which is **only** for use when converting set-ups from v.3.2 and earlier to the present format. This facility is launched from the MIKE 11 menu under Start -> Programs ->MIKE 11 -> Mike 11 convert. The start up window has one pull down menu File which lists a number of conversion possibilities. Choose the appropriate format conversion and browse the file to be converted.

Note: When converting v.3.2 network-files (.RDF) all relevant cross section files (.pst, .ix0, .ix1) must be located in the same directory as the .RDF file.

B.1.3 Converting simulation results to text files

The application res11read is designed for converting one or more MIKE 11 result files to a text file (ascii). Thus the tool may be used as a conversion tool for subsequent post-processing of the results.

As for ‘pfsmerge’ the application is launched from a dos prompt. The syntax is:

...\RES11READ Option(s) Res11FileName1 ... Res11FileNameN OutputFileName

where

...\ denotes the full path to the application located in the bin director of the installation.

Res11FileName1 ... Res11FileNameN is the list of .res11 files to convert.

OutputFileName is the name of the output file (ascii).

Finally one or more of the options below should be used:

- xy: X-Y coordinates and levels for all grid points.
- xyh: X-Y coordinates and levels for all h-points.
- xyq: X-Y coordinates and levels for all Q-points.
- xyxsec: X-Y coordinates and levels for all h-points with cross sections.
- raw: Raw data for cross sections.
- sim: Content of the .sim11 file used for the simulation.
- minX: Minimum values in grid points for item no X.
- maxX: Maximum values in grid points for item no X.
- xsecids: Cross section IDs.
- usermarks: User defined marks.
- items: List of dynamic items.
- allres: All results of the simulation.
- someresFILE: Some results are written to the output file (selection in FILE).
- compareFILE: Compare results (selection in FILE).
- silent: Writing to prompt is cancelled. Used in conjunction with one or more of the other options.
- MessageFILE: Return file with 0 or 1 for Compare results (Returning FILE).
- DHIASCII: Additional option for suppressing header information - in DHI standard format. Should be used in conjunction with one or more of the above.
- FloodWatch: Flood Watch comma separated Matrix format.

For the option someresFILE the format of the FILE is:

```
ItemNumber   Chainage  Rivername
  1           100       MAIN
  1           200       MAIN
```

Figure 12.3 Format for use in the file used for the someresFILE option.