Tutorial Hydrology in polders

SOBEK-Rural RR module

March, 2007

3.9230.00

Tutorial Hydrology in polders

SOBEK-Rural RR module

March, 2007

Contents

1 Tutorial Hydrology in polders

1.1 General

In this tutorial the basic principles of working with the Rainfall-Runoff module of SOBEK-Rural are explained step by step and you will be guided to set-up a simple schematisation. Rainfall-Runoff offers a wide range of options. This tutorial however only shows a limited number of them. After finishing the tutorial, one will have enough experience to continue oneself. Some experience on working with the WINDOWS Operating System is required.

The tutorial contains:

1. setting up a simple model;
2. computing;
3. viewing the results;
4. extending your model.

The tutorial does not explain all options in all windows that will 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 by browsing through the menu system.

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_RR’.

  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.

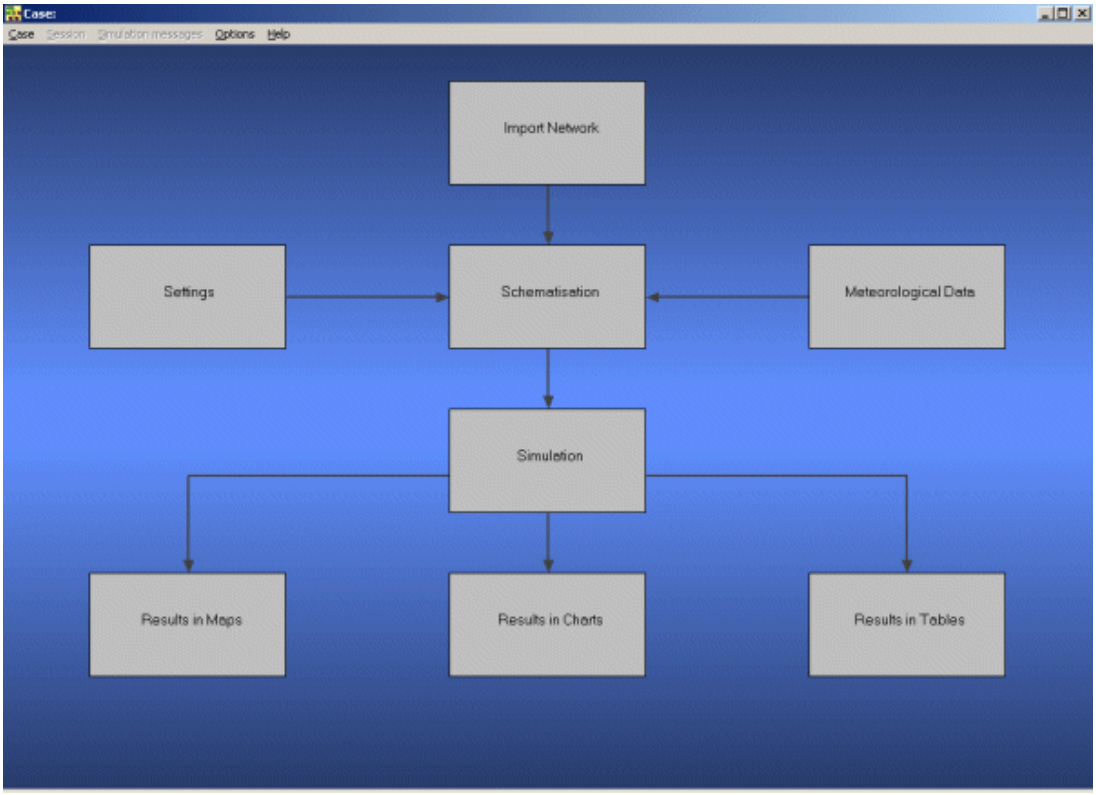
- Click the <OK> button.

You have added a new project with the name ‘T_RR’. 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 scenarios 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 are visible:

5. Import Network;
6. Settings;
7. Meteorological Data;
8. Schematisation;
9. Simulation;
10. Results in Maps;
11. Results in Charts;
12. 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:

13. administration of cases (which data are related to which cases);
14. checking whether the model calculations for the cases are performed in the predefined order;

15. logging the actions of the Case Manager (including view and print);
16. 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 the <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:

17. yellow: the task can be executed;
18. green: the task has been executed at least once and can be executed again;
19. red: the task cannot be executed until all preceding tasks have been executed.

When the task is being executed the task block is purple. You can execute a task by double-clicking 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, which means that this task block must be executed.

- Execute the task block ‘Import Network’ by double-clicking.

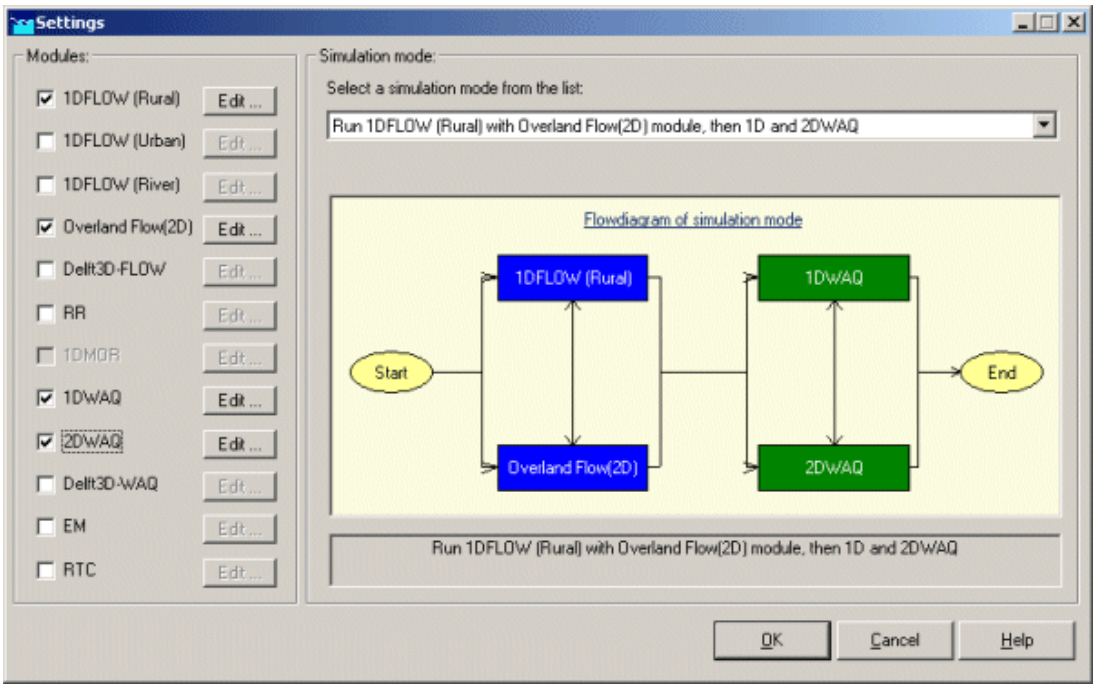
In this task block the origin of the schematisation can be defined. Schematisations, used in SOBEK, can be either imported from a database or set-up from scratch.
If a schematisation is already available in the standard exchange format it can easily be imported to SOBEK. Links with data formats can be made upon 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 the <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 this task block.
Depending on the set of modules that you purchased, some may be disabled (grey), and some may be enabled.



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.

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.

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.

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’.

Start editing the settings for the Rainfall-Runoff module:
- Double-click the ‘Settings’ task block.

The number of active modules depends on which modules you bought. In this tutorial we will only activate the Rainfall-Runoff module, the other modules are inactive.
- Unselect all the selected modules if any.
- Select the 'RR' module.
- Press the <Edit...> button of the 'RR' module.

You have to define certain settings, such as the
- Click the 'Time Settings' tab.

- Set the 'time step in computation' to 30 minutes
  (type '30' in the 'min' edit box and '0' in the others).
- Select the 'Output Options' tab (located on the right side of the window).
- Define 'output time step' to 30 minutes.
- Activate the following output options:

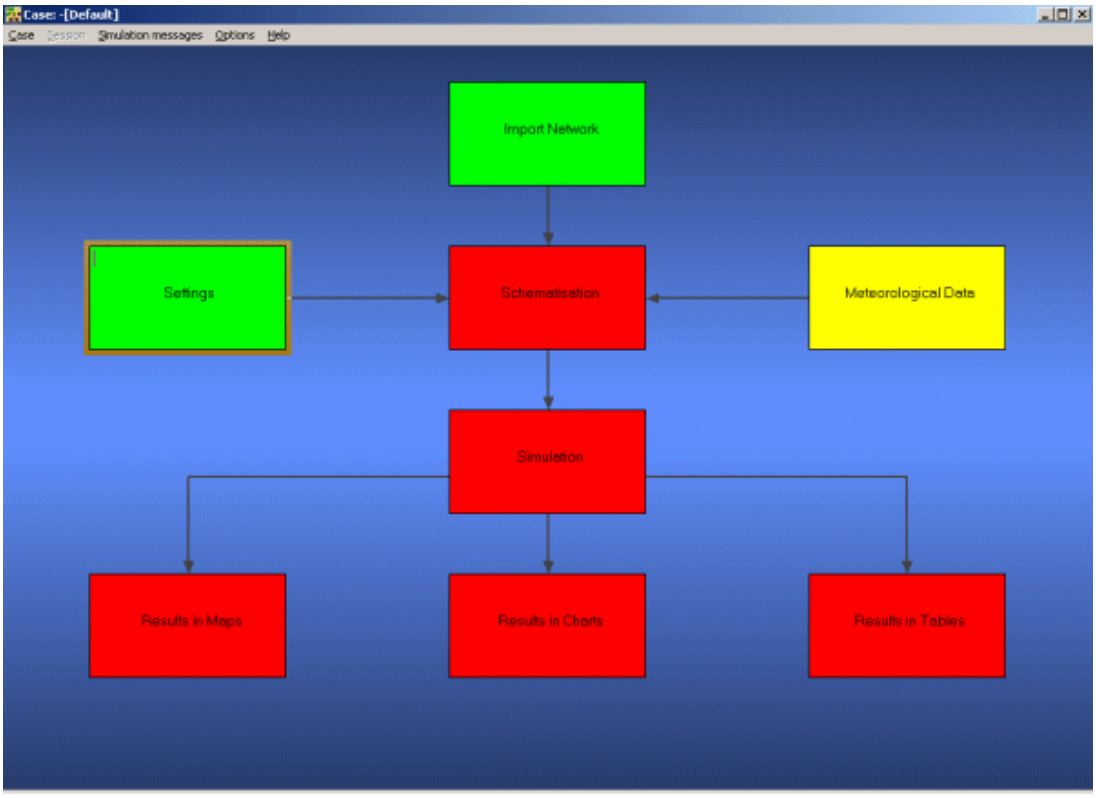
  - Flows at links
  - Open water node
  - Structure node
  - Unpaved node

- Press the <OK> button.

For this guided tour all remaining default values are fine.
Did you notice that you did not have to define the begin and end of the simulation period? Rainfall-Runoff will automatically derive this from the rainfall data. The rainfall data are incorporated in the 'Meteorological Data' task block. The next section deals with this task block.

- Press the <OK> button to exit this task. Your settings are then saved.

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

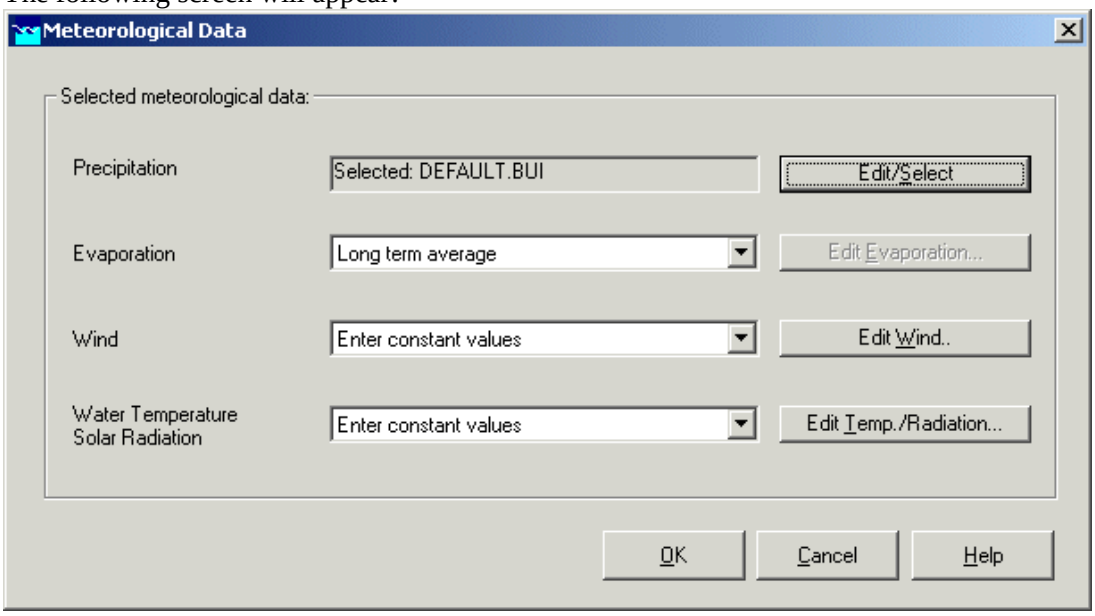


1.6 Task block 'Meteorological Data'

SOBEK-Rural simulations require meteorological input data, i.e. precipitation data and evaporation data.
The evaporation data are linked to the time series of the precipitation data. The simulation period is determined by the start- and end dates of the precipitation data.

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

The following screen will appear:



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

- Click the <Edit/Select> button next to the Precipitation box.
- Click the <Select event> button.
- Select the rainfall event named 'DEFAULT.BUI'.
- Press <OK> button.
- Press <OK> button again to leave the 'edit precipitation data' window.
- Select 'Long term average' 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'

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 the model by clicking the upper button <Edit model>.

- Click the <Edit model> button.

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:

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

Generally speaking, NETTER has two edit modes. The first mode is the mode to set-up the schematisation. The second edit mode is the mode for editing the attribute data. In this mode you provide attributes for 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.

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

The View menu contains commands for 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 with the left mouse button 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 fitting all schematisation objects in to the NETTER window.

The  button redraws the view fitting all GIS map layers in to 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.

- 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.

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

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

In the edit network mode, all edit network functions and network objects for the selected module are available.

- Select and the ‘General’ edit network functions to place the General functions toolbar anywhere on 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 node objects to place the ‘Rainfall-Runoff Model‘ node objects toolbar anywhere on your screen.
- Select and the ‘Rainfall-Runoff Model‘ link objects to place the reach objects toolbar anywhere on your screen.

Please customize the captions of the Edit network toolbar by clicking 'View' - 'Toolbars' - 'Customize...'. Caption only, Icon only and Icon and Caption are the available options.

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.

Now you can start drawing you application.

- Select the button, RR-Unpaved node.

- Select the  button to select the function 'Add node'.
- Enter 'Unpaved1' in both input fields.
- Click the <OK> button.
- Locate the mouse at a position where you want to add the RR-Unpaved 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, RR-Open Water node.
- Select the  button.
- Enter 'Open water1' in both input fields.
- Click the <OK> button.
- Click on the left-mouse button again to actually add the node on your screen.
- Select the  button, RR-link.
- Select the  button to select the function 'Connect nodes'.
- Click with the left mouse button on the RR-Unpaved node and drag to the RR-Open Water node while keeping the button down. Release the left mouse button.
- Select the  button, RR-Unpaved node.
- Select the  button, RR-link.
- Select the  button to select the function 'Add connect'.
- Enter 'Unpaved2' in both input fields.
- Click the <OK> button.
- Click with the left mouse button on the map where the second RR-Unpaved should be placed and drag to the RR-Open Water node while keeping the button down. Release the left mouse button.

Now the two unpaved area nodes are connected to the open water node.

Downstream of an unpaved area node should be an open water or a boundary node. The option ‘defined direction’ can be used to see the downstream site. The defined direction can be viewed by selecting ‘Options’-‘Network Data...’-‘Links’-‘Defined’. Note that the arrow is pointed towards the open water node. This means that downstream of the unpaved area node is the open water node and the unpaved area discharges on the open water node.

- Select the  button, RR - Pump node.
- Select the  button.
- Enter 'Pump1' in both input fields.
- Click the <OK> button.
- Click on the left-mouse button again to add the node on your screen.
- Select the  button, RR - Boundary node.
- Select the  button.
- Enter 'Boundary1' in both input fields.
- Click the <OK> button.

- Click on the left-mouse button again to add the node on your screen.
- Select the  button, RR-link.
- Select the  button to select the function 'Connect nodes'.
- Click with the left mouse button on the RR - Open Water node and drag to the RR - Pump node while keeping the button down. Release the left mouse button.
- Click with the left mouse button on the RR - Pump node and drag to the RR - Boundary node while keeping the button down. Release the left mouse button.

The schematisation has been set-up. The next step is to define the attribute data of the schematisation.

You will provide the following data step-by-step:

Unpaved area node

- Area per crop = 280 ha grass;
- Ground water area = 400 ha;
- Surface level = 1 m above reference level (constant value);
- Soil type = sand_maximum (μ = 0.117 per m);
- Depth groundwater layer = 5 m. This information is only relevant for computing of salt concentration, thus for computing the volume of this node. The option to compute salt concentrations can be turned on in the "settings task block", but is turned off in this tutorial;
- Initial groundwater level = equal to target level open water or level at boundary node;
- Storage, Maximum on land = 3 mm;
- Storage, Initial = 0 mm;
- Infiltration capacity = 10 mm/hr;
- Computation option for drainage = Hellinga - de Zeeuw
- Reaction factor Surface runoff = 100 1/day;
- Reaction factor Horizontal inflow = 0.05 1/day;
- Reaction factor Drainage 0 - 1 m alpha = 0.4 1/day;
- Reaction factor Drainage 1 - 2 m alpha = 0.4 1/day;
- Reaction factor Drainage 2 - 3 m alpha = 0.4 1/day;
- Reaction factor Drainage 3 - infinity alpha = 0.4 1/day;
- Seepage = 0 mm/day.

- Unselect the  button, Edit Network, to close the edit network mode.
- Select the 'Unpaved1' node.
- Click right mouse button.
- Select 'Model data' - 'Rainfall Runoff Model'.
- Select the 'Area' tab.
- In the group box 'Area per crop', select [ha] as the unit.
- Click the <Table...> button.
- Enter '280' for the crop 'grass'.
- Click the <OK> button.
- Check the check box 'Use different area for ground water computations:'.
- Enter the value '400'.
- In the group box 'Surface level', select 'constant value'.
- Enter the value '1'.
- Select the 'Soil' tab.
- Select 'sand_maximum (μ = 0.117 per m)'.
- Type '5' in the input box 'Thickness ground water layer'.

- Type '1.5' in the input box 'Maximum allowed ground water level:'
- In the 'Initial groundwater level' group box, select 'equal to initial target level open water or level at boundary node'.
- Select the 'Storage' tab.
- Type the name 'storage1' in the combo box.
- Click the <Define> button.
- Select [mm] as the unit.
- Enter the value of 3 mm for maximum storage on land.
- Enter the value of 0 mm for initial storage.
- Then press the <Save> button.
- Then press the <OK> button (You have added a Storage definition. Accept this as a new definition ?).
- Select the 'Infiltration' tab.
- Type the name 'infiltration1' in the text box.
- Click on the <Define> button.
- Select [mm/hr] as the unit.
- Enter the value for infiltration capacity of 10 mm/hr.
- Click the <Save> button.
- Click the <OK> button (You have added an Infiltration definition. Accept this as a new definition ?).
- Select the 'Drainage' tab.
- Select the radio button 'De Zeeuw-Hellinga' in the 'Computation option for drainage' group box.
- In the 'Definition' group box, enter the name 'drainage1'.
- Click the <Define> button.
- Enter '100' as the surface runoff reaction factor [1/day].
- Enter '0.05' as the horizontal inflow reaction factor.
- In the 'Reaction factor [1/day]' group box, check the top check box. A table appears in which the reaction factor data can be entered.
- To define the top drainage layer from 0 - 1 m below surface, enter '1' in the input box.
- Enter '0.4' as the reaction factor for this layer.
- Enter the following data:

Reaction factor Drainage 1 - **2** m	**0.4** 1/day
Reaction factor Drainage 2 - **3** m	**0.4** 1/day
Reaction factor Drainage 3 - infinity	**0.4** 1/day

- Click the <Save> button.
- Click the <OK> button (You have added a definition. Accept this as a new definition ?).
- Select the Seepage tab.
- Type the name 'seepage1'.
- Click on <Define> button.
- Select the option constant for seepage.
- Enter the value 0.
- Click the button <Save> to save this set of parameters.
- Click the <OK> button (You have added a Seepage definition. Accept this as a new definition ?).
- Click the <OK> button.

For the second RR - Unpaved node data you will use the Multiple Data Editor.

- Select the  button, Select by rectangle.

- Select the two RR - Unpaved 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' - 'Rainfall Runoff Model'.
- Select 'RR-UnPaved'.
- Select 'Unpaved area'.
- Select the row of 'Unpaved1'.
- Select 'Edit' - 'Copy'.
- Select the row of 'Unpaved2'.
- Select 'Edit' - 'Paste'.
- Select 'File' - 'Save Data'.
- Select 'File' - 'Exit'.

Open water

- Select the open water node.
- Click the right mouse button.
- Select 'Model data' - 'Rainfall-Runoff Model'.
- Select the 'Surface' tab.
- Enter the 'Bottom level' at -1 m above datum.
  This information is only relevant for computing of salt concentration, thus for computing the volume of this node.
- Select 'constant area'.
- Choose 'ha' for area unit.
- Enter a surface area of 20 hectares.
- Select the 'Management' tab.
- Select 'Fixed target level [m above datum]'.
- Enter 0 as the fixed target level.
- Enter 0 m above datum as the maximum permissible level.
- Select the 'Seepage' tab.
- Type the name 'seepage open water1' in the combo box.
- Click the <Define> button.
- Enter a seepage of 0 mm/day.
- Then press the <Save> button.
- Click the <OK> button (You have added a Seepage definition. Accept this as a new definition?).
- Click the <OK> button.

Pump

- Select the RR - Pump node.
- Click right mouse button.
- Then select 'Model data' - 'Rainfall Runoff Model'.
- Select the 'Options' tab.
- Select 'Normal' as the 'Pump type' (Note that the upstream level is checked only).
- Select the 'Pump' tab.
- Choose 'm³/min' as the capacity unit.
- In the group box 'Low capacity', enter the capacity of 30.
- In the group box 'Low capacity', click the <Table (Day)> of 'Switch on/off levels'.
- Enter the following values:

Date [dd/mm/yyyy]	Time [hh:mm:ss]	Switch **on** level during the **day** - target level [m]	Switch **off** level during the **day** - target level [m]

01/01/2002	00:00:00	0.01	-0.01
31/12/2002	23:59:00	0.01	-0.01

- Click the <OK> button.
- In the group box 'High capacity', enter the additional capacity of 0.
- Click the <OK> button.

Boundary node
- Select the RR - Boundary node
- Click the right mouse button.
- Select 'Model data' - 'Rainfall Runoff Model'.
- Select the 'Boundary' tab.
- Select the 'Fixed boundary' option.
- Enter 0.5 m above datum as the fixed boundary level.
- Click the <OK> button.

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

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'.
- 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'.

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.
- Select the item 'Open water nodes' in the Active Legend.
- Select the node 'Open water1' by clicking it.
- Click the  button on the 'View Data' window.

Now analyse your results!

Suppose, you want to analyse the open water level and the groundwater level. Therefore it should be useful to depict both variables in one graph.

- Do not close the graphic server.
- Select the item 'Unpaved nodes' in the Active Legend.
- In the 'View data' window, select the item 'Groundw. Level [m]'.
- Select the 'Unpaved1' node.
- Click the  button in the 'View data' window.

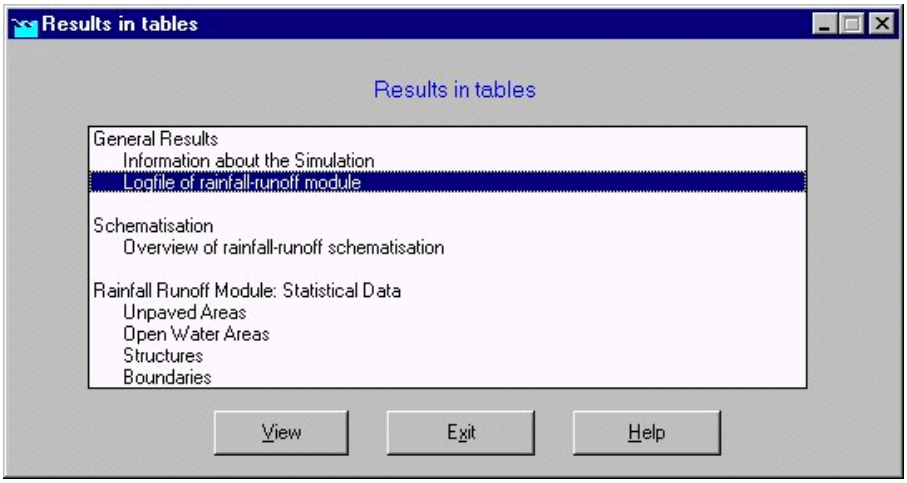
As you see a graph appears containing two variables!

- Select 'File' - 'Exit' to close the Graph Server.
- Select 'File' - 'Exit' to close 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 'Results in tables' task block.



- 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'.
- Click the <Exit> button.

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 'System Balance per timestep'.
- Click the <View> button.
- Select the parameters 'Rainfall' - 'Storage Unpaved' - 'Storage OpenWater' - 'Boundaries out' by using the <Ctrl> key.
- Select the location 'Total RR system'.
- 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 Extending your model

In this exercise we are using a combined schematisation of the SOBEK-Rural RR module and the SOBEK-Rural 1DFLOW module. This part is only possible if you have finished the tutorial 'Hydrodynamics in open water (SOBEK-Rural 1DFLOW module)' and the end results of the tutorial are available on the computer you are now using.

The RR model determines the runoff of various types of areas each with their characteristic influence on the dynamics of the rainfall-runoff process. The different area types can be described by various types of so-called runoff-nodes. The computed runoff is used as input for a one-dimensional hydrodynamic flow model. The 1DFLOW model computes the water levels and flows in a network of open canals.
The exchange of data between the two models can be both sequentially and simultaneously. In the sequential mode the RR model can be considered as a pre-processor for the 1DFLOW model while in the simultaneous mode real on-line interaction can be taken into account.

Running 1DFLOW and RR simultaneously

- Double-click the 'Settings' task block.
- Press the <Edit...> button of 'RR'.
- Select the tab 'Time Settings'.
- Enter the time step in computation: 10 minutes.
- Select the tab 'Output options'.
- Enter the output time step : 10 minutes in the respective edit box.

- Click the <OK> button.
- Turn on the checkbox '1DFLOW (Rural)'.
- In the 'Simulation mode:' group box, select the mode 'Run RR (Rainfall-Runoff) and 1DFLOW (Rural) module simultaneously'.
- Press the <Edit...> button of '1DFLOW (Rural)'.
- Select the tab 'Time Settings'.
- Enter the time step in computation: 10 minutes.
- Select the radio button 'simulation period derived from meteorological data'.
- Select the tab 'Simulation settings'.
- Select the radio button 'unsteady calculation'.
- Select the tab 'Initial data'.
- Select the radio button 'define local values in <Edit network>.
- Select the tab 'Output options'.
- Enter the output time step : 10 minutes in the respective edit box.
- Press the <OK> button.
- Press the <OK> button to exit the 'Settings' task and save your settings.
- Double-click the 'Schematisation' task block.
- Click the <Edit model> button.
- Select the  button, Edit Network, to start the edit network mode.

To give the whole RR schematisation unique identifiers we will add a preposition for all links.

- Select the  button, Select by rectangle.
- Select the whole RR schematisation by clicking on the map and drag while keeping the button down. Release the left mouse button.
- Select the  button to select the function 'Change IDs'.
- Enter the preposition 'RR-'.
- Uncheck the 'Nodes' check box.
- Check the 'Branches' check box.
- Click the <OK> button.

Now we will import the schematisation of the tutorial 'Hydrodynamics in open water'.

- Select 'File' - 'Import Case...'.
- Select 'Case_one' of the project 'T_CHANN'.
- Click the <OK> button.
- Click the <Yes> button (Do you want to combine the new Network with the existing network?).
- Click the <Yes> button (The Network layer was changed. Do you want to save the changes?).
- Select the file 'network.ntw'.
- Press the <Save> button.
- Click the <Yes> button.

Now we will connect the two schematisations.

- Select the  node, Flow - RR Connection on Channel.
- Select the  button to select the function 'Add node'.
- Enter '1DFLOW - RR connection1' in both input fields.
- Click the <OK> button.
- Locate the mouse at a position where you want to add the Flow - RR Connection on Channel node and click the left-mouse button again to actually add the node.
- Select the  button to select the function 'Delete node'.

- Select the 'Boundary1' node.
- Press the <Yes> button (Are you sure you want to delete RR - Boundary: RR-Boundary1?).
- Select the  button, RR-link.
- Select the  button to select the function 'Connect nodes'.
- Click with the left mouse button on the RR - Pump node and drag to the '1DFLOW - RR connection1' node while keeping the button down. Release the left mouse button.
- Select the  node, RR - Unpaved.
- Select the  button to select the function 'Add node'.
- Enter 'Unpaved3' in both input fields.
- Click the <OK> button.
- Locate the mouse at a position where you want to add the RR - Unpaved node and click the left-mouse button again to add the node.
- Select the  node, Flow - RR Connection on Channel.
- Select the  button to select the function 'Add node'.
- Enter '1DFLOW - RR connection2' in both input fields.
- Click the <OK> button.
- Locate the mouse at a position where you want to add the Flow - RR Connection on Channel node and click the left-mouse button again to actually add the node.
- Select the  button, RR-link.
- Select the  button to select the function 'Connect nodes'.
- Click with the left mouse button on the 'Unpaved3' node and drag to the '1DFLOW - RR connection2' node while keeping the button down. Release the left mouse button.

- Select 'Tools' - 'Validate network by model' - 'Rainfall Runoff Model'.
- Click the <OK> button (No errors detected).
- Select the  button to exit the edit network mode.

- Select the '1DFLOW - RR connection1' node.
- Click right mouse button.
- Select 'Model data' - 'Rainfall Runoff Model'.
- Select the 'Boundary' tab.
- Select the radio button 'Variable boundary'.
- Select the radio button 'Online from Flow Module'.
- Click the <OK> button.
- Do the same for the '1DFLOW - RR connection2' node.
- Select the  button, Select by rectangle.
- Select the whole RR 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 'RR-UnPaved'.
- Select 'Unpaved area'.
- Select the row of 'Unpaved1'.
- Select 'Edit' - 'Copy'.
- Select the row of 'Unpaved3'.

- Select 'Edit' - 'Paste'.
- Select 'File' - 'Save Data'.
- Select 'File' - 'Exit'.
- Select  button.
- Select the menu item 'File' - 'Exit', to leave NETTER.
- Click the <Check Rainfall-Runoff Model> button.
- Click the <Perform Check> button.
- Click the <View Log> button.
- Select 'File' - 'Exit'.
- Click the <OK> button.
- Click the <OK> button.
- Click the <Check Flow Model> button.
- Click the <Perform Check> button.
- Click the <OK> button.
- Click the <OK> button.
- Click the <OK> button.
- Select the menu item 'Case' - 'Save As...'.
- Enter the name 'Extended'.
- Click the <OK> button.

- Double-click the task block 'Simulation'.

- Double-click 'Results in Maps' task block.
- Select the item 'Lateral Flows' in the Active Legend.
- Select the nodes '1DFLOW - RR connection1' and '1DFLOW - RR connection2' by using the <Ctrl> key.
- Click the  button on the 'View Data' window.

Now analyse your results!

1.13 Epilogue

In this tutorial the most important aspects of working with SOBEK has been discussed. 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!