Tutorial Hydrodynamics in open water

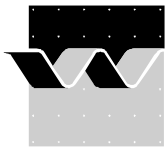
SOBEK-Rural 1DFLOW

August 2007

Tutorial Hydrodynamics in open water

SOBEK-Rural 1DFLOW

August 2007



WL | delft hydraulics

Contents

1 Tutorial Hydrodynamics in open water

1.1 General

In this tutorial the basic principles of working with the 1DFLOW module of SOBEK-Rural are explained step by step and you will be guided to set-up a simple network and to extend this network with new elements. This tutorial will only show a limited number of the large amount of options. It will teach the basic principles of working with the 1DFLOW module of SOBEK-Rural and give you enough experience to continue on your own. Some experience in working with the Microsoft® Windows® operating system is required.
The tutorial contains:

1. setting up a simple network;
2. editing boundary conditions;
3. editing profiles.

The tutorial does not explain all options in all windows that appear. Once you get the hang-and-feel of the modelling system, you may wish to browse through the options not dealt with in the tutorial.

1.2 Getting started

- Click the Windows <Start> button.
- Select the 'Programs' menu.
- Select the 'Delft Hydraulics' menu.
- Select the 'SOBEK' menu item (SOBEK210).
- Click the 'SOBEK' icon.
- In the SOBEK main window: select the menu item 'Options' - 'SOBEK Options'.
- Select the tab 'Background Map'.
- Select the file 'Tutorial1.map'.
- Press <OK> button to save and close SOBEK Options.
- Double-click the 'New Project' button.
- Type the name 'T_CHANN'.

  The program converts all the characters into upper case. If a project with the same name already exists, the user has to enter a different name here.

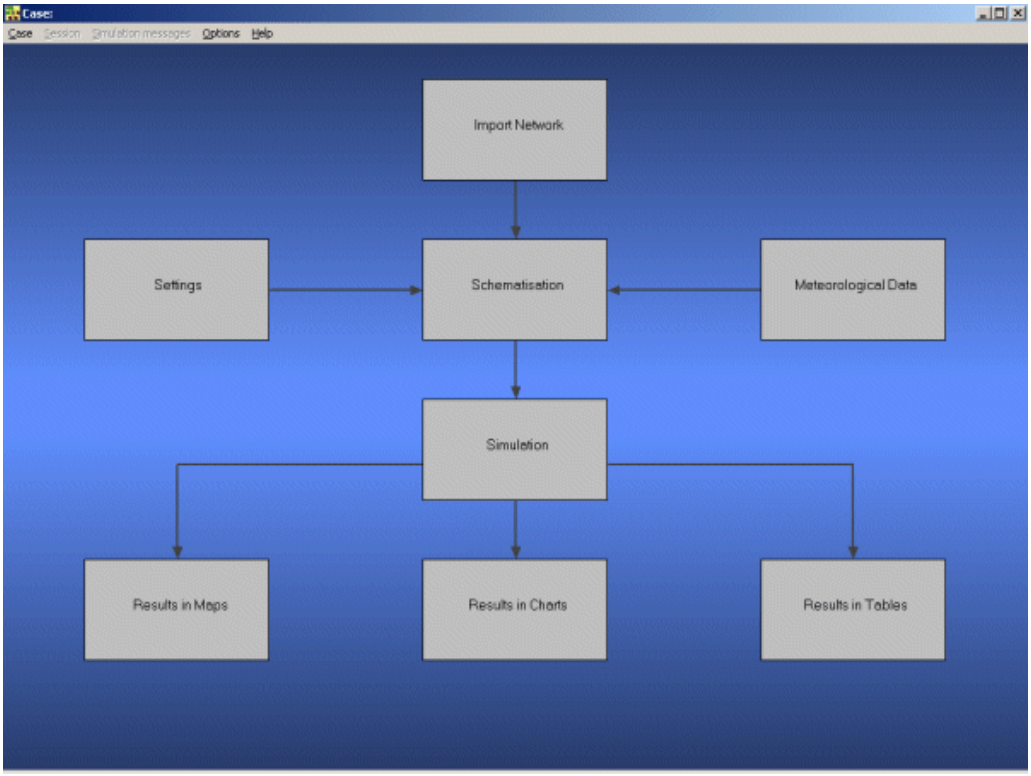
- Click the <OK> button.

You have added a new project with the name 'T_CHANN'. You are now asked: do you want to work with this project?

- Click the <Yes> button.

1.3 Case management

The screen of the so-called "case manager" appears. This tool automatically keeps track of cases and the related files. For instance: you might want to save different scenario's within a project as cases with different names. This is organized through the case manager.



The case manager screen..

On the screen a number of blocks will appear:

1. Import Network;
2. Settings;
3. Meteorological Data;
4. Schematisation;
5. Simulation;
6. Results in Maps;
7. Results in Charts;
8. Results in Tables.

Each block represents a specific task. A task can be a model, a set of linked models, the selection of a scenario or strategy, or a (graphical) presentation tool. The arrows between the blocks represent the relations between the tasks. When an arrow is pointing from block "A" to block "B", the task of block B can only be executed after the task of block A is finished.

The Case Manager has the following tasks:

9. administration of cases (which data are related to which cases);
10. checking whether the model calculations for the cases are performed in the predefined order;
11. logging the actions of the Case Manager (including view and print);

12. providing access to the computational framework through a user interface, so the user can:
    manipulate a case (read, save, delete, etc.);
    choose and run predefined tasks (modules);
    view and check the status of all tasks;
    view the relation between the various tasks.

When the Case Manager screen appears first after you have added a project all task blocks are grey. To activate the task blocks you have to open the default case of this new project:

- Select the menu option 'Case' - 'Open'.
- Select 'Default' from the list.
- Click <OK> button.

Another method is to double-click one of the grey task blocks and select ‘Default’.
Once you have opened the default case the task blocks are no longer grey, but one of the following colors:

13. yellow: the task can be executed;
14. green: the task has been executed at least once and can be executed again;
15. red: the task cannot be executed until the preceding task has been executed.

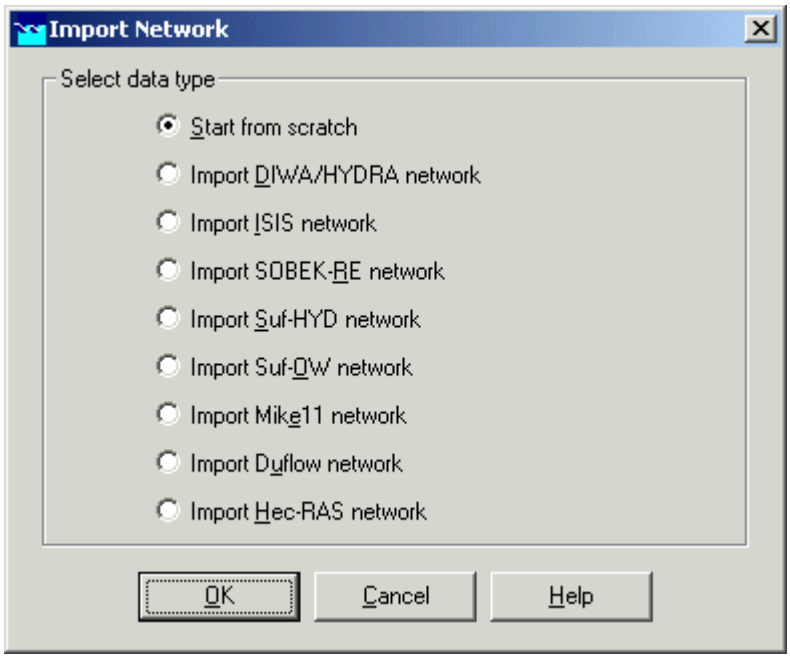
When the task is being executed the task block is purple. You can execute a task by double-clicking on the task block. When you select a yellow or green task block, the color will change to purple and then change to green again when the task is finished.

Now, we will discuss each task block.

1.4 Task block 'import network'

The color of this task block is yellow, which means that this task block is still waiting to be executed.

- Execute the task block 'Import Network' by double-clicking it. The block then turns purple and the Import network window will pop up:



The import network window.

In this task block the origin of the schematisation must be defined. Schematisations, used in SOBEK, can be either imported from a database or set-up from scratch. If a schematisation is already available in the standard exchange format it can easily be imported from the database to SOBEK. Links with data formats can be custom made on request. For that reason some radio buttons might be turned grey.

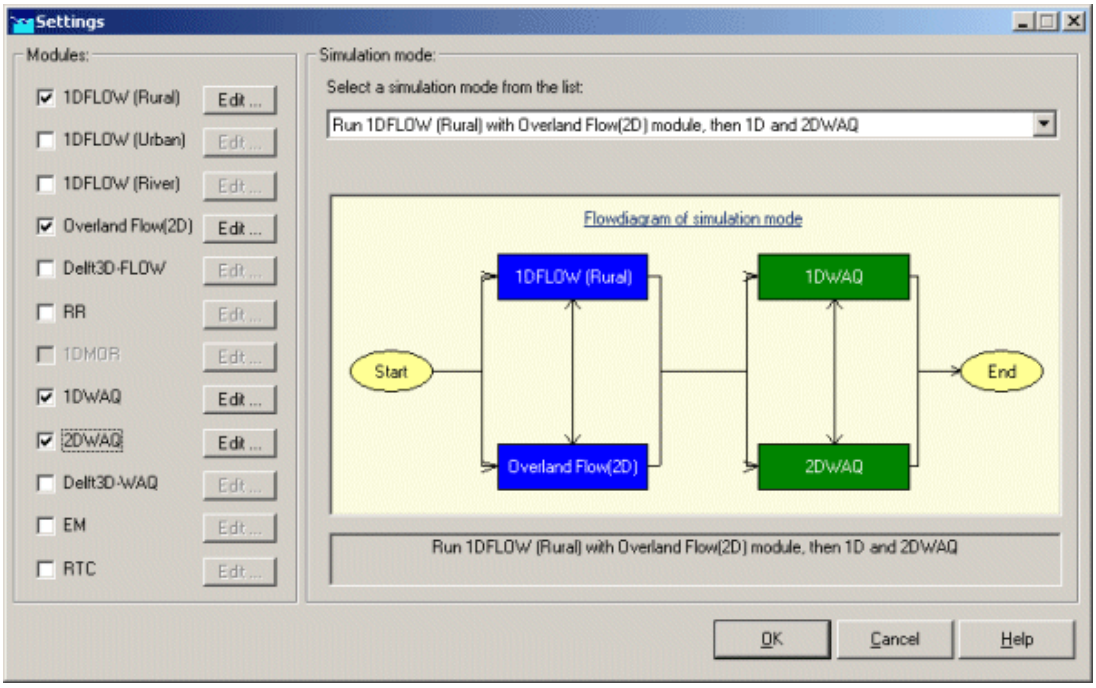
Let's set up a schematisation from scratch.

- Select the radio button 'Start from scratch'.
- Press <OK> button.

Notice that you're back in the Case Manager now and that the task block 'Import Network' has turned green.

1.5 Task block 'Settings'

The 'Settings' task block is used to select the SOBEK modules that you want to use for your project. Also computational parameters such as calculation time steps, simulation period and initial water levels can be set in the 'Settings' task block. Depending on the set of modules that you purchased, some of them may be disabled (grey), and some may be enabled.



The settings window.

Hydrodynamics

SOBEK-Rural 1DFLOW
The SOBEK-Rural 1DFLOW module is a sophisticated module that can be used for the simulation of one-dimensional flow in irrigation and drainage systems. It is a tool that can be used to simulate and solve problems in regional water management, such as irrigation construction, drainage, automation of canal systems, dredging and flood protection. This module can be used stand-alone or in combination with other modules, for example the SOBEK-Rural RR module (Rainfall-Runoff).

SOBEK-Urban 1DFLOW
The SOBEK-Urban 1DFLOW module is a sophisticated module for the simulation of one-dimensional flow in wastewater and storm water systems. It is a tool that can be used to simulate and solve problems in urban drainage systems such as determination of urban drainage capacities including treatment plants, assessment of sewer overflow frequency and design of detention basins. The SOBEK-Urban 1DFLOW module can also be used in combination with the SOBEK-Rural 1DFLOW module, the SOBEK-Urban RR (Rainfall-Runoff) module and other modules. One of the competitive advantages is the combination with the SOBEK-Rural 1DFLOW module for environmental study on receiving waters.

SOBEK-River 1DFLOW
The SOBEK-River 1DFLOW module is a sophisticated module that can be used for the simulation of one-dimensional water flow in river systems and estuaries. It is a tool that can be used to simulate and solve problems in river water management such as flood protection, flood-risk assessment, real-time forecasting, dam break analysis, navigation and dredging. This module can be used stand-alone or in combination with other modules.

Hydrology

The RR (Rainfall-Runoff) module is a module that can be used for the simulation of rainfall-runoff processes. This module is a part of a large family of modules which can be linked. The list of modules includes (amongst others) SOBEK-Rural 1DFLOW module, SOBEK-Urban 1DFLOW module and RTC (Real Time Control) module. The RR module is frequently used in combination with the SOBEK-Rural 1DFLOW and SOBEK-Urban 1DFLOW modules. It is then possible to either to perform calculations for both modules simultaneously or sequentially.

Real Time Control

The RTC (Real Time Control) module is a module that can be used for the simulation of complex real time control of hydraulic systems. It can be applied to rainfall-runoff, hydraulics and water quality computations. In that case the rainfall-runoff and water quality computations are run simultaneously with the hydrodynamics computations, thus incorporating full interaction between all processes.

Water Quality

The above mentioned modules can also be used in combination with modules for simulating water quality processes (1DWAQ module, 2DWAQ module and/or EM (EMission) module).

Thus, several combinations of modules are possible. Depending on the problems to be solved you can set the desired combination. The modules can easily be selected via the task block 'Settings'.

- Double-click the 'Settings' task block. Its colour changes and the settings window appears.
- Unselect all the selected modules if any.
- Select the '1DFLOW (Rural)' module.
- Press the <Edit...> button of '1DFLOW (Rural)'.

You have to define a number of settings.

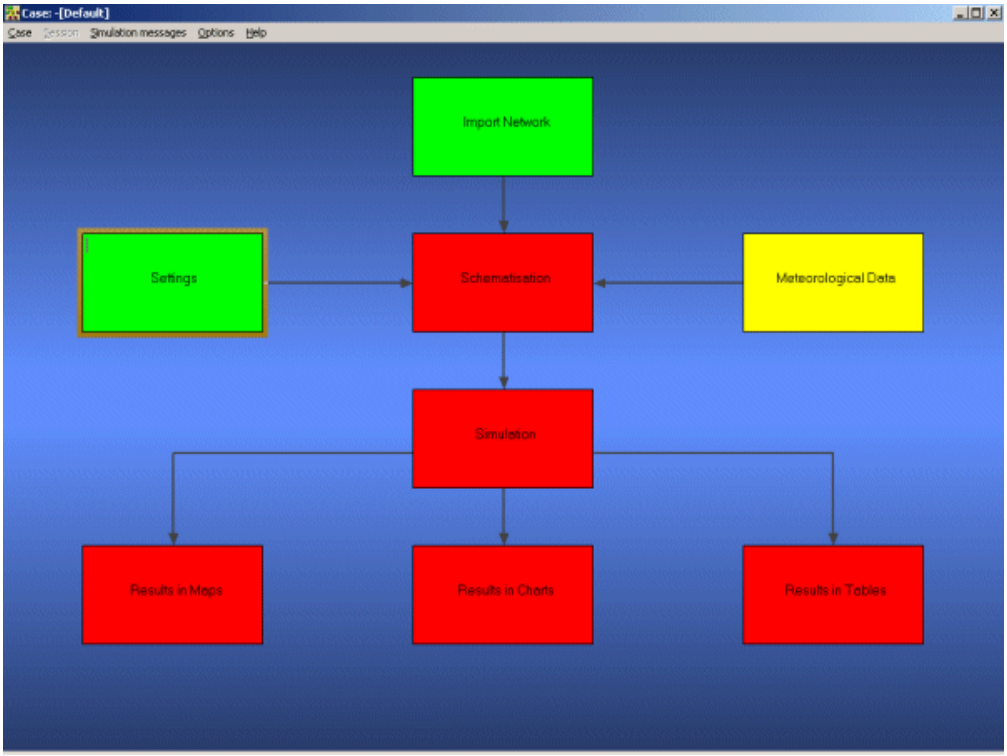
- Select the tab 'Time settings'.
- Enter the time step in the computation: 10 minutes in the 'min' edit box.
- Select the radio button 'Simulation period defined as below'.

- Enter the year of the start time of the simulation: 2006.
- Enter also the 'Month', 'Day', 'Hour', 'Min' and 'Sec' data of the start time of the simulation: 1, 1, 1, 0, 0.
- Enter the end time of the simulation: 2006, 1, 1, 10, 0, 0 in the respective edit boxes.
- Select the tab 'Output options'.
- Enter the output step: 30 minutes in the respective edit box.

Note that the simulation period data and the time step can also be changed by clicking the arrows left of the edit boxes.

- Select the tab 'Initial data'.
- Select the radio button ‘define local values in <Edit Network>’.
- Press the <OK> button to return to the 'Settings' window.
- Finally press the <OK> button, to save your settings and return to the Case Manager.

You should now see the following screen, which indicates that both the 'Settings' task and the 'Import Network' task have been completed and that the Meteorological data task should still be performed.



The case manager window after completing the 'settings' and 'import network' tasks.

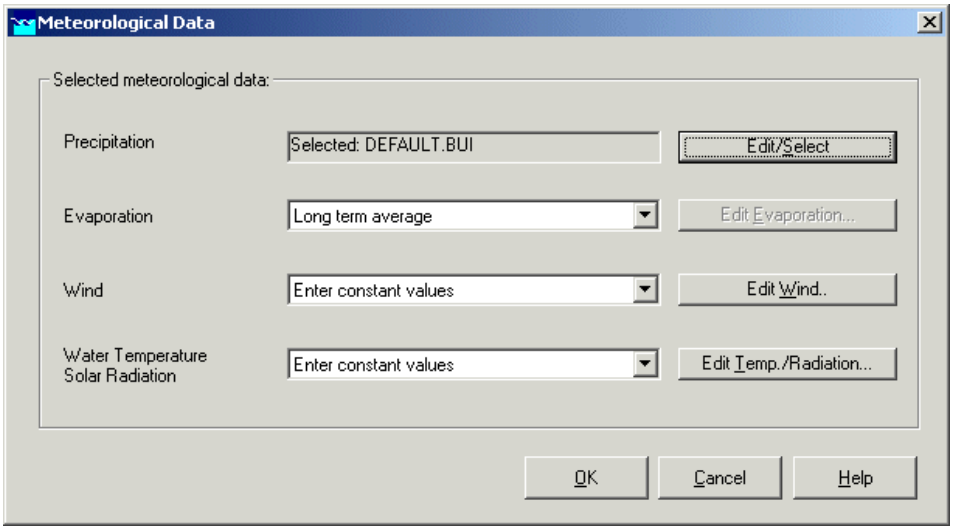
1.6 Task block 'Meteorological data'

SOBEK Rural simulations require meteorological input data, i.e. precipitation data, evaporation data and wind data. The Meteorological data task block provides precipitation and evaporation data to the RR (Rainfall-Runoff) module and wind data to the 1DFLOW and Overland Flow (2D) modules. For simplified rainfall-runoff processes precipitation data can optionally be provided to the 1DFLOW and Overland Flow (2D) modules.

As this tutorial deals with hydrodynamics, the 'Meteorological data' task block is of minor importance. We will not include wind effects nor rainfall on the channel.

- Double-click the 'Meteorological Data' task block of the Case Manager.

The following screen will appear:



The meteorological data window.

In the dialog that pops up you can see how precipitation, evaporation and wind data are defined.

- Click <OK> to leave the Meteorological data window.

Now you have finished defining the meteorological data. Notice that this task block has turned green too!

1.7 Task block 'Schematisation'

A schematisation can easily be set up with the help of the network editor. You will set up a simple schematisation.

- Double-click the 'Schematisation' task block of the Case Manager.

You can choose to edit a new or an existing model by clicking the upper <Edit Model> button.

- Click <Edit model>.

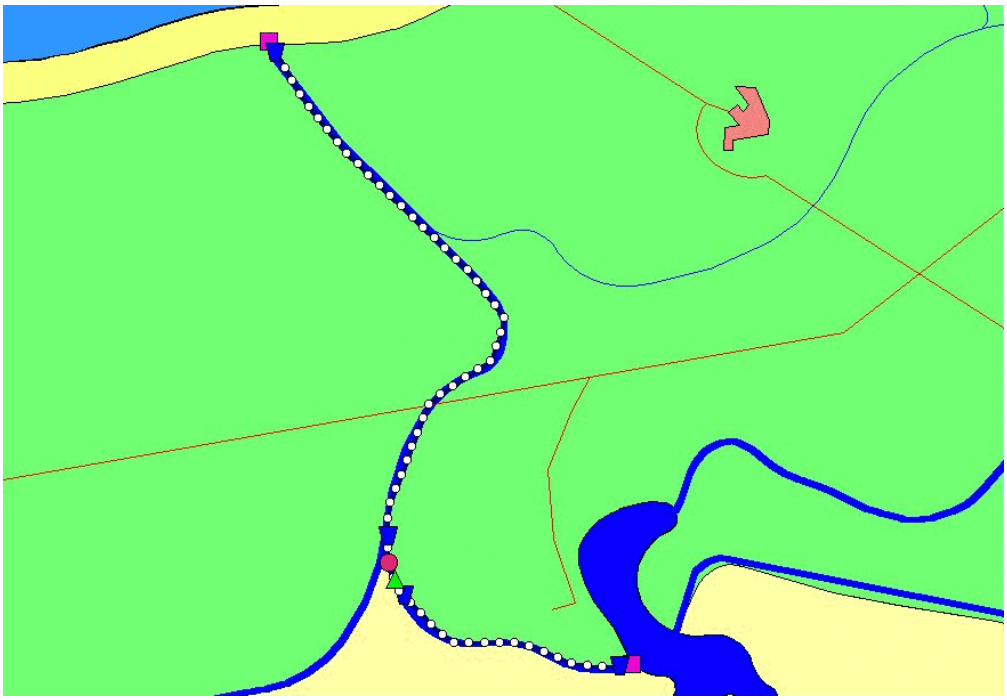
When the option <Edit Model> of the 'Schematisation' is selected, the network editor starts. The network editor is called NETTER and is a component of the Delft Hydraulics Decision Support System (Delft-DSS) tools. NETTER offers the possibility to set-up the schematisation on top of a background GIS map. NETTER also offers advanced analysis tools to show model results linked to the schematisation and provide the user with full printing facilities to make high quality prints.

Within NETTER you can do the following:

1. Interactively and graphically prepare a schematisation;
2. Generate schematisations upon GIS map Layers;
3. Carry out schematisation operations: search for a certain node, show node numbers and names, show link numbers, etc.;
4. Carry out map operations: zooming in, zooming out, (de)activating map layers, colouring of map layers, adding title information on the map, etc.;
5. View results of simulation models for schematisations created in NETTER;
6. Print maps or schematisations.

Generally speaking, NETTER has two edit modes. The first mode is the mode to set-up the schematisation, e.g. by adding new nodes. The second edit mode is the mode for editing the attribute data. In this mode you provide the attributes of the schematisation objects. For example, a pump station must have a pump capacity and switch on/off levels.

In this exercise you will work on a simple schematisation.



The schematisation to be created in this tutorial.

In order to focus on a small part of the map, you can use the zoom functionalities.

The View menu contains commands to zoom in, zoom out, centre the window, move the window and show all schematisation or map layers.
The  button allows you to zoom in on any part of the "active main window".
The  button allows you to zoom out by shrinking the displayed part of the "active main window".
The  button allows you to centre a schematisation or map GIS object. When choosing this command and then clicking on an object NETTER, redraws the map centring the chosen object to the NETTER window.
The  button allows you to shift the view by clicking the mouse anywhere in the NETTER window and dragging the view to another position.
The  button redraws the view while fitting all schematisation objects into the NETTER window.
The  button redraws the view while fitting all GIS map layers into the NETTER window.
The  button restores the view to the previous zoom state.
The  button restores the view to the state before the last 'Show Previous' command.

You are now asked to zoom in on the area between the lake and the sea.

- Select  button.
- Move the mouse pointer to the main window.
- Click and hold down the left mouse button, while dragging the pointer across the main window. The size of the rectangle determines the magnification.
- Release the left mouse button.

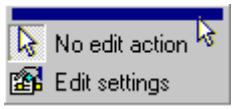
Now, you will build a simple schematisation. This schematisation consists of a small open channel with a weir. We will now build this schematisation.

- Select the  button, Edit Network, to start the edit network mode.

When you have selected the edit network mode all edit network functions and network objects for the selected module will be available.

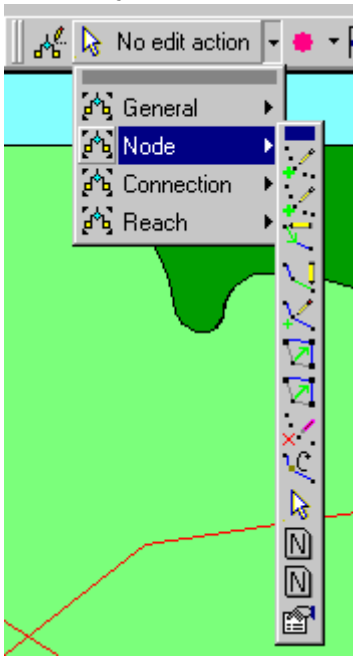
- Select  from the 

section and click the 'General' edit network functions to unveil the General functions toolbar and move it to anywhere on your screen by clicking the upper section of the selected toolbar and dragging it:



Click this section of a toolbar to drag it to your screen!

- Select  and the 'Node' edit network functions to place the Node functions toolbar anywhere on your screen.



- Select  and the 'Connection' edit network functions to place the Connection functions toolbar anywhere on your screen.
- Select  and the 'Reach' edit network functions to place the Reach functions toolbar anywhere on your screen.
- Select  from the  section and place the node objects toolbar anywhere on your screen.
- Select  from the  section to place the reach objects toolbar anywhere on your screen.

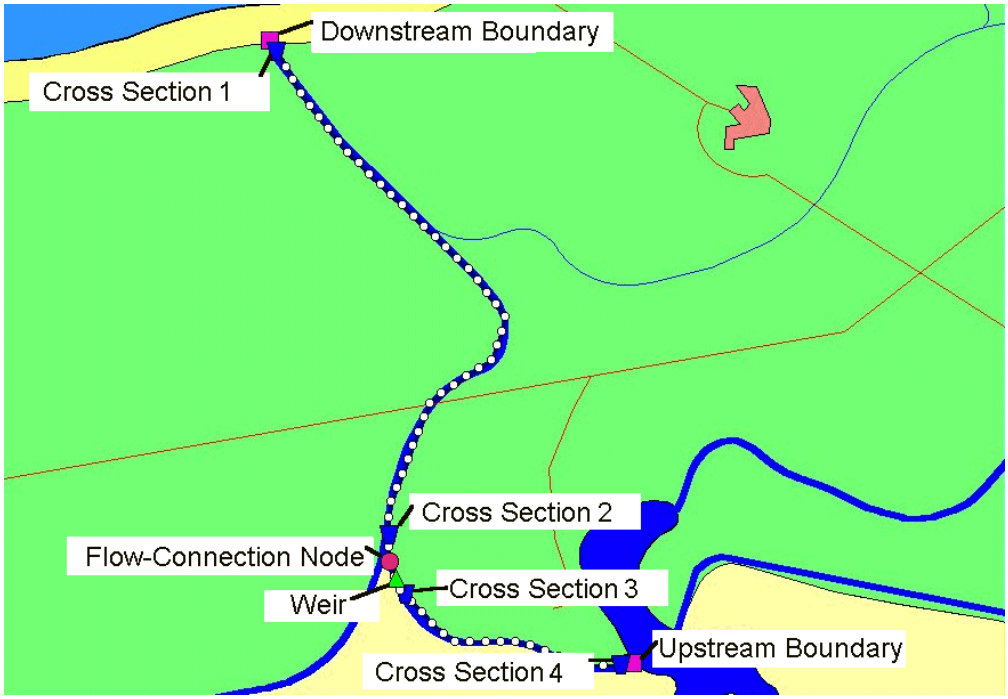
If you desire more information explaining the large amount of objects, you can customize the toolbars by clicking 'View' - 'Toolbars' - 'Customize...'. Caption only, Icon only and Icon and Caption are the available options. 'Icon only' means that you for the selected toolbar you will only see the icons. Choosing Icon and Caption will also place a label explaining each symbol.

It is possible to define the identifiers (or ID's) of the nodes and branches (links) automatically or manually.

- Select the  button in the 'General' tool bar, Edit settings, to go to the edit network options.
- Select the tab 'Node'.
- In the 'ID' group box, select the radio button 'Manual'.
- In the 'Name' group box, select the radio button 'Manual'.
- Select the tab 'Link'.
- Check if the ID and name will be set automatically.
- Click the <OK> button.

Create a schematisation:

Now you can start drawing your application. You will create a schematisation that is similar to the one below. The schematisation will involve a river section that connects a lake to the coast:



Create a network similar to this one.

- Select the  node, Flow-Boundary.
- Select the  button to select the function 'Add node'.
- Enter 'Upstream Boundary' in both input fields.
- Click the <OK> button.
- Locate the mouse at a position where you want to add the Upstream Boundary node and click the left-mouse button again to actually add the node.

In order to see the identifiers on the map please:

- Click the  button in the Active Legend or select the menu item 'Options' - 'Network Data...'.
- Select the tab 'Node'.
- Select the radio button 'Name'.
- Press the <OK> button.

- Select the  button again.
- Enter 'Downstream Boundary' in both input fields.
- Click the <OK> button.
- Add the Downstream Boundary node as before.
- Select the  node, Flow-Connection Node.
- Select the  button again.
- Enter 'Flow-Connection Node' in both input fields.
- Click the <OK> button.
- Click on the left-mouse button to actually add the Flow-Connection node on your screen.
- Select the  button, Flow-Channel.
- Select the  button to select the function 'Connect nodes'.
- Click with the left mouse button on the upstream boundary node, which you have already added and drag to the connection node while keeping the button down. Release the left mouse button.
- Click with the left mouse button on the connection node and drag to the downstream boundary node while keeping the button down. Release the left mouse button.

Now the two boundary nodes and the connection node are connected. The option ‘defined direction’ can be used to see the positive defined direction.

- Select 'Options' - 'Network Data...'.
- Select the tab 'Branch'.
- In the data group 'Show Direction', select the radio button 'Defined'.
- Click the <OK> button.

The model will need a calculation grid. Before generating this grid automatically the generation of names will be switched off.

- Select the  button in the 'General' tool bar, Edit settings, to show the edit network options window.
- Select the tab 'Node'.
- In the 'Name' data group, select the radio button 'No Names'.
- Press the <OK> button.

Several options are available to generate a grid.

- Select the  button to select the function 'Calculation grid all reaches'.
- Select the 'Split Vector' option.
- Select the node type 'Flow - Calculation Point' from the drop down list.
- Then enter '100' in the length edit box to set the calculation grid to a 100 m length.
- Click the <OK> button.

The vector layer:
To show schematizations with high performance, NETTER shows by default connections between two Flow - nodes in a straight line. However the length between the nodes may differ from the distance between the nodes in a straight line. The actual length between Flow - nodes is stored in the vector layer. You can edit the length in this layer.

- Select the  button, 'Edit Reach Vectors', to edit a selected reach vector.
- Select the reach.
- Select the  button to show the coordinates.
- Select the  button to add a coordinate.
- Click with the left mouse button on the reach to actually add a coordinate on your screen and while keeping the button down drag the new coordinate to the new location.
- Add and drag other coordinates of the selected reach.
- De-select the  button to stop adding coordinates.
- Select the other reach.
- Select the  button to add a coordinate again.
- Click with the left mouse button on the reach to actually add a coordinate on your screen and while keeping the button down drag the new coordinate to the new location.
- Add and drag other coordinates of the selected reach.
- De-select the  button, 'Edit Reach Vectors', to stop editing the reach vector.

Add reach objects:

- Select the  button in the 'General' tool bar, Edit settings, to go to the edit network options.
- Select the tab 'Node'.
- In the group box 'Name', select the radio button 'Manual'.
- Press the <OK> button.

- Select the  node, Flow - Cross Section node.
- Select the  (add node) function.
- Enter 'Cross Section 1' in both input fields.
- Click the <OK> button.
- Click on the left-mouse button to actually add the Flow-Cross Section node on your screen, near the 'Downstream boundary' node.
- Add the other three Flow-Cross Section nodes.
- Select the  node, Flow - Weir node.

- Select the  function.
- Enter 'Weir' in both input fields.
- Click the <OK> button.
- Click on the left-mouse button to actually add the Flow-Weir node on your screen.

A schematisation has been set-up. The next step is to define the attribute data of the schematisation. Therefore you have to be in 'model attribute data' mode.

Editing the boundary data:
Now, we will set the attribute data for the boundary nodes. The upstream node will be a discharge boundary, whereas the downstream node will be a water level boundary.

- Select the downstream boundary node.
- Click with your right mouse button.
- Select 'Model data' - 'Flow Model'.
- Select 'water level (h)'.
- Enter a constant value of '0'.
- Click the <OK> button.
- Select the upstream boundary node.
- Click with your right mouse button.
- Select 'Model data' - 'Flow Model'.
- Select 'flow (Q)'.
- Enter a constant value of '50'.
- Click the <OK> button.

Editing the cross section data:
For the calculation of the hydraulic conductivity, every reach needs to contain at least one Flow - Cross Section node. On this node type the bed level and profile can be defined. SOBEK offers the functionality to make re-usage of profile definitions. On each flow-cross section node you can either create a new profile definition, or select a profile definition that you had already created for another node!

- Select the downstream Flow-Cross section node *Cross Section 1*.
- Click with your right mouse button.
- Select 'Model data' - 'Flow Model'.
- Select 'Location' tab.
- Set 'Bed level' to '-2'.
- Set 'Surface level' to '3'.
- Select 'Cross section' tab.
- Select 'Trapezium' cross section type.
- Enter 'My profile 1' in 'Cross section: ' field.
- Press the <Define dimensions> button.
- Set 'Slope' to '1'.
- Set 'Bottom width' to '20'.
- Set 'Maximum flow width' to ‘26'.
- Press <Save dimensions>.
- Click the <OK> button (You changed the profile definition name, do you want to add it as a new definition?).

Now we will add friction data. Note that you can choose to submit a global value or a local value. The global value is used in all reaches for which no local value was defined. The local values are used for the entire reach on which that cross section is located. If you choose to submit local values, make sure that the "use local value for this cross section" box is checked and that the "local values" are shown in the 'Show' combo box.

- Select 'Friction' tab.
- Select 'Use local value(s) for this cross section'.
- Select 'Local value(s)' in the 'Show' combo box.
- Select 'Chezy (C)' for type friction (Bed).
- Enter '35' for constant value.
- Select 'Initial Value' tab.
- Select 'Use local initial value for this reach'.
- Enter '2' for 'Initial water level', local value.
- Enter '0' for 'Initial flow in positive direction', local value.
- Click the <OK> button.

We will add the data for the remaining Flow-cross section nodes by using the Multiple Data Editor.

cross section node	bed level	surface level	profile definition	friction	initial water level/depth
Cross Section 2 (second from downstream)	-1	4	My profile 1	local, Chèzy 35	2m water level
Cross Section 3 (third from downstream)	-1	4	My profile 1	local, Chèzy 35	3m water depth
Cross Section 4 (fourth from downstream)	5	9	My profile 1	local, Chèzy 35	3m water depth

- Select the  button, 'Select by rectangle'.
- Select the whole schematisation by dragging a rectangle around it.
- Click right mouse button.
- Select 'Model data' - 'Flow Model'.
- Select 'Flow - Cross Section'.
- Select 'Cross Section'.
- Select the column 'Cross Section'.
- Click right mouse button.
- Select 'Replace'
- Select 'My profile 1' from the list.
- Press <OK> button.
- Enter '-1' as Reference Level [m AD] of Cross Section 2.
- Enter '4' as Surface Level [m AD] of Cross Section 2.
- Enter '-1' as Reference Level [m AD] of Cross Section 3.
- Enter '4' as Surface Level [m AD] of Cross Section 3.
- Enter '5' as Reference Level [m AD] of Cross Section 4.
- Enter '9' as Surface Level [m AD] of Cross Section 4.

All input data of the Multiple Data Editor can be viewed in Graphs.

- Select the column 'Surface Level [m AD].
- Click right mouse button.
- Select 'Graph'.
- Close the graph.

The data may also be shown on the map.

- Select the column 'Reference Level [m AD].
- Click right mouse button again.
- Select 'Show on Map'.
- Click 'Cross Section' of 'Model data' in the Active Legend.
- Click on the  button in the Active Legend or select the menu item 'Options' - 'Network Data...'.
- Select the tab 'Node'.
- Select the radio button 'Data value'.
- Press the <OK> button.
- After viewing the data on the map, click the  button in the Active Legend.
- Select the tab 'Node'.
- Select the radio button 'Name'.
- Press the <OK> button.

Now we will add the friction data.

- In the Multiple Data Editor, select 'Friction'.
- Press the <Yes> button (Do you want to save the data?).
- Select 'Chezy'.
- Enter '35'.

And last but not least we will add the initial values.

- Select 'Initial values'.
- Press the <Yes> button (Do you want to save the data?).

To get a good insight which objects you edit in the Multiple Data Editor, SOBEK offers the functionality to mark these objects.

- In the column 'Reach ID' of the Multiple Data Editor, select the reaches and view on the map which reach has been marked until the upstream reach has been selected.
- Select 'Depth' as the type of the initial level for the upstream reach.
- Enter '3'.
- Select 'File' - 'Exit' of the Multiple Data Editor.
- Press the <Yes> button (Do you want to save the data?).

Editing the weir data:

- Select the Flow-Weir node.
- Click right mouse button.

- Select 'Model data' - 'Flow Model'.
- Enter ‘20’ for the flow-weir's crest width.
- Keep the weir's crest level at it's default value of 0.7 m.
- Click the <OK> button.

Saving the network and the model:
The schematisation has been setup. Now we will save the map settings, the schematisation, leave the task block 'Schematisation' and save the case.

- Select 'File' - 'Save' - 'Map'.
- Select  button in NETTER.
- Select the menu item ‘File’-’Exit’, to leave NETTER.
- Click the <OK> button in the schematisation window.

Now only your schematisation has been saved in NETTER. The whole case must be saved too!

- Select the menu item ‘Case’-’Save As’.
- Enter the name ‘Case_one’ to save the case.
- Click the <OK> button.

1.8 Task block 'Simulation'

The next step in the modelling process is to perform the calculations.

- Double-click the task block 'Simulation'.

You will see a window appearing, showing a simulation status bar. After the simulation has successfully finished, this window will disappear again, and the 'Simulation' task block in the case manager will turn green.

1.9 Task block 'Results in Maps'

Results in maps gives you a clear impression of the results in time. The program NETTER is used in this task block. Since NETTER also is used to set up a schematisation, it will be easy for you, being an experienced user now, to view the results.

- Double-click 'Results in Maps' task block to analyse the results.

Plotting the water levels.

- Select 'Results at nodes' in the Active Legend.
- Select a node.
- Click the right mouse button.
- Select 'Show Graph'.
- Select another node.
- Click the right mouse button.
- Select 'Show Graph'.

The Graph Server window will show the data plot of the two selected nodes.

- Select 'File' - 'Exit' to close the Graph Server.

To get a quick overview of the results data, you can make the nodes change colour and size, according to their data value.

- Choose 'Options' - 'Network data' in the menu bar.
- Click the 'All Data' tab.
- Activate the 'width' checkbox for 'Show node data'.
- Click the <OK> button.
- Use the  buttons to animate the change of water levels in the model.

Plotting discharges
Notice that water levels are calculated at nodes, whereas discharges are calculated at reach segments. To plot discharges in a graph you will therefore have to select reach segments:

- Select 'Results at reach segments' in the Active Legend.
- Select a reach segment.
- Click the right mouse button.
- Select 'Show Graph'. Note: if both the 'Show Graph' option and the  button in the View Data window do not appear, you've probably selected a *node*, in stead of a *reach segment*. Zoom in a little to make sure you select a reach segment.

Now analyse your results!

Animating the flows:
The direction of the flows through the model can be animated on the map:

- Make sure that you selected the 'Results at reach segments' from the Active Legend.
- Zoom in on a small part of the water system.
- Choose 'Options' - 'Network data' in the menu bar.

- Click on the tab 'Branch'.
- Select 'All data' of the 'Show Direction' topic.
- Select the 'Arrow flow' box.
- Click <OK> button.
- Use the  buttons from the 'View data' window to animate the flows.

Note: If you do not see arrows moving trough the network, the arrows might be too small compared to the reach thickness. You can enlarge them by changing the size of the arrows, after selecting 'Options' - 'Network options...' and pressing the button <Links...>.

User Defined Output

To reduce the number of user actions SOBEK offers the User Defined Output options in NETTER.

- Select 'Tools' - 'Output options'.
- Select 'Flow module' from the 'Module' list.
- Select 'water level' from the 'Output title' list.
- Select 'maximum' from the 'Function' list.
- Press <Add> button.
- Select 'Flow module' from the list again.
- Select 'water depth' from the list.
- Select 'minimum' from the list.
- Press the <OK> button.

Now with only one user action the maximum water levels can be viewed on the map.

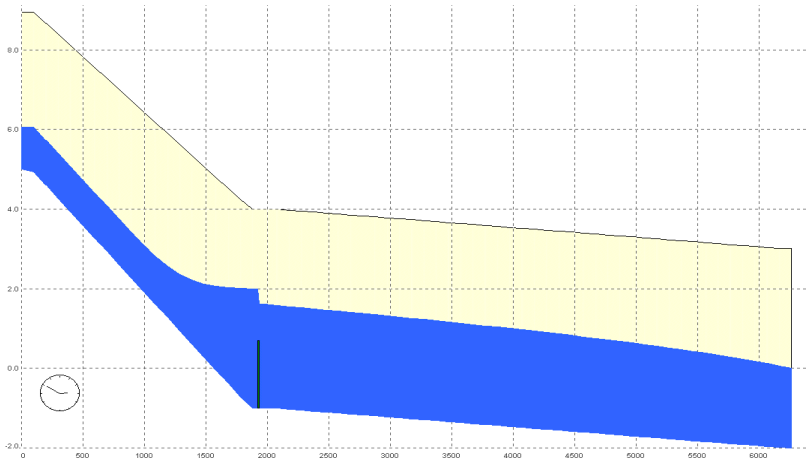
- Select 'water depth, minimum' in the Active Legend under User Defined Output.

The data is also available by selecting 'Results at nodes' in the Active Legend, in the 'View Data' window, selecting 'Waterdepth [m]' from the list, and in the menu bar, selecting 'Options' - 'Data Statistics' - 'Minimum'.

Creating a Side view animation

- Select the upstream Flow - Boundary node while holding down the <SHIFT> key. Keep the <SHIFT> key pressed. Then click the downstream Flow-Boundary node. Now, you can release the <SHIFT> key.
- Click the right mouse button (on the selection) and select ‘Side view’.
- Press the <OK> button of the Set up animation.
- Click  to watch the animation.

The animation will look like this:



An example of a side view animation.

In the side view the different network objects can easily be distinguished: the image depicts the surface level (upper line), bed level (bottom line) and structures (thick vertical line). Various options are available to plot object labels in the side view.

You can also select objects in Side view and view the input and output data of a selected object.

- Select the Flow - Weir structure in Side view.
- Click the right mouse button.
- Select 'Show Info'.
- Press <Close viewer>.
- Click the right mouse button again.
- Select 'Show Graph'.
- Multiple select 'Discharge [m3/s]' - 'Waterlevel up [m AD]' and 'Waterlevel down [m AD]'.
- Press <OK> button.
- Select 'File' - 'Exit' of the Graph Server to close the graph.

You can also load other results to be viewed in Side view.

- Select 'View' - 'User Data' - 'Load other results..'.
- Select 'Results at reach segments'.
- Press the <OK> button.
- Select 'Velocity [m/s]'.
- Press <OK> button.
- Click  to watch the animation.

Note that you may load any other results available for the selected objects. For example water quality results, if available.

- Select 'File' - 'Exit' in the side view window [SOBEK Side view].
- Select 'File' - 'Exit' to close NETTER.

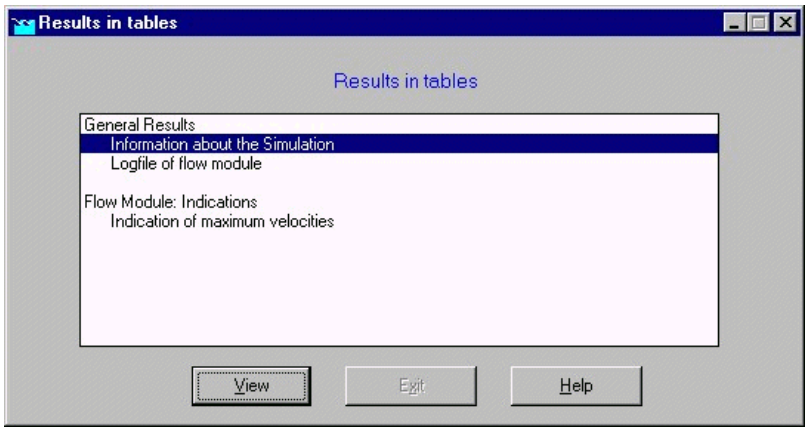
Do not forget to save the case!

- Select the menu item ‘Case’-’Save’.

1.10 Task block 'Results in Tables'

The 'Results in tables' task block provides detailed reports about the simulation.

- Double-click 'Results in tables' task block.



The results in tables window.

- Select ‘Information about the Simulation’;
- Select 'View' and view the results. Important information regarding the water balance of your computation and the total balance error are given in this file (amongst others);
- Select 'File' - 'Exit' from the 'View' window;
- Click the <Exit> button in the 'Results in tables' window.

1.11 Task block 'Results in Charts'

In the task block 'Results in Charts' the user can easily depict result data in one graph.

- Double-click on the 'Results in Charts' task block.
- Select 'Water Balance'.
- Click the <View> button.
- Select the parameter 'Volume (1000 m^3)'.
- Press the <All> button of Locations.
- Press the <All> button of Timesteps.
- Press the <Graph> button.
- Select 'File' - 'Exit' to close the 'Graph Server' window.
- Press the <Exit> button to close ODS_VIEW.
- Press the <Exit> button to close the task block 'Results in Charts'.

1.12 Linkage node

To easily add branches to a main drainage system SOBEK offers the so-called Flow - Linkage node functionality. To show this functionality we will add a torrent to the existing schematisation.

- Double-click the 'Schematisation' task block of the Case Manager.
- Click the <Edit model> button.

The Tutorial map folder contains a map layer of the torrent.

- Select 'Map' in the Active Legend.
- Select 'Torrent' from the list of layers.
- Click on the  button to move the selected layer to the bottom of the list. Now this layer will be on top.
- Press <OK> button.

Now we will add the Flow - Linkage node.

- Select the  button, Edit Network, to start the edit network mode.
- Select the  button, Flow - Linkage node.
- Select the  button to select the function ‘Add node’.
- Enter 'Linkage node' in both input fields.
- Click the <OK> button.

- Click on the left-mouse button again to actually add the Flow - Linkage node on your screen.
- Select the  node, Flow-Boundary, by clicking on it.
- Select the  button to select the function ‘Add node’.
- Enter 'Upstream Torrent' in both input fields.
- Click the <OK> button.
- Click on the left-mouse button again to actually add the Upstream Torrent node on your screen.
- Zoom in to get a good view on the Flow - Linkage node and the Flow - Boundary node.

- Select the  button, Flow-Channel, by clicking on it.
- Select the  button to select the function ‘Connect nodes’.
- Click with the left mouse button on the upstream torrent node, which you have already added and drag to the Flow - Linkage node while keeping the button down. Release the left mouse button.
- Select the  button, 'Edit Reach Vectors'.
- Edit the vector layer to introduce the correct distances.
- Select the  button in the 'General' tool bar, Edit settings, to show the edit network options window.
- Select the tab 'Node'.
- In the 'Name' data group, select the radio button 'No Names'.
- Press the <OK> button.
- Select the  button to select the function 'Calculation grid all reaches'.

- Click the <OK> button.

SOBEK offers a powerful tool to validate your network.

- Select 'Tools' - 'Validate network by model' - 'Flow Model'.

The following message appears in the Network Validation window: "A Flow reach must contain a Profile node".

- Select the message in the Network Validation window. Now the torrent reach will be selected on the map.
- Press the <Finished> button.
- Select the  button in the 'General' tool bar, Edit settings, to show the edit network options window.
- Select the tab 'Node'.
- In the 'Name' data group, select the radio button 'Manual'.
- Press the <OK> button.
- Select the  node, Flow - Cross Section node.
- Select the  (add node) function.
- Enter 'Cross Section Torrent1' in both input fields.
- Click the <OK> button.
- Click on the left-mouse button to actually add the Flow-Cross Section node on your screen, near to the Upstream torrent boundary.
- Select the  (add node) function.
- Enter 'Cross Section Torrent2' in both input fields.
- Click the <OK> button.
- Click on the left-mouse button to actually add the Flow-Cross Section node on your screen, near to the Linkage Node.

- Select the  button, Select by rectangle, by clicking on it.
- Select the part of the schematisation which includes the new Cross Section nodes by clicking on the map and drag while keeping the button down.
- Release the left mouse button.
- Click right mouse button.
- Select 'Model data' - 'Flow Model'.
- Select 'Flow - Cross Section'.
- Select 'Cross Section'.
- Select the column 'Cross Section'.
- Click right mouse button.
- Select 'Replace'
- Select 'My profile 1' from the list.
- Press <OK> button.
- Enter '15' as Reference Level [m AD] of Cross Section Torrent1.
- Enter '20' as Surface Level [m AD] of Cross Section Torrent1.
- Enter '0' as Reference Level [m AD] of Cross Section Torrent2.

- Enter '2' as Surface Level [m AD] of Cross Section Torrent2.

Now we will add the friction data.

- Select 'Friction'.
- Press <Yes> button.
- Select 'Chezy'.
- Enter '35'.

And last but not least we will add the initial values.

- Select 'Initial values'.
- Press <Yes> button.
- Select 'Depth' as the type of the initial level for the upstream reach. Note that the current reach will be selected in NETTER.
- Enter '1.25'.
- Select 'File' - 'Exit' of the Multiple Data Editor.
- Press <Yes> to save the data.

- Select the 'Upstream Torrent' node on the map.
- Click right mouse button.
- Select 'Model data' - 'Flow Model'.
- Select the radio button 'flow (Q)'.
- Enter the value '10'.
- Press the <OK> button.

Saving the network and the model:
The schematisation has been setup. Now we will save the map settings, the schematisation, leave the task block 'Schematisation' and save the case.

- Select 'File' - 'Save' - 'Map'.
- Select  button in NETTER.
- Select the menu item ‘File’-’Exit’, to leave NETTER.
- Click the <OK> button in the schematisation window.

Now only your schematisation has been saved in NETTER. The whole case must be saved too!

- Select the menu item ‘Case’-’Save As’
- Enter the name ‘Case_two’ to save the case.
- Click the <OK> button.

Please run the simulation and view the results. Please use the Side view option to view the Linkage node functionality.

1.13 Epilogue

In this tutorial the most important aspects of working with SOBEK have been discussed. Extended documentation can be found in the SOBEK online help. The online help can be found next to the SOBEK start icon. Since you have gained experience now it's not that difficult to find out other options and possibilities of SOBEK which not have been discussed here. Good luck!