Tutorial Hydrodynamics in sewers

SOBEK-Urban 1DFLOW + RR modules

September 2007

Tutorial Hydrodynamics in sewers

SOBEK-Urban 1DFLOW + RR modules

September 2007

Contents

1   Tutorial Hydrodynamics in sewers

1.1   General

In this tutorial the basic principles of working with the SOBEK-Urban 1DFLOW module and RR (Rainfall-Runoff) module are explained step by step and you will be guided to set-up a simple network on your own and to extend this network. This tutorial will only show a limited number of the large number of options. It will teach the basic principles of working with SOBEK-Urban 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 schematisation, using the Dutch Standard Exchange Format (Suf-HYD);
2. analyzing results;
3. extending the schematisation.

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 on the Windows <Start> button.
- Select the 'Programs' menu.
- Select the 'Delft Hydraulics' menu.
- Select the 'SOBEK' menu item (SOBEK210).
- Click on the 'SOBEK' icon.
- 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_SEWER'.

  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_SEWER'. 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 trough the case manager.
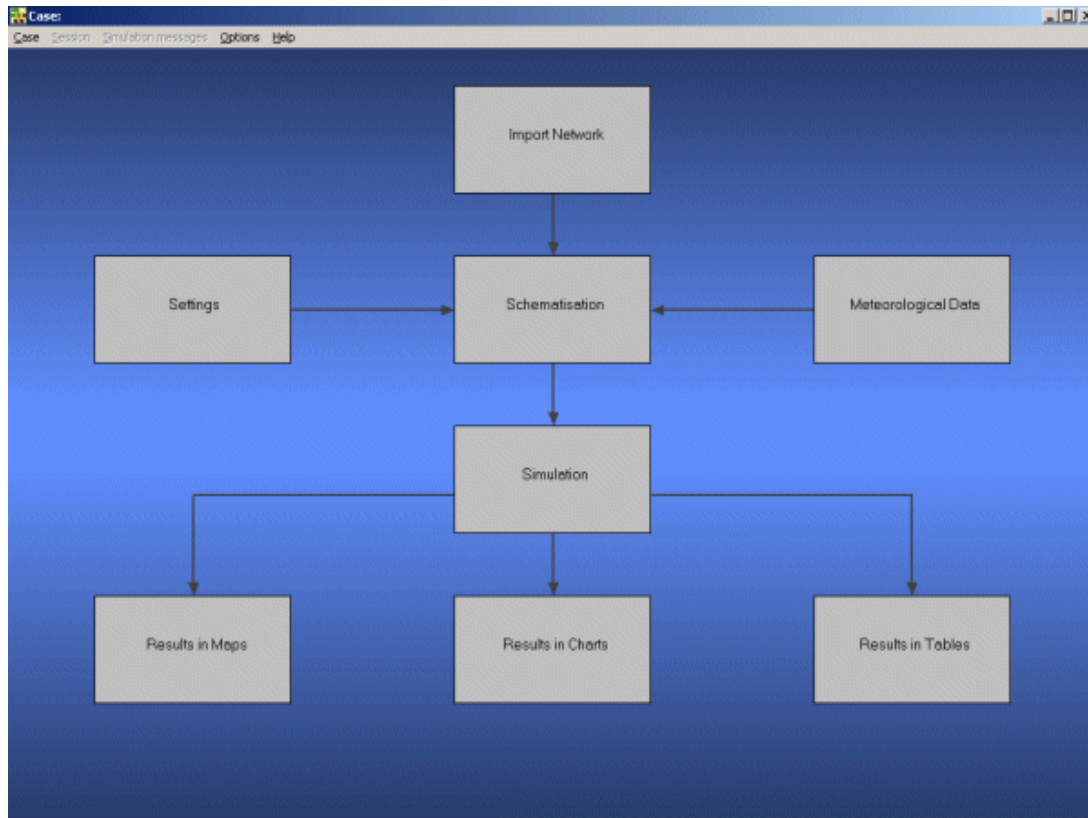
The case manager screen.

On the screen a number of blocks are visible:

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, or a set of linked models, or 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:

1. administration of cases (which data are related to which cases);
2. checking whether the model calculations for the cases are performed in the predefined order;
3. logging the actions of the Case Manager (including view and print);
4. 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 click on 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:

1. yellow: the task can be executed;
2. green: the task has been executed at least once and can be executed again;
3. 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.
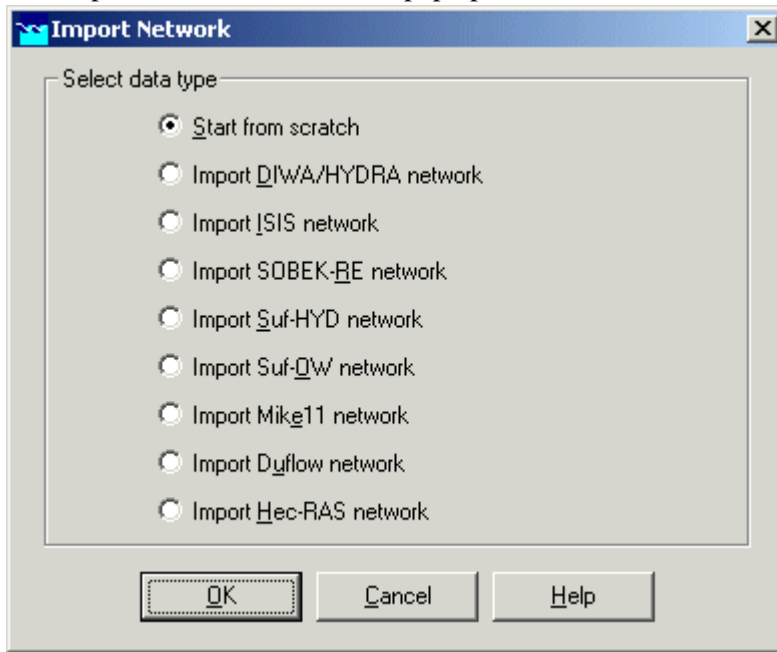Now, we will discuss each task block.

1.4   Task block 'Import network'

The color of this task block is yellow, so this means that this task block must be executed.

- Execute the task block 'Import Network' by double-clicking.

    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 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 import a SUF-HYD network. SUF_HYD is the Dutch standard exchange file format for data regarding sewer systems.

- Select the radio button 'Import Suf-HYD network' by pointing at the corresponding radio button with the mouse pointer and by clicking the left mouse button.
- Press <OK> button.
- Press <OK> button.
- Select the file TUTORIAL.HYD.
- Press <Open> button.
- Press <Yes> button. View the messages in the log file that is shown.
- Select 'File' - 'Exit' to close the file.

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 may be disabled (grey), and some may be enabled.
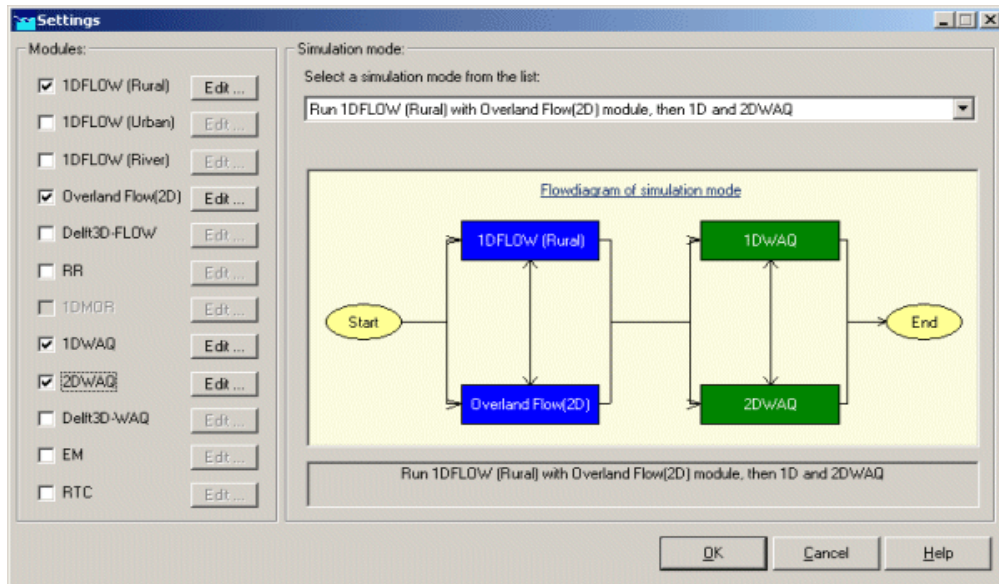
The settings window.

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.
- Unselect all the selected modules if any.
- Select the 'RR' module.
- And select the '1DFLOW (Urban)' module.
- Press the <Edit...> button of 'RR'.
- Enter the time step in computation: 1 minute.
- Select the radio button 'Simulation period will be derived from meteorological data' by pointing at the corresponding radio button with the mouse pointer and clicking the left mouse button.
- Select the 'Output options' tab.
- Enter the output time step in computation: 1 minute.
- Press the <OK> button to return to the 'Settings' window.

- Press the <Edit...> button of '1DFLOW (Urban)'.
- Enter the time step in computation: 1 minute.
- Click on the tab 'Initial data' with the left mouse button.
- Select the radio button 'initial depth in channels [m]'.
- Enter the value 0 .
- Select the 'Output options' tab.
- Enter the output time step in computation: 1 minute.
- Select the tab 'Branches' of 'Output parameters'.
- Select 'water level gradient'.
- 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 screen depicted below, indicating that both the 'Settings' task and the 'Import Network' task have been completed and that the Meteorological data task should still be performed.
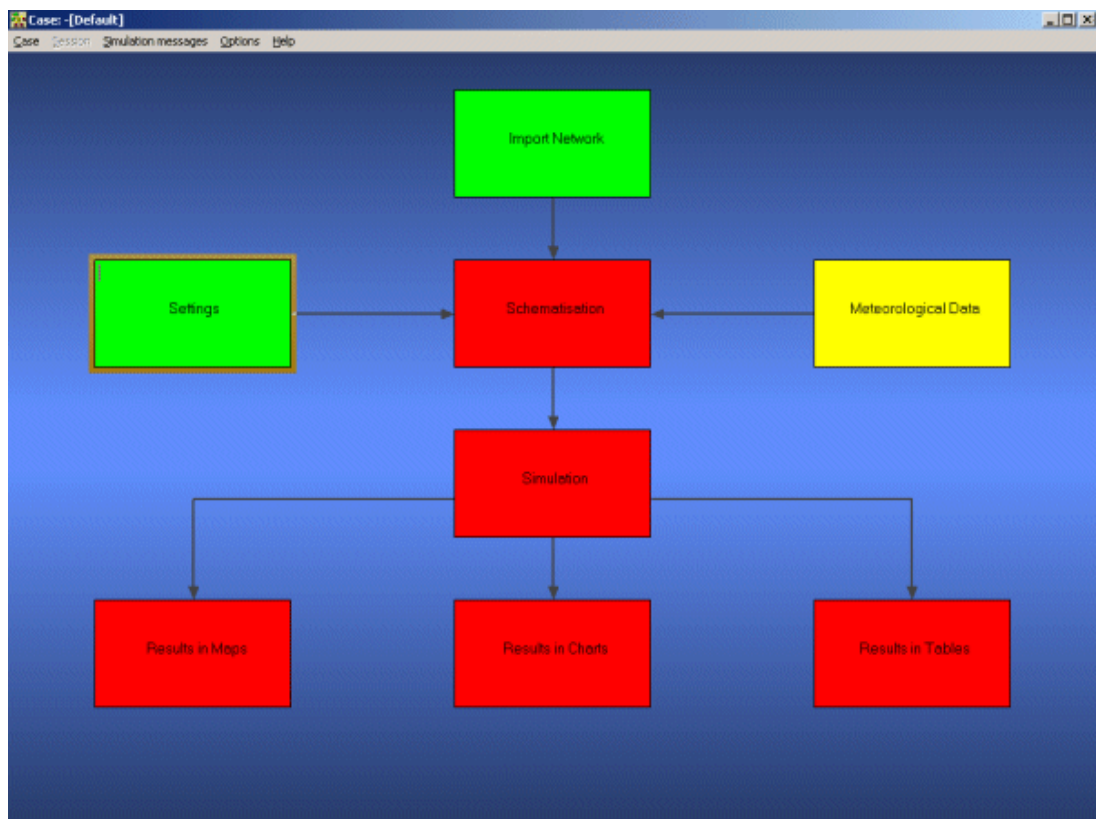
The case manager after completing the 'settings' and 'import network' tasks.

1.6   Task block 'Meteorological data'

SOBEK-Urban simulations require meteorological input data, i.e. precipitation data and evaporation data.
The evaporation data are connected with the time series of the precipitation data. The simulation period is determined by the start- and end date of the precipitation data.

- Double-click the 'Meteorological Data' task block of the Case Manager. The following screen will appear:
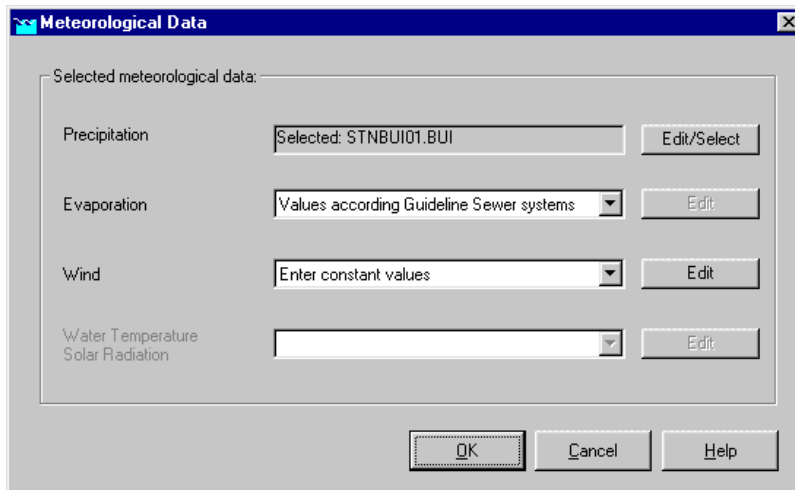
The meteorological data window.

Now you can select and edit the precipitation data and evaporation data.

- Click the <Edit/Select> button next to the Precipitation box.
- Press the <Select event...> button.
- Select the rainfall event named 'STNBUI07.BUI'.
- Click the <OK> button.
- Click <OK> again to leave the 'edit precipitation data' window.
- Select 'Values according Guideline Sewer systems' for evaporation.
- Click the <OK> button to leave the Meteorological Data window.

Now you're ready defining the meteorological data. Notice that this task block has turned green too!

1.7   Task block 'Schematisation'

- Double-click the 'Schematisation' task block of the Case Manager.
- 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 attached 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 of 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 on one or more pages which fit together.

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 give attributes to 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.
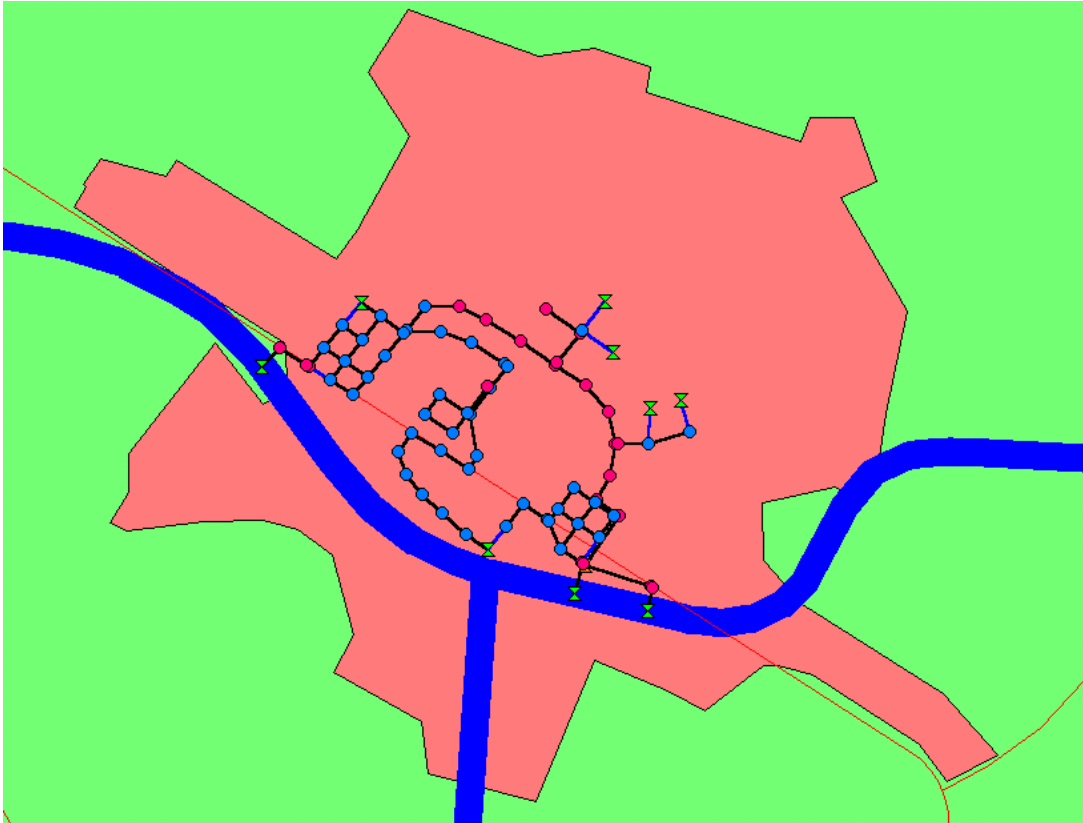
The schematisation to be extended in this tutorial.

Several zoom options are available. 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 center a schematisation or map GIS object. When choosing this command and then clicking with the left mouse button on an object, NETTER redraws the map centering 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 fitting all schematisation objects into the NETTER window.

The button redraws the view fitting all GIS map layers into the NETTER window.

The button restores the view of the map before the last zoom command was given.

The button restores the view of the map before the last 'Show Previous' command was given.

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

Now you will zoom in to the city.

- Select [🔍] button.
- Move the mouse pointer to the main window.
- Click and hold the left mouse button, make a rectangle by dragging the pointer across the main window. The size of this rectangle determines the magnification.
- Release the left mouse button.

The schematisation, showing a part of the city's sewer system, including attribute data, was directly imported from a Suf-HYD file (standard file format used for sewer network data). We will view and adjust the schematisation later on. Now we will save the schematisation; leave the task block 'Schematisation'; save the case and go to the task block 'Simulation'.

- Select 'File' - 'Save' - 'Map'.

- Select the [💾] button, to save the **network**.

To get a good insight of the schematisation it is possible to show all the input data on the map or in graphs.

- Select the [▫] button, Select by rectangle, by clicking on it.
- Select the whole schematisation by clicking on the map and drag while keeping the button down. Release the left mouse button.
- Click right mouse button.
- Select 'Model data' - 'Rainfall Runoff Model'.
- Select 'Flow - Manhole with Runoff'.
- Select 'Sewerage Runoff'.
- Select the column 'Open Sloped area [m2]'.
- Click the right mouse button.
- Select 'Show on Map'.
- Select 'Sewerage Runoff' in the Active Legend.
- Click on the [⬚] button in the Active Legend or select the menu item 'Options' - 'Network Data...'.
- Select the 'All Data' tab.
- Select 'Width' of 'Show branch data'.
- Select 'Width' of 'Show node data'.
- Press the <OK> button.

The following figure appears.
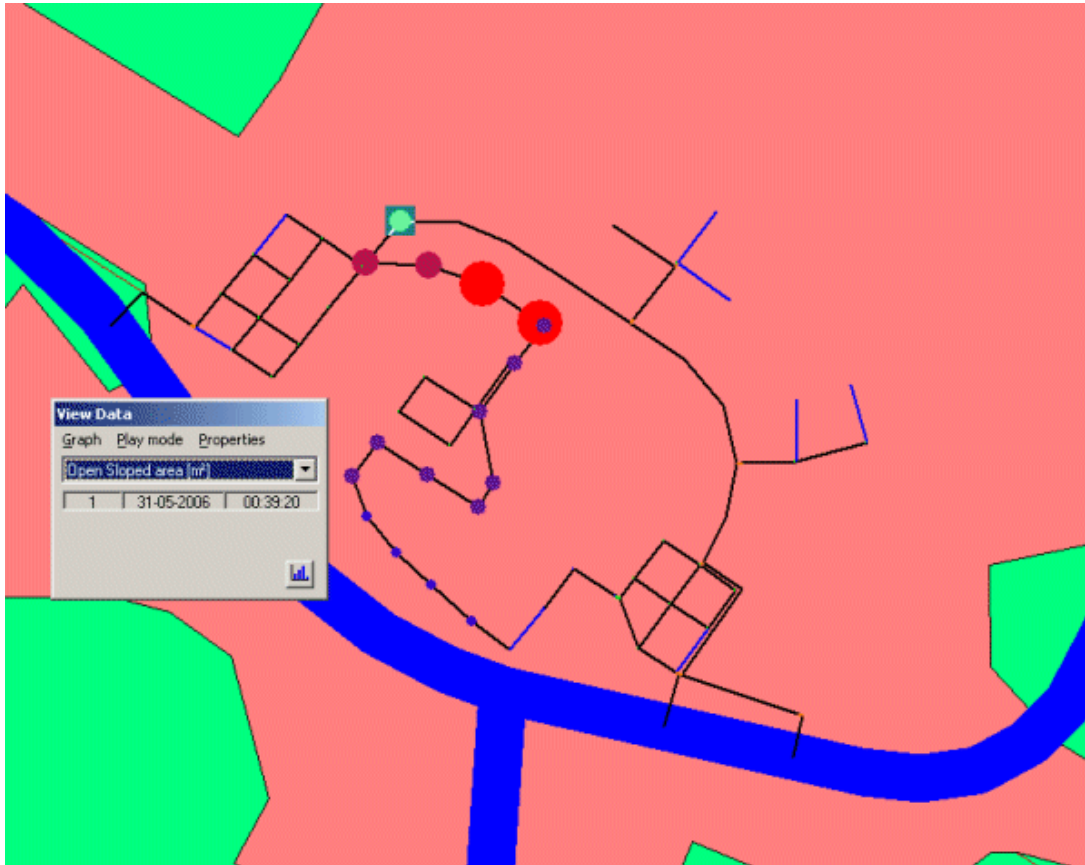
In this way all input data available in the Multiple Data Editor can be viewed on the map
with colour and size of the objects according to their data value!

Another way to get good insight in the schematisation is make a side view.

- Click on the ▦ button in the Active Legend.
- Select the 'Node' tab.
- Select 'ID'.
- Press the <OK> button.

- Click simultaneously the <SHIFT> key on the keyboard and select the Flow - External
  Weir with the ID 0-10003 with the left mouse button. Keep the <SHIFT> key pressed.
  Then click the Flow - External Weir with the ID 0-8002. Now, you can release the
  <SHIFT> key. (Note: in older versions of SOBEK,the <Ctrl> key should be used)
- Click the right mouse button (on the selection).
- Select 'Side view'.

In the side view the different network objects can easily be distinguished: the image depicts
the surface level (upper line), pipe diameter (middle and bottom line), manholes (vertical
lines), structures (thick vertical line). Various options are available to plot object labels in
the side view.
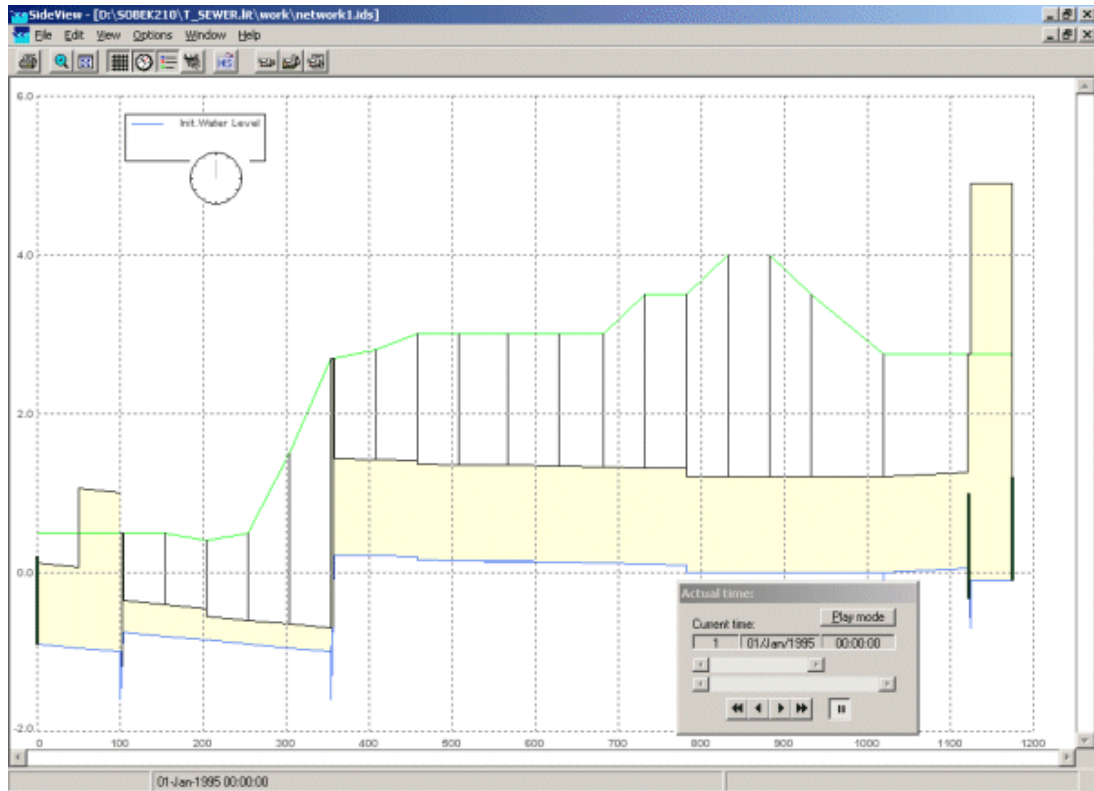
You may re-use this side view path.

- Select 'File' - 'Save'.
- Enter the file name 'Path1.ids'.
- Press the <Save> button.
- Select 'File' - 'Exit'.

- Click on the ⬛ button in the Active Legend.
- Select the 'Node' tab.
- Select 'None'.
- Press the <OK> button.

And last but not least you can make a storage graph.

- Select 'Tools' - 'Storage Graph'.
- Select 'whole network'.
- Select 'use storage in nodes'.
- Press the <Process> button.

The following figure appears.
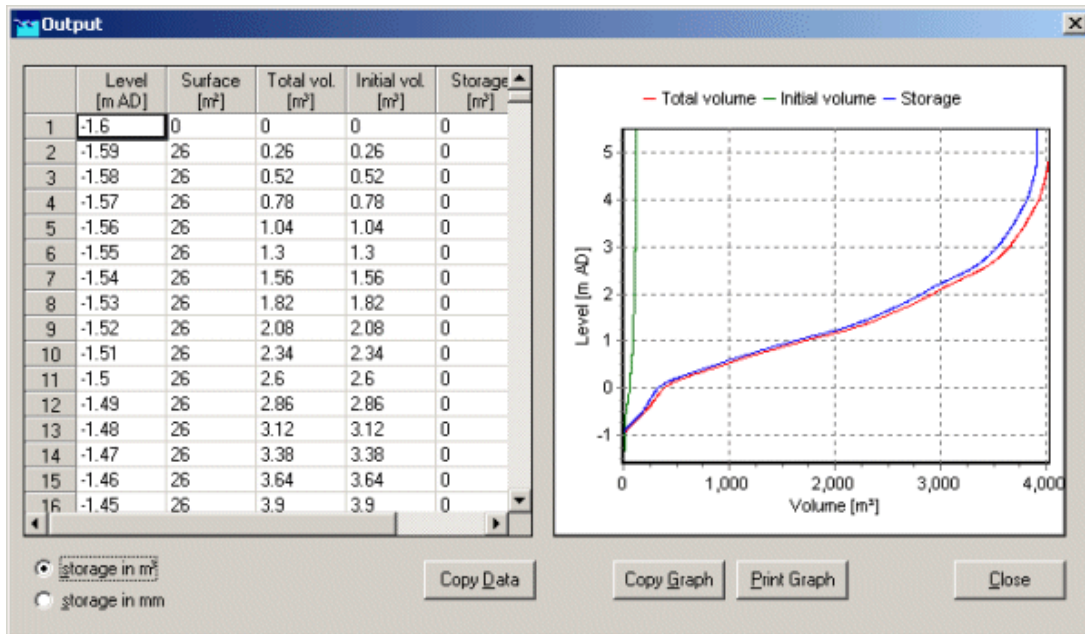
- Click the <Close> button.
- Press the <Exit> button.

To do the simulation you have to leave NETTER.

- Select the menu item 'File'-'Exit', to leave NETTER.
- Click the <OK> button.

- 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 bar appearing, showing the progression from the simulation. After the simulation has finished, the Results in Maps, Results in Charts and Results in Tables task blocks will become yellow.

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.

Creating a side view animation

- Select 'Tools' - 'Side view'.
- Select the file 'Path1.ids'.
- Press the <Open> button.
- Press the <OK> button of the Set up Animation.
- Click ▶ to watch the animation;

If you would like to add other side views in the same SideView application, just select another path on the map, click the right mouse button, select 'Side view' and press the <OK> button.

- Select 'File' - 'Exit'.

User Defined Output

- Select 'Tools' - 'Output options'.
- Select 'global SOBEK' from the list button 'Use definition as defined in: '.
- Press the <Add> button a few times until the total number of rows is 4.
- Fill in the data as shown in the figure below.
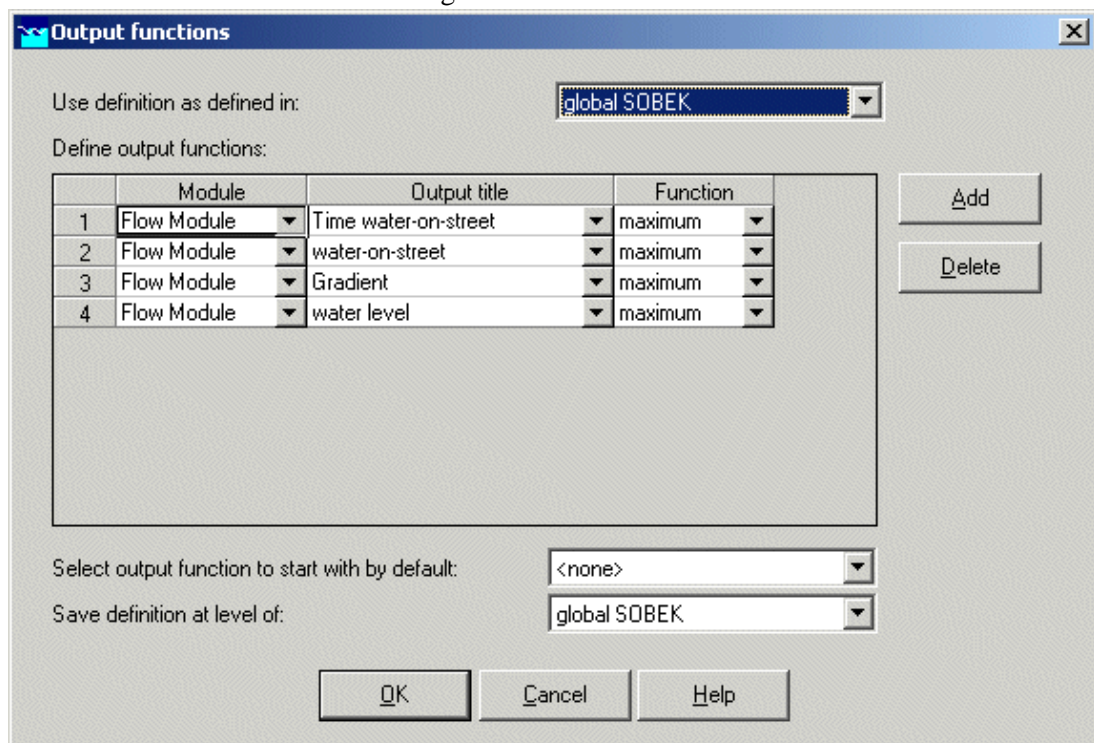
- Press the <OK> button.
- In 'User Defined Output' of the Active Legend, select 'Time water-on-street, maximum'.

The following figure appears.

Plotting water levels

- Select the item 'Results at nodes' in the Active Legend.
- Select several nodes by using simultaneously the <CTRL> key on the keyboard.
- In the 'View Data' window, select the item 'Waterlevel [m AD]'.
- Click the ▙ button on the 'View Data' window.
- Select 'File' - 'Exit' of the 'Graph Server' window.

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 pipe.
- In the 'View Data' window, select the item 'Discharge [m^3/s]'.
- Click the ⬛ button on the 'View Data' window.
- Select 'File' - 'Exit' of the 'Graph Server' window.

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'.
- Select 'Options' - 'Network Data...'.
- Select the 'Branch' tab.
- Under the 'Show Direction' topic select 'All Data'.
- Click the 'arrow flow' box.
- Select the 'All Data' tab.
- Un-select the 'Width' of 'Show branch data'.
- Click the <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...>.

- Select 'File' - 'Exit' to leave NETTER.

1.10 Task block 'Results in Tables'

The 'Results in tables' task block provides detailed reports about the simulation and the input and the output data.

- Double-click the 'Results in Tables' task block.
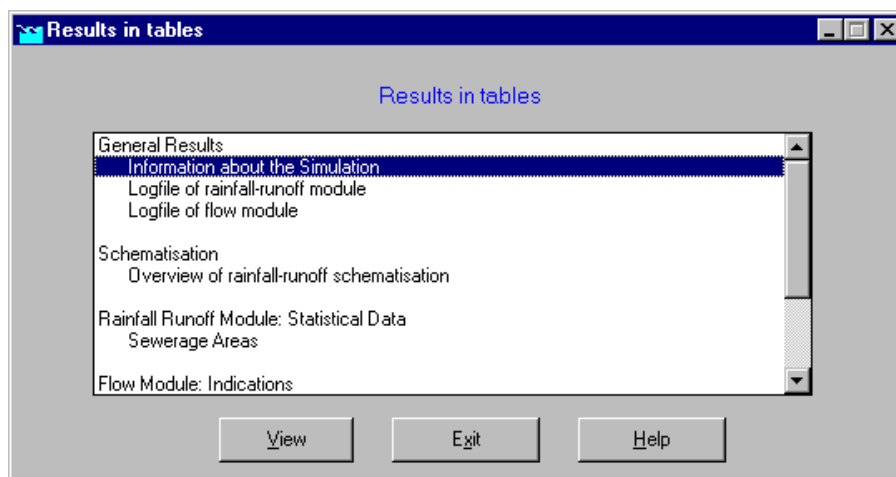
The results in tables window.

- Select 'Information about the Simulation'.
- Select '<u>V</u>iew' 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 '<u>F</u>ile' - 'E<u>x</u>it'.
- Click the <E<u>x</u>it> button in the 'Results in tables' window.
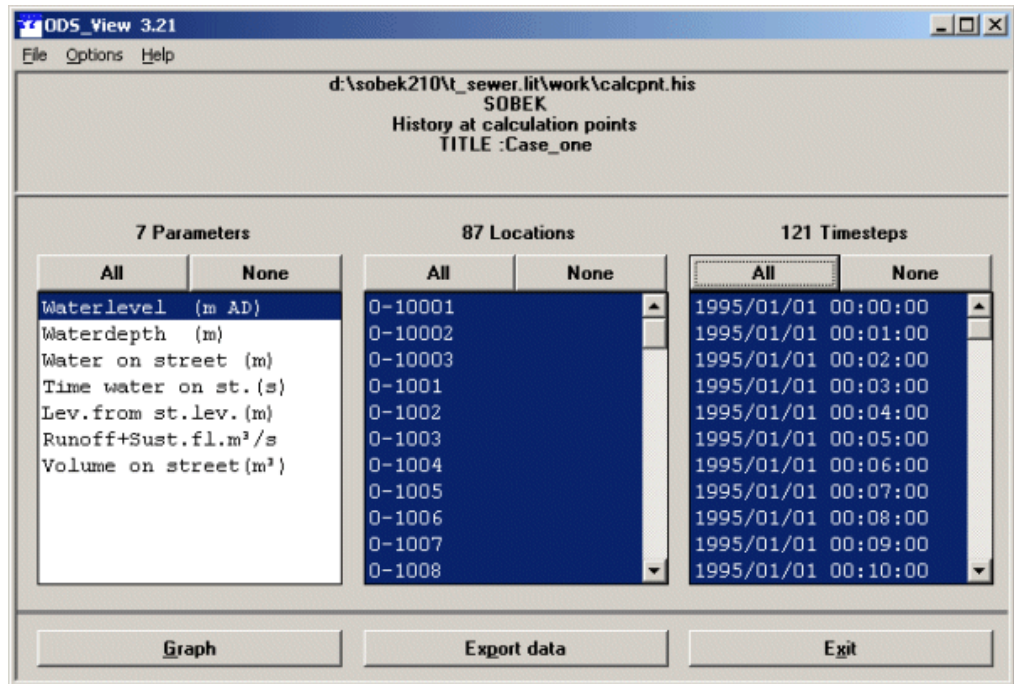
1.11 Task block 'Results in Charts'

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.
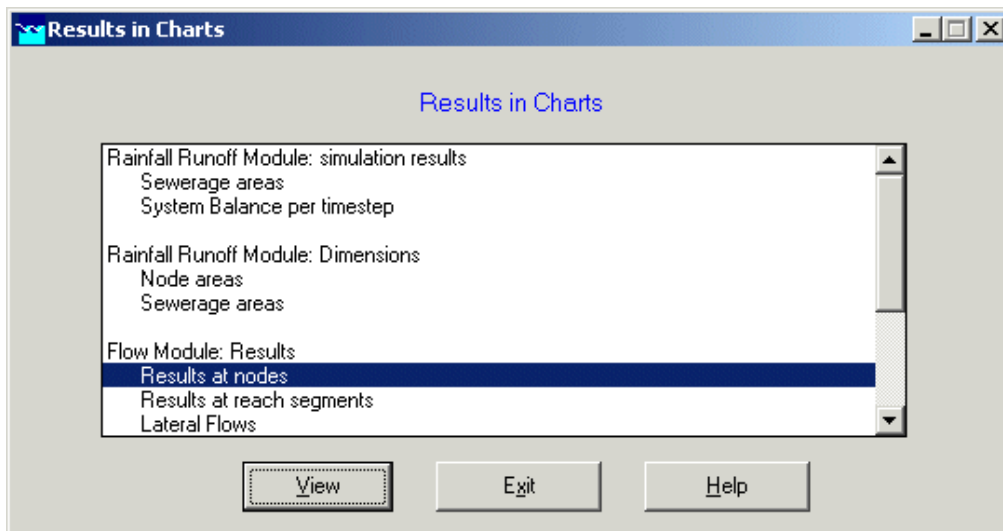
The results in charts window.

- Select 'Results at nodes'.
- Click the <View> button.

The ODS_VIEW window.

To plot a graph of your model results, select one or more parameters in the left box, select one or more locations for which you want to view results, and select the time span that you want to view results for.

Use the <Ctrl> or <Shift> keys to select more than one item within a box.

After creating a graph, feel free to explore the wide range of possibilities for display management. Adjust the axes according to your wishes, apply different graph templates, plot one parameter on the left axis and another parameter on the right one, etc. etc.
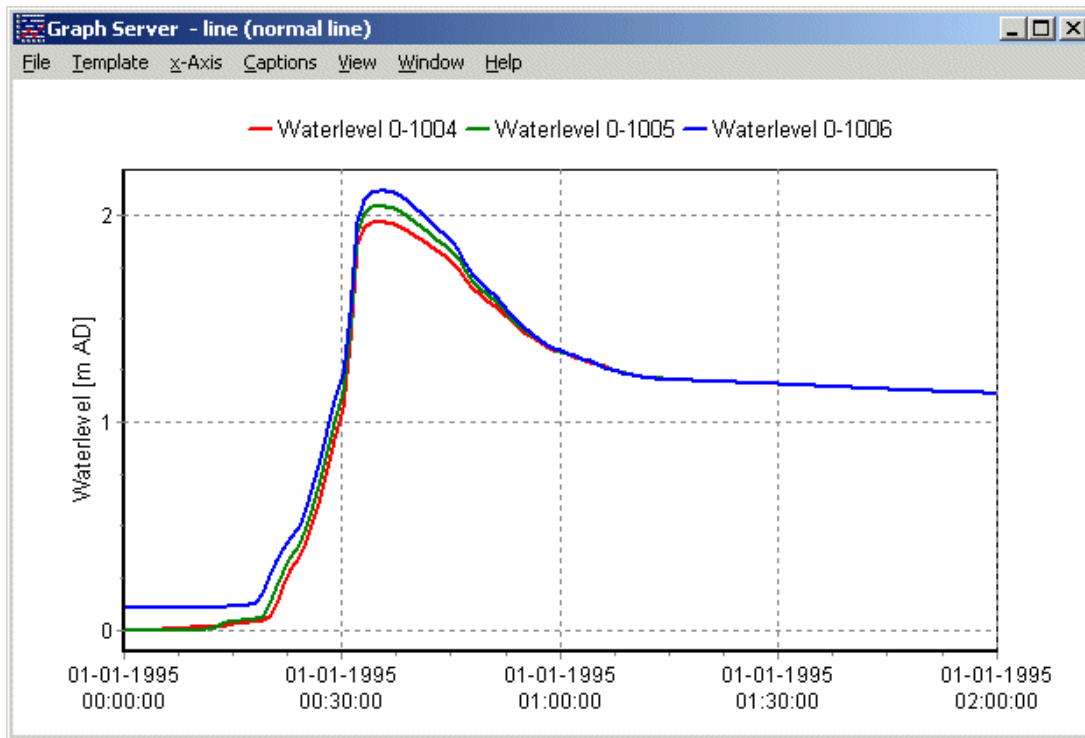
An example of a graph created in the 'Results in Maps' task block.

One can also choose to export the data to different file formats such as a spreadsheet. To do so, click the <Export data> button in the 'Results in Charts' window.

- Select the parameter 'Waterlevel [m AD]'.
- Select the location 0-1004, 0-1005 and 0-1006.
- 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'.

- Select 'Case' - 'Save' to save the case.

1.12 Case Analysis Tool

In 'Case_one' we used the rain event 'STNBUI07.BUI'. The total rainfall in this event is 19.8 mm. To show the impact of a much larger rainfall event we will select the rainfall event 'STNBUI10.BUI. The total rainfall in this event is 35.7 mm.

- Double-click the task block 'Meteorological Data'.
- Press the <Edit/Select> button.
- Press the <Select event...> button.
- Select 'STNBUI10.BUI' from the list.
- Press the <OK> button.
- Press the <OK> button.
- Press the <OK> button.
- Double-click the task block 'Schematisation'.
- Press the <OK> button.
- Double-click the task block 'Simulation'.

- Select 'Case' - 'Save as...'.
- Enter the name 'Case_two'.
- Press the <OK> button.
- Select 'Case' - 'Exit'.

- Select 'Projects' - 'Case Analysis Tool'.
- Select 'T_SEWER' from the list.
- Press the <OK> button.

- Select the tab 'Cases'.
- Select 'Case_one' from the cases list.
- Select the item 'FLOW: results of nodes'.
- Select 'Case_two' from the cases list.
- Select the item 'FLOW: results of nodes'.

- Check 'Case_one' as the Reference Case.

- Select the tab 'Locations'.
- Press the button [>>] to select all locations.

- Select the tab 'Parameters'.
- Double click on the item 'Water level [m AD] FLOW: results at nodes'.

- Select the tab 'Functions'.
- Click 'CaseFunctions'.
- Double click 'Difference with base case'.

- Press the button [ ].

- Select 'Options' - 'Data Statistics' - 'Maximum'.

The following figure appears. This figure shows the maximum difference between the two cases.
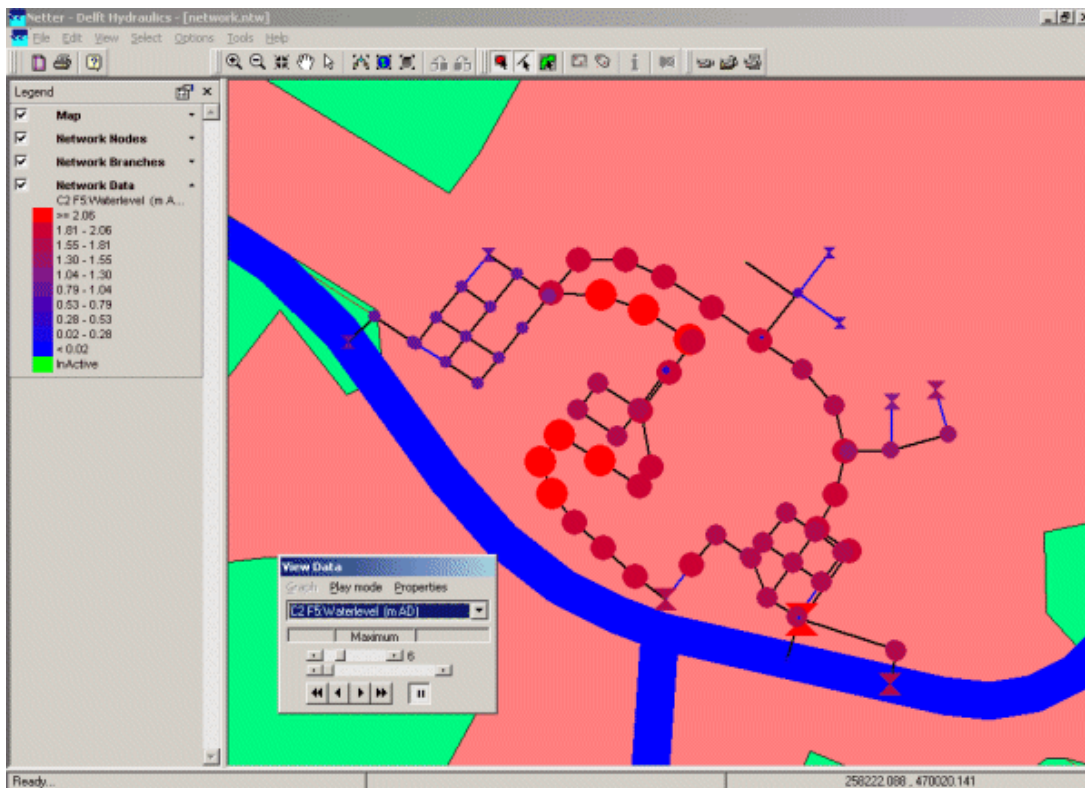
- Select 'File' - 'Exit'.
- Select 'Application' - 'Exit'.
- Press the button <No>.

1.13 Series simulation based on independent rainfall events

To simulate a long period in time, for example 50 years, it may take a lot of time. If you are only interested in the sewer overflows, the method of independent rainfall events may be of interest for you since it reduces the computation time and focuses on the events that matters. SOBEK offers the functionality to run in series mode (multiple independent rainfall events). It offers also structure statistics functionality, such as Total Volume [m3], Volume / Year [m3], Net Spill Time [min].

- Select 'Projects' - 'Open Project'.
- Select 'T_SEWER' from the list.
- Press the <OK> button.
- Select 'Case' - 'Open as new'.
- Select 'Case_two'.
- Enter the name 'T_SEWER.RKS'.
- Press the <OK> button.

- Double click the 'Meteorological Data' task block.
- Press the <Edit/Select> button.
- Press the <Select series...> button.
- Select the item 'T_SEWER.RKS' from the list.
- Press the <OK> button.
- Press the <OK> button.
- Press the <OK> button.

- Double click the 'Schematisation' task block.
- Press the <OK> button.

- Double click the 'Simulation' task block.
- Press the <Yes> button, to select the mean output option.

- Double click the 'Results in Maps' task block.
- Select 'Tools' - 'Structure Statistics'.
- Press the <Process> button.

The following figure appears:

	ID	Type	Direction	Capacity [m²/hr]	Crest Level [m AD]	Width [m]	Heigh [m]	Total Volume [m²]	Volume/Year [m²]	Freq./year [-]	Brut.SpillTime [min.]	Net.SpillTime [m]	Vol.1:1 yr [m²]
1	0-10003	External weir	pos.		0.20	5.00		208.12	208.12	3.00	64	64	126.75
2	0-2013	External weir	pos.		2.90	1.00		3410.60	3410.60	7.00	176	176	1522.79
3	0-4003	External weir	pos.		0.20	3.00		409.91	409.91	6.00	160	138	217.17
4	0-6004	External weir	pos.		1.85	2.00		1224.11	1224.11	10.00	454	454	276.32
5	0-6006	External weir	pos.		1.85	2.00		1223.92	1223.92	10.00	454	454	276.23
6	0-7004	External weir	pos.		3.00	2.00		1663.24	1663.24	10.00	420	418	469.84
7	0-7005	External weir	pos.		3.25	1.00		0.00	0.00	0.00	0	0	0.00
8	0-8002	External weir	pos.		1.20	5.00		14470.93	14470.93	8.00	348	344	4342.72
9	0-9002	External weir	pos.		4.00	5.00		1656.05	1656.05	10.00	732	728	184.30

- Select the cell with ID '0-8002'.
- Click the right mouse button.
- Select 'Show Graph'.
- Scroll to event number 6.

The following figure appears:
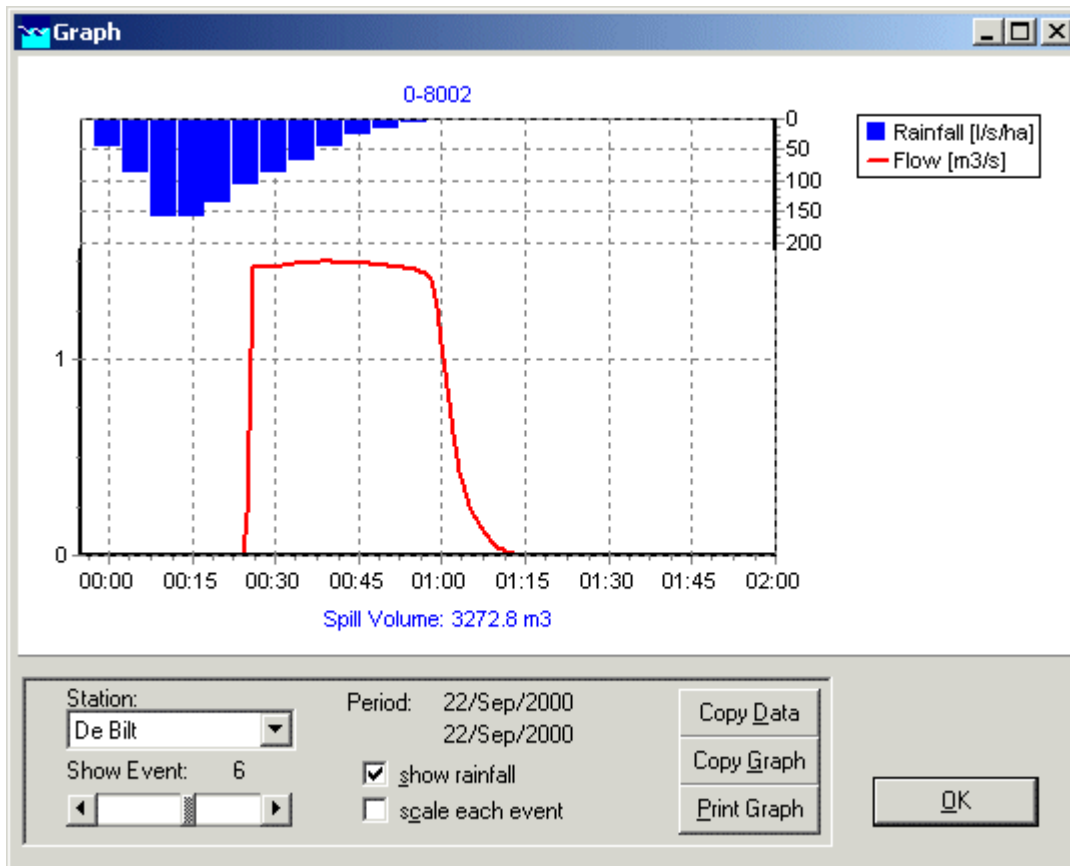
- Press the <OK> button.
- Select 'File' - 'Exit'.
- Press the <Exit> button.
- Select 'File' - 'Exit'.
- Select 'Case' - 'Save'.

1.14 Task block 'Schematisation' extending your schematisation

Now, you will extend the simple schematisation.

- Select 'Case' - 'Open as new'.
- Select 'Case_two'.
- Enter the name 'Extended case'.
- Press the <OK> button.
- Double-click the 'Schematisation' task block.
- Press the <Edit model> button.
- Select the ![button] button, Edit Network, to switch to 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 the ![button] button from the ![No edit action] section and the 'General' edit network functions to unveil the 'General functions' toolbar and move it to anywhere on your screen by clicking the upper part of the selected toolbar and dragging it:
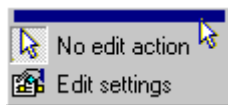
- Select the ![button] button and the 'Node' edit network functions to place the 'Node functions' toolbar anywhere on your screen.
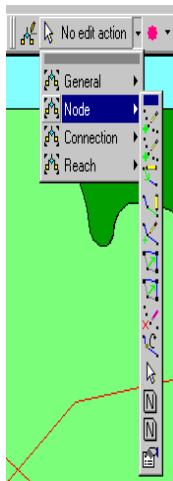
- Select the ![button] button and the 'Connection' edit network functions to place the 'Connection' functions toolbar anywhere on your screen.
- Select the ![button] button and the 'Reach' edit network functions to place the 'Reach functions' toolbar anywhere on your screen.

- Select ▾ from the ⬤ section and the place the objects toolbar anywhere on your screen.

- Select ▾ from the ▭ section and place the objects toolbar anywhere on your screen.

If you desire more information explaining the large amount of objects, you can customise the toolbars by clicking 'View' - 'Toolbars' - 'Customize...'. Caption only, Icon only and Icon and Caption are the available options. 'Icon only' means that, for the selected toolbar, you will only see the icons, but no label explaining them. 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 and manually.

- 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 'ID', select the radio button 'Manual'.
- In the group box 'Name', select the radio button 'Manual'.
- Select the tab 'Link'.
- In the group box 'ID', select the radio button 'Manual'.
- In the group box 'Name', select the radio button 'Manual'.
- Click the <OK> button.

In order to see the identifiers on the map please:

- Click on the 🖼️ button in the Active Legend or select the menu item 'Options' - 'Network Data...'.
- Select the tab 'Node'.
- Select the radio button 'ID'.
- Press the <OK> button.
- Select 'Select ' - 'Search...' from the menu bar.
- Enter '0-7003'.
- Press the <OK> button.
- Zoom in towards node '0-7003' .

Now you can start drawing the extended schematisation with node '0-7006' and link '0-7006--7003'.
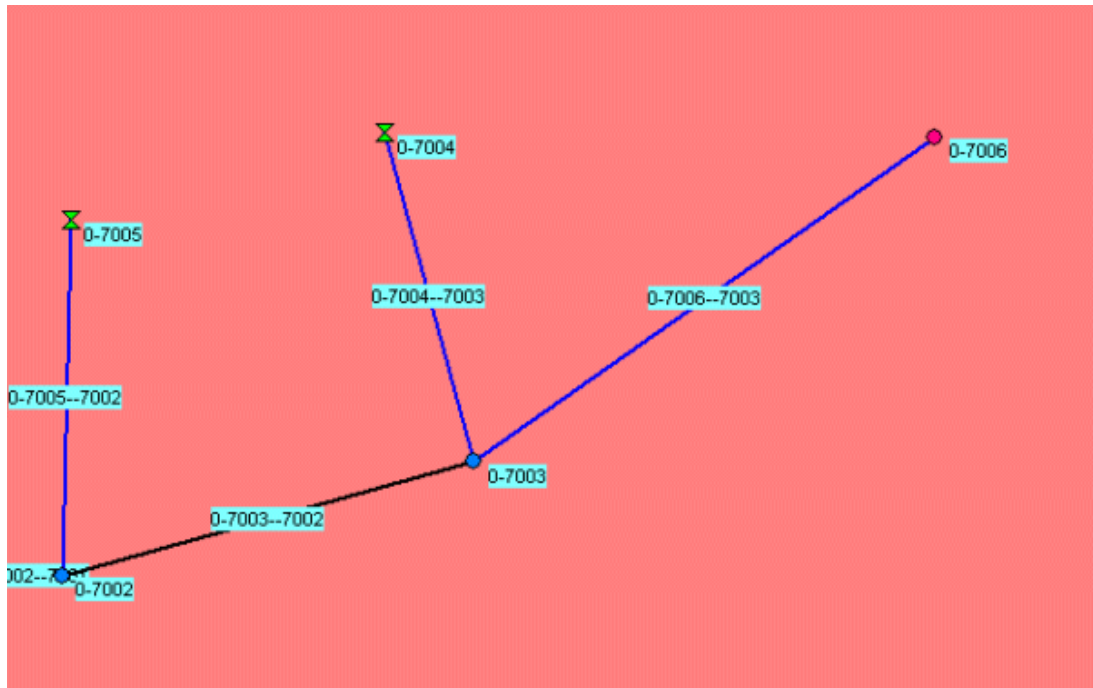
- Activate the [icon] button, Flow-Manhole, by clicking on it.

- Activate the [icon] button, Flow-Pipe with Runoff, by clicking on it.

- Activate the [icon] button, 'Add connect, by clicking on it.
- Enter '0-7006' as the ID of the Node definition.
- Enter '0-7006' as the Name of the Node definition.
- Enter '0-7006--7003' as the ID of the Link definition.
- Enter '0-7006--7003' as the Name of the Link definition.
- Press the <OK> button.
- Next, click the desired location on the screen to actually add the node, keep you mouse button down while dragging to node '0-7003' and release the mouse button.

Now the new Flow-Manhole object is connected to the simple schematisation. The option 'defined direction' can be used to see the positive defined direction. The defined direction can be viewed by selecting 'Options'-'Network Data...'-'Branch'-'Defined'.

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 [icon] button, Edit Reach Vectors', to edit a selected reach vector.
- Select the reach.
- Select the [icon] button to show the coordinates.
- Select the [icon] 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.

- Unselect the [icon] button, Edit Reach Vectors to leave the 'vector layer' mode.

- Unselect the [icon] button, Edit network, to leave the 'edit network' mode.

The schematisation has been extended. The next step is to define the attribute data of the schematisation. Therefore you have to switch to the model attribute data mode.

Editing node (manhole) data

- Select the Flow-Manhole '0-7006'.
- Click with your right mouse button.
- Select 'Model data' - 'Flow Model'.
- Select the tab 'Bottom Level'.
- Enter '1.6' for the Bottom Level.
- Enter constant value of '4' of the storage area at Bottom Level.
- Select the tab 'Surface level'.
- Select 'Reservoir' of the Water on street type.
- Select the constant value option.
- Enter '3.5' of the Street level.
- Enter '500' for the storage area at street level.
- Press the <OK> button.

Editing reach (pipe) data

- Select the new Flow-Pipe with Runoff object.
- Click with your right mouse button.
- Select 'Model data' - 'Flow Model'.
- Select the tab 'Cross section'.
- Select Cross sections type 'Round'.
- Select the Cross section 'Round 800 mm'.
- Select the tab 'Location'.
- Press the <Get Levels> button.
- Click the <OK> button.

- Select the new Flow - Pipe '0-7006--7003', the Flow - Pipe '0-7004-7003 and the Flow - Pipe '0-7005--7002' by clicking simultaneously the "Shift" key on the keyboard and the objects one by one with the left mouse button. Now, you can release the "Shift" key.
- Click with your right mouse button.
- Select 'Model data' - 'Rainfall-Runoff Model'.
- Select the item Flow - Pipe with Runoff' in the tree.
- Select 'Sewerage Runoff'.
- Select the column 'Open Sloped area [m2]'.
- From edit menu, select 'Edit Column' - 'Distribute catchment area'
- Enter the value '2000'.
- Click the <OK> button.
- Select [icon] button.

- Select 'File' - 'Exit'.
- Select 'Tools' - 'Validate network by model' - 'Flow Model'.
- Press the <OK> button.
- Select 'Tools' - 'Validate network by model' - 'Rainfall Runoff Model'.
- Press the <OK> button.
- Select ⊟ button.
- Select the menu item 'File'-'Exit', to leave NETTER.
- Press the button <Check Rainfall-Runoff Model> to check all attribute data.
- Press the button <Perform Check>.
- Press the <OK> button.
- Press the <OK> button.
- Press the button <Check Flow Model>.
- Press the <Options...> button.
- Select 'Give error message on missing data'.
- Select 'Keep unused profile definitions'.
- Select 'Keep unused structure definitions'.
- Press the <OK> button.
- Press the button <Perform Check>.
- Press the <OK> button.
- Press the <OK> button.
- Press the <OK> button.

- Select 'Case' - 'Save'.

Please start the simulation and analyze the results.

1.15 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!