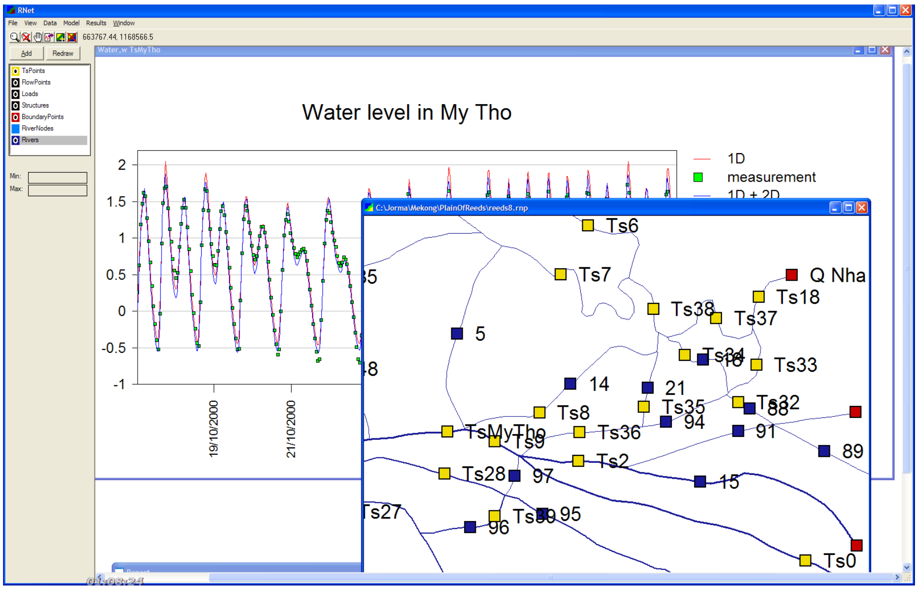
Mekong River Commission

Hybrid 1D/2D/3D model user guide



MRC Information and Knowledge Management Programme

Hydrological, Environmental and Socio-Economic Modelling Tools for the Lower Mekong Basin Impact Assessment/ FINDS

August 2008

SYKE
FINNISH ENVIRONMENT INSTITUTE

Finnish Environment Institute
in association with
EIA Centre of Finland Ltd.

Hydrological, Environmental and Socio-Economic modelling Tools for the Lower Mekong Basin Impact Assessment/ FINDS

Information and Knowledge Management Programme (IKMP)

August 2008

Jorma Koponen, Hannu Lauri, EIA Ltd.

EIA Ltd.
Tekniikantie 21 B
02150 Espoo
Finland
Tel: +358 9 70018680
Fax: +358 9 70018682
www.eia.fi
jorma.koponen@eia.fi

This document can be referred as follows:

MRC/WUP-FIN, 2008. Hybrid 1D/2D/3D model manual. Hydrological, Environmental and Socio-Economic modelling Tools for the Lower Mekong Basin Impact Assessment/ FINDS. Mekong River Commission and Finnish Environment Institute Consultancy Consortium, Vientiane, Lao PDR. 42 pp.

TABLE OF CONTENTS

Table of Contents

ACRONYMS AND ABBREVIATIONS

1D	one dimensional, e.g. river channel network
2D	two-dimensional, e.g. shallow well-mixed lake
3D	three-dimensional, e.g. lake, coastal or floodplain areas with varying properties in the horizontal and vertical directions
EIA	Environmental Impact Assessment
EIA Ltd.	Environmental Impact Assessment Centre of Finland (www.eia.fi)
GIS	Geographical Information System
GUI	Graphical User Interface
HBV	name of the lumped hydrological model
LU	Land Use
MRC	Mekong River Commission (www.mrcmekong.org)
RLGis	River Life GIS, a programme used together with the EIA 3D model
Rnet	EIA 1D model used separately or combined with the EIA 3D model
WQ	Water Quality
WUP-FIN	Lower Mekong modelling Project under Water Utilization Programme of Mekong River Commission (www.eia.fi/wup-fin)

1 OVERVIEW ON THE 1D/2D/3D MODEL CHARACTERISTICS

1.1 MODEL SYSTEMS AND CHARACTERISATIONS IN 1D, 2D AND 3D

Models in general can be divided into 4 categories according to their dimensionality:

0. 0D models don’t describe spatial variability of simulated variables. 0D models can be used to simulate for instance small mixed ponds or stirred reactors
1. 1D models are used typically for river and channel networks. They describe variation of flow, sediment concentration and other variables in the longitudinal direction. In special cases 1D models can be used in vertical direction for instance for vertically stratified lakes.
2. 2D models describe variation of simulated variables in both horizontal dimensions (or is special cases in the vertical and one horizontal dimension). 2D models are typically used for flood wave/ pulse simulation on the floodplain, coastal wetting/ drying and in shallow lakes.
3. 3D models describe the calculation variables in all spatial directions. Typical application areas include lakes, reservoirs, floodplains, rivers, coastal and sea areas where processes are dominantly 3-dimensional or the variable values wary considerably .

The disadvantage of 2D and 3D models is, that description of a large channel network is not practical with them. 1D models are better adapted for that task. On the other hand 1D models for floodplains and other 2D/3D systems don’t capture all important characteristics and tend to become rather complicated in their schematisations. The obvious **solution for model limitations is to use combined 1D/2D/3D model** which utilises the best features of all models.

The **MRCS modelling environment** consists of a number of separate and coupled models. Under the **DSF** (Decision Support Framework) three models are used: **ISIS 1D** hydrodynamic, sediment and salinity model for the Mekong mainstream and main tributaries, **SWAT hydrological** model for the Mekong catchment area for simulating rainfall/ runoff and hydrological processes and **IQQM water resources and allocation** model. Additional tools include **EIA 1D/2D/3D modelling tools** and **JICA MIKE11 1D** model application for the Cambodian floodplains.

ISIS, SWAT, IQQM and WUP-FIN modelling tools are connected off-line. For instance SWAT results are calculated firs providing information on watershed flows into to the Mekong, ISIS calculates after that the consequent flow in the river system and finally Tonle Sap 3D model uses ISIS Tonle Sap River flows as a boundary value. Off-line coupling doesn’t work in areas where there is significant interaction between the different model domains. For such cases one needs a coupled model simulating the interaction between the channel and river network and the floodplain, reservoirs or lakes. The EIA combined 1D/2D/3D model is a system where the coupled flow, sediment and water quality can be simulated. In this document it is called **EIA-123D**.

In the EIA-123D the 1D characteristics are simulated with the **RNet** river network model that solves one-dimensional river flow and water quality in a river network. The solution is based on solving simplified St. Venant equations numerically. The river system is described to the model as a river network constructed from one-dimensional rivers, and river nodes that connect individual rivers to a network. The 2D and 3D characteristics are solved by the **EIA-23D** model. It is a comprehensive hydrodynamic and water quality model. It is described in detail in Model Report and User Manual. The combined model, EIA-123D, can take its boundary values from measurements, hydrological models or basin-wide model systems such as DSF.

The connection algorithm is devised to be flexible in the sense that basically any 1D model with capability to communicate outside itself can be coupled with the floodplain model. The structure of the model system is shown in figure 1.

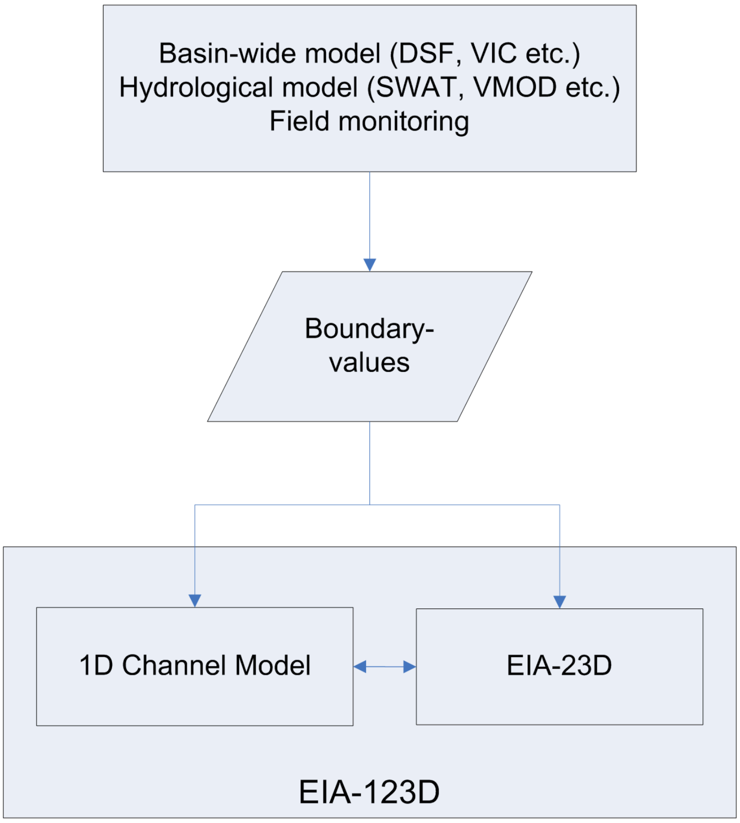


Figure 1. EIA-123D combined model simulation scheme overview.

1.2 CHALLENGES OF THE COMBINED 1D/ 2D/ 3D MODELLING

The challenges specific for the combined 1D/ 2D/ 3D modelling can be divided into data and stability/ computational time related issues. Both can hamper practical model implementation. It is common that combined models run exceedingly slow – even slower than in real-time - that is one day simulation takes more than a day to run! (Green 2007)

The data issues and their solutions in the EIA-123D modelling system can be summarised as:

- Setting up the 1D sub-model takes lot of resources for complex system such as Plain of Reeds or the whole Vietnamese delta → (i) import required information from existing 1D models or couple existing 1D model with the floodplain model, (ii) use automated and graphical user interface assisted model input data processing.

- Setting up floodplain model can be also a resource consuming affair in large and complex areas → automated and graphical user interface assisted use of GIS and control structure data for model input data processing.
- Data is lacking especially on control structures → As model grids, river networks and data structures map directly to map coordinates and model floodplain processes translate directly to real world phenomena, remote sensing data can be used to clarify control structures directly or indirectly through flooding patterns.

Stability and computational time requirements are tied together. Often stability problems can be overcome by diminishing model time step, but this comes on the expense of computational time. Stability issues connected with the model coupling and their solutions are discussed in chapter 5.1. Similarly specific issues in the sub-models have to be dealt with for the system to be able to run with reasonable processing times. As an example, small water depths and handling of flooding and wetting in both the channel and floodplain models requires specific attention.

Other issues connected with the combined model running include data exchange between the models, synchronization of sub-model execution, maintenance of mass-balances, limiting of numerical errors arising from use of very large or small parameter or variable values and software development related problems such as module linking and error handling in multi-language programming (in case of the combined model Fortran – 2D/3D model -, C – interactive animation window during simulation - and C++ - RNet 1D model).

2 1D RNET MODEL EQUATIONS AND SOLUTION PRINCIPLES

2.1 MODEL EQUATIONS

The model is based on one-dimensional St.Venant equations, written for the flow (Q) and flow cross section (A) in metric units. For friction computation the Manning's formulation is used.

$$\frac{\partial Q}{\partial x} + \frac{\partial A}{\partial t} - q_L = 0 \qquad \text{(mass)}$$

$$\frac{\partial Q}{\partial t} + \frac{\partial (Q^2 / A)}{\partial x} + gA\frac{\partial h}{\partial x} = gA\left(S_0 - S_f\right) \qquad \text{(momentum)}$$

$$S_f = \frac{n^2 Q|Q|}{A^2 R^{4/3}} \qquad \text{(friction)}$$

Q = flow, m3/s

A = flow cross section, m2

q_L = side flow per unit length, m2/s

g = gravitational constant, 9.81 m2/s

h = water depth, m

S_0= bottom slope

S_f = friction term

R = A/P

P = wet perimeter, m

n = Manning’s friction factor, about ~ 0.03 (unitles)

If river cross section is rectangular, following formulations can be used.

$$\sigma = b$$
$$A = bh$$
$$P = b + 2h$$
$$h = \frac{A}{b}$$
$$R = \frac{A}{P} = \frac{bh}{b + 2h}$$

σ = river width at the water surface

b = river bottom width

τ = river bank slope

For trapezoidal cross section following formulations can be used:

$$\sigma = b + 2\tau h$$
$$A = h\left(b + \tau h\right)$$
$$P = b + 2h\sqrt{1 + \tau^2}$$
$$h = \sqrt{\left(\frac{b}{2\tau}\right)^2 + \frac{A}{\tau}} - \frac{b}{2\tau}$$
$$R = \frac{A}{P} = \frac{h(b + 2\tau h)}{b + 2h\sqrt{1 + \tau^2}}$$

Kinematic approximation is : $Q = \frac{A^{5/3}\sqrt{S_0}}{nP^{2/3}}$

If cross section is rectangular, A=bh, we obtain

$$\frac{\partial}{\partial x}\left(g\frac{A^2}{2b}\right) = gA\frac{\partial h}{\partial x}$$

The momentum conservation equation can be then written in the following form

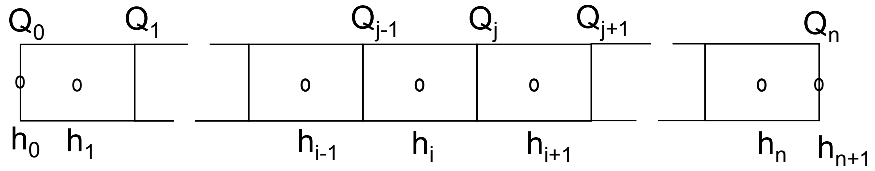
$$\frac{\partial Q}{\partial t} + \frac{\partial}{\partial x}\left(\frac{Q^2}{A} + g\frac{A^2}{2b}\right) = gA\left(S_0 - S_f\right)$$

2.2 SOLUTION METHOD

Let's define the grid box bottom level z(i) in the same point as water level h. The we can write

$$S^{0,i+1/2} = - dz/dx = -(z(i+1)-z(i))/\Delta x$$

The model grid is staggered as follows



Implicit discretation

$$\alpha_i \frac{h_i - h_i^0}{\Delta t} + \frac{Q_{i-1/2} - Q_{i+1/2}}{\Delta x} = q_i^L$$

$$\frac{1}{gA^0_{i+1/2}} \left(\frac{Q_{i+1/2} - Q^0_{i+1/2}}{\Delta t} \right) + \frac{h_{i+1} - h_i}{\Delta x} = - \frac{z_{i+1} - z_i}{\Delta x} - S^0_{f,i+1/2}$$

$$\alpha_i = \left(\frac{\partial A}{\partial h} \right)_i$$

By rearranging and assigning $\lambda=\Delta t/\Delta x$, and $q^s=\lambda q^L$

$$- \lambda Q_{i-1/2} + \alpha_i h_i + \lambda Q_{i+1/2} = \alpha_i^0 h_i^0 + \lambda q_i^S$$

$$- \lambda h_i + \frac{1}{gA^0_{i+1/2}} Q_{i+1/2} + \lambda h_{i+1} = - \lambda \left(z_{i+1} - z_i \right) - \Delta t S^0_{f,i+1/2} + \frac{1}{gA^0_{i+1/2}} Q^0_{i+1/2}$$

This can be written in tridiagonal matrix form:

1		2		3		4		..		n-1		n			
h_1	Q_1	h_2	Q_2	h_3	Q_3	h_4	Q_4			h_{n-1}	Q_{n-1}	h_n	Q_n	RHS	
B_{11}	C_{11}													RHS_{11}	Mass
A_{12}	B_{12}	C_{12}												RHS_{12}	Mom.
	A_{21}	B_{21}	C_{21}											RHS_{21}	Mass
		A_{22}	B_{22}	C_{22}										RHS_{22}	Mom.
			A_{31}	B_{31}	C_{31}									RHS_{31}	Mass
				A_{32}	B_{31}	C_{31}								RHS_{32}	Mom.
					A_{41}	B_{41}	C_{41}							RHS_{41}	Mass
						A_{42}	B_{42}	C_{42}						RHS_{42}	Mom.
							..	..	..					…	…
									A_{n-1}	B_{n-1}	C_{n-1}				Mass
										A_{n-1}	B_{n-1}	C_{n-1}			Mom.
											A_{n1}	B_{n1}	C_{n1}	RHS_{n1}	Mass
												A_{n2}	B_{n2}	RHS_{n2}	Mom.

To solve this Thomas algorithm can be used. Nonlinearity is taken into account by iterating the solution few (=3) times, so that S_f and A are updated from Q and h, after which the computation is done again.

2.3 BOUNDARY VALUES

Boundary conditions, left border:

Time dependen flow $Q_{in}(t)$:

$RHS_{11} = RHS_{11} - \Delta t/\Delta x * Q_{in}(t)$

(first equation is mass conservation)

Water level:

Add momentum conservation equation as first equation.

boundary grid box size is $\Delta x/2$

z(0) is bottom level at the boundary.

b(0) is bottom width at the boundary

A is computed using the boundary h-value

$$\frac{2}{gA_b^0}Q_b + \lambda h_1 = -\lambda\left(z_1 - z_0\right) - 2\Delta t S_{f,b}^0 + \frac{2}{gA_b^0}Q_b^0 + \lambda h_b$$

The equation matrix start:

0	1		2		3		..			
Q_0	h_1	Q_1	h_2	Q_2	h_3	Q_3		RHS		
B_{02}	C_{02}							RHS_{02}		Mom.
A_{11}	B_{11}	C_{11}						RHS_{11}		Mass
	A_{12}	B_{12}	C_{12}					RHS_{12}		Mom.
		A_{21}	B_{21}	C_{21}				RHS_{21}		Mass
			A_{22}	B_{22}	C_{22}			RHS_{22}		Mom.
				A_{31}	B_{31}	C_{31}		RHS_{31}		Mass

$RHS_{02} = 2*RHS_{02}+\Delta t/\Delta x*(z_1-z_0+h_{in}(t))$

$B_{02}= 2*B_{02}$

Boundary conditions, right border

Water level:

Last equation is momemtum conservation, grid box size is Δx/2

$$-\lambda h_i + \frac{2}{gA_R^0}Q_{i+1/2} = -\lambda\left(z_R - z_i\right) - 2\Delta t S_{f,R}^0 + \frac{2}{gA_R^0}Q_{i+1/2}^0 - \lambda h_R$$

Time-dependent water level $h_{out}(t)$ at the right boundary

$RHS_{n2} = 2*RHS_{n2}+\Delta t/\Delta x*(z_R-z_{n2}-h_{out}(t)))$

$B_{n2}= 2*B_{n2}$

Flow at right boundary:

Last equation is mass conservation

$RHS_{n1} = RHS_{n1}-\Delta t/\Delta x*Q_{out}(t)$

If flow depends on water level, this condition is used. Flow is computed using the previous time step water level.

Internal boundaries at nodes

A: side flow connection

Side river and main river: the side river boundary condition is water level, main river side flow is set using the side river flow

pj.qs(n)=-sj.q(0);

sj.h(0)=pj.h(n);

B: river node using water levels

All rivers either beging or end at a river node

For all rivers boundary condition is water level

The water level at node point is computed be supposing the flow sum at the node point to be equal to zero (by conservation of mass).

Equations:

$$\sum_j Q_j = 0$$

$$\frac{\partial Q_j}{\partial t} + gA_j \frac{\partial h_j}{\partial x} = gA_j(S_{0,j} - S_{f,j})$$

Right boundary flow is incoming and positive. From the momentum equation below we obtain (grid box size Δx/2):

$$Q_{R,j} = Q_{i1-1,j} = -\frac{gA_{i1}^0\lambda}{2}\left(z_X - z_{i1-1} + h_X - h_{i1-1}\right) - \Delta tgA_{i1}^0S_{f,i1}^0 + Q_{i1-1}^0$$

For left boundary we can write

$$Q_{L,j} = Q_{0,j} = -\frac{gA_0^0\lambda_1}{2}\left(z_1 - z_X + h_1 - h_X\right) - \Delta tgA_0^0S_{f,0}^0 + Q_0^0$$

In the crossing point

$$\sum_j Q_{R,j} - \sum_k Q_{L,k} = 0$$

Now we can solve h_X

=>

$$h_X = \frac{\sum_j \left[\left(-\frac{gA_{i1}^0\lambda_{i1}}{2}\left(z_X - z_{i1-1} - h_{i1-1}\right) - \Delta tgA_{i1}^0S_{f,i1}^0 + Q_{i1-1}\right)_{R.j} + \left(\frac{gA_0^0\lambda_0}{2}\left(z_1 - z_X + h_1\right) + \Delta tgA_0^0S_{f,0}^0 - Q_1\right)_{L,j}\right]}{\sum_j \left(\frac{gA_{i1}^0\lambda_{i1}}{2}_{R,j} + \frac{gA_0^0\lambda_0}{2}_{L,j}\right)}$$

where $\lambda_i=\Delta t/\Delta x_i$.

2.4 DETERMINATION OF CROSS-SECTION AREAS, WATER DEPTHS AND WETTED PERIMETERS

Piecewise linear approximation

Piecewise linear approximation is obtained from cross section data. Cross sections are defined with (x,z) , that is distance from river bank and corresponding bed elevation. Interface VIV-software obtains channel properties for any water level and model grid cell with the following procedure:

1. derives linear approximation of the bottom contour based on the cross section data
2. divides section into 10 layers (see figure below)
3. calculates cross section area and wetted perimeter for each water level at the layer boundaries
4. calculates average bottom elevation for the cross sections
5. writes elevations, areas and wetted perimeters to a rnd-file for each cross section.

The 1D river model reads the rnd-file values and interpolates them for each calculated water level. In the horizontal direction values are interpolated linearly for model grid cells in between cross sections.

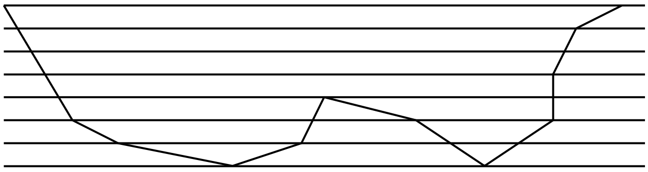


Figure 2. Method of piecewise cross sections: linear approximation of bottom contour and division of cross-section into layers..

Trapezoidal approximation

Trapezoidal approximation works well for many smaller rivers. The approximation is however not appropriate for places where relatively narrow dry season river channel expands to the surrounding floodplain during high water season. Because of this trapezoidal approximation is not used in the Mekong Delta applications.

The river cross section is a modified trapezoid (1).

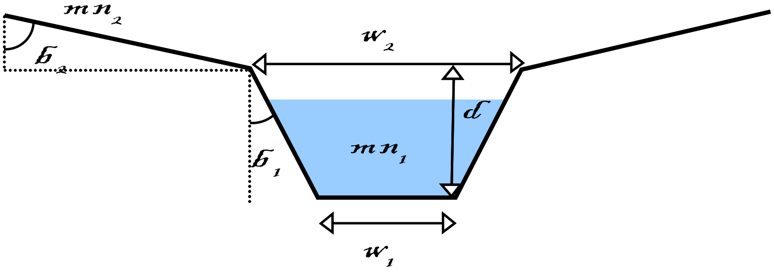


Figure 3. River cross section.

River parameters (figure 3):

d = river bank height, m
w_1 = river bottom width, m
w_2 = river width at bank height = $w_1 + 2d \tan(b_1)$, m
b_1 = river bank slope
b_2 = floodplain slope
mn_1 = Manning’s friction parameter for river channel
mn_2 = Manning’s friction parameter for floodplain

River cross section area is calculated from the water height:

$$
\begin{array}{ll}
A = y\,(w_1 + y \tan(b_1)), & y < d \qquad (39) \\
A = d\,(w_1 + d \tan(b_1)) + (y - d)\,((w_2 + (y-d) \tan(b_2)), & y > d
\end{array}
$$

y = river depth, m

At the moment the definition of the river cross sections is simplified. Only the w1 and b1 parameters can be defined.

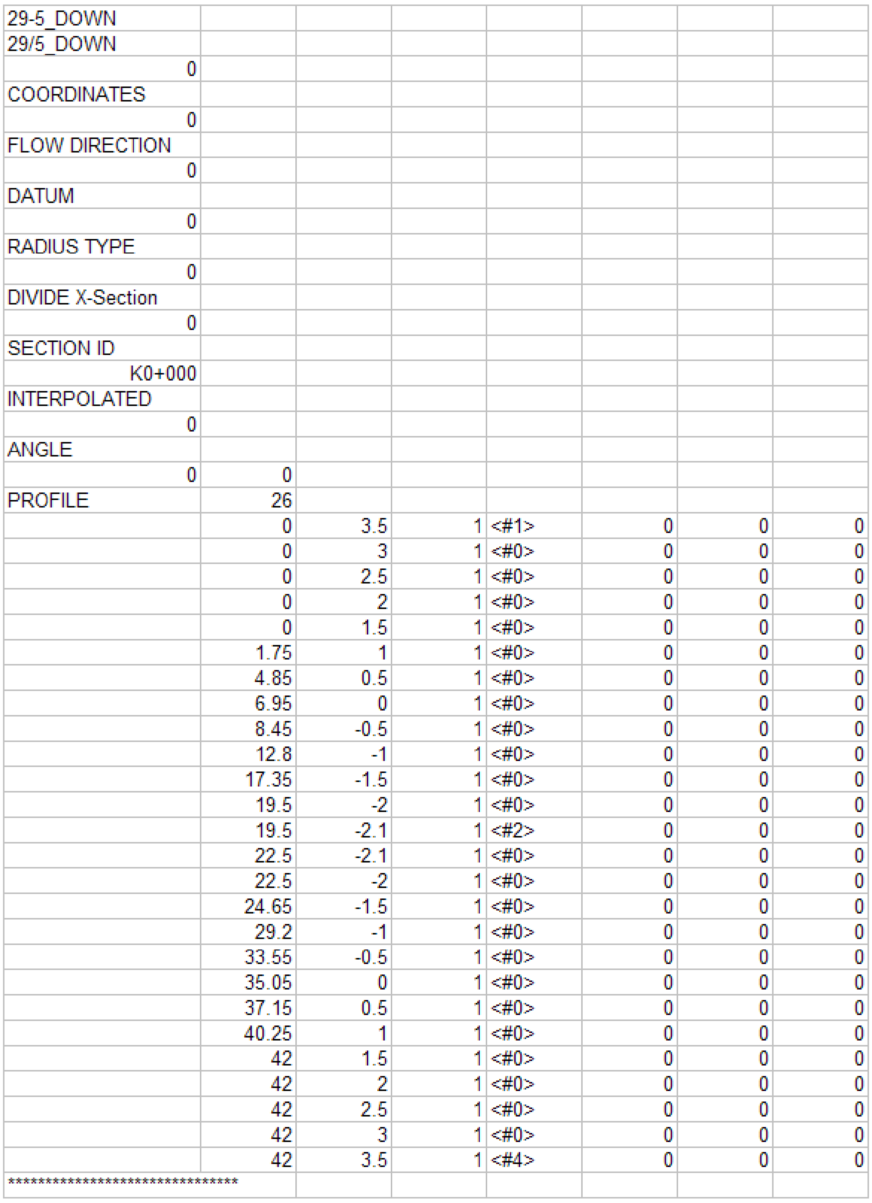
3 SETTING-UP THE 1D RNET MODEL

3.1 SETTING UP MODEL SOFTWARE

If older versions have been installed, clean-up your machine of older versions by (i) selecting *EIAModels/Remove EIAviv* in Windows Programs-menu, or (ii) removing through Windows *Start/ Settings/ Control Panel / Add or Remove Programs*. Also, if possible, application directories should be cleaned of exe- (l.exe), ip- and frm-files as they can interfere with software functioning.

Run VivSetup.exe for software installation. By default this sets up user software and model executables in *c:\EIModels\VIV* – folder.

3.2 IMPORTING MODEL SCHEMATISATION



29-5_DOWN							
29/5_DOWN							
0							
COORDINATES							
0							
FLOW DIRECTION							
0							
DATUM							
0							
RADIUS TYPE							
0							
DIVIDE X-Section							
0							
SECTION ID							
K0+000							
INTERPOLATED							
0							
ANGLE							
0	0						
PROFILE	26						
	0	3.5	1	<#1>	0	0	0
	0	3	1	<#0>	0	0	0
	0	2.5	1	<#0>	0	0	0
	0	2	1	<#0>	0	0	0
	0	1.5	1	<#0>	0	0	0
	1.75	1	1	<#0>	0	0	0
	4.85	0.5	1	<#0>	0	0	0
	6.95	0	1	<#0>	0	0	0
	8.45	-0.5	1	<#0>	0	0	0
	12.8	-1	1	<#0>	0	0	0
	17.35	-1.5	1	<#0>	0	0	0
	19.5	-2	1	<#0>	0	0	0
	19.5	-2.1	1	<#2>	0	0	0
	22.5	-2.1	1	<#0>	0	0	0
	22.5	-2	1	<#0>	0	0	0
	24.65	-1.5	1	<#0>	0	0	0
	29.2	-1	1	<#0>	0	0	0
	33.55	-0.5	1	<#0>	0	0	0
	35.05	0	1	<#0>	0	0	0
	37.15	0.5	1	<#0>	0	0	0
	40.25	1	1	<#0>	0	0	0
	42	1.5	1	<#0>	0	0	0
	42	2	1	<#0>	0	0	0
	42	2.5	1	<#0>	0	0	0
	42	3	1	<#0>	0	0	0
	42	3.5	1	<#4>	0	0	0
*******************************							
]

Figure 4. Example of MIKE11 cross-section definition file.

The steps for importing the model schematisation from a MIKE11 Ascii cross-section data files, example in Figure 4:

1. Click rnet.ip-file. This opens up the RNet user interface.
2. The first step is to import necessary data to the user interface.
3. Add ipd-file defining the river channel paths, lengths and names (for instance reeds_net4.ipd in the GIS-folder). Use “Add”-button on the upper left hand corner of the user interface. (figure 5)
4. Select the GIS layer you just added from the GIS-layer list (figure 6)
5. Select “Data/ Create data from pllayer”. This will begin a process by which you have to specify rive name and length attributes. The user interface will also query the MIKE11 cross-section file name (assume file type txt). (figure 7).
6. With the assumed reeds_net4.ipd-file the “Select rive name attribute” is the fifth field (TOPO_ID 80 S), “Select river length attribute” the third (BRS_END 15 R). The cross-section file is for instance Xsec.txt in the GIS-folder. The TOPO_ID field in the ipd-file defines the name of each river section. The corresponding cross-sections are sought with the name in the cross-section file.
7. After the query the user interface has created three additional layers for river nodes, channels and names (figure 8).

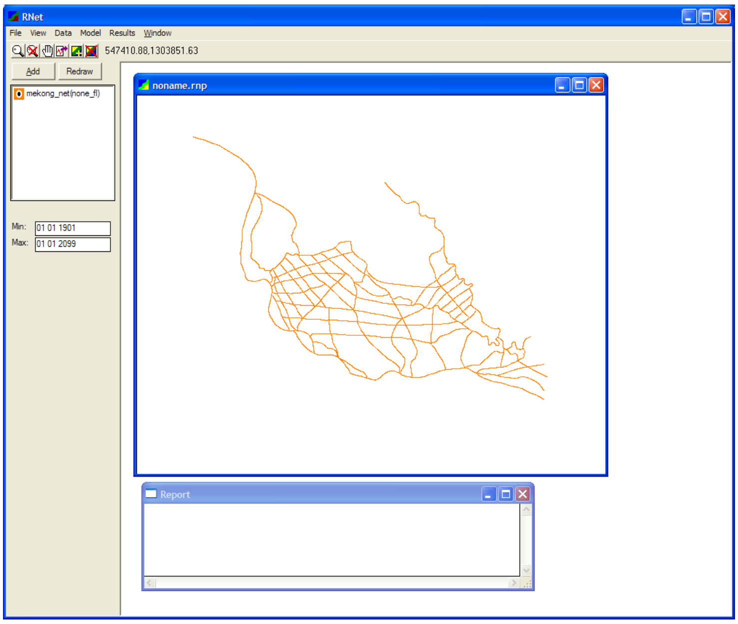
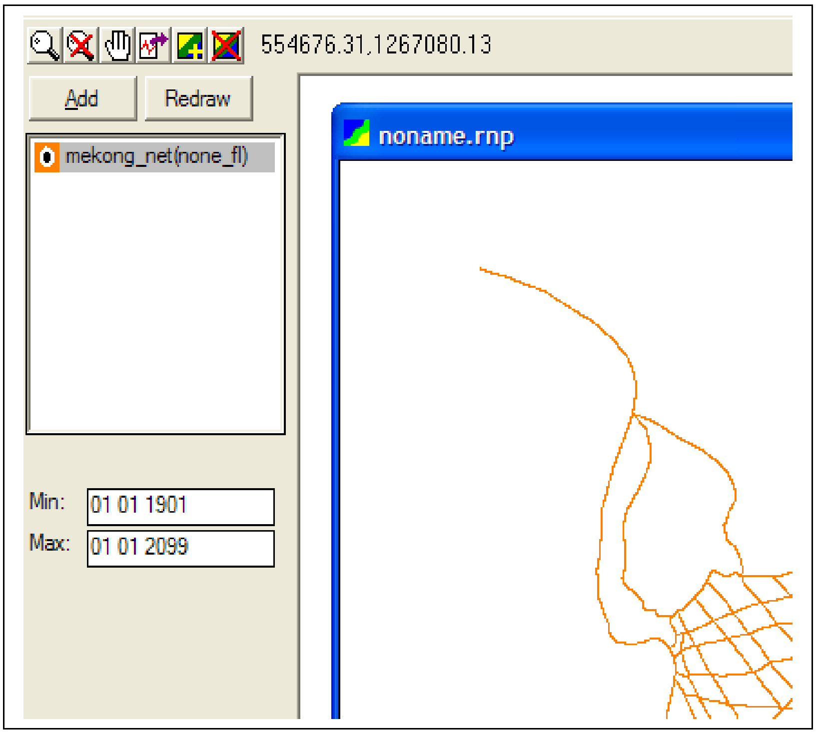


Figure 5. Addition of the ipd-river path file.



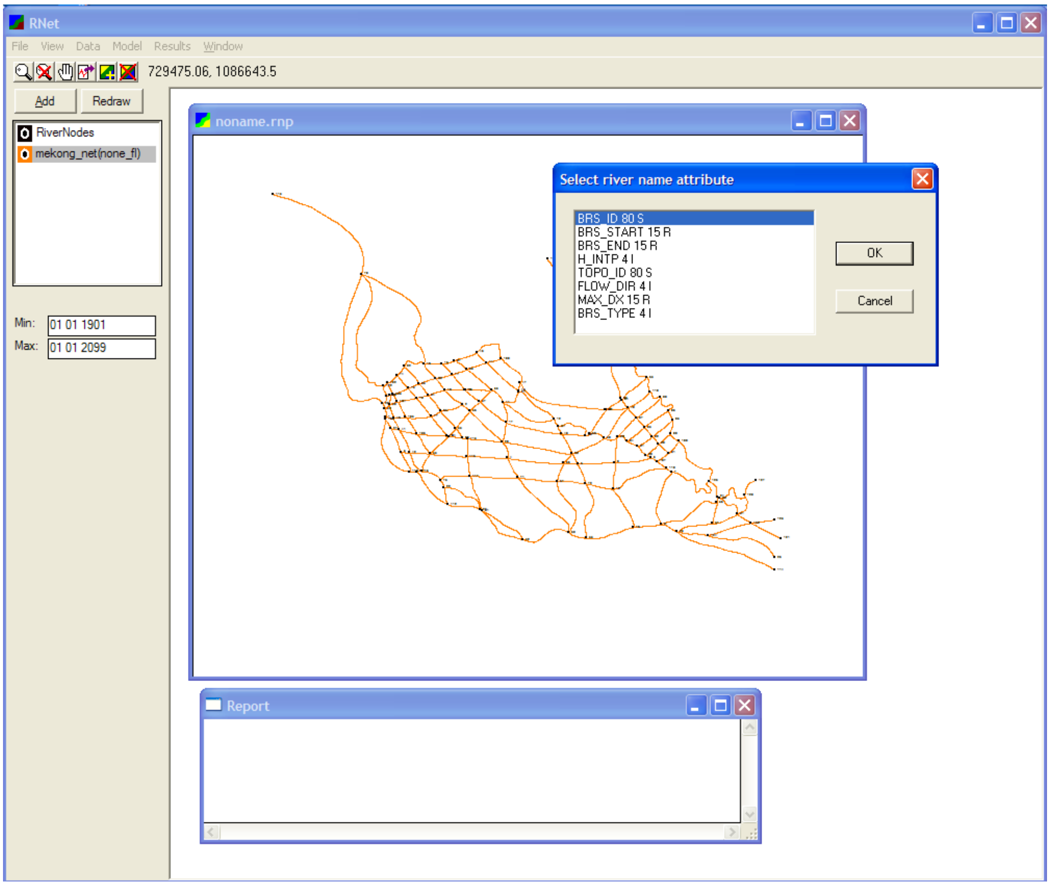
554676.31,1267080.13

mekong_net(none_fl)

Min: 01 01 1901

Max: 01 01 2099

Figure 6. Selection of the river network polygon layer.



729475.06, 1086643.5

RiverNodes
mekong_net(none_fl)

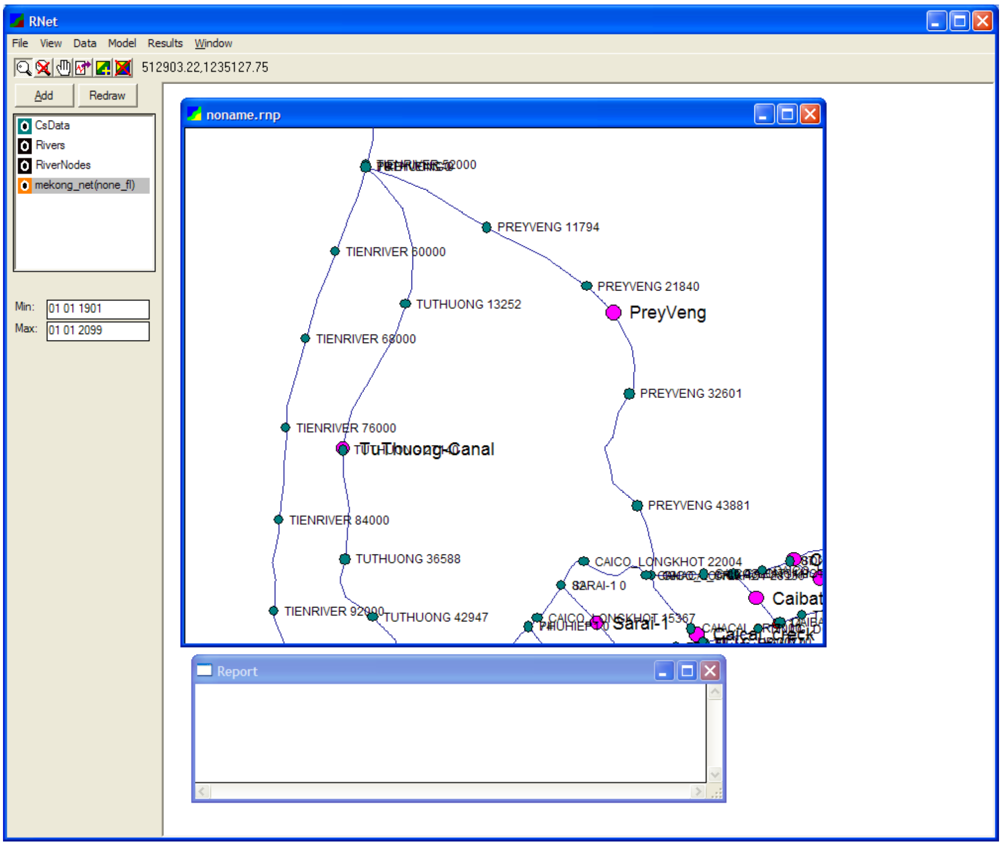
Min: 01 01 1901

Max: 01 01 2099

Select river name attribute

BRS_ID 80 S
BRS_START 15 R
BRS_END 15 R
H_INTP 4 I
TOPO_ID 80 S
FLOW_DIR 4 I
MAX_DX 15 R
BRS_TYPE 4 I

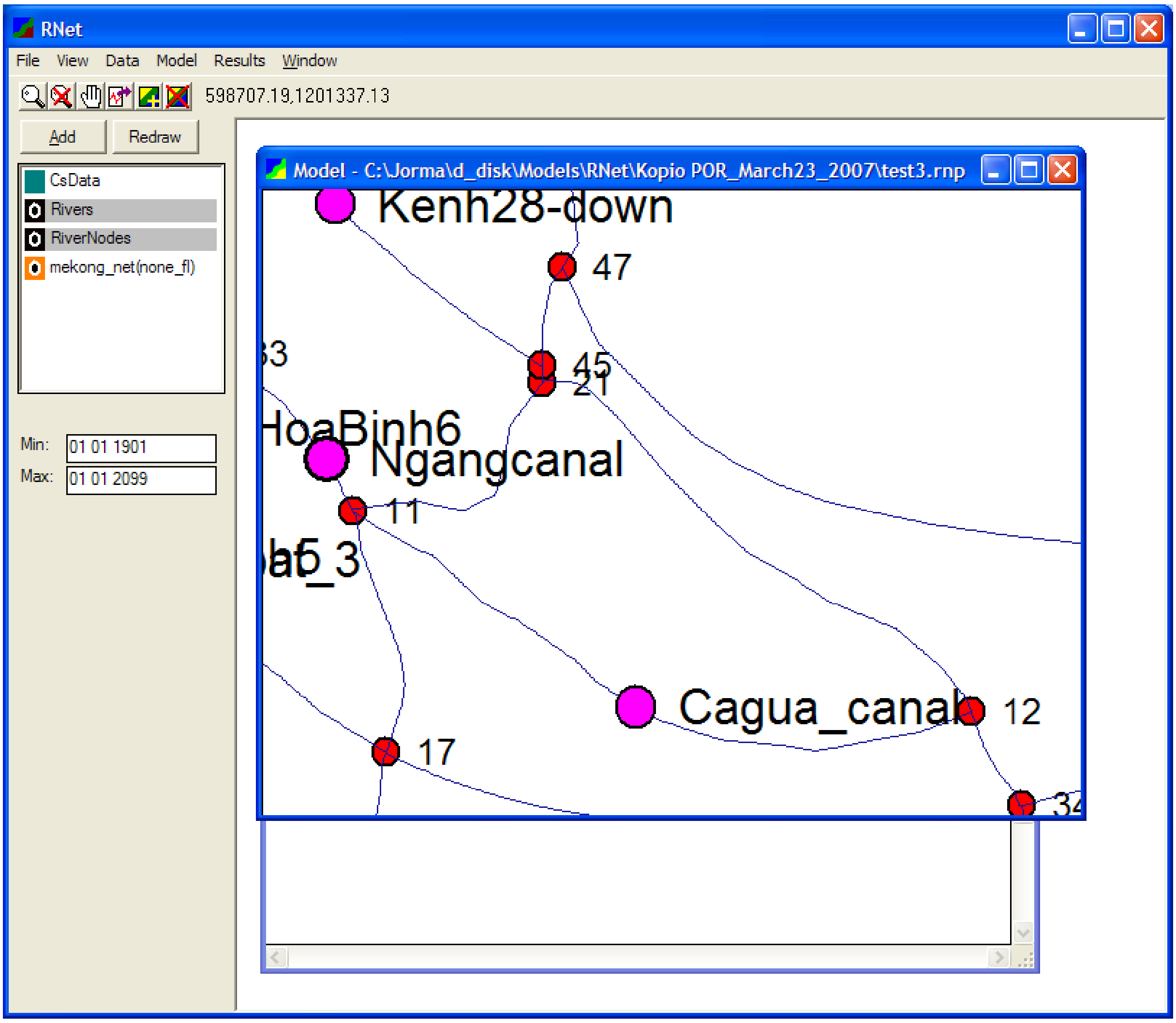
Figure 7. Querying river network characteristics for grid generation.



Min: 01 01 1901
Max: 01 01 2099

Figure 8. River schematisation

8. The next step is to create model grid file (model schematisation). Select Rivers and RiverNodes layers (figure 9). After that select Data/"Make grid file from layer data" and define the name of the grid file. The type of the file is rnd.
9. Define the grid file to be used by the 1D model: Data/Files and define the uppermost Gridfile file.
10. Save the new 1D model as ("File/Save as") as type *.rnp. This is the model definition file that the 1D model will use.



Min: 01 01 1901

Max: 01 01 2099

Figure 9. Selection of layers (Rivers + RiverNodes) for river schematisation file creation.

3.3 MODIFICATION OF THE MODEL SCHEMATISATION FILE (*.RND)

River.rnd file contains

- list of rivers
- list of river nodes and sections with section characteristic information

For each river, data consists of:

Data name	Example value	Explanation
river	40	river identifier
left	59	left node identifier
right	9	right node identifier
gridsz	21	number of grid boxes

length	10607.7	total length of river
xpos	633294.75	x-coord of river middle point
ypos	1174221.5	y-coord of river middle point
path matrix	86, 2	Npoints+1, 2 coordinates; this is a river path in map coordinates
	85, 0	number of path points (Npoints); 0 is for filling
	633914, 1179472	pathpoint1 x,y
	633642, 1178040	pathpoint2 x,y
	...	etc
csdata matrix	2, 7	trapezoidal section definitions: number of defined cross sections in a river, 7 values for each section
	0,-2.17,2.74.,4.04,0.0145,0.92,1.15	section1 data
	7331,-3.38,0.1,3.05,0.02,1.34,2.34	section2 data
dadata matrix	22, 4	Piecewise linear section definitions: number of defined cross sections in a river, 4 values for each section
	0, -2.17, 0.0145, 10	Section1 data
	0.332, 1.2948, 3.307984, 0	Section1, water level1 data
	0.664, 2.5896, 6.615967, 0	Section1, water level2 data
	...	etc
node	4	river node identifie
xpos	588897.125	node x-coordinate
ypos	1170770	node y-coordinate
zpos	0	node bottom elevation or 0
	...	etc

Path matrix: contains river middle line as set of points. Starts from middle of left node, and end at middle of right node. Must contain at least 2 points (left and right node points).

Cross section data (“csdata”) consists of following data:

- distance from left node (m)
- bottom elevation (m)

- river bottom width (m)
- bank slope
- manning's friction (typically 0.01 - 0.05)
- bank/dike/highest height, left bank (m)
- bank/dike/highest height, right bank (m)

“Csdata” is not used in the Plain of Reeds application.

Depth-area data (“dadata”) cross section data consists of:

- distance from left node (m)
- average bottom elevation (m)
- manning's friction (typically 0.01 - 0.05)
- number of water elevation data

For each water level following information is provided:

- water elevation (m)
- cross section area (m^2)
- wetted perimeter (m)
- 0 (a reserved value for future model versions).

To add a river between existing nodes:

Add a new river at the end of list of rivers. Set id to id of last existing river +1, and fill out the rest of the data. At least one cross section is required, and at least two points (start and end) must be in the river path, and there coordinates must be equal to the corresponding node coordinates.

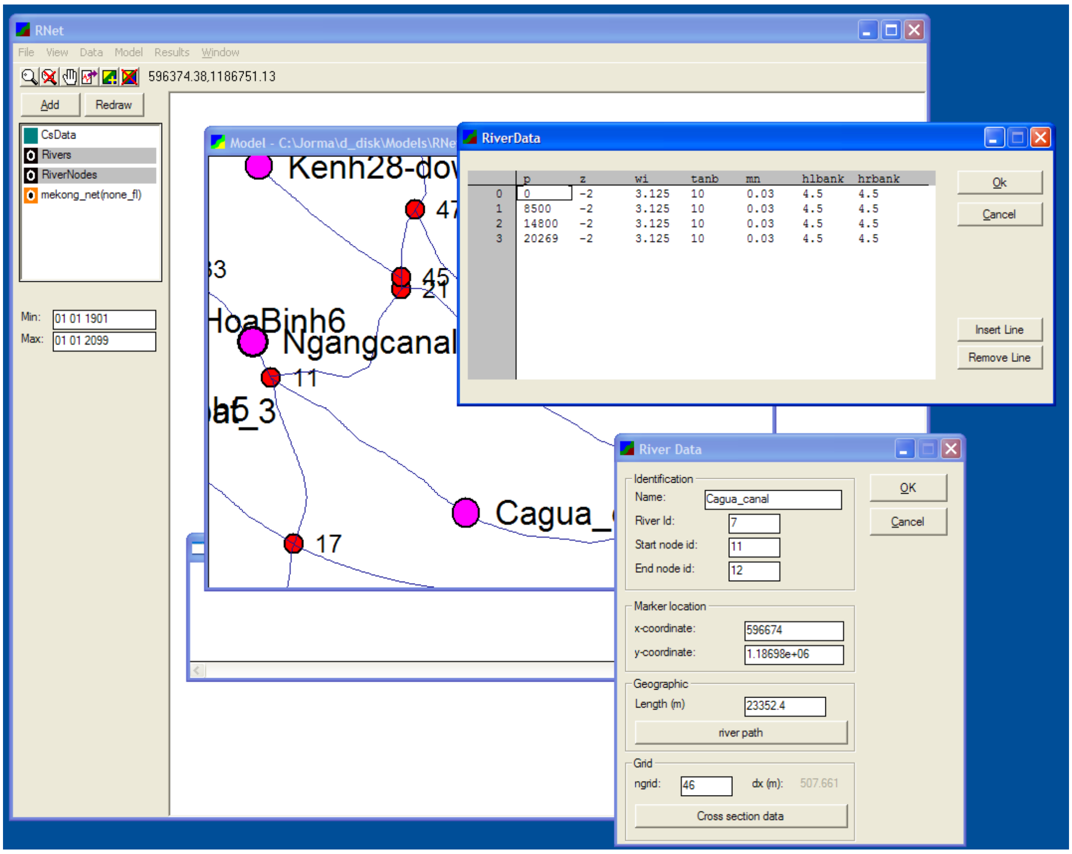
To add a river with new river node:

First add a new river node to node list proceed as above.

To remove a river:

Remove the river from the list. No id changes are required. If some river node is left with no connection to river network remove those nodes as well.

Modification of the river segment and node data by user interface



596374.38,1186751.13

Min: 01 01 1901
Max: 01 01 2099

	p	z	wi	tanb	mn	hlbank	hrbank
0	0	-2	3.125	10	0.03	4.5	4.5
1	8500	-2	3.125	10	0.03	4.5	4.5
2	14800	-2	3.125	10	0.03	4.5	4.5
3	20269	-2	3.125	10	0.03	4.5	4.5

Identification
Name: Cagua_canal
River Id: 7
Start node id: 11
End node id: 12

Marker location
x-coordinate: 596674
y-coordinate: 1.18698e+06

Geographic
Length (m) 23352.4

Grid
ngrid: 46
dx (m): 507.661

Figure 10. Editing of the 1D grid.

Figure 10 shows an example of the user interface. User can select a river node or river. In the figure Cagua-canal has been selected. The dialog windows can be used to change channel parameters including width, slope, elevation, path and manning (at the moment available only for trapezoidal approximation; not applicable to Delta). Also the number of cross sections (actual calculation points) can be altered.

3.4 MODEL PARAMETER FILE (*.RNP)

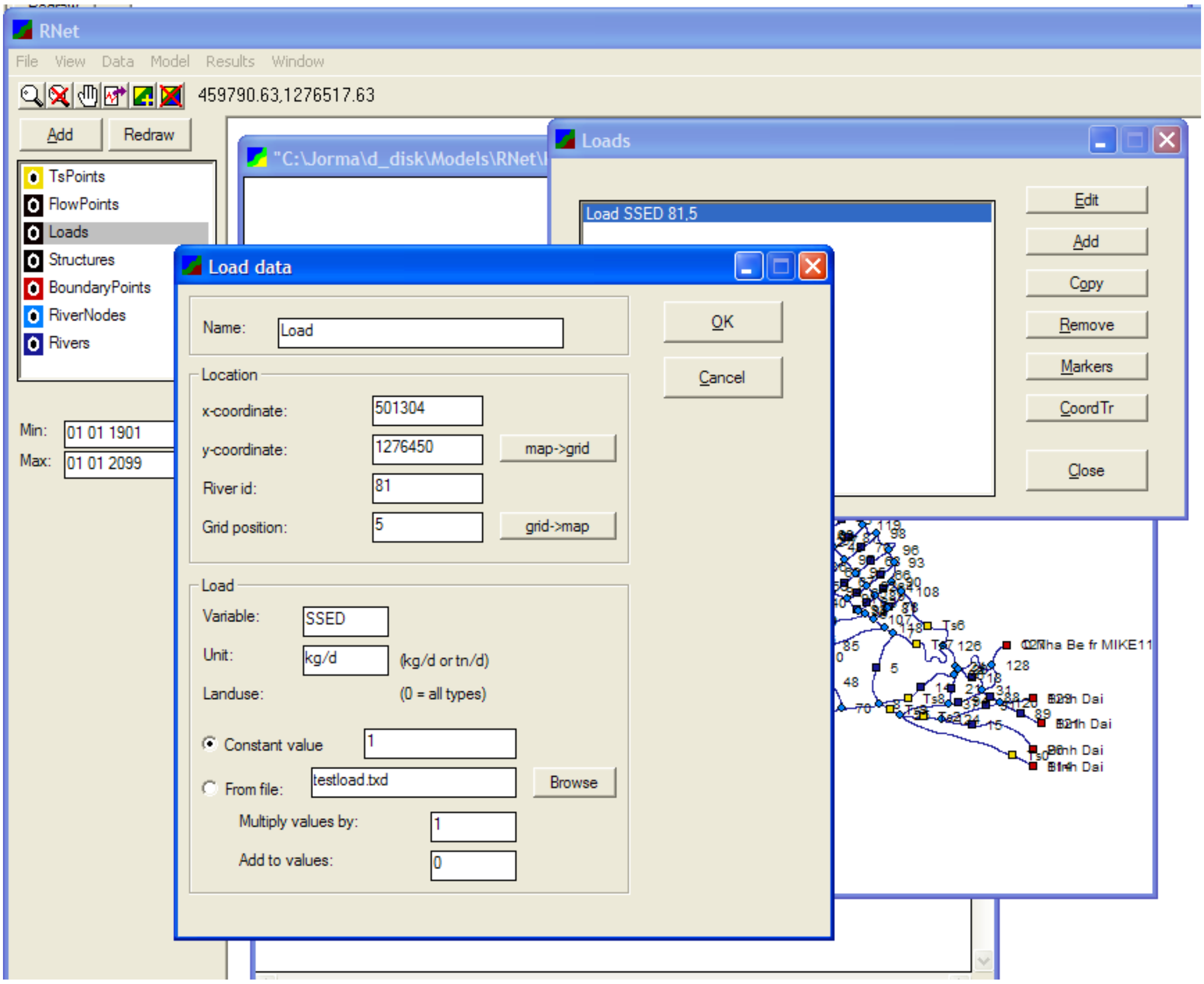
This file defines the model parameters and boundary conditions. Below an example file with explanations. Typically the model user interface creates this file.

File contents:	Explanation:
// rnet parameter file	Comments
// testcase1	
appname "RnetApplication"	application name
gridfile "test3.rnd"	Grid file name
outfile "test3_out.txd"	Output report file
tsfile "test3_ts.txd"	Timeseries output file
fifile "test3_fiels.txd"	Field output file
initfile "none"	Initial state file, "none" if no file
initwl 0	Initial water level
inith 1	Initial water depth
endfile "test3_end.rns"	End state file
msgfile "test3.err"	Error file
frmfile "rnetpar.frm"	Parameter formatting file
exefile "rnet.exe"	Model executable
tspoint list()	list of time series output points
wqvars list()	list of water quality variables
loadts list()	list of water quality loads
flowts list()	list of discharge points
bpoint list()	list of discharge and elevation boundary values
sdata list()	number of additional flows
startdatestr "19830101"	computation starttime YYYYMMDD (this is not in use in the combined 1D/2D/3D model)
enddatestr "19841231"	computation end time YYYYMMDD (not in use in the combined 1D/2D/3D model)
dtriver 30	river computation timestep, seconds
dttsout 3	timeseries output timestep, hours
dtfiout 3	field output timestep, hours
dateout 24	date to screen during computation step, hours
markersz 0.03	marker size

At the moment the 1D model user interface provides possibility to change following parameters (after creation and save of the model, open it again to activate all options):

- computation time step, dtriver, in seconds Model/”Computation parameters”.
- output time steps for timeseries, fielddata and screen date output in hours (Model/”Output timesteps”)
- input and output file selection
- time series points with the addition button (button with a +-sign)
- time series, discharges, water quality load, structures, water level boundaries, and discharge boundaries with the GIS layers (figure 11).

In order to use the GIS-layer method, selected appropriate layer, right click with a mouse and select Properties from the menu.



459790.63,1276517.63

Min: 01 01 1901

Max: 01 01 2099

Load SSED 81.5

Name: Load

x-coordinate: 501304

y-coordinate: 1276450

River id: 81

Grid position: 5

Variable: SSED

Unit: kg/d (kg/d or tn/d)

Landuse: (0 = all types)

● Constant value 1

○ From file: testload.txd

Multiply values by: 1

Add to values: 0

Figure 11. GIS layer method for defining water quality load.

3.5 DEFINITION OF HYDRAULIC STRUCTURES

Flow modifying structures can be inserted to the model to Q grid locations. A structure in the model generally defined as a point in which the flow is not defined by the flow equations, but by structure specific formula that computes the flow through the structure given the water level on both sides of the structure.

The model equations are modified simply, by setting the Q equation coefficient as follows: Aij = 0, Bij=1, Cij=0, RHSij=Qx, where Qx is computed from structure specific formula using water level data from the previous computation step.

Formulas for structures:

Horizontal sharp-edged weir (Kindsvater-Carter rectangular weir equation):

$$Q = C_e \, 2/3 \, (2g)^{0.5} \, b \, h^{1.5}$$

Q = water flow rate, m3/sec

b = width of the weir opening (m)

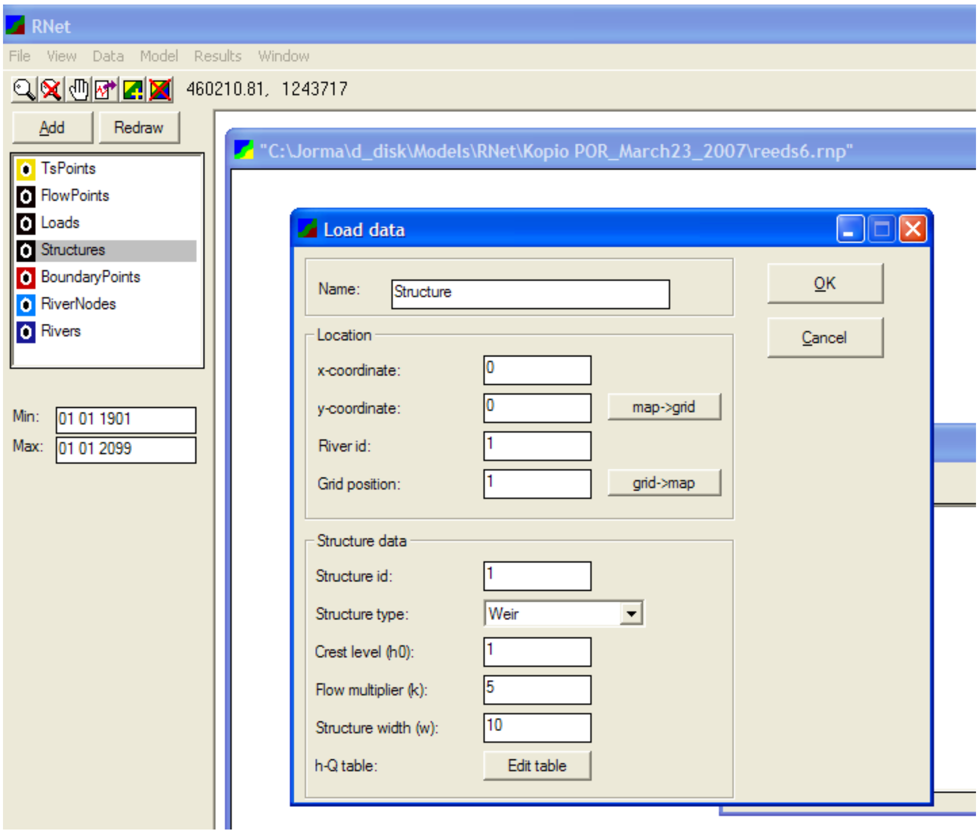
C_e = Discharge coefficient, about 0.6-0.8, averages to about 0.62

g = gravitational constant, 9.81

h = height of the water over the weir (m), measured behind the weir edge

($k = C_e \, 2/3 \, (2g)^{0.5} = 1.83$)

Editor for the hydraulic structures is shown in figure 12.



460210.81, 1243717

"C:\Jorma\d_disk\Models\RNet\Kopio POR_March23_2007\reeds6.rnp"

Min: 01 01 1901
Max: 01 01 2099

Name: Structure
x-coordinate: 0
y-coordinate: 0
River id: 1
Grid position: 1
Structure id: 1
Structure type: Weir
Crest level (h0): 1
Flow multiplier (k): 5
Structure width (w): 10
h-Q table:

Figure 12. Definition of structures through the user interface.

4 USE OF HYDRAULIC STRUCTURES IN THE FLOODPLAIN 2D/3D MODEL

This chapter presents the definition of control structures in the floodplain part (that is the 2D/ 3D model) of the IKMP-1D+2D+3D model. Other aspects of the floodplain set-up are discussed in detail in the 3D user manual.

Control structures are located in the middle of grid cells. They don't change grid cell volume, that is it is assumed that the control structure dimensions are small compared to the grid cell size. If the volume change needs to be accommodated, the model grid has to be changed accordingly.

Only a limited number of controls have been realised in the model compared to the multitude of all possibilities. However, the model input, data and processing structures have been devised in a way that allows easy implementation of any new required structure.

4.1 3D MODEL INPUT METHOD THROUGH THE CONTROL.DAT-FILE

With this option the controls are given in the control.dat-file. The file has to be created in the same folder where the model is run, by default wq-directory. There are 4 possible control structure types:

2 = DIKE (WEIR FLOW)

3 = WATER LEVEL GATE

4 = TIME GATE

5 = DAM/ DIKE BREAK

The format of the control.dat-file is:

NCO : number of control points

IC1,JC1,IC2,JC2, TYPE: control structure start and end points in model grid coordinates and control structure type (2 – 5)

TYPE 2:

DIR,HDIKE1,HDIKE2: dike direction (see below), elevation at the bottom of the dike (if 0 model grid elevation is used), height of the dike from the foot of the dike

TYPE 3:

WL, QX, QY: gate open water level, x-discharge, y-discharge

TYPE 4:

D,M,Y,H : gate start date

D,M,Y,H : gate stop date

QX, QY: x-discharge, y-discharge

TYPE 5:

D,M,Y,H : dike break start date

D,M,Y,H : dike break stop date

BRTIM,BRINW,BRFIW,BRINH,BRFIH: break time (h), initial width, final width, initial height, final height

rnp

For TYPE 2 the dike orientation types are:

1 - 2 weir overflow in both x- and y-directions (corner)

3 weir overflow in x-direction (u-flow)

4 weir overflow in y-direction (v-flow)

An example of a control.dat-file is given below:

3	*:3 control structures specified*
3,3 4,3, 2	*: IC1,JC1 (structure start grid coordinates)*
	: IC2,JC2 (end coordinates)
	:Control Type (2 for weir and dike overflow)
4, 0, 5	*:Orientation, (4 signifying that the flow is in y-direction*
	: HDIKE1, HDIKE2 (elevation and height of the dike)
30,40,31,40 3	*: Water level gate*
2. 4000. 0.	*: WL, Qx, Qy (water level has to rise to 2 m for the gate to open, x-discharge 4000 m^3/s and y-discharge 0 m^3/s)*
40,41,40,41 4	*:Time gate*
1 7 1999 13.4	*:gate open start date*
31 12 9999 12.	*:gate close date*
0. 10000.	*:x- and y-discharges*

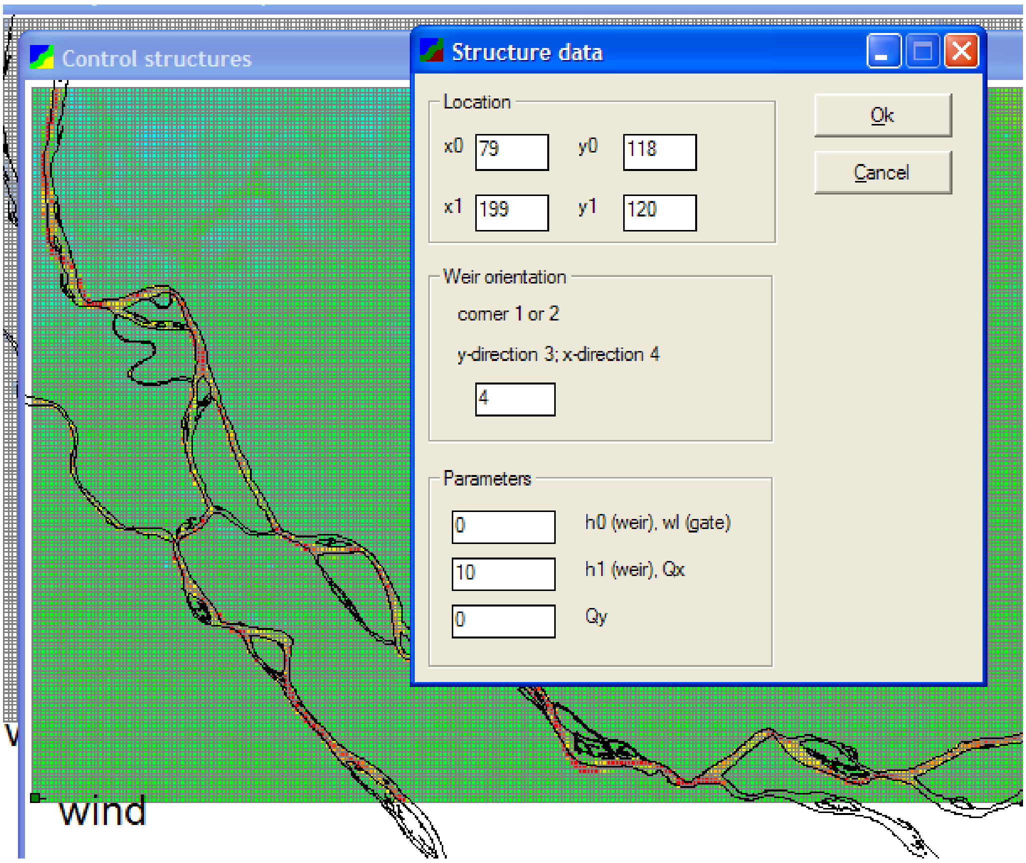
4.2 INPUT METHOD THROUGH A BIL GIS-FILE

When GIS-data on dikes is available, it can given to the model in BIL-format (Binary Inter-Leaved format). At the moment the individual dike height can't be specified with this method and it is assumed that there is no overflow over the specified dikes.

The resolution of the BIL-file has to be the same as in the model. If nested models are used there must be a separate BIL-file for each nested grid. The code for a dike (raster value) is 4000. Other numbers mean no dike. The raster can be created by GIS-software such as ArcGis with appropriate spatial extensions. After GIS-processing the raster layer has to be checked and corrected by hand in order the dikes to be continuous and not block river channels.

4.3 INPUT METHOD THROUGH MODEL USER INTERFACE

Dikes (or embankments in general) and gates can be defined through a map and control structure table editor (figure 14). It can be found in the user interface under Model/”control structures”-menu. After user has pointed out the area and specified the type for the structure, the user interface asks secondary data such as dike orientation, dike height and gate flow. The requested control structure information is the same as explained in chapter 4.1. At the moment dialogue windows for type 2 and 3 structures only have been realised, that is for dikes and water level gates.



Location
x0 79
y0 118
x1 199
y1 120

Weir orientation
corner 1 or 2
y-direction 3; x-direction 4
4

Parameters
0 h0 (weir), wl (gate)
10 h1 (weir), Qx
0 Qy

Figure 14. Dialogue window for specification of dikes, dikes and gates.

5 COUPLING OF THE 1D AND 2D/3D MODELS

5.1 BASIC COUPLING PRINCIPLES

The two component models of the combined EIA-123D model system are presented in the previous chapters. The set-ups and controls described in those chapters are sub-model specific. The remaining two chapters discuss the calculation principles and controls that are needed for the combined running of the two sub-models and are common for the both sub-models.

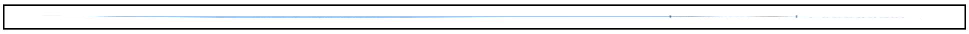


Figure 15. Schematic representation of the river channel (1D) and floodplain (2D/3D) connection. The connection zone can contain a control structure (in figure a dike) or exist without specific control structure.

Figure 15 present the basic principle of coupling. 1D river channel and 2D/3D floodplain models are coupled through a connection zone. The connection zone can be a dike, gate, bridge opening etc. or floodplain area without any control structure. The 2D/3D domain is not restricted to floodplain - as well lakes, reservoirs or coastal areas can be handled with the model.

The flow in the connection zone depends on the control structure. The formulas for dikes, weirs and floodplain without structures are presented here. For gates design discharges are defined. Hydraulic formulas for gates, culverts etc. are easily added when necessary. The *rectangular broad crested weir formula* (Chow, 1959) is used both weirs and dikes. It is an appropriate approximation for embankments, dikes and actual broad crested weirs when it can be assumed that the elevation of the weir is sufficient to create critical flow conditions over the weir. Formula can be used for wide range of conditions, the range defined as $0.08 < h_1/L < 0.5$. *h1* is the elevation of the upstream water surface above the weir and *L* length of the weir crest (the length defined in the direction of the overflow). The broad crested weir formula is:

$$Q = C_d \sqrt{gb}\left(\frac{2}{3}H\right)^{\frac{2}{3}}$$

$$H = h_1 + \frac{V_1^2}{2g}$$

$$C_d = \frac{0.65}{(1 + H/H_w)^{\frac{1}{2}}}$$

Here *g* is the acceleration of the earth's gravity, *b* is the width of the weir (in the transversal direction compared to the weir flow), *H* is the energy of the upstream flow measured relative to the weir elevation, V_1 average velocity of flow upstream of the weir and H_w the height of the weir above the upstream channel/ floodplain. The weir equations are solved iteratively by first assuming value for V_1.

The broad crested weir formula can be used locally for the connection zone also when the water level raises above the weir crest on both sides of the weir until the *h1/L* ratio reaches 0.5. However, in the larger scale the connection zone starts to act as local disturbance compared to the scale of the model solution. For instance, when river channel width is 50 m and floodplain grid cell size 100 m, a embankment width is less than 10 % compared to the solution scale. Because of this, it is defined that when the water level reaches 0.1 m above weir crest on both sides of the weir the solution method is changed into Manning's equation. (In case of the 2D/3D model and floodplain embankment, the solution method switches to the usual Navier-Stokes or diffusive wave solution.) Manning's equation is also used in now control structure case. The Manning's equation is given as (see Karvonen 2007):

$$v = \frac{1}{n} R^{2/3} S_f^{\ 1/2}$$

$$h_f = L\frac{n^2 v^2}{R^{4/3}} = L\frac{n^2 Q^2}{A^2 R^{4/3}} .$$

In these equations:

h = water depth (flow depth) (m)

A= area of the flow cross-section (m^2)

P = wetted perimeter (m), length of line in contact with water

R = hydraulic radius =A/P (m) (in the model approximated as h)

z = elevation above a reference datum (m)

L = characteristic distance between connecting river section and floodplain cell (m)

S_o = bottom slope (m/m); $\Delta z_B = LS_o$

h_f = friction loss (m)

S_f = friction slope (slope of the energy line) (m/m); $S_f = h_f/L$ (in case of uniform flow equals bottom slope = $\Delta z_B/L$)

Q = discharge (m^3 s^{-1})

v = velocity (average) =Q/A (m s^{-1}).

The L is given in the model as the floodplain model grid size. A seemingly more accurate treatment would calculate the actual distance. However, the approximation sets the connection flow scale the same as the floodplain flow, and probably is a better choice.

The problem of too straightforward application of the above flow formulas leads to problems because they don't take into account available flow volumes on both sides of the control structure or the connecting floodplain section. The problem can be overcome with small enough simulation time steps which limit flow volumes transferred during each time step. However, the simulation may become very time consuming. Because of this the model limits water volume transferred during each time step so that on the upstream part of the flow water level doesn't go below the dike crest or floodplain level (figure 15). This doesn't totally eliminate stability problems, because if on the receiving side the area receiving water is small water level may go up too much during one time step and increasing unstable oscillations may result. Filters to eliminate this type of oscillation were tried, but they result in unrealistic model behaviour, and have not been implemented in the current version.

5.2 COUPLED MODEL RUNS AND MASS BALANCES

The parallel running of the 1D and 2D/3D sub-models is flexible in the sense that each model can have independent time step. This poses specific demands for maintaining water volumes and simulation variable mass.

The overall synchronization and mass balances of the different models are maintained by running synchronization time behind the actual execution of the sub-models. The synchronization routine executes a sub-model when the synchronization time is larger than the cumulative sub-model time. After each execution, the sub-model time is increased with the sub-model time-step. The synchronization is further complicated by the fact that the 2D/3D sub-model has different time step for each of its processes. The time-step to be used in the synchronization is selected to be water volume transfer time-step. Water volumes to be transferred are maintained as storage and time counters. For instance if river simulation time step is much smaller than the floodplain one and flow is from river segment to floodplain, the counter adds water volumes to the volume counter each river time step. Similarly the time counter is added. When the floodplain model is executed it transfers water volumes and accumulation times into discharges and adds water volumes corresponding to the floodplain time step to the floodplain. Counters are not allowed to transfer more water than is available in them.

The water volume transfer time-step is used also to transfer mass (concentrations) of salinity, water quality and sediment variables between the river and the floodplain, although internally the time step for 2D/3D mass advection differs from the water volume transfer time-step. Concentrations can’t be used for counters, and they are converted to mass which after river-floodplain mass transfer has to be converted back to concentration. If the floodplain model is 3D, the mass from each vertical layer has to be accounted for. When mass is transferred into the floodplain model it is assumed that flow velocity into each layer is the same and the layer concentrations are modified accordingly taking care that mass is maintained.

5.3 CREATION OF THE COUPLING FILE

The combined 1D/2D/3D model requires a file that identifies the river sections placements in relation of the 2D/3D grid. The file can be created by the following procedure:

1. Select river segment and river node layers (Rivers, RiverNodes) in the RNet model user interface.
2. Select Data/”Export grid connection file” from the user interface.
3. The user interface will ask first to what file the connection information should be written. Specify RNETGRD.DAT, which is read by the combined model by default.
4. User interface will ask specifications for the 3D grid (figure 16). Specify the 3D grid origin coordinates in the same coordinate system as the river network. Specify also 3D grid width and number of grid points in x- and y-directions. In case of the base Plain of Reeds 500 m grid the parameters would be: 518750 1133450 500 363 174.
5. The user interface will calculate the connections rather long time. The resulting output file format is:

32 1 0.5 1 2 (river number, river section number, left bank dike/bank height, right dike/bank height, number of 3D-model grid cells the river section connects to)

*20563 0.1872192395 (3D grid cell number = (i-1)*jc + j, where jc is the number of rows in the grid; ratio of the river section in the specified 3D cell)*

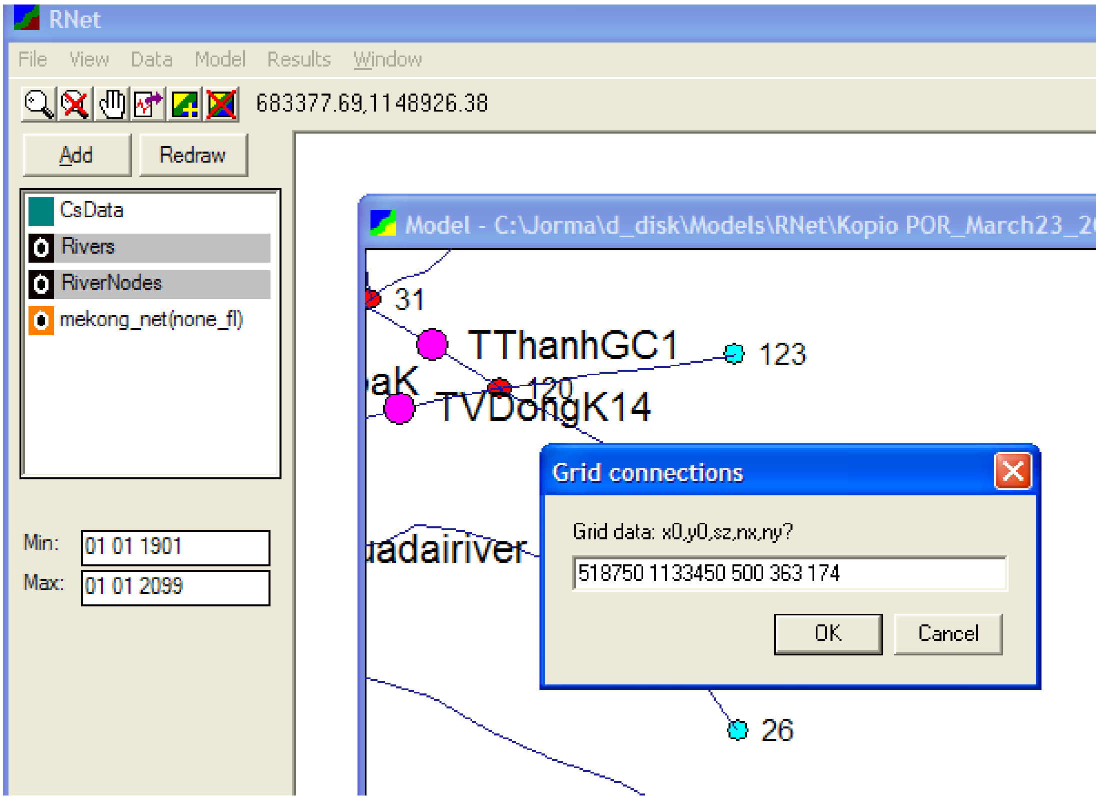
20737 0.8127807456

32 2 0.5 1 3

20737 0.259372386

20910 0.6695159097

```
    20911 0.07111183104
32   3 0.5 1 2
    20910 0.3313231896
    21084 0.6686768402
      etc.
```



683377.69,1148926.38

Min: 01 01 1901

Max: 01 01 2099

Grid connections

Grid data: x0,y0,sz,nx,ny?

518750 1133450 500 363 174

Figure 16. Specifying the 1D-3D connection file. Dialogue for 3D model origin and grid size.

If one wants to connect river to both sides of the floodplain separately, one has to add 3D grid cells to left and right hand sides of a river. At the moment the model takes maximum of the two bank height values as the dike/bank height. A more advanced description is possible where the 3D grid cell location in relation of the right and left banks is taken into account.

5.4 COUPLING OF THE FLOODPLAIN AND RIVER FLOW IN THE MODEL

There are two basic ways how water flows between the river and floodplain systems:

1. over river banks
2. through control structures

The limiting river bank elevation is defined by the maximum cross section bank elevation defined in the RNETGRD.DAT-file (see previous chapter). Currently the Plain of Reeds maximum elevations may be underestimates, at least in certain locations, as they have been derived from a 1D model which is not sensitive to the elevations.

Control structures include both static and dynamic structures. So far dikes/banks and gates have been implemented, but it is easy to include any structure as data becomes available. The control structures are given to the model in RFCONTROL.DAT-file. Its file format resembles that of the RNETGRD.DAT:

81 93 0. 0. 1 (river number, river section number, left dike/bank height, right dike/bank height, number of 3D-model grid cells the river section connects to)

103 37 1 (3D model grid cell i and j indexes; control type)

10. 0.1 1. (control information)

At the moment left and right dike/bank heights don't have impact on the calculation.

The control structure information for weirs (type = 1) is:

- weir width (m)
- weir crest elevation (m)
- scaling

The control structure information for gates (type = 2) includes only gate flow (m^3/s). Additional possibilities for implementation include specification of gate operation water levels and times.

It is possible to eliminate river bank overflow completely and give exchange only through the control structures defined in RFCONTROL.DAT-file. This is done by specifying model executable I1d2d3d_hw.exe (hw = high walls) in user interface menu “Source data/ Application setup/ Model executable”.

5.5 MODEL CALIBRATION

In the 1D model only Manning coefficient is available for a calibration variable. With the trapezoidal approximation the Manning coefficient can be mostly easily defined through the grid editor by selecting the river and editing the Manning value (figure 10). This is not applicable for the Delta applications, and user needs to edit rnd-file for calibration.

Manning coefficient together with the calculated flow velocity determines directly friction in each river cross-section, and indirectly water levels and flow dynamics.

The calibration procedure for the combined model is:

1. calibration of 1D model separately focusing on low water season
2. checking and possible calibration of the 2D/3D model
3. combined calibration of the two models focusing on high water season.

Calibration results can be seen in figure 13. The green squares are measurements, red line is 1D model result and blue line combined 1D/2D model result. The Manning coefficient in 1D has been calibrated to 0.0145 through whole river system. The combined model gives better results indicating that river-floodplain interaction impacts water elevations in the high water season. In the low water season combined model gives same results as the 1D model.

It is expected that model results will be improved even further when following developments take place:

- addition of control structures, mostly gates, to the model
- checking of river bank elevations; this data has been obtained from a 1D model that is not sensitive to the high river bank elevations and it is suspected that data has not been carefully checked in this respect
- more detailed Manning calibration distinguishing different types of channels

- calibrating magnitude of the channel – floodplain interaction; at the moment scaling of the bank flow is set to 10%.

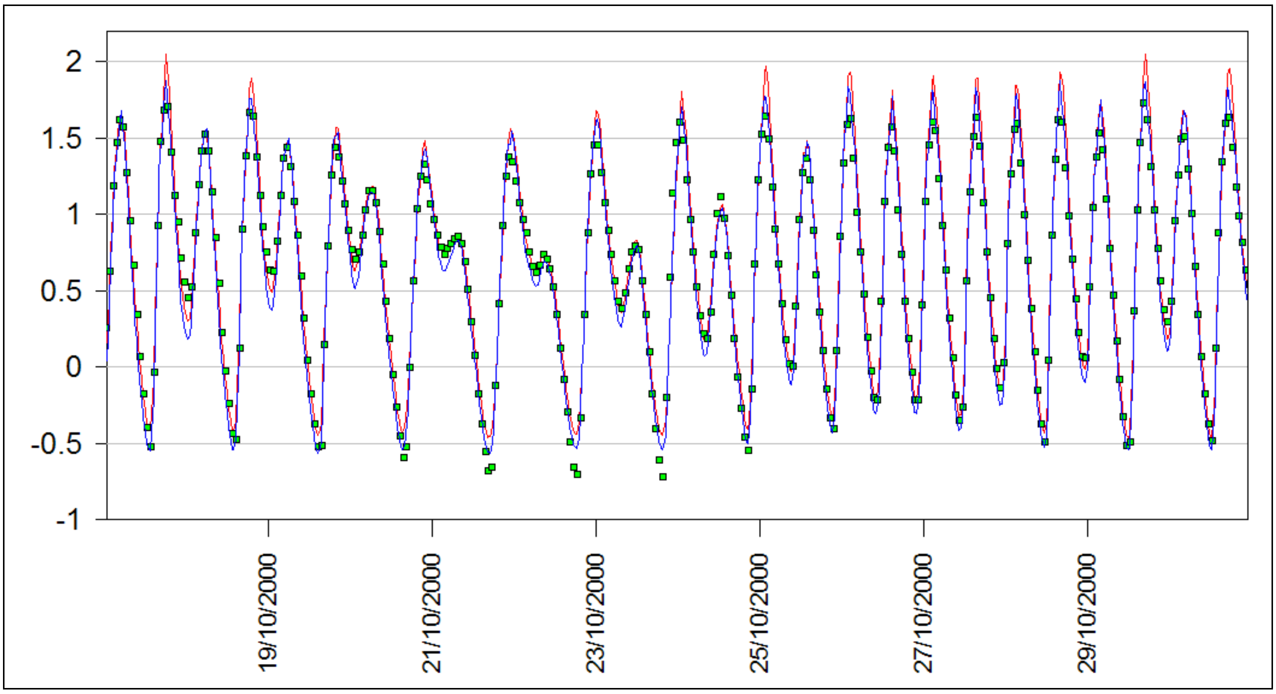


Figure 13. Comparison of measured (green boxes) water elevations in My Tho with computed values (red 1D, blue 1D+2D).

6 RUNNING THE COMBINED MODEL

After the 1D and 2D/3D sub-models have been set-up and the connection files (see previous chapter) have been defined, the combined model is ready to run.

The combined model user interface functions in every way in the same manner as the separate 2D/3D model interface. The only exception is the default model execution folder where model input and output files are created. In case of the combined model all input-, output- and executable files are located in the same folder. This is because the model executable requires common folder for both sub-models.

During the run-time, animations show only the floodplain model results (figure 17 and 18). After model run the 1D model results can be viewed through the RNet user interface or through clicking the timeseries file which opens timeseries drawing software.

Some examples of the 2D/ 3D floodplain model post-processing results are shown in figures 19 – 22. The case in figures includes a flood pulse travelling through the channel network (1D) as well as overland flow from Cambodia (3D). In many places there is out- or inflow from/ into the channels (calculation method 2 described in Chapter 5.3). The colours signify sediment concentration. Sedimentation is included in the simulation.

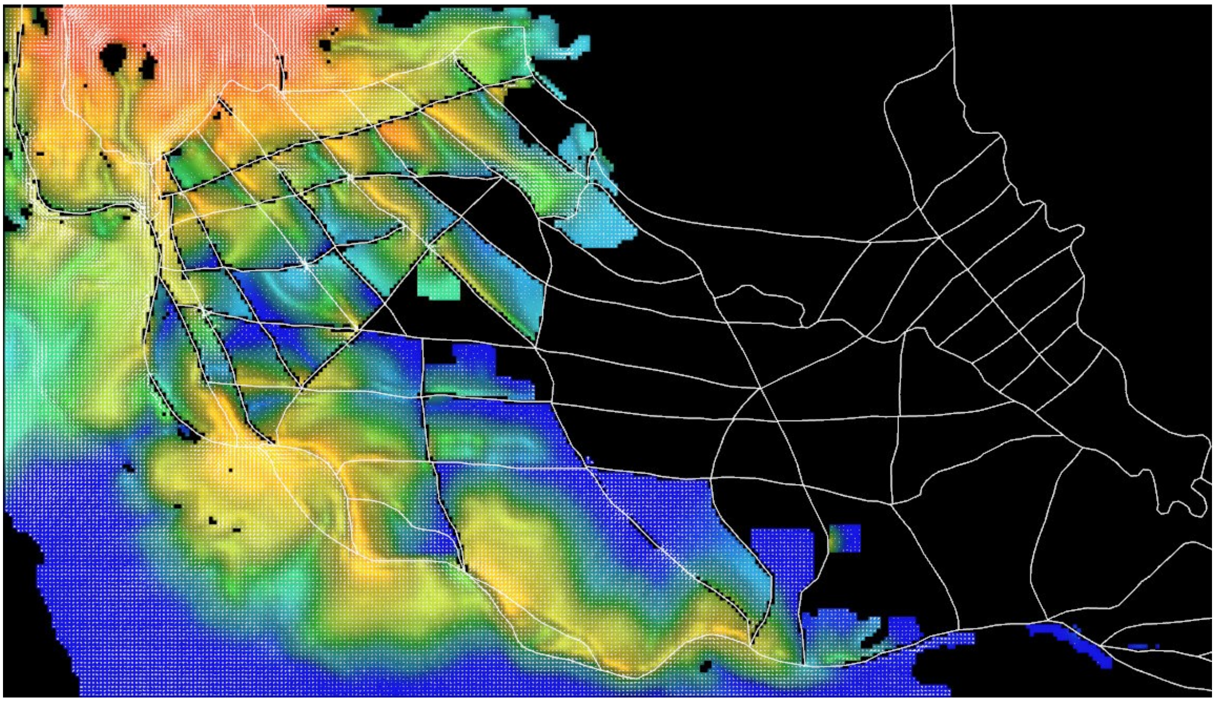


Figure 17. Model animation window during simulation. colour signifies sediment concentration. Calculation method number 2 (see Chapter 5.3) used. Two channel embankments (roads) edited in the rnetgrd.dat-file. Overland flow from Cambodia and flood pulse travelling in the river network.

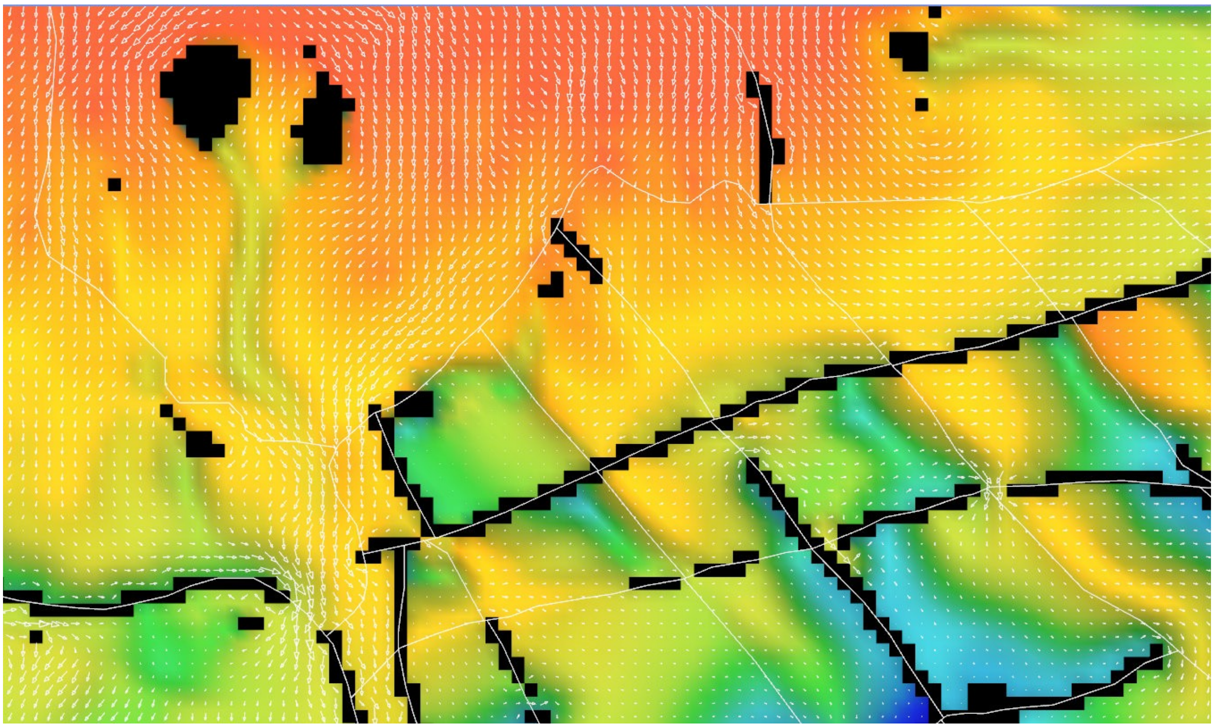


Figure 18. Zoomed view of the animation window showing flow arrows and sediment concentrations.

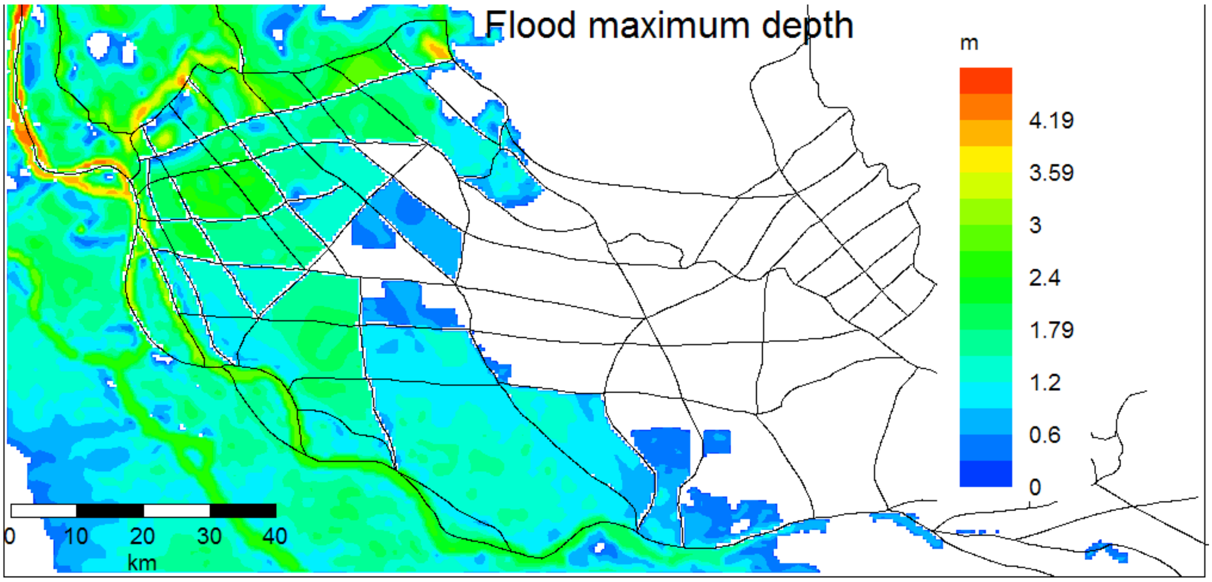


Figure 19. Floodplain model post-processing results showing flood maximum depth.

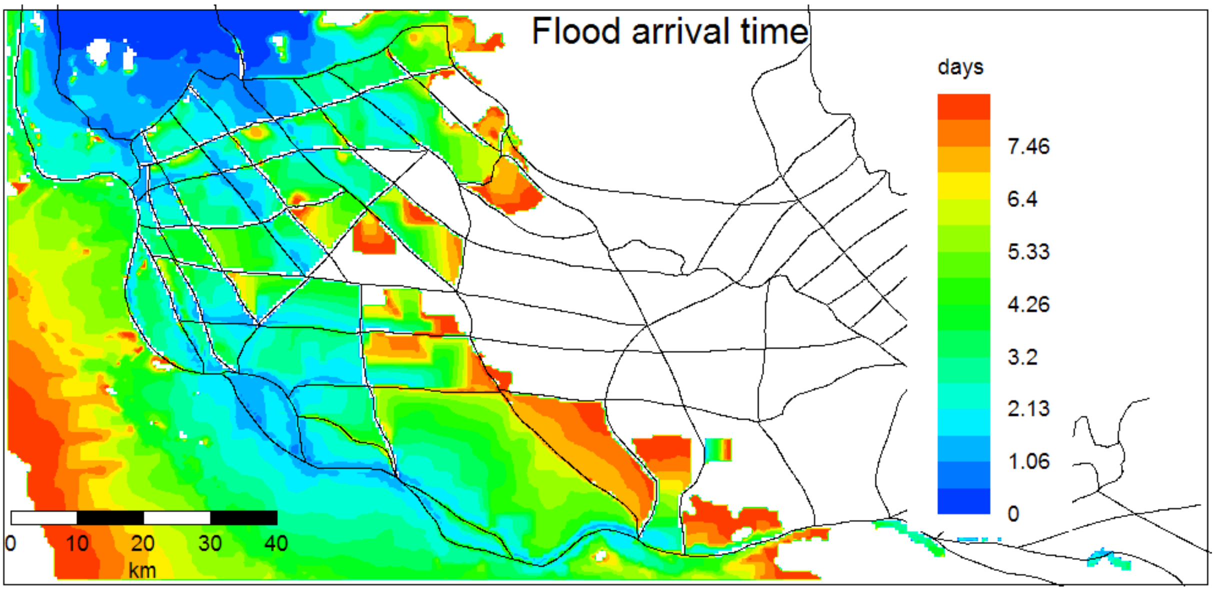


Figure 20. Floodplain model post-processing results showing flood arrival time.

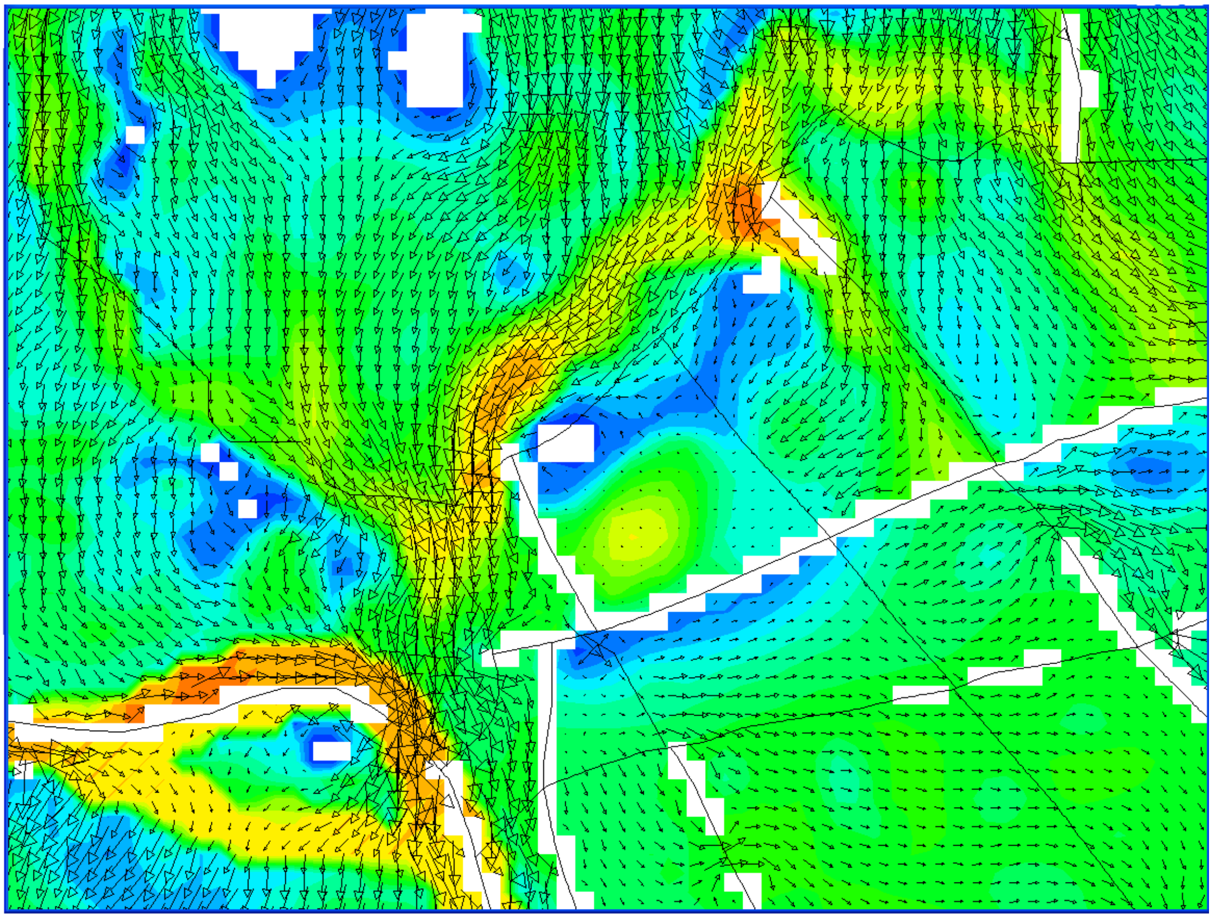


Figure 21. Floodplain model post-processing results showing close-up of water depths and floodplain flows.

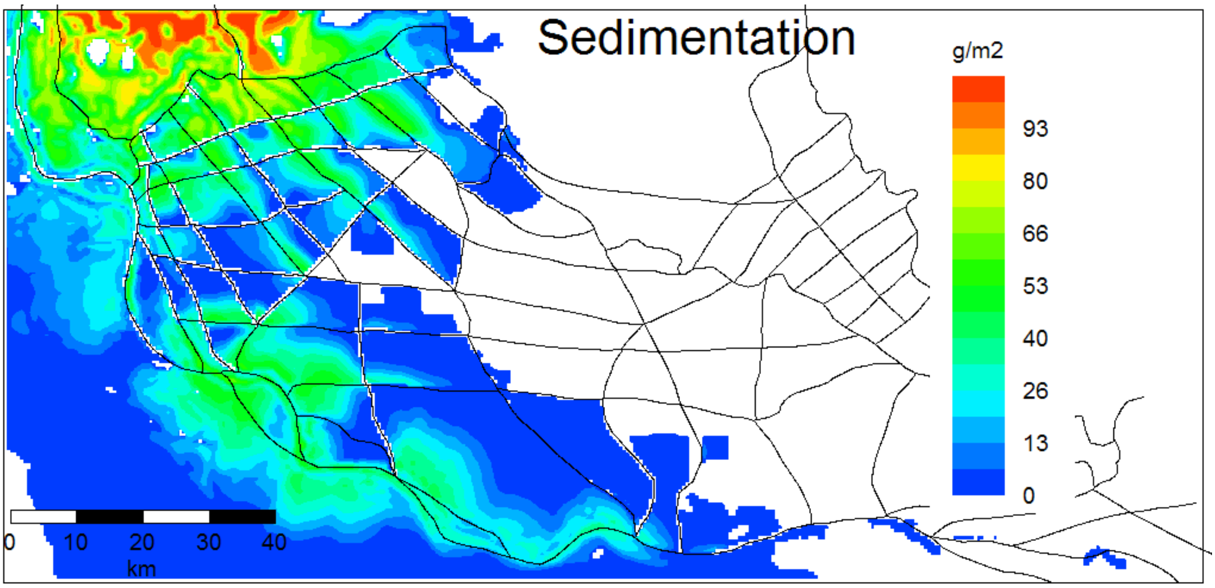


Figure 22. Floodplain model post-processing results showing net sedimentation.

Box 1. Overview for the model developers and requirements.

Name of software:	EIA-123D Model
Developer:	Jorma Koponen, Markku Virtanen, Hannu Lauri, and Arto Inkala (of about 20 other developers) Environmental Impact Assessment Centre of Finland Ltd. (EIA Ltd.) Tekniikantie 21 B, 02150 Espoo, Finland Tel. +358-9-70018680 Fax. +358-9-70018682 E-mail: koponen@eia.fi / virtanen@eia.fi
Version history:	First 2D version 1975, first 3D version 1983, last major revision in 2002. 2007 major change in the flooding model. RNet-1D model developed since 1997. First combined 1D/2D/3D model in 2006, enhanced version in 2007.
Hardware required:	Can be run in all types of computers from PCs to supercomputers where appropriate compilers are available.
Software required:	Can be run on all platforms (most user friendly version requires Windows-platform)
Program language:	FORTRAN (model and basic graphics), C (run-time interactive animation) and C++ (graphical user interface and RNet model)
Program size:	About 4 MB. Graphical user interface 0.7 MB (sizes are for Windows)
Availability:	Available free within cooperative projects or as part of an application
More information:	www.eia.fi

Development work has required about 100 man years and three decades. Model system has been used in more than 300 research, engineering and consultancy studies.

The software runs exceptionally fast compared to many other simulation models. With other software combined 1D/2D modelling can require more time than passes in real-time.

EIA-123D and EIA-23D are probably the only 3D floodplain models existing.

If the model user interface setup program is not working or the user interface doesn't work properly, the reason can be following:

- You don't have the administrator rights when installing the software
  → sign in to the computer as administrator and install again
- You don't have enough space in your hard disc
  → release space in your hard disc and try to install again
- Uninstallation of previous version of the software has not been successful
  → re-initiate uninstall either through control panel or through EIA remove menu and delete l.exe-, frm- and ip-files in the application directories.
- There are errors when user interface is used especially after software crashes
  → check by going to Task manager and there processes. If you see a Viv-processes, end them and re-initiate application again.

If the model simulation is stuck but consumes full processor power:

- The 3D OpenGL graphics drivers and hardware acceleration cause in some computers problems
  → In Windows go to Settings/ Control Panel/ Display/ Settings/ Advanced/ Troubleshoot/. Set Hardware Acceleration one notch above "None".

If the model simulation gives unrealistic results or crashes:

- The most common cause is too long time-steps
  → Test and reduce time steps one by one until model runs properly