

DHI Software

MIKE 11

A modelling System for Rivers and Channels

User Guide (vol 2)



1 / 126

DOCUMENT CLASSIFICATION
REVIEW WAIVER PER
CLASSIFICATION OFFICE

ADMIN RECORD
DHI Software 2000

SW-A-004550



Please Note

Copyright

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

Limited Liability

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

IN NO EVENT SHALL DHI OR ITS REPRESENTATIVES (AGENTS AND SUPPLIERS) BE LIABLE FOR ANY DAMAGES WHATSOEVER INCLUDING, WITHOUT LIMITATION, SPECIAL, INDIRECT, INCIDENTAL OR CONSEQUENTIAL DAMAGES OR DAMAGES FOR LOSS OF BUSINESS PROFITS OR SAVINGS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION OR OTHER PECUNIARY LOSS ARISING OUT OF THE USE OF OR THE INABILITY TO USE THIS DHI SOFTWARE PRODUCT, EVEN IF DHI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. THIS LIMITATION SHALL APPLY TO CLAIMS OF PERSONAL INJURY TO THE EXTENT PERMITTED BY LAW. SOME COUNTRIES OR STATES DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL, SPECIAL, INDIRECT, INCIDENTAL DAMAGES AND, ACCORDINGLY, SOME PORTIONS OF THESE LIMITATIONS MAY NOT APPLY TO YOU. BY YOUR OPENING OF THIS SEALED PACKAGE OR INSTALLING OR USING THE SOFTWARE, YOU HAVE ACCEPTED THAT THE ABOVE LIMITATIONS OR THE MAXIMUM LEGALLY APPLICABLE SUBSET OF THESE LIMITATIONS APPLY TO YOUR PURCHASE OF THIS SOFTWARE.'

Printing History

May 2000
Edition 2000



HydroDynamic Parameter Editor

3



6 HYDRODYNAMIC PARAMETER EDITOR

6.1 Introduction

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. 161)
- Add. Output (p. 163)
- Flood Plain Resistance (p. 165)
- Initial (p. 166)
- Wind (p. 167)
- Bed Resistance (p. 168)
- Bed Resistance Toolbox (p. 172)
- Wave approx (p. 173)
- Default values (p. 175)
- User Def. Marks (p. 177)
- Mixing Coefficients (p. 179)
- W. L. Incr.- Curve (p. 180)
- W. L. Incr.- Sand Bars (p. 182)

6.2 Quasi Steady

6.2.1 Computational parameters

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

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.

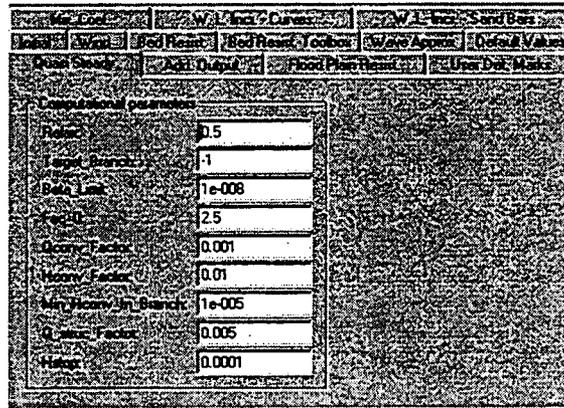


Figure 6.1 The Quasi Steady property page.

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.

6.3 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.

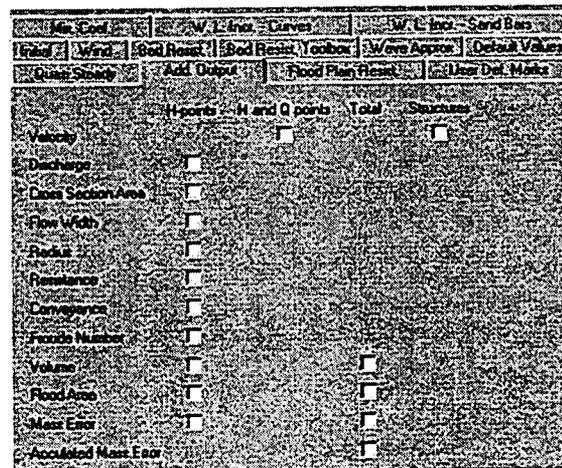


Figure 6.2 The additional output property page.

**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 interpreted 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:

$$F = \frac{Q}{A \sqrt{g \frac{A}{b_s}}} \quad (6.1)$$

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 neighboring Q-points.

Total: The total surface water area for the river system.

Mass Error

Mass error measured in m^3/s . 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.

6.4 Flood Plain Resistance

Flood plain resistance numbers are applied when using the morphological sediment transport modules. 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.

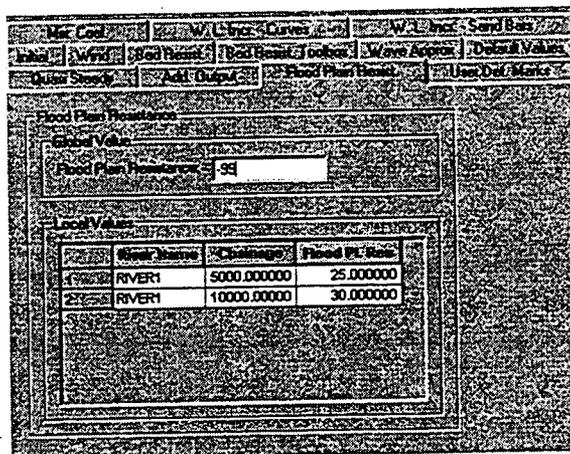


Figure 6.3 The Flood Plain Resistance property page.

6.5 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.

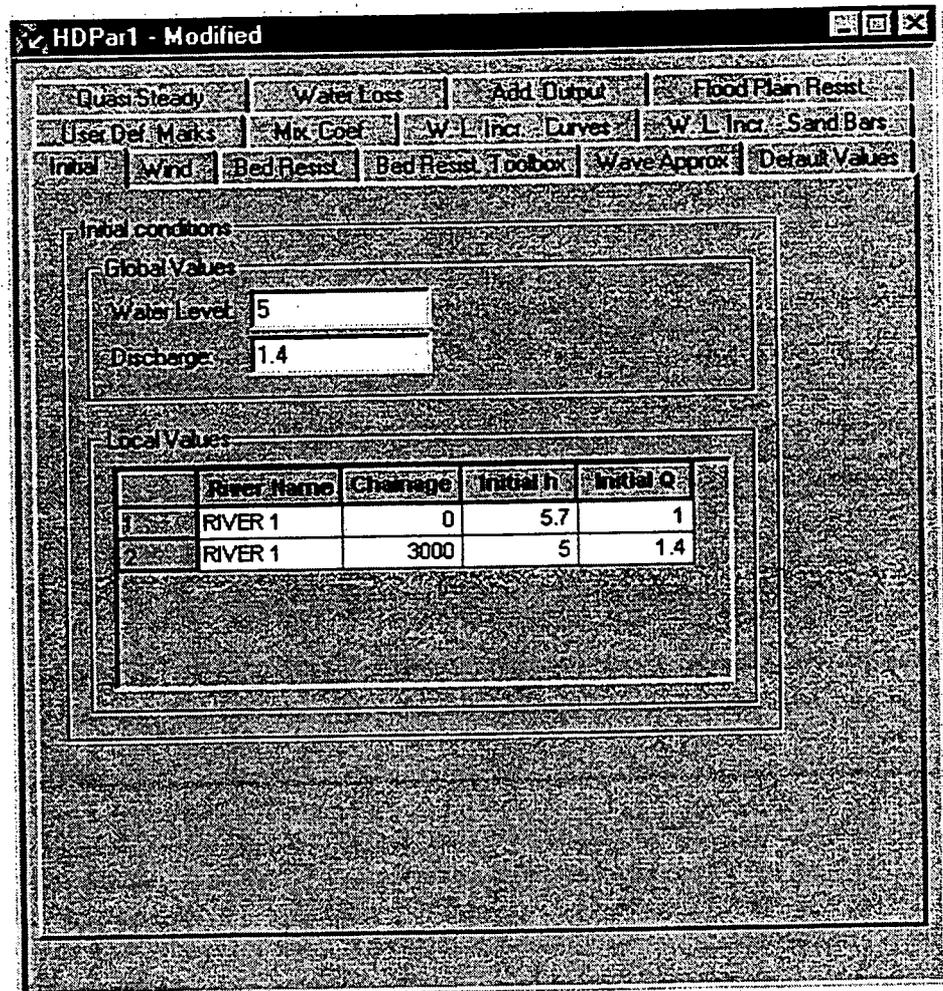


Figure 6.4 Initial value tab

6.6 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.



	River Name	Changeage	Factor
	RIVER 1	0.000000	0.7
	RIVER 1	5000.000000	0.3

Figure 6.5 Wind tab.

6.7 Bed Resistance

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

6.7.1 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 Mannings 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.

12



HDPPar1 - Modified

Quasi Steady | Water Loss | Add Output | Flood Plain Resist.
User Def. Mark | Mix. Coef. | W/L Inc. Curves | W/L Inc. Sand Bars
Initial | Wind | Bed Resist. | Bed Resist. Toolbox | Wave Approx. | Default Values

Approach:
 Uniform Section
 Triple zone

Resistance Formula:
Manning (M)

Global Values:
Resistance Number: 30

Local Values:

	River Name	Change	Resistance
1	RIVER 1	0	30
2	RIVER 1	5000	25
3	RIVER 1	14000	20
4	RIVER 1	21000	30

Figure 6.6 Uniform approach for implementation of the bed resistance.

6.7.2 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

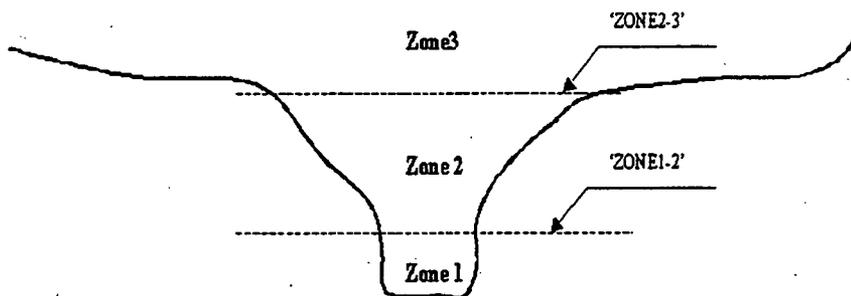


Figure 6.7 Triple Zone division of cross section

Zone separator lines must be defined in the User Defined Markers page (see description in Activation of Bed resistance Triple Zone Approach (p. 177)).

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

6.7.3 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 vegetations influence on the flow resistance has been performed in a few danish gauging-programmes. These are referred to in •.

124



6.8 Bed Resistance Toolbox

Name	Coverage	a	b	Min. Vel.	Max. Vel.
Global	100%	-1	0	0.025	0.5

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 combobox:

- Not Active

Bed resistance values used in the computation are those specified in the Bed Resistance page (Uniform or Triple zone approach)

- $n = 1/M = a \cdot \ln(VR)^b$

The bed resistance is calculated as a function of $\ln(\text{velocity} \cdot \text{Hydraulic Radius})$

- $n = 1/M = a \cdot D^b$

The bed resistance is calculated as a function of the Water depth.

- $n = 1/M = a \cdot V^b$

The bed resistance is calculated as a function of the velocity.

- Table (Velocity, Resistance value)

A User defined table of resistance value as a function of actual velocity can be defined. The bed resistance value applied in the simulation will



be the interpolated value from this table, depending on the actual velocity.

Note. To define the first line in the table, click the 'Velocity' bar in the upper half of the page. Thereafter, press the <TAB> button and a new line will be present in the grid in the upper part of the page.

All features (equations and table) can be defined both globally and local.

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 do not go below or above values considered reasonable by the user for the specific setup.

6.9 Wave approx

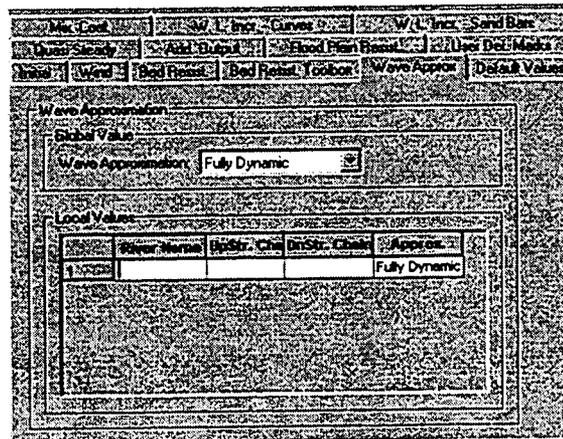


Figure 6.9 The Wave Approximation property page.

There are 4 possible flow description available in MIKE 11. The flow descriptions can be selected globally for the system and/or locally for indi-

76



vidual 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).

6.9.1 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.

6.9.2 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.

6.9.3 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.



6.10 Default values

Computation Scheme			
Delta	0.5	Zeta Min	0.1
Delh	0.01	Zeta Max	0
Delhs	0.1	InterIMax	10
Alpha	1	Ketcher	1
Theta	1	Jitter Size	100
Dh Mod	0.0001	Jitter Angle	-1
Dh Mod2	0.01	Jitter Exp	-1

Flow Compatibility: Water Level

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 1.7 Coefficients, HD default parameters (p. 24) in the Reference Manual).

6.10.1 Computation Scheme

Delta

The 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 $5 \cdot \text{Delh}$ 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 alternative 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 1.28.1 Suppression of convective acceleration term (p. 101) in the Reference Manual.

6.10.2 Switches

Node Compatibility

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

6.11 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 (linear interpolation will be performed on stretches between chainages and marker levels defined in this page).

6.11.1 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 6.11 shows an example where a single point marker has been defined ('Main Bridge' at RIVER1, chainage 1500) and triple zone separator lines has been defined in RIVER1 in the reach from chainage 0.0 to 5000.

The screenshot shows the 'HDPar1.HD11' window with various tabs like 'Info', 'Wind', 'Bed Resist', etc. The 'User Def. Marks' tab is active, displaying a list of marks and a data table.

Mark title
1 MAIN BRIDGE
2 ZONE1-2
3 ZONE2-3

	River Name	Chainage	MAIN BRID	ZONE1-2	ZONE2-3
1	RIVER1	0.000000		1	1.5
2	RIVER1	1500.000000	2.3	1.2	1.7
3	RIVER1	5000.000000		0.9	1.6

Figure 6.11 Example of defining User Defined Marks



6.12 Mixing Coefficients

Figure 6.12 The Mixing Coefficients property page.

Used only in conjunction with the Quasi 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.

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.



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.

22



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

23

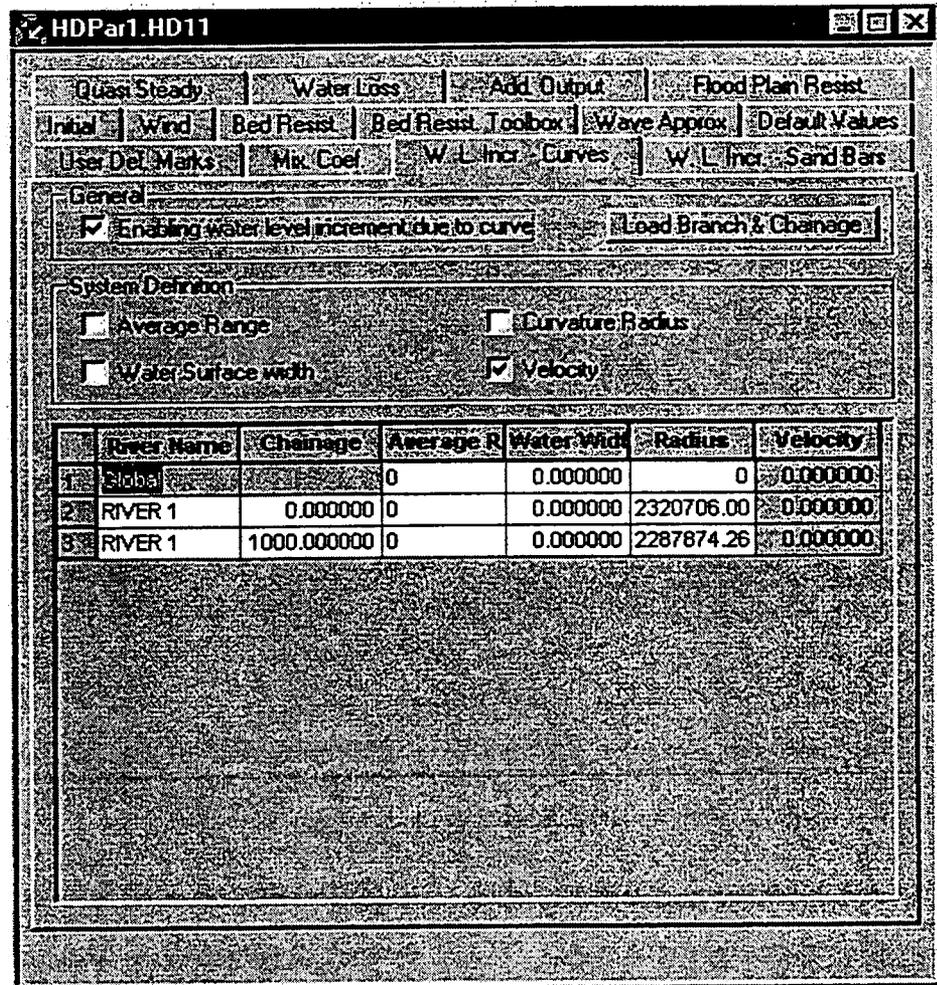


Figure 6.13 Water level increment due to curves.

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 be ticked.

Load Branch and Chainage button

This button activates a window with three choices

- Load the Branch and Chainage from Cross Section editor.

24



- 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.

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.

6.14 W. L. Incr.- Sand Bars

Used **only** in conjunction with the Quasi 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 6.14 with all the different features all of which are described below.

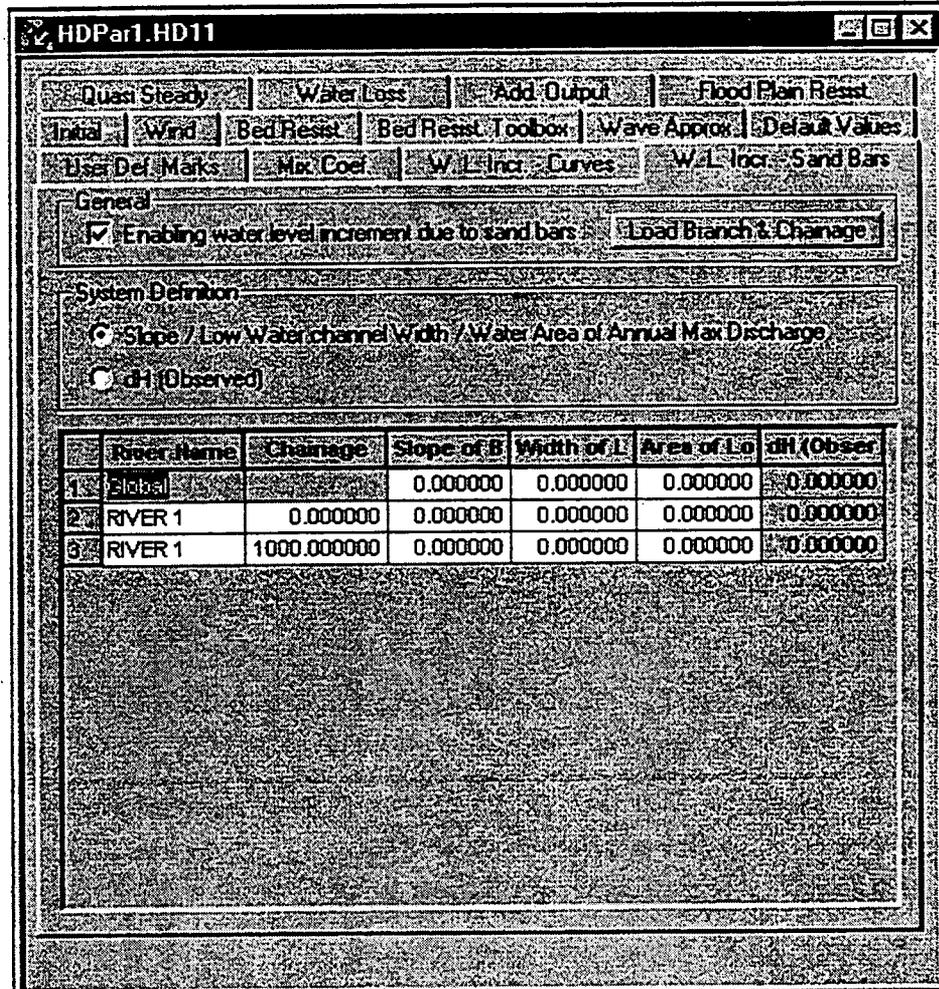


Figure 6.14 Water level increment due to sand bars.

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).

26



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.

27



Advection Dispersion Editor

28



7 **ADVECTION DISPERSION PARAMETER EDITOR**

7.1 **Introduction**

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.

7.1.1 **Advection Dispersion module (AD)**

The advection-dispersion (AD) module is based on the one-dimensional equation of conservation of mass of a dissolved or suspended material, i.e. the advection-dispersion equation. The module requires output from the hydrodynamic module, in time and space, in terms of discharge and water level, cross-sectional area and hydraulic radius.

The Advection-Dispersion Equation (*p. 188*) 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.

7.1.2 **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.

7.1.3 **Cohesive Sediment Transport module (CST)**

The cohesive sediment transport (CST) module also forms part of the AD module. In contrast to the non-cohesive sediment transport (NST) module,



the sediment transport cannot be described by local parameters only because the settling velocity of the fine sediments is very low. The cohesive module uses the AD module to describe the transport of the suspended sediment. Erosion/deposition is modelled as a source/sink term in the advection-dispersion equation. The erosion rate depends on the local hydraulic conditions whereas the deposition rate depends on the concentration of the suspended sediment and on the hydraulic conditions.

The module can also be used when resuspension of sediment affects water quality. This is because the resuspension of cohesive sediment often gives rise to oxygen depletion due to the high organic content and associated oxygen demand (COD) in the cohesive sediment. Likewise resuspension of cohesive sediment can give rise to heavy metal pollution since heavy metals adhere to the sediment.

7.1.4 *Advanced Cohesive Sediment Transport module (Advanced 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.

7.1.5 *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:

$$\frac{\partial AC}{\partial t} + \frac{\partial QC}{\partial x} - \frac{\partial}{\partial x} \left(AD \frac{\partial C}{\partial x} \right) = -AKC + C_2q \quad (7.1)$$

where

C : concentration (arbitrary unit)

D : dispersion coefficient (m^2/s)



- A : cross-sectional area (m^2)
- K : linear decay coefficient (s^{-1})
- C_2 : source/sink concentration
- q : lateral inflow
- x : space coordinate (m)
- t : time coordinate (s)

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.



7.2 Sediment layers

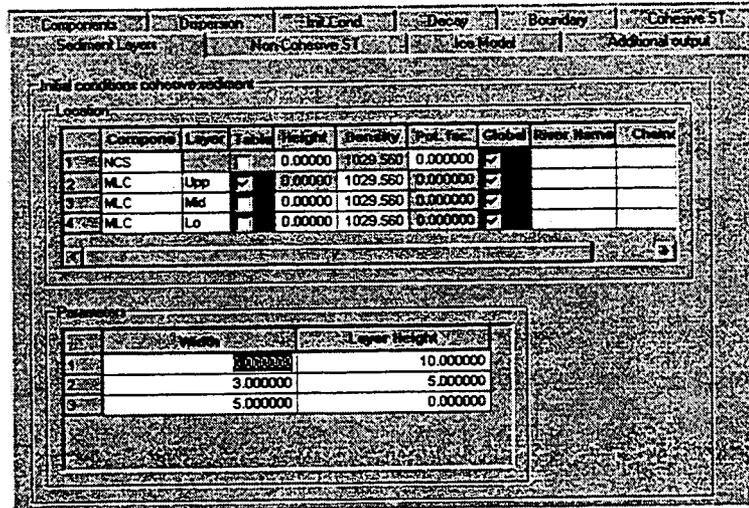


Figure 7.1 The Sediment Layers property page.

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. 191) a more detailed initial condition can be specified using a width-Height table, see Parameters (p. 191).

32



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. 192) 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.

7.2.1 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).

33



7.3 Non-cohesive ST

Data				
Component	Value 1	Value 2	Value 3	Value 4
NCS	0.001000	1.00000	<input checked="" type="checkbox"/>	

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.

7.4 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

35



- 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.

7.5 *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.



	Dispersion		Irr Cond		Decay		Boundary		Cohesive ST	
	Sediment Layer	Non Cohesive ST			1st Order Model					Additional output
	Total	Total Accumulated	Grid	Grid Accumulated						
Mass	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>						
Mass balance	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>						
1. order decay	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>						
Mass in branches	<input type="checkbox"/>	<input type="checkbox"/>								
Transport, total			<input type="checkbox"/>	<input type="checkbox"/>						
Dispersive transport			<input type="checkbox"/>	<input type="checkbox"/>						
Convective transport			<input type="checkbox"/>	<input type="checkbox"/>						

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.



7.6 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'.

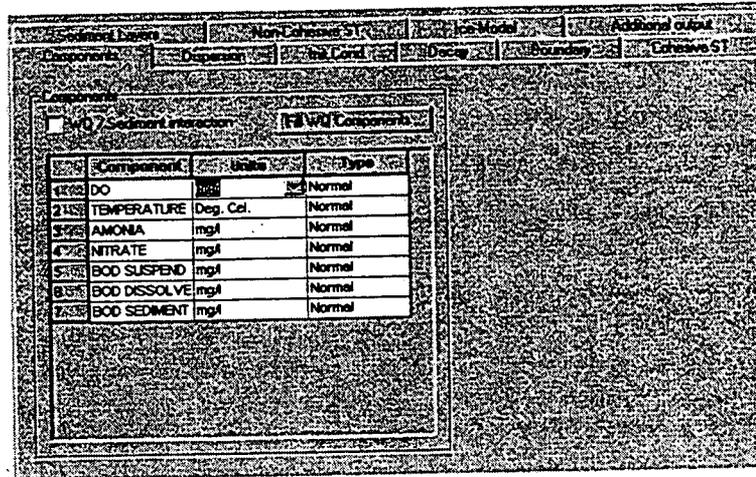


Figure 7.4 The component property page.

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.

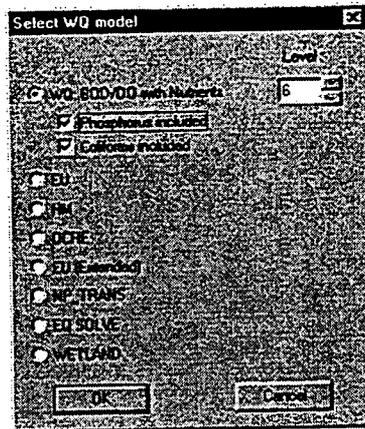


Figure 7.5 Selecting different WQ model components.

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/m³:** Microgram per cubic meter.



- **mg/m³**: Milligram per cubic meter
- **g/m³**: Gram per cubic meter
- **kg/m³**: 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. 196). 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.

7.7 Dispersion

The dispersion coefficient, D , is described as a function of the mean flow velocity, V , as shown below.

$$D = aV^b \quad (7.2)$$

Where a is the dispersion factor and b the dispersion exponent. Typical value ranges for D : 1-5 m²/s (for small streams), 5-20 m²/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.



River Name	Channel Length	Dispersion Coefficient	Exponent	Minimum Coeff.	Maximum Coeff.
RIVER1	10000.000	15.000000	1.000000	5.000000	25.000000
RIVER1	20000.000	15.000000	1.000000	5.000000	25.000000

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 where 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 $10 \text{ m}^2/\text{s}$ (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 ($D = 15V$, $5 < D < 25$)

7.8 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.



Initial conditions					
Component	Concentration	Global	River Name	Chainage	
COMP1	10	<input checked="" type="checkbox"/>			
COMP2	2	<input checked="" type="checkbox"/>			
COMP2	2	<input type="checkbox"/>	RIVER1	10000	
COMP2	7	<input type="checkbox"/>	RIVER1	20000	
COMP2	7	<input type="checkbox"/>	RIVER1	25000	

Figure 7.7 The initial conditions property page.

Initial values 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. 196).

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.

7.9 Decay

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

$$\frac{dC}{dt} = KC \quad (7.3)$$

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.

Component	Decay const (K)	Global	River Name	Chainage
1 COMP2	1.000000	<input checked="" type="checkbox"/>	RIVER1	10000.0000
2 COMP2	2.000000	<input type="checkbox"/>	RIVER1	20000.0000
3 COMP2	2.000000	<input type="checkbox"/>	RIVER1	20000.0000

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. 196).

Decay const

Here the value of the decay constant are entered.

474



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.

7.10 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

7.10.1 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

45



(during flow reversal at the boundary), the boundary condition concentration is adjusted according to:

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

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,

$$\frac{\partial^2 C}{\partial k^2} = 0 \quad (7.5)$$

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:

$$\frac{\partial C}{\partial k} = 0 \quad (7.6)$$

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

7.10.2 Entering the data

River Name	Chainage	Type	K Factor
MAIN	0.000000	Open, Transport	0.000000
MAIN	30000.0000	Open, Concentration	0.100000
TRIBUTARY	0.000000	Open, Transport	0.000000
DEADEND	0.000000	Closed	0.000800

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. 204).
- Open, Concentration, see description at Open concentration boundary (p. 203).
- Open, Transport, see description at Open transport boundary (p. 204).



K-mix

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

7.11 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.

7.11.1 Single Layer Cohesive Model

Component	Type	Global	River Name	Channel	C offset	Settling
1	Single	<input checked="" type="checkbox"/>			10	4.5
4						

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/m².**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/m².**Erosion coefficient**The erosion coefficient is applied linearly in the erosion expression. Typical range: 0.20 - 0.50 g/m²/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.



7.11.2 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/m².

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/m² for layer 1 and 0.20 - 0.50 N/m² for layer 2 and 3.

Erosion coefficient

The erosion coefficient is applied linearly in the erosion expression. Typical range: 0.20 - 0.50 g/m²/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/m²/s,

51



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.

7.11.3 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:

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

where,

S is the source term in the advection dispersion equation

C is the concentration of the suspended sediment (kg/m³)

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

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

τ is the critical shear stress for deposition (N/m²)



τ_{cd} is the bed shear stress (N/m²)

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

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

where,

ρ fluid density (kg/m³)

g acceleration of gravity (m/s²)

M the Manning number (m^{1/3}/s)

h flow depth (m)

V flow velocity (m/s)

Substituting the bed shear stress into the deposition equation results in the following equation:

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

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.

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

where

S source term in the advection dispersion equation



- τ bed shear stress (N/m²)
- τ_{ce} critical shear stress for erosion (N/m²)
- M^* erodibility of the bed (g/m²/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:

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

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:

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

where

- S_d rate of deposition (kg/m²/s)
- τ critical shear stress for deposition (N/m²)
- c suspended sediment concentration (kg/m³)

All deposited material is added to sub-layer 1.

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

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



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:

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

where,

S_E rate of erosion ($\text{kg/m}^2/\text{s}$)

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

56



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.

Coliforms and phosphorus are optional at all model levels.

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.

Coliforms and phosphorus are optional at all model levels.

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.

Coliforms and phosphorus are optional at all model levels.



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.

Coliforms and phosphorus are optional at all model levels.

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.

Coliforms and phosphorus are optional at all model levels.

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 and phosphorus are optional at all model levels.

Besides, the model can include Coli bacteria (faecal and total) and/or phosphorus on each of the six levels.

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.



8.2.1 Arrhenius

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

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

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.

8.2.2 Degradation at the bed

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 baseline sediment oxygen demand of 20°C is specified. The unit is g O₂/day. The baseline sediment oxygen demand is the basic demand of oxygen 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.

8.2.3 Bed/sediment (level 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 baseline sediment oxygen demand of 20°C is specified. The unit is g O₂/day. The baseline sediment oxygen demand is the basic demand of oxygen 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/m²/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.

8.2.4 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 veloc-

60



ity 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 # 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.

8.2.5 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 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.

Three parameters for nitrogen contents are required in the case of modeling 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 O₂ 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 O₂ used.

In summary:

First field

Global value for the release of ammonia at BOD decay (gNH₄⁺-N/gO₂)



Second field

Global value for the uptake of ammonia in plants proportional to the net photosynthesis ($\text{gNH}_4\text{-N/gO}_2$)

Third field

Global value for the ammonia uptake in bacteria proportional to their degradation of BOD ($\text{gNH}_4\text{-N/gO}_2$)

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

8.2.6 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 $\text{gNH}_4\text{-N/gBOD}$.

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 $\text{gNH}_4\text{-N uptake/gO}_2$ released.

In the last field the uptake of ammonia-nitrogen by bacteria must be specified relative to their uptake of oxygen. The unit is $\text{gNH}_4\text{-N uptake/gO}_2$ used.

In summary:

First field

WJ



Global value for the release of ammonia at BOD dissolved decay (gNH₄⁺-N/gO₂)

Second field

Global value for the release of ammonia at BOD suspended decay (gNH₄⁺-N/gO₂)

Third field

Global value for the release of ammonia at BOD bed decay (gNH₄⁺-N/gO₂)

Fourth field

Global value for the uptake of ammonia in plants proportional to the net photosynthesis (gNH₄⁺-N/gO₂)

Fifth field

Global value for the ammonia uptake in bacteria proportional to their degradation of BOD (gNH₄⁺-N/gO₂)

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

8.2.7 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 0-order reaction, supplemented with film 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 O₂ / g NH₄⁺-N.

63



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

8.2.8 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 bacteria films 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 0-order reaction, supplemented with film 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.

8.2.9 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.

$$\text{Coli decay} = C_{\text{coli}} K \Theta_T^{T-20} \Theta_S^{\text{SAL}} \Theta_I^I \quad (8.2)$$

Global values for the first order decay rates, K , for faecal and total coliforms, respectively, are specified in the first two fields, at 20 °C, 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.



Global values	
1 st order decay (total) (1/day)	0.700
1 st order decay (1/day)	0.800
Temperature coefficient of decay rate	1.090
Salinity coefficient of decay rate	1.006
Light coefficient of decay rate	7.400
Light coefficient (1/m)	1.400
Salinity (per thousand)	0.000

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

$$I = I_0 \frac{(1 - \exp(-\eta Z))}{\eta Z} \quad (8.3)$$

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

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

8.2.10 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 O₂.

In the second field, uptake of phosphorus by plants per g O₂ 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.

65



8.2.11 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 O₂ for dissolved, suspended and bottom BOD, respectively.

In the last field, uptake of phosphorus by plants per g O₂ 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.

8.2.12 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.

8.2.13 P. exchange with the bed

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

cel



There are three parameters to specify on this page, dealing with phosphorus 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.

8.2.14 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 $\text{kJ}/\text{m}^2/\text{hour}$.
- 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 $\text{kJ}/\text{m}^2/\text{day}$.

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

67



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.

8.2.15 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

Equation for reaeration constant

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.

68



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:

$$\frac{dC}{dt} = K_2(C_m - C) - R + P(t) \quad (8.4)$$

where

C oxygen concentration (mg/l)

C_m oxygen concentration at saturation (mg/l)

R respiration ($\text{gO}_2/\text{m}^3/\text{day}$)

$P(t)$ photosynthetic production ($\text{gO}_2/\text{m}^3/\text{day}$)

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.

69



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. 231).

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. 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.

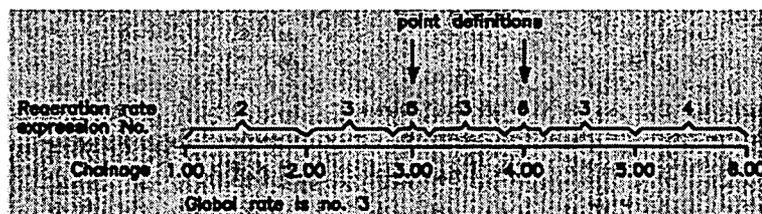


Figure 8.1 Combination of reaeration expressions

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

70



river stretch has to be split into sections at each side of the point definition. The order of the definitions is arbitrary.

8.2.16 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 square of 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 $(\text{gO}_2/\text{m}^3)^2$. The BOD decay increases at very low O_2 concentrations due to the depression of bacteria under anaerobic conditions.

The decay of BOD is calculated as

$$\text{Degradation} = \text{BOD} K_1 \Theta^{T-20} \frac{\text{DO}^2}{K_S - \text{DO}^2} \quad (8.5)$$

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

8.2.17 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.

71



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

$$K_2 = au^b h^c I^d \quad (8.6)$$

K_2 reaeration constant at 20°C (g O₂/m²/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. 228) 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

$$K_2 = 27185u^{0.931} h^{-0.692} I^{1.09} \quad (8.7)$$

The Thyssen expression is recommended for calculations in small streams.

2 O'Conner Dubbins expression

$$K_2 = 3.9u^{0.5} h^{-1.5} \quad (8.8)$$

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

3 Churchill expression

$$K_2 = 5.233uh^{-1.67} \quad (8.9)$$

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

72



Custom expressions

4

$$K_2 = a_3 u^{b_3} h^{c_3} I^{d_3} \quad (8.10)$$

5

$$K_2 = a_2 u^{b_2} h^{c_2} I^{d_2} \quad (8.11)$$

6

$$K_2 = a_1 u^{b_1} h^{c_1} I^{d_1} \quad (8.12)$$

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

74



Eutrophication Editor

75



9 EUTROPHICATION EDITOR

9.1 Introduction

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:

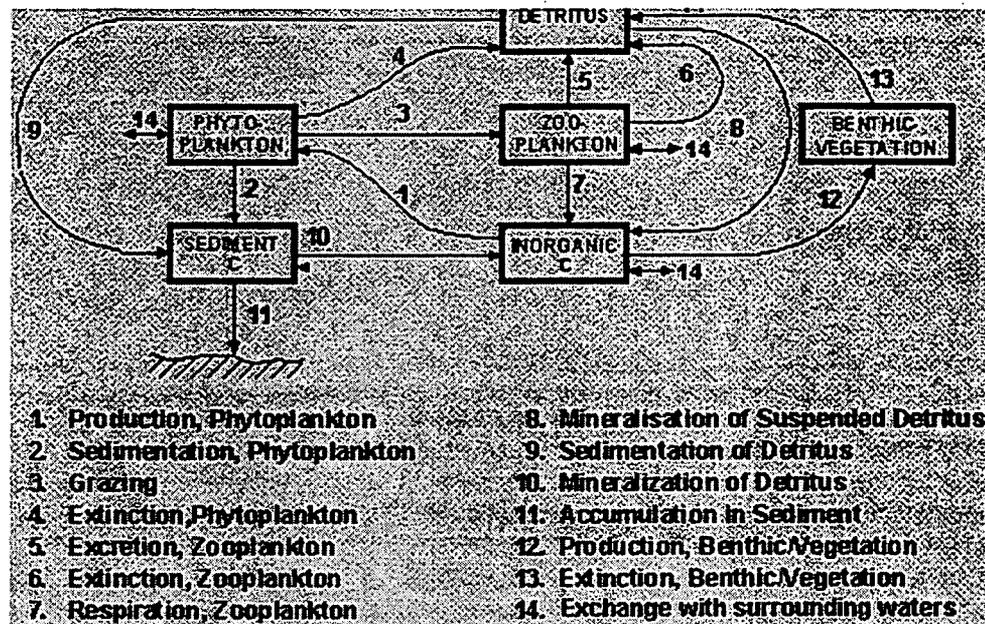


Figure 9.1 State variables and processes in the EU model exemplified by the cycling of Carbon (C).



See also EU Processes (p. 238) and Required Data for the EU Model (p. 240).

9.2 Background

9.2.1 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		

**Table 1:**

Oxygen consumption in sediments:	ODSC		
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 Introduction (p. 237) and Required Data for the EU Model (p. 240).

9.2.2 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. 241), 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.

9.2.3 EU Forcing Functions

The Required Data for the EU Model (p. 240) 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.

9.2.4 Guidelines for Selection of Time Step

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

80



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. 246) (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.

9.2.5 EU Results

The result from a model simulation, i.e. the concentration distributions of the 12 state variables will be stored in the .res11 file as for any other AD simulation. Additionally, a number of EU Processes (p. 238) 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:

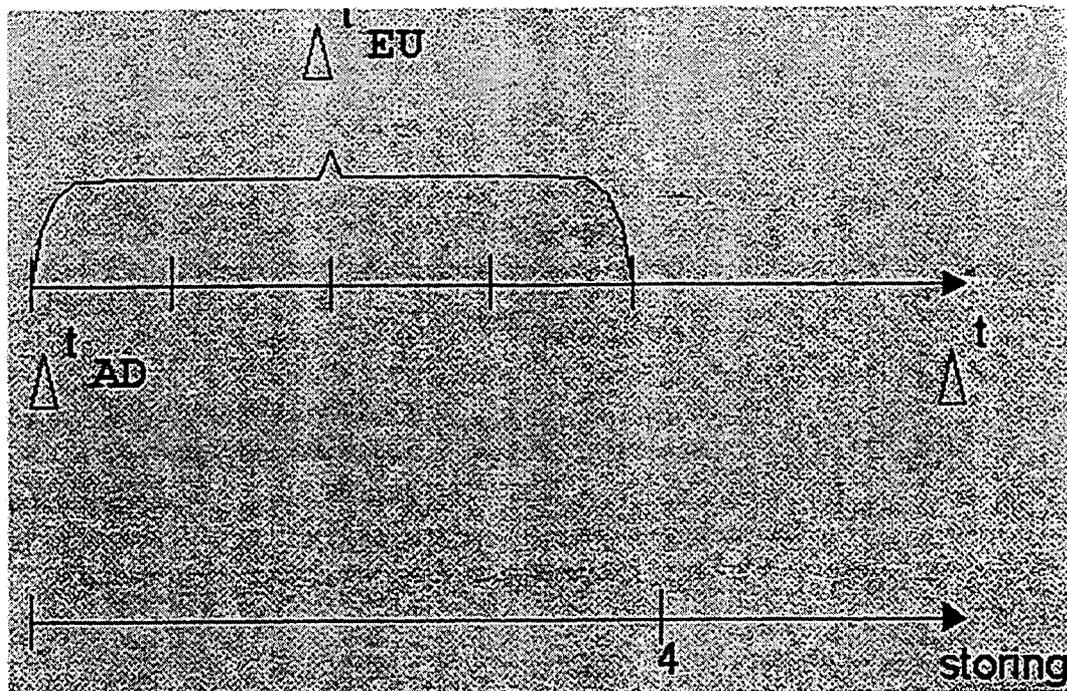


Figure 9.2 Storing frequency

$$\Delta t_{EU} = 3 \text{ hrs} = 180 \text{ min}$$

$$\Delta t_{AD} = 180, 90, 60, 30, 15, 5 \text{ min. and others}$$

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.).

9.3 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. 244)
- Oxygen (p. 244)
- Phytoplankton (p. 245)
- Sediment (p. 245)
- Zooplankton (p. 245)
- Benthic vegetation (p. 245)
- Detrius (p. 246)
- General model parameters (p. 246)

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

9.3.1 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. 243).

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

9.3.2 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. 243).

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



9.3.3 *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. 243).

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

9.3.4 *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. 243).

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

9.3.5 *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. 243).

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

9.3.6 *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.

84



For general information on the EU Parameters, see EU Property Pages (p. 243).

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

9.3.7 *Detritus*

This property page offers 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. 243).

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

9.3.8 *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. 241).

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



Sediment Transport Parameter Editor

86



10 SEDIMENT TRANSPORT PARAMETER EDITOR

10.1 Introduction

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. 251)
- Transport model (p. 252)
- Calibration factors (p. 257)
- Data for graded ST (p. 258)
- Preset distribution of sediment in nodes (p. 260)
- Passive branches (p. 260)
- Initial dune dimensions (p. 261)

10.1.1 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.

10.1.2 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)

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.



10.2 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 m and 2500 m, where the local grain diameter varies linearly between 1.2 and 1.5 mm. and between 2500 m and 4400 m where the grain diameter varies linearly between 1.5 and 1.1 mm. At the same chainages, the standard deviation varies linearly between 1.2, 1.2 and 1.0.

	River Name	Chainage	Grain diam.	St. deviation
1	RIVER1	1000.00000	1.200000	1.200000
2	RIVER1	2500.00000	1.500000	1.200000
3	RIVER1	4400.00000	1.100000	1.000000

Figure 10.1 Example of implementation of local grain diameter.



10.3 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 appropriate transport model and adjust the transport parameters if required.

The screenshot shows a software window titled "ST-River1.ST11" with several tabs: "Data for Graded S...", "Preset Distribution of Sediment in Nodes", "Passive Branches", "Sediment Grain Diameter", "Transport Model", and "Calibration Factors". The "Transport Model" tab is active.

Model type

- Total Load (Acker and White)
- Bed Load and/or Suspended Load
 - Bed Load (Engelund and Fredsoe)
 - Suspended Load (Van Rijn)

Model Parameters

Spec. Gravity	2.65
Kin Viscosity	1 $\times 10^{-6}$
Beta	0.85
Theta Critical	0.056
Gamma	
Acker White	BD35

Calculation of

- Bottom Level
 - dH/dZ: Back water
 - PSI: 0.9
 - F1: 0.9
 - Fac: 1.5
 - Porosity: 0.35
- Bed Shear Stress
 - Method (M): Manning (M)
 - Minimum: 10
 - Maximum: 100
 - Omega: 1

Storage

- Bed / Suspended load
- Total sediment volumes in each grid point
- Graded sediment volumes in each grid point

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 Fredsøe' 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.

10.3.1 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 m²/s. That is, if a value of 0.000001 m²/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 is only applicable for Engelund-Fredsøe and the van Rijn models.

- Total sediment volumes in each grid point



Feature not implemented in the computational kernel of MIKE 11 2000.

- Graded sediment volumes in each grid point

Feature not implemented in the computational kernel of MIKE 11 2000.

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.

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 define which shear stress / resistance type formulation is used throughout the simulation (Manning's M, Manning's n or Chezy).

The feature is not implemented in the computational kernel of MIKE 11 2000, so modifying this selection box will have no effect on the simulation results.

91

**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.

10.3.2 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 10.8.

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.

92



Smart - Jaeggi Factors

Dimensionless sed. transport:

$$a1 * \left[\left(\frac{D90}{D30} \right)^{a2} * |^{a3} * C^{a4} * \text{theta}^{a5} * (a6 * \text{theta}^{a7} - \text{thetacr})^{a8} \right]$$

Coef. 1 (a1)	<input type="text" value="4"/>	Coef. 2 (a6)	<input type="text" value="1"/>
Exp. 1 (a2)	<input type="text" value="0.2"/>	Exp. 4 (a5)	<input type="text" value="0.5"/>
Exp. 2 (a3)	<input type="text" value="0.6"/>	Exp. 5 (a7)	<input type="text" value="1"/>
Exp. 3 (a4)	<input type="text" value="1"/>	Exp. 6 (a8)	<input type="text" value="1"/>
Uniformity of sediment (D90/D30)	<input type="text" value="1.34"/>		
Angle of repose (degrees)	<input type="text" value="33"/>		
Slope corr. form	<input checked="" type="checkbox" value="Bottomlevel"/>		

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.

10.3.3 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 ASCII 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.

10.4 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 m.



ST-River1.ST11

Data for Graded ST | Present Distribution of Sediment in Nodes | Passive Branches

Sediment Grain Diameter | Transport Model | Calibration Factors

Global Data

Factor 1:

Factor 2:

	River Name	Channelage	Factor1	Factor2
1	RIVER1	1000.0000	1.500000	0.750000
2	RIVER1	4000.0000	1.200000	0.800000

Figure 10.4 Calibration factors dialog

10.5 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.

95



The effects of shielding can also be included by setting a check mark in the 'Shielding of particles' checkbox.

The percentage contribution and transport rate of each fraction can be stored in the 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.

ST-River1 ST11

Sediment Grain Diameter Transport Model Calibration Factors

Data for Graded ST Preset Distribution of Sediment in Nodes Passive Branches

Global Data

Min. depth of active layer:

Init. depth of passive layer:

Shielding of particles

Save sed. transport each fraction

Save fraction values: Active layer Passive layer

	Fraction Number	Fraction Diameter	Frac. Value Active Layer	Frac. Value Passive Layer	Global	River Name	Change
1	1	1.000000	25.000000	10.000000	<input checked="" type="checkbox"/>		
2	2	1.500000	30.000000	25.000000	<input checked="" type="checkbox"/>		
3	3	2.300000	25.000000	30.000000	<input checked="" type="checkbox"/>		
4	4	3.000000	20.000000	35.000000	<input checked="" type="checkbox"/>		

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

96



10.6 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:

$$Q_m^{n+1} = \frac{K_m Q_m^n}{\sum_{\substack{\text{downstream} \\ \text{branches}}} K_i Q_i^n} \sum_{\substack{\text{upstream} \\ \text{branches}}} Q_j^{n+1} \quad (10.1)$$

Where

Q_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.

10.7 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.



ST-River1.ST11

Sediment Grain Diameter Transport Model Calibration Factors

Data for Graded ST Preset Distribution of Sediment in Nodes Passive Branches

	River Name	UpStr. Chainage	DownStr. Chainage
1	RIVER1	6000.000000	10000.000000
2	RIVER2	2000.000000	3000.000000

Save result in passive branches

Figure 10.6 Passive branched property page.

10.8 Initial dune dimensions

When selecting the Engelund-Fredsoe transport model the dune height and length are computed when calculation of bottom shear stress is included. The dune dimensions can be specified as applicable globally and locally. If dune dimensions are specified for local application, these values will be used instead of any globally specified values.

Figure 10.7 shows an example where the global dune height has been set to 0.25 m, and the global dune length has been set to 12.50 m. These values will be used in the entire river network, except in the reach 'RIVER1', between chainage 5.000 km and 10.000 km, where the dune height varies linearly between 0.25 m and 0.40 m.



ST-River1.ST11

Sediment Grain Diameter | Transport Model | Calibration Factors | Data for Graded ST

Preset Distribution of Sediment at Nodes | Passive Branches | Initial Dune Dimensions

Global Values

Height: 0.25

Length: 125

River Name	Channelage	Height	Length
RIVER1	5000.0000	0.250000	12.500000
RIVER1	10000.000	0.400000	12.500000

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.

99



Flood Forecasting Editor

100



11 FLOOD FORECASTING EDITOR

11.1 Introduction

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.

11.2 Basic definitions

11.2.1 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 11.2.2, Simulation Mode. The length of the Forecast Period is always specified in the Forecast Menu, see section 11.3.1

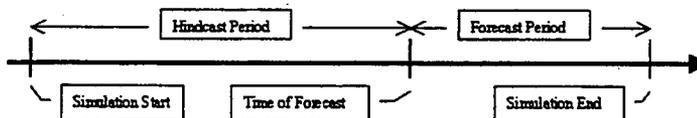


Figure 11.1 Definition of ToF

101



11.2.2 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.

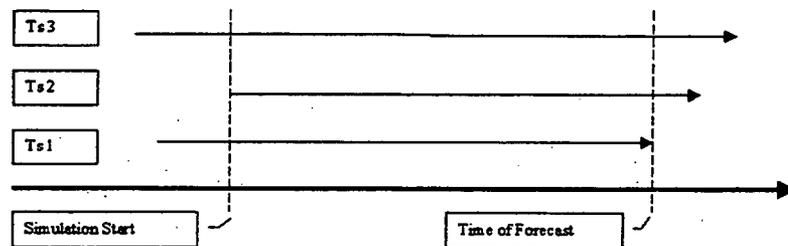


Figure 11.2 Definition of Hindcast Period and ToF

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.

102

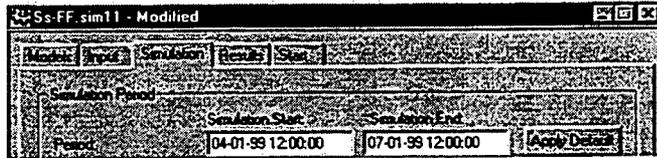


Figure 11.3 Definition of Hindcast period in historical mode

The forecast period is defined in the Forecast Menu.

11.3 Forecast

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

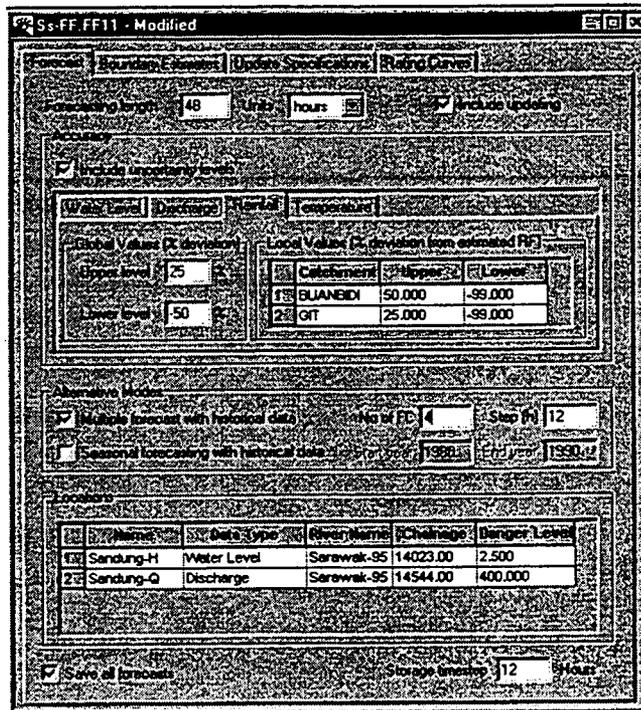


Figure 11.4 Basic Forecast Definitions

103



11.3.1 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**

11.3.2 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 11.5.

11.3.3 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 11.4.4.

11.3.4 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 11.2.2.

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.

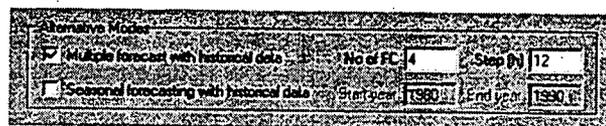


Figure 11.5 Selection of Historical Mode

104



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).

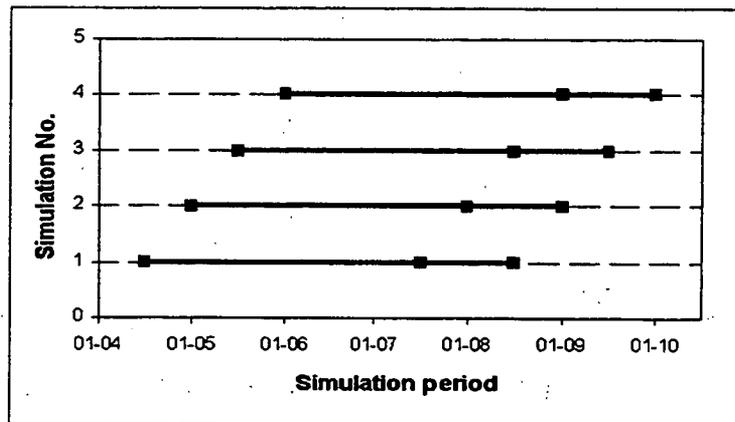


Figure 11.6 Multiple simulations in Historical Mode

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 11.2.2, 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

11.3.5 Location of forecast stations

Forecast points are specified as shown in Figure 11.7 below

No.	Name	Data Type	River Name	Chainage	Danger Level
18	Sending-H	Water Level	Serawak-95	14023.00	2,500
23	Sending-Q	Discharge	Serawak-95	14544.00	400,000

Save all forecasts Storage Interval: 12 Hours

Figure 11.7 Location of Forecast Points

106



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.

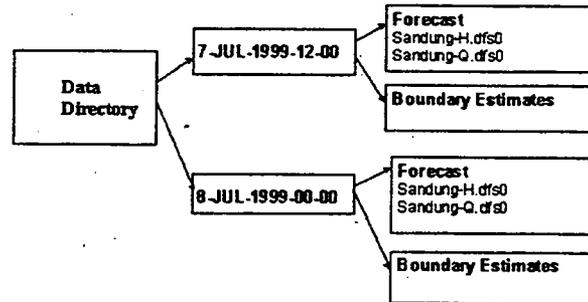


Figure 11.8 Forecast data directory structure

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.

11.4 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 11.4.3.

Figure 11.9 shows the Boundary Estimates menu.

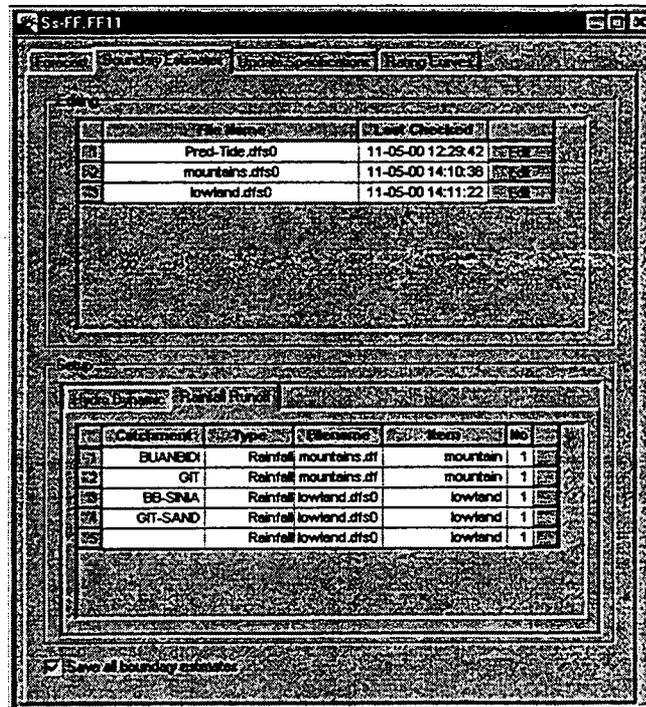


Figure 11.9 Boundary Estimates

11.4.1 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.

11.4.2 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.

11.4.3 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 Figures 11.10 - 11.14.

Omit a boundary condition.

A boundary condition timeseries 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

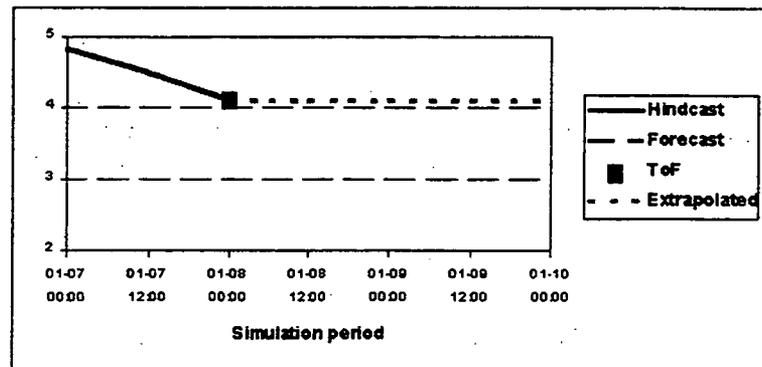


Figure 11.10 Extrapolation from value at ToF

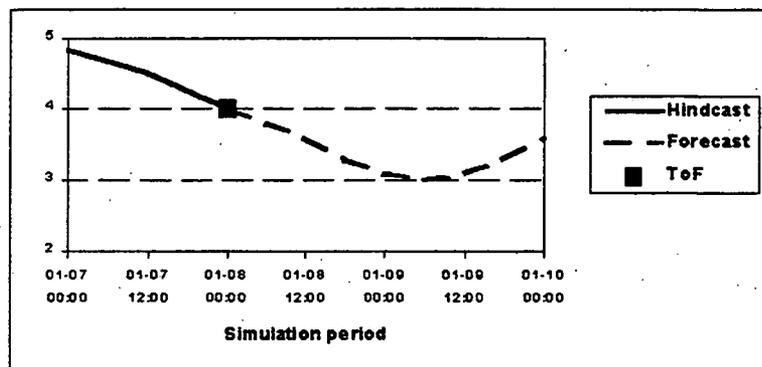


Figure 11.11 Estimated boundary conditions as specified.

109

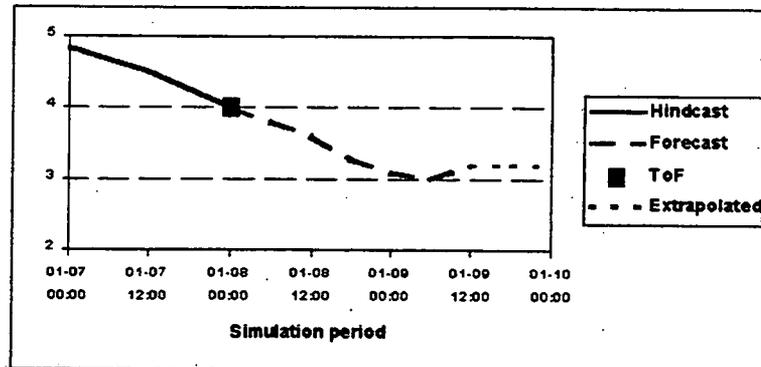


Figure 11.12 Extrapolation of Estimated boundary conditions

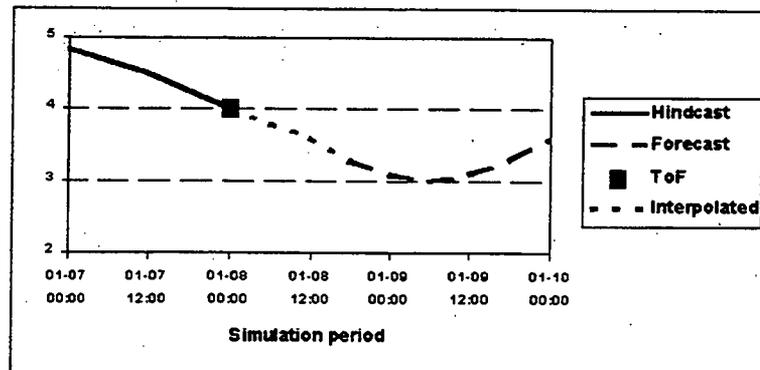


Figure 11.13 Interpolation of Estimated boundary condition

110

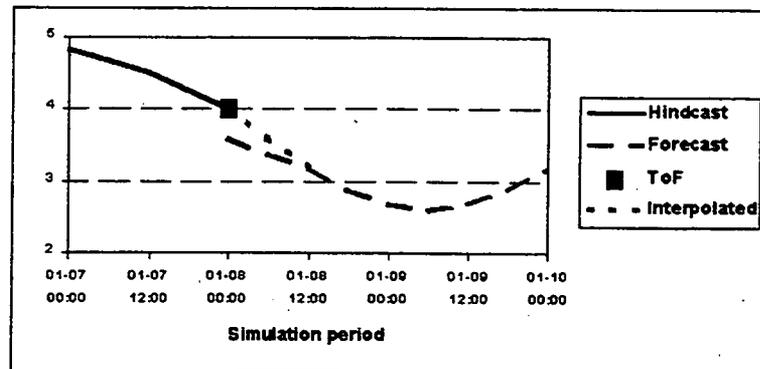


Figure 11.14 Discontinuity at ToF

11.4.4 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 11.3.5 and Figure 11.15 below.

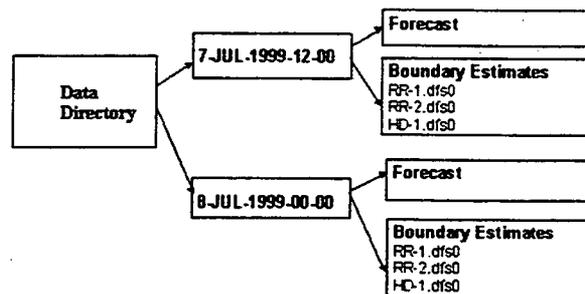


Figure 11.15 Estimated boundary directory structure

11.5 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.



Station	River Name	Chainage	Data Type	File Name	Iterations	Frequency
1	Kiri-97	264650.00	Water Level	C:\Projects\S	1	Iteratio
2	steal-97	886870.00	Water Level	C:\Projects\S	1	Iteratio
3	Pedil-97	694295.00	Water Level	C:\Projects\S	1	Iteratio
4	noten-97	590403.00	Water Level	C:\Projects\S	1	Iteratio
5	Selang-97	795919.00	Water Level	C:\Projects\S	1	Iteratio

Figure 11.16 Update Specification

11.5.1 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 $q^{n+1/2}$ is calculated and applied as the updating discharge.

No. of iterations

If a river branch includes a number of update points the specified No. of iterations should be equal to or larger than this number. For large rivers with few update points it may increase the update efficiency to use an even larger number of iterations. Different numbers of iterations should be tested before operational forecasting is initiated. A larger number will increase the accuracy but also increase the required calculation time.

Frequency

Frequency of updating, i.e. the number of MIKE 11 HD time steps between data observations in the time series used for updating.

11.5.2 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

11.5.3 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

113



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

114



Batch Simulation Editor

115



12 BATCH SIMULATION EDITOR

12.1 Introduction

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.

12.2 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, timestep, 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 filenames have been defined by the user in the Batch Simulation Editor, will the definitions in the Base Sim11-file be modified.

Filename and path to the base Sim11 file must be defined in the 'Base Simulation File' field (Use the '...' button to browse for the Base Sim11 file on your computer).

Define parameters to adjust in batch simulation

The user must define the number of simulations to be performed in the batch simulation by specifying a number in the 'Number of simulations' field. 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 treeview 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.

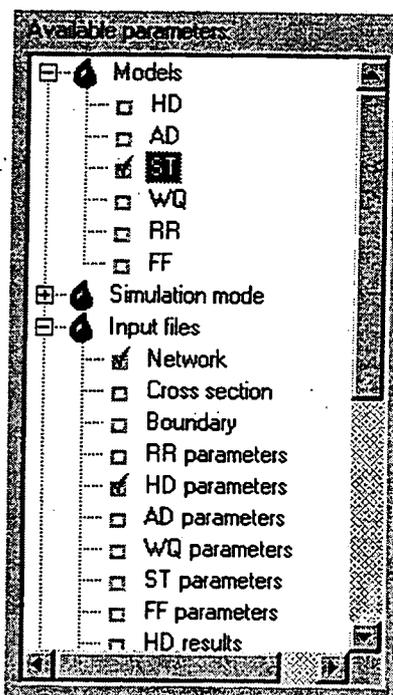


Figure 12.1 Treeview from the Batch Simulation Editor dialog for selecting batch simulation parameters

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.

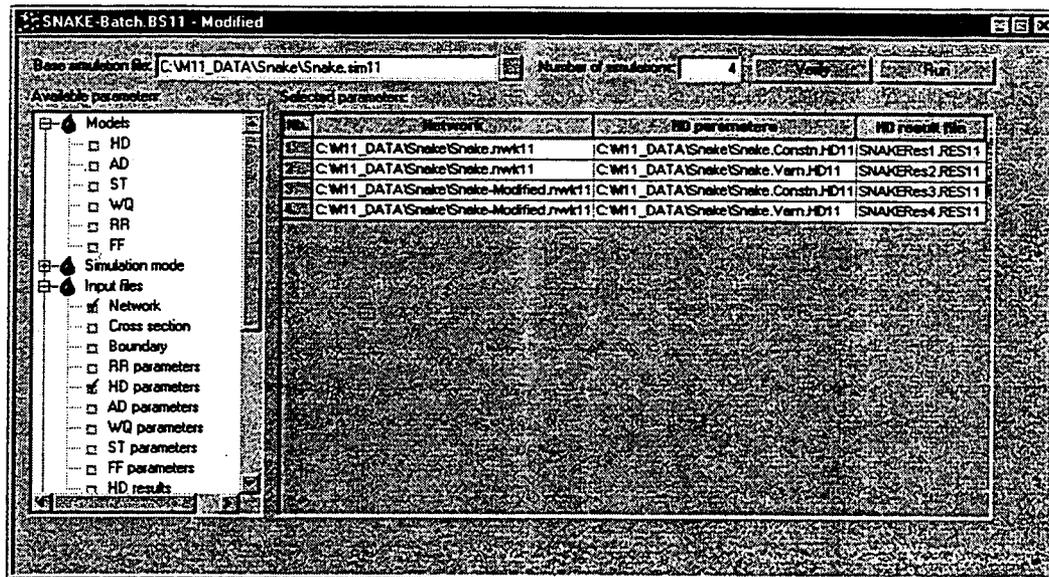


Figure 12.2 Example of Batch Simulation setup.



APPENDIX A
Flow resistance and vegetation

120



A.1 FLOW RESISTANCE AND VEGETATION

A.1.1 Introduction

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 vegetations influence 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.1.2 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 Figure A.1.1

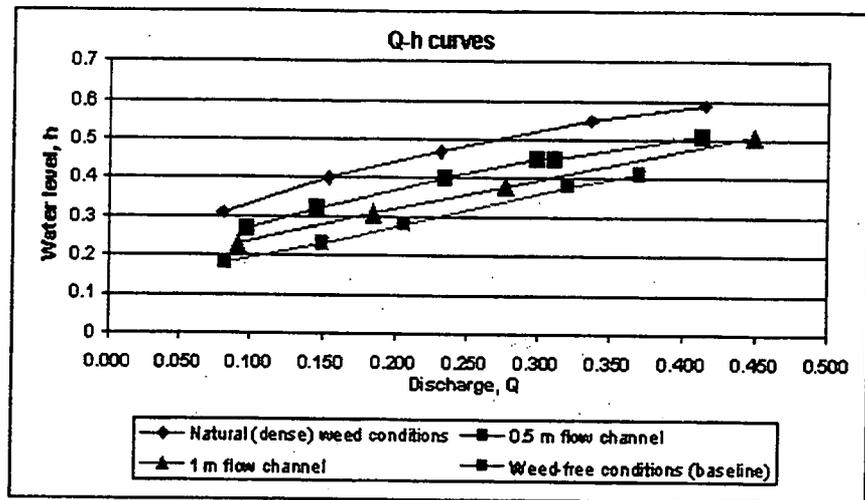


Figure A.1.1 Q-h curves determined for varying flow channel width

Calculated Manning numbers (Manning's M) are presented in Figure A.1.2 as a function of Discharge, Q . From this figure, it can be seen, that the flow resistance in a weed-filled stream can be up to 4 times larger compared to weed-free conditions in the same stream.

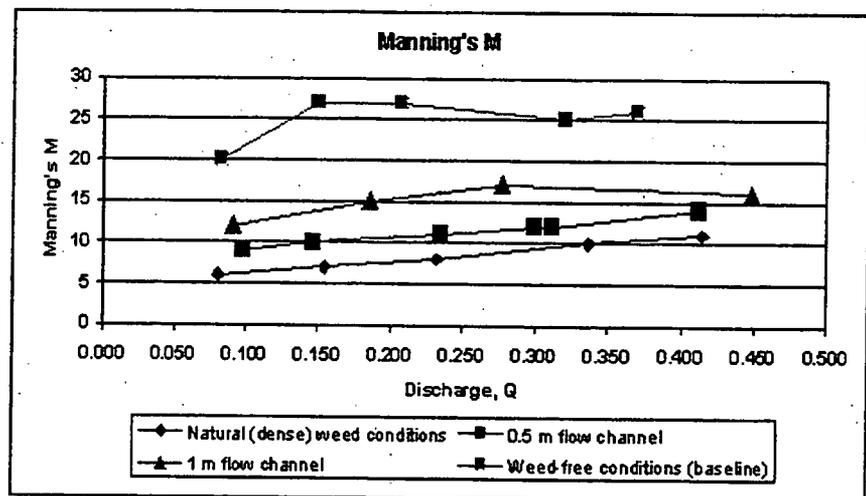


Figure A.1.2 Mannings M calculated as a function of Discharge, Q

122



A.1.3 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: enkeltblad et pindsvineknop). The experiment included a series of measurements with varying weed density. Figure A.1.3 shows the results from the measurements. Manning's n is plotted against the product; Velocity, V , times the hydraulic radius, R , for two different densities of weed (defined by mass of dry material per area) and a complete weed-free situation. From the results it can be seen, that the flow resistance varies with a factor of 4 to 6 from a weed-free channel to a situation with very dense vegetation (325 g dry material/m²).

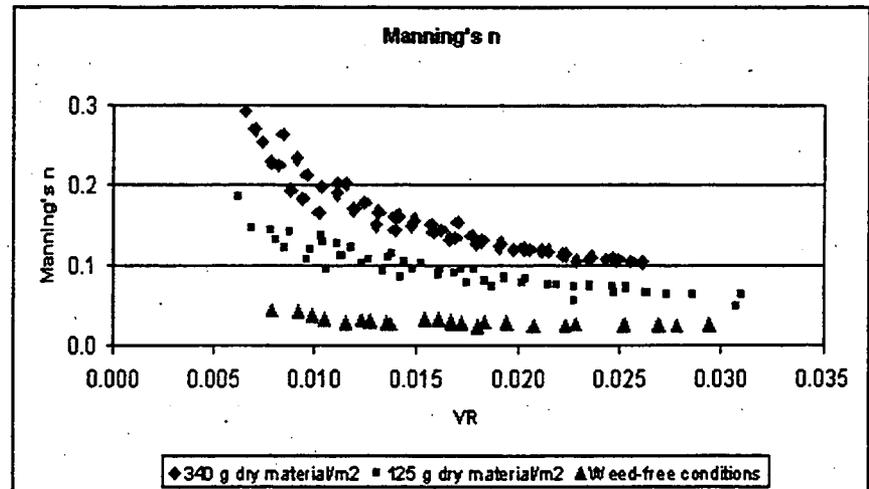


Figure A.1.3 Manning's n vs VR (VR : Velocity times Hydraulic Radius)

Jensen /3/ discusses the possible correlation of flow resistance and hydraulic parameters and presents arguments, stating that the variation in flow resistance can be correlated to the product, VR for a specific weed density by the following equation:

$$n = a \ln(VR) + b \quad (\text{A.1.1})$$

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; Simsted

123 A •



\bar{A} , 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:

$$n = aD_{\eta}^b \quad (\text{A.1.2})$$

where a and b are coefficients as described for equation (A.1.1) and D_{η} is the hydraulic depth calculated from:

$$D_{\eta} = \frac{A}{B} \quad (\text{A.1.3})$$

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.1.4 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 $\text{m}^{1/3}/\text{s}$ at small water depths up to 30 $\text{m}^{1/3}/\text{s}$ for large water depths. Figure A.1.4 shows the calculated Manning numbers as a function of water depth. For comparison expressions of the form (A.1.2) have been fitted to the data.

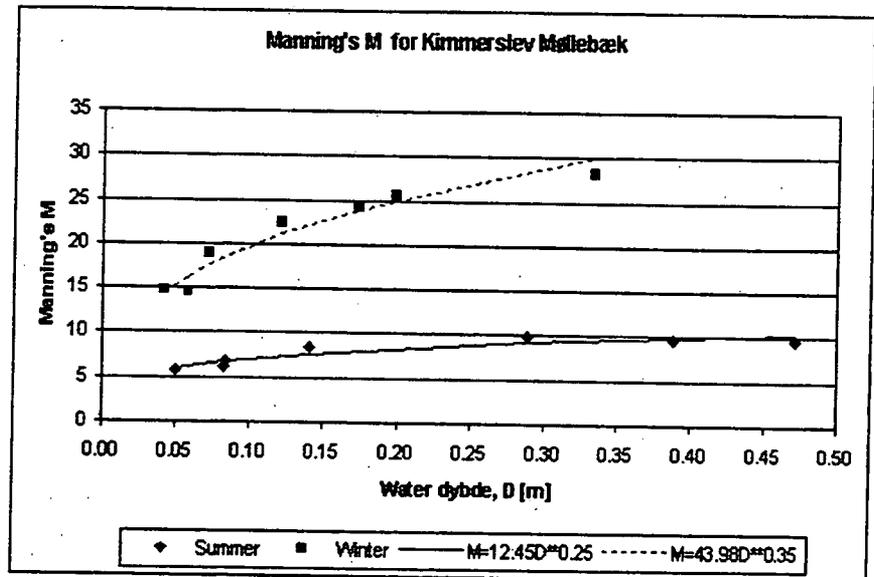


Figure A.1.4 Manning's M for Kimmerslev Møllebæk in summer and winter period. Results calculated with the formulas of the form $M = \alpha D^{\beta}$ are also included.

A.1.5 Experiments in 'ArnÄ'

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. $10 \text{ m}^{1/3}/\text{s}$, increasing to approx. $15 \text{ m}^{1/3}/\text{s}$ in August 1990 as a result of weed cutting. Thereafter Manning's M increases during winter to a value of approx. $25 \text{ m}^{1/3}/\text{s}$. From april it is found, that Manning's M starts to drop and ends at approx. $10 \text{ m}^{1/3}/\text{s}$ in late summer.

These results - an annual variation in Manning's M between approx. $10 \text{ m}^{1/3}/\text{s}$ and $25 \text{ m}^{1/3}/\text{s}$ - are identical to the variations observed in 'Kimmerslev Møllebæk'.



A.1.6 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 Waterresources*, 1996 (In Danish)

- /3/ Jensen, K.R.:

"Undersøgelse af Vandløbsvegetationens Hydrauliske Indflydelse." *Afgangprojekt, AUC, 1992*

"Investigation 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". *Afgangprojekt, AUC, 1990*

"Flow channels in weed-filled streams". *B.Sc. thesis from University of Aalborg, 1990.*