1D/2D modelling suite for integral water solutions

SOBEK

Deltares systems

Hydrodynamics, Rainfall Runoff and Real Time Control

Deltares

Enabling Delta Life

User Manual

SOBEK

Hydrodynamics, Rainfall Runoff and Real Time Control

User Manual

Version: 1.00
SVN Revision: 55373

April 18, 2018

SOBEK, User Manual

Published and printed by:
Deltares
Boussinesqweg 1
2629 HV Delft
P.O. 177
2600 MH Delft
The Netherlands

telephone: +31 88 335 82 73
fax: +31 88 335 85 82
e-mail: info@deltares.nl
www: https://www.deltares.nl

For sales contact:
telephone: +31 88 335 81 88
fax: +31 88 335 81 11
e-mail: software@deltares.nl
www: https://www.deltares.nl/software

For support contact:
telephone: +31 88 335 81 00
fax: +31 88 335 81 11
e-mail: software.support@deltares.nl
www: https://www.deltares.nl/software

Contents

List of Figures

List of Tables

List of Symbols

Symbol	Unit	Description
A_F	m^2	Cross sectional flow area
A_R	m^2	Rainfall runoff area
A_S	m^2	Cross sectional storage area
A_T	m^2	Cross sectional total area (i.e. $A_F + A_S$)
A_z	m^2	Surface area
C	$m^{1/2}/s$	Chézy coefficient
C_d	-	Drag coefficient
C_{wind}	-	The wind friction coefficient.
f_d	-	Global design factor for overall extreme high or low load (rainfall)
g	m/s^2	Acceleration due to gravity (≈ 9.81)
h	m	Total water depth
h_0	m	average water depth in a branch
i_R	mm/s	Intensity of rainfall
i_s	mm/s	Intensity of seepage
O	m	Wetted perimeter
Q	m^3/s	Discharge
q_{lat}	m^2/s	Lateral discharge per unit length
R	m	Hydraulic radius
t	s	Time
w_f	m	Cross sectional width at water level
x	m	Cartesian coordinate along channel
y	m	Cartesian coordinate across channel
z_b	m	Bed level, positive up
ϕ_{wind}	deg	Angle between the wind direction and the local channel direction
ρ_w	kg/m^3	Water density
ρ_{air}	kg/m^3	Density of air
τ_{wind}	N/m^2	Wind shear stress

1 Introduction

1.1 About SOBEK

SOBEK is named after the ancient Egyptian crocodile river god. Crocodiles were believed to have predictive powers, as they would lay their eggs just above the level of the upcoming Nile flood.

SOBEK is an integrated software package for river, urban or rural management. Seven program modules work together to give a comprehensive overview of waterway systems keeping you in control. Its integrated framework also means that SOBEK can link river, canal and sewer systems for a total water management solution. This program is very easy to configure and quick to learn. It guides you in obtaining correct model descriptions. The graphically oriented interface makes it more user friendly than similar types of software. SOBEK is designed to interface with your existing software. It can download information from a variety of standard data formats and GIS systems. SOBEK is based on high performance computer technology. That means it can handle water networks of any size - big or small. And because of the robustness of the numerical core, your computer will not crash no matter how complicated the simulation.

1.2 Product Info

SOBEK, your answer to changing conditions
Water systems are influenced by many different factors. Heavier than expected, rainfall, high winds and pollution loads can have an impact on the water system. SOBEK 's unique integrated format means that you can check the effectiveness of measures taken to keep your system running at peak efficiency. The manual or automatic operation of pumps, sluice gates, weirs, storage tanks and other structures can all be incorporated into your model, giving you a realistic picture of how your system behaves in extreme scenarios. Results are displayed as maps, graphs, tables and animations helping you to analyse and communicate your ideas.

Product lines
SOBEK has three basic product lines covering any fresh water management situation in River, Urban and Rural systems alike. Each product line consists of different modules to simulate particular aspects of the water system. These modules can be operated separately or in combination. The data transfer between the modules is fully automatic and modules can be run in sequence or simultaneously to facilitate the physical interaction.

Module / feature	SOBEK-Rural	SOBEK-Urban	SOBEK-River
Hydrodynamics			
1DFLOW	✓	✓	✓
Overland Flow module (2D)	✓	✓	
Hydrology			
RR	✓	✓	
Morphology			
1DMOR (incl Sediment Transport)			✓
Water Quality			
1DWAQ	✓		✓
2DWAQ			
Emission module			
Real Time Control			
RTC simulation module	✓	✓	

All product lines use the same interface components. The interfaces of the urban and rural product lines are fully integrated. So, when you know how to work with one module, you know how to work with them all. The user interface is used to prepare your schematisation, control your input data, check for possible schematisation errors, and helps you to analyse both input data and all computed parameters.

Open interface

SOBEK has been developed by Deltares in partnership with the National Dutch Institute of Inland Water Management and Wastewater Treatment (RIZA), and the major Dutch consulting companies. SOBEK is the concrete result of our experience with governments, city authorities, water boards, private consulting companies and research institutes. Because so many people have helped to develop SOBEK, we've made the program as open as possible.

SOBEK is designed to interface with your existing software. It has a transparent ASCII database and can download information from a variety of standard data formats and GIS systems, so there's no need for extra typing. The program is very easy to configure and quick to learn. It checks extensively for data input and schematisation errors. The GIS oriented interface makes it more user-friendly than most other programs, especially for larger networks. The computer screen automatically adjusts itself to your needs and expertise so you can organise your water system information and tasks.

Application areas

Product line	Area of interest
SOBEK-Rural	Irrigation construction, rehabilitation, modernization
A	Drainage and flood protection
A	Long-term and real-time operation of multiple reservoirs
A	Real-time control and automation of canal system
SOBEK-Urban	Determination of urban drainage capacities, including treatment plants
A	Assessment of sewer overflow frequency
A	Design of detention basins
A	Real-time control of urban drainage systems
A	Environmental study on receiving waters
SOBEK-River	Navigation
A	Flood protection, flood-risk assessment
A	Water pollution studies
A	Estuaries with fresh and salt water
A	Sand mining, sediment and morphology studies

The integrated approach makes SOBEK a valuable instrument for flood forecasting, navigation, optimising drainage systems, controlling irrigation systems, reservoir operation, sewer overflow design, groundwater level control, river morphology regulation, and water quality control. The integrated approach also means that SOBEK can combine river systems, urban systems and rural systems for a total water management solution.

Size and complexity doesn't matter
SOBEK is based on high-performance computer technology. That means it can handle water networks of any size (big or small) and complexity.

It automatically configures itself to the type of water system you're modelling. All numerical computations have self-selecting time steps and are extremely robust.

1.3 Introduction to SOBEK-Rural

SOBEK-Rural gives regional water managers a high-quality tool for modelling irrigation systems, drainage systems, natural streams in lowlands and hilly areas. Applications are typically related to optimizing agricultural production flood control, irrigation, canal automation, reservoir operation, and water quality control. SOBEK-Rural can also answer questions about increased pollution loads in response to growing urbanisation. SOBEK-Rural offers the support you need for effective planning, design and operation of new and existing water systems.

The software calculates (easily, accurately and fast) the flow in simple or complex channel networks, consisting of thousands of branches, cross sections and structures. You can define all types of boundary conditions, as well as define lateral inflow and outflow using time series or standard formulae. For more detail, the rainfall run-off process of urban areas and various types of unpaved areas can be modelled, taking into account land use, the unsaturated zone, groundwater, capillary rise and the interaction with water levels in open channels. For water quality and environmental problems the Water Quality module offers almost unlimited possibilities.

The graphic display superimposes your network over a (GIS or aerial photo) map of the area so you can see the canals, reservoirs, weirs, pumping stations, treatment plants, urban and rural areas at a glance. Animation options show the direction of flow through the network and by varying the thickness and colours of selected network elements, all input and computed parameters can be visualised. By clicking on the map you can draw your network and adjust any detail to suit your needs. The network can also be viewed from the side. Side view allows you to print the design and watch the water profiles and real-time operation of structures in detail.

SOBEK-Rural incorporates four modules:

- ✓ Hydrodynamics
- ✓ Hydrology
- ✓ Water Quality
- ✓ Real-Time Control

1.4 Introduction to SOBEK-Urban

SOBEK-Urban offers a comprehensive modelling tool for simple or extensive urban drainage systems consisting of sewers and open channels. SOBEK-Urban allows you to design new urban areas or analyse and improve existing ones. You can use it to find out what measures will prevent drainage congestion, street flooding and water pollution from sewer overflows. The return period of street flooding and sewer overflows can be analysed using long time series of rainfall data or storm events.

The models help you to find out how the performance of the urban drainage system can be improved by a better operation of the pumps gates and weirs. The impact of treatment plants and sewer overflows on the receiving water can be analysed by combining everything into one model. It's the ideal tool for the design, management and renovation of urban sewer systems.

SOBEK-Urban models the rainfall run-off process for various types of paved and unpaved areas. It doesn't matter whether the urban drainage system consists of open channels and sewer pipes, storage tanks and reservoirs. Even street flow can be modelled with SOBEK-Urban.

The application handles all kinds of cross sections, control structures and any network configuration (branched and looped). The size and complexity of the system doesn't matter either. The computation is extremely fast, also for large networks (e.g. 25 000 sewer pipes and manholes in one model). On top of that, SOBEK-Urban offers virtually any real-time control option for pumps, weirs and gates in the urban system.

The graphic display superimposes your network over a (GIS or aerial photo) map of the area so you can see sewer pipes, manholes, canals weirs and pumping stations at a glance. By clicking on the map you can draw your network and adjust any detail to suit your needs. Animation options show the direction of flow and by varying the thickness and colours of network elements all input and computed parameters can be visualised. The network can also be viewed from the side and allows you to print the design and watch the filling and drying process in detail.

The software conforms to strict Dutch guidelines for sewerage calculations.

SOBEK-Urban consists of three modules:

- ✓ Hydrodynamics
- ✓ Hydrology
- ✓ Real-Time Control

1.5 Introduction to SOBEK-River

SOBEK-River is the product line designed for simple and complex river systems and estuaries. It simulates the water flows, the water quality and morphological changes in river systems, estuaries and other types of alluvial channel networks. The networks can be branched or looped. SOBEK-River is able to work with complex cross-sectional profiles consisting of various sub-sections.

The Windows-based interface makes it easy to use and have the Windows look and feel that you're used to. Direct on-screen display of the river network gives you an overall view of your system.

SOBEK-River works with one or more of the following modules:

- ✓ Hydrodynamics
- ✓ Morphology
- ✓ Water Quality

1.6 Support

General information

If you have a question about SOBEK for which you cannot find the answer in the manual, you can contact SOBEK Support at Deltares.

You should have the following information ready:

- ◇ the version number of SOBEK (visible in the upper-left corner of the window after the program is started);
- ◇ the type of hardware you are using;
- ◇ the operating system you are using;
- ◇ the exact wording of any message that appeared on your screen (write it down or take a screenshot);
- ◇ a description of what happened and what you were doing when the problem occurred;
- ◇ a description of how you tried to solve the problem;
- ◇ whether you are able to reproduce the problem by repeating what you did when the problem occurred.

It may also be necessary to send the project data to SOBEK Support. The best way to do this is to close SOBEK, zip the relevant project folder (*.lit) and send it using an e-mail. If the project is too large to be included as an attachment in an e-mail, SOBEK Support can provide the credentials to an ftp account where the data can be uploaded.

Security software
FlexNet

Our software uses license managing software by Flexera Software. If you want to know more about the security software, see the following link: http://www.flexerasoftware.com/products/flexnet-publisher.htm

In case you have additional questions about the security software, Flexera Software support can be found here: http://support.flexerasoftware.com/

It may be useful to seek out Flexera support if you are using various software packages using different security systems.

The dongle manufacturers
If you want to know more about new and old dongle manufactures, see the link below. In this document you can find links to drivers and other tools. It is also a comprehensive manual. Note: You do not need to install any drivers linked to in this document, the program setup will install the correct dongle drivers automatically as part of the Deltares License Manager Installation. These links are provided for troubleshooting purposes only:

http://www.flexerasoftware.com/webdocuments/PDF/faq_FLEXid_dongles.pdf

A direct link to the drivers for the dongles used in our software:

http://www.aladdin.com/support/hasp/enduser.aspx

1.7 About Deltares

Deltares
Enabling Delta Life

In 2008, four renowned Dutch organisations decided to pool their knowledge and expertise. WL | Delft Hydraulics, GeoDelft, TNO's Subsurface and Groundwater unit and parts of Rijkswaterstaat (the Dutch Directorate-General for Public Works and Water Management) joined together to set up Deltares, an independent institute for the development, dissemination and application of knowledge concerning water, soil and subsurface. The result is an organisation superbly well-equipped to address complex, integrated issues relating to water, soil, subsurface management and spatial planning in deltas, coastal areas and river basins.

Clients
Our national and international clients include government authorities, policy makers and administrators responsible for the short- and long-term governance of bodies of water and related infrastructures. Our clients are also in the private sector: for example, among multilateral agencies, consulting engineers, contractors, and in industry. They all have one thing in common: the need for solid, practical advice. They come to us because they know that we understand their concerns and are prepared to approach their questions from independent, fresh perspectives.

What we can do for you
Deltares brings together a long-standing reputation for excellence in hydrology, hydraulics, morphology, water quality and ecology. Construction and design matters related to offshore, coasts, harbours, estuaries, rivers and canals, and industry — also our forte — are approached in a manner tuned to the specific requirements of the client. In addition, we operate at the policy level by delivering decision support and carrying out environmental impact

assessments in the above mentioned working areas.

We have a full range of experimental facilities and computer programs, most of which have been developed and validated by our experts in residence. On the basis of a sound understanding of the processes involved, all water systems can be simulated by us, numerically, experimentally, or through a combination of the two. At Deltares, transfer of technology and know-how is an inherent part of our approach. This is done through a variety of training courses and seminars, and on the job.

More demanding
Demands of increased production and economic growth are frequently coming in direct — and public — conflict with environmental concerns. Balancing the needs of one with those of the other, not only for today, but also for the future, is often expressed in the phrase "sustainable development".

No simple matter, it makes our clients' jobs more demanding. In turn, they demand more of us, not only for construction — and design — related issues, but also for far-branching policy and management concerns. They expect and get from us optimal performance: a multidisciplinary, scientifically rigorous approach linked to cost-conscious good business sense.

1.8 Notation

In this manual the following conventions in type style are used:

Type style	Used for
Button	Buttons (on the screen or on the keyboard) that perform the indicated action or give access to a corresponding window. The text can be located on the button itself or next to the button.
Window	Name of a window, shown in the name bar. Also used for subwindows, indicating a group of buttons and/or data fields.

2 Getting started

2.1 Starting SOBEK

To get a good insight of the capabilities of SOBEK please use the tutorials provided with this SOBEK release.

The following tutorials are available:

- section 4.1, Tutorial Hydrodynamics in open water (SOBEK-Rural 1DFLOW module)
- section 4.2, Tutorial Hydrodynamics in sewers (SOBEK-Urban 1DFLOW + RR modules)
- section 4.3, Tutorial Hydrodynamics - 1D2D floodings (SOBEK-Rural 1DFLOW + Overland Flow modules)
- section 4.4, Tutorial Hydrology in polders (SOBEK-Rural RR module)
- The SOBEK-Rural 1DWAQ tutorial can be found in the D-Water Quality User Manual (D-WAQ UM, 2013)

The following manuals provide information regarding this SOBEK release:

- This manual (The Hydrodynamics, Rainfall Runoff and Real Time Control User Manual),
- The D-Water Quality User Manual (D-WAQ UM, 2013),
- The D-Water Quality Processes Library Description
- The D-Water Quality Processes Tables (D-WAQ PLT, 2013).

2.2 Free trial options

If you do not have a license file, SOBEK can still be used in Free Trial Copy mode. In this mode the following schematization restrictions apply:

- SOBEK-Rural 1DFLOW (100 node limit)
- SOBEK-Rural Overland Flow (2D) module - 2DFLOW (1000 active 2D cell limit)
- SOBEK-Rural RR (5 node limit)
- SOBEK-Urban 1DFLOW (100 node limit)
- SOBEK-Urban RR (100 node limit).

The Water Quality module cannot be used in Free Trial Copy mode.

While in Free Trial Copy mode, the user has the full functionality of the Task blocks: *Import*, *Meteo*, *Results in Maps*, *Results in Charts* and *Results in Tables* at her/his disposal.

The following Tutorials can be followed in Free Trial Copy mode:

- 4.1, Tutorial Hydrodynamics in open water (SOBEK-Rural 1DFLOW module)
- 4.2, Tutorial Hydrodynamics in sewers (SOBEK-Urban 1DFLOW + RR modules)
- 4.3, Tutorial Hydrodynamics - 1D2D floodings (SOBEK-Rural 1DFLOW + Overland Flow modules)
- 4.4, Tutorial Hydrology in polders (SOBEK-Rural RR module)

3 Installation manual

3.1 Introduction

Deltares has developed a unique, fully integrated modelling framework for a multi-disciplinary approach and 1D-2D computations for river systems and estuaries, irrigation and drainage systems and wastewater and storm water systems. It can carry out simulations of flows, rainfall-runoff, sediment transports, water quality, morphological developments and ecology. It has been designed for experts and non-experts alike. The framework is composed of several modules, grouped around a mutual interface, while being capable of interacting with one another.

3.1.1 System requirements

Operating Systems:

◇ Windows Server 2008
◇ Windows 7
◇ Windows 8.1
◇ Windows 10

Administrative privileges are required in order to install and use the software.

Minimum hardware specifications:

◇ 1,5 GHz Intel Core processor or equivalent
◇ 2 GB of RAM
◇ 2 GB free hard disk space
◇ a graphics adapter with 128 MB video memory and screen resolution of 800x600

Preferred hardware specifications:

◇ 3 GHz Intel Core processor or equivalent
◇ 6 GB of RAM, 64-bit Windows Operating System
◇ 20 GB free hard disk space
◇ a graphics adapter with 256MB video memory and a screen resolution of 1024x768

SOBEK is a 32-bit application. On 32-bit Windows operating systems, SOBEK can use a maximum of 2GB RAM (address space). On 64-bit Windows operating systems, SOBEK can use a maximum of 4GB RAM.

Regional Settings format in Windows: SOBEK should be installed on machines using a dot (.) as decimal separator symbol and a comma (,) as digit grouping symbol. For example as English format.

3.2 Deltares License Manager

For information about the Deltares License Manager, including installation manual and Frequently Asked Questions, please visit:

https://publicwiki.deltares.nl/display/LMADMIN/Deltares+License+Management

In order to use SOBEK, the license manager must be installed.

3.3 Installing SOBEK

This document describes the steps that should be taken to install SOBEK on your personal computer.

The installation consists of two parts:

- The installation of the Deltares Software License Manager
- The installation of the SOBEK software package

The licensing of your software is managed by the Deltares Software License Manager (DS_Flex), which is based on the FlexNet Publisher licensing system. It provides a user friendly environment in which a range of license constructions can be arranged, varying from single user licenses to multiple user network licenses.

Note that your license file may not be suitable for all mentioned license constructions.

- Log on to the PC. You need to log on as **administrator** (or a similar account with similar user privileges).
- Start the program setup (named SETUPSOBEK.EXE or similar), from the installation CD or the location where you downloaded it to.

Now you are lead through the setup procedure by a graphic user interface:

The first dialog is the SOBEK Welcome screen:

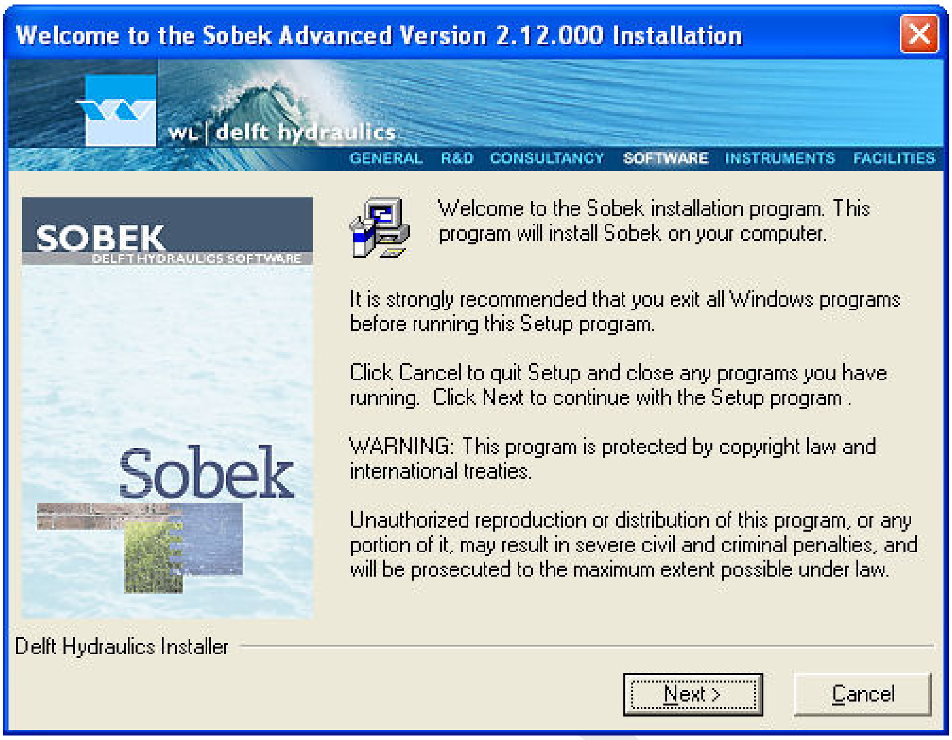


***Figure 3.1:** SOBEK **welcome** window*

◇ Click the *Next* button to go to the next dialog.

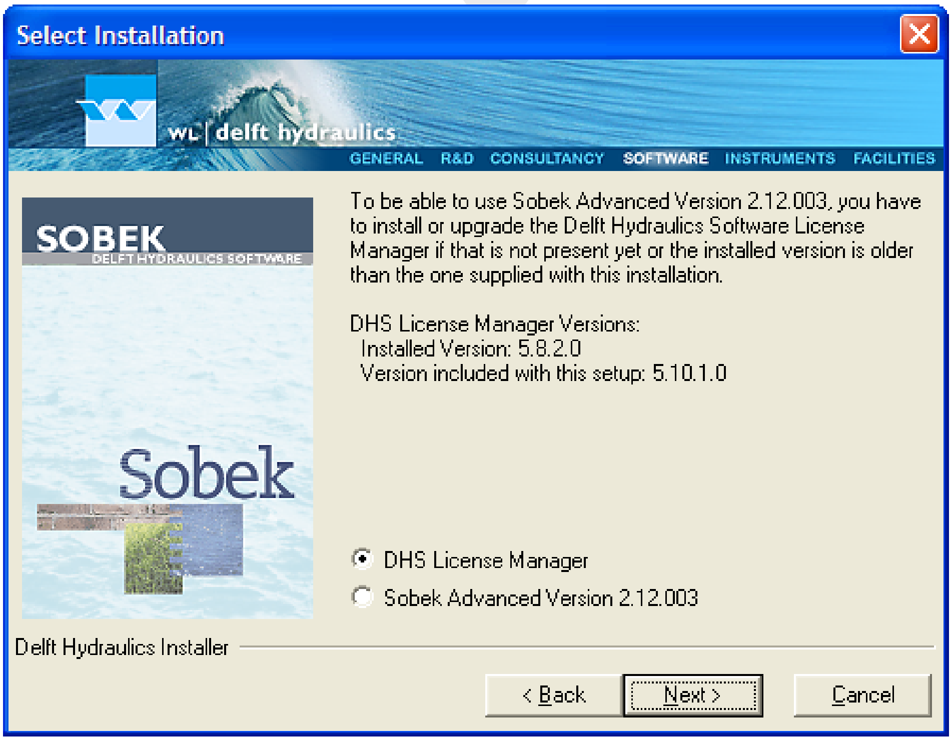


***Figure 3.2: Select installation** window, Deltares Software License Manager selected*

This dialog gives the opportunity to choose to install the license manager or to install SOBEK.

In this example the version of the Deltares Software License Manager is newer than the already installed one. It is highly recommended to use always the most recent version of the Deltares Software License Manager. If the license manager has not been installed yet, the installed version will be displayed as 'None'.

Note:
The Deltares Software License Manager must always be installed in order to use SOBEK. Even when trying out the Free Trial functionality of the software. Failure to install the license manager will result in a SOBEK Startup error. (Run-time error '53'. File not found: wlauth40.dll)

◇ Click the *Next* button to install the Deltares Software License Manager.

The actual installation of the Deltares Software License Manager is described in a separate paragraph. After it has been installed, the following situation will be displayed:

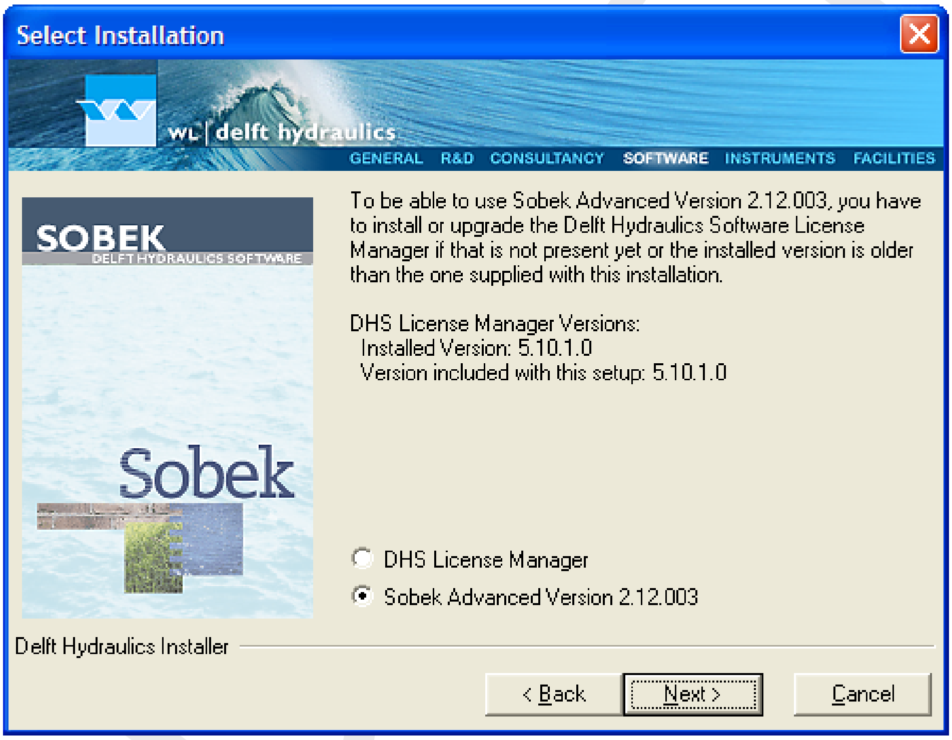


***Figure 3.3: Select installation** window, SOBEK selected*

◇ Select the SOBEK option if not selected yet.
◇ Click the *Next* button to install SOBEK.

If there are already one or more SOBEK versions installed, the following dialog will appear:

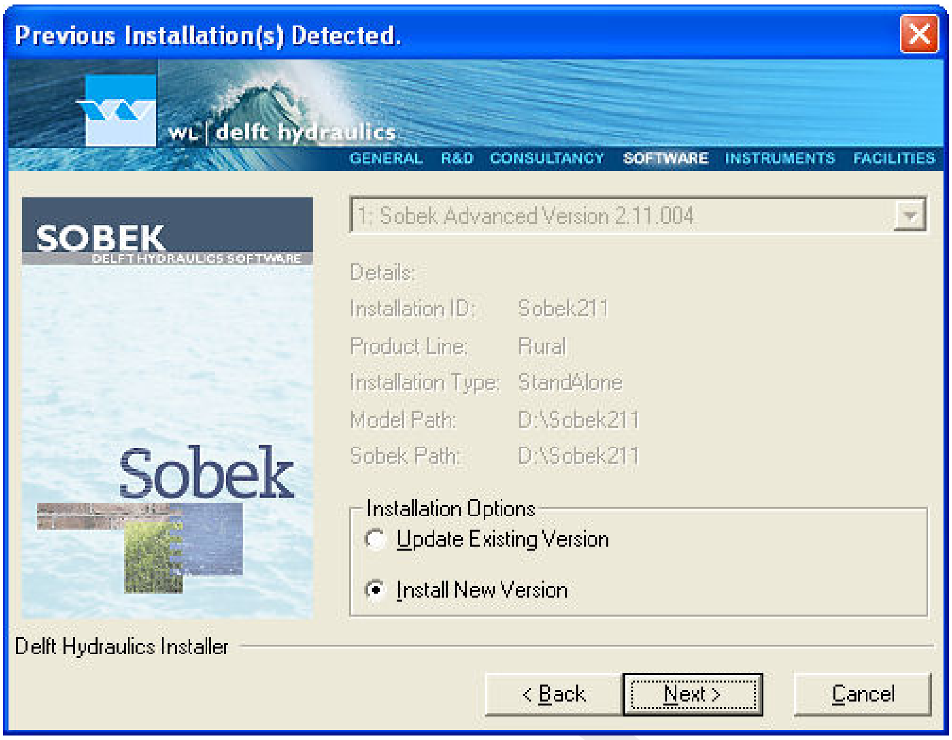


***Figure 3.4: Previous Installation(s) Detected**, Install New Version selected*

Here you can select the Update option and select a SOBEK version which you want to update. In that case all the options of that installation will be taken from the selected installation, like Installation ID, Product Line, Installation Type and the directories where the files are stored. Ensure that in case of network installations the installation path is accessible and writable.

When you select to update an existing version, after clicking the *Next* button the Miscellaneous Options dialog will appear. This dialog will be explained later.

When the Install New Version option is selected, clicking the *Next* button will activate the following dialog:

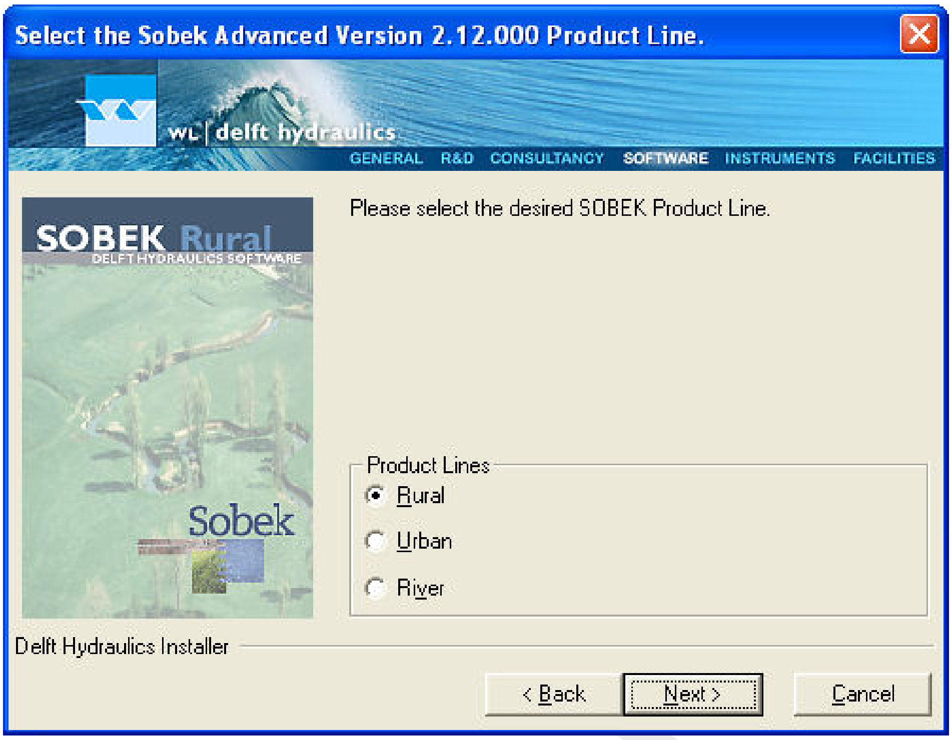


Figure 3.5: *SOBEK installation, select Product Line*

In this dialog the desired SOBEK Product Line can be selected. You can choose between Rural, Urban and River.

Choosing one of these options will have the following consequences:

◇ Typical Rural, Urban or River default numerical parameters will be selected for new projects. Numerical parameters can always be changed later in the Settings menu.
◇ A different background image for the SOBEK startup screen will be used.

Note:
Choosing between Rural, Urban and River has no consequences as to which modules you can use. Which modules you can use is determined entirely by your license file.

◇ Click the *Next* button to go forward to the Installation ID dialog.

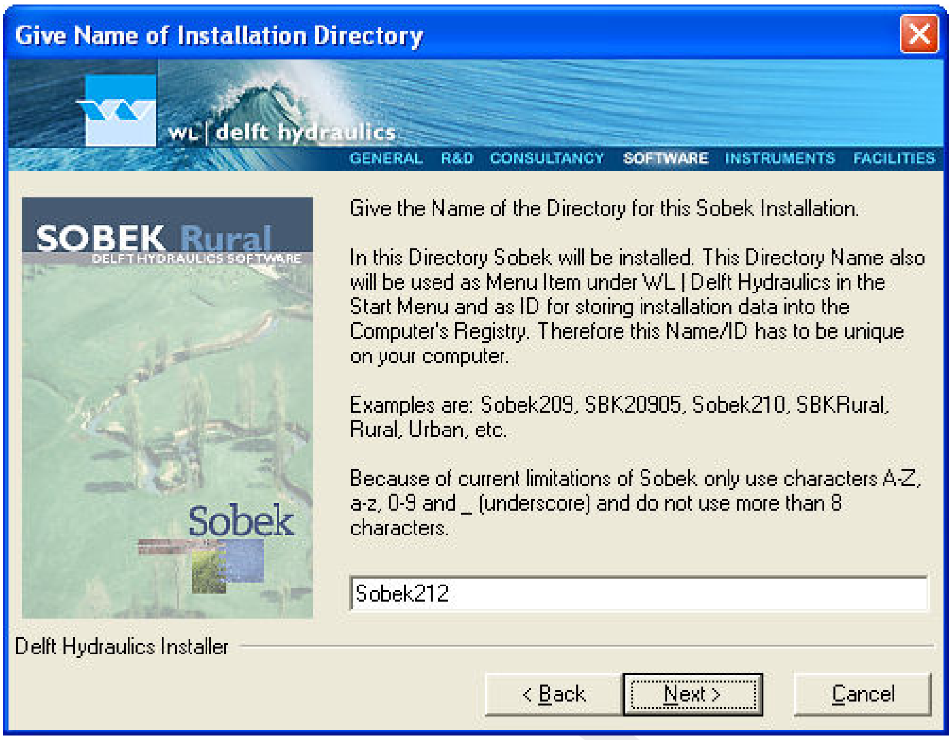


Figure 3.6: *Enter SOBEK installation directory*

Here you must specify the name of the directory where SOBEK will be installed. This directory name also will be used as entry in the Windows Start Menu under Delft Hydraulics and as ID for storing installation data into the computer's registry. You can install multiple versions of SOBEK on the same computer, but you have to use a unique folder name for each installation.

The Installation ID is limited to 8 characters and may only contain A–Z, a–z, 0–9 and '_' (underscore).

◇ Click the *Next* button to go to the Drive Selection dialog.

In the dialog below you must select the drive where the SOBEK system will be installed. The actual directory will be <*Drive*:\Install_ID>

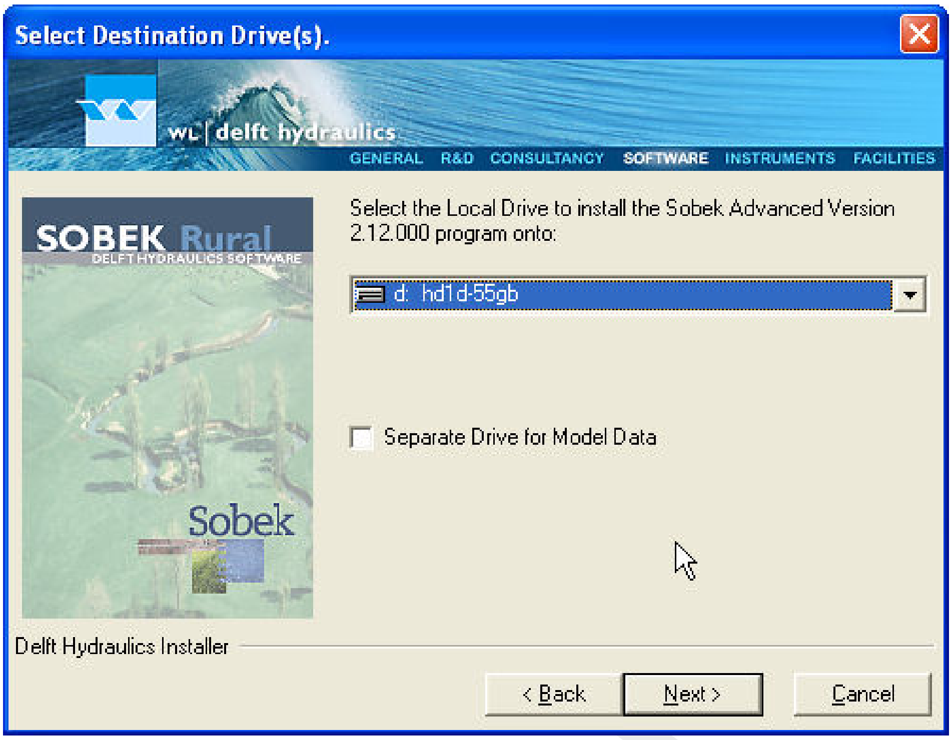


Figure 3.7: *Select destination drive(s)*

In some case it is desired or even required to separate the programs and the model data. In that case you can mark the checkbox and then select separate drives for the programs and the model data. In the example below (using the Installation ID from the earlier example dialog) the programs will be stored in <C:\Sobek214> and the model data in <D:\Sobek214>.

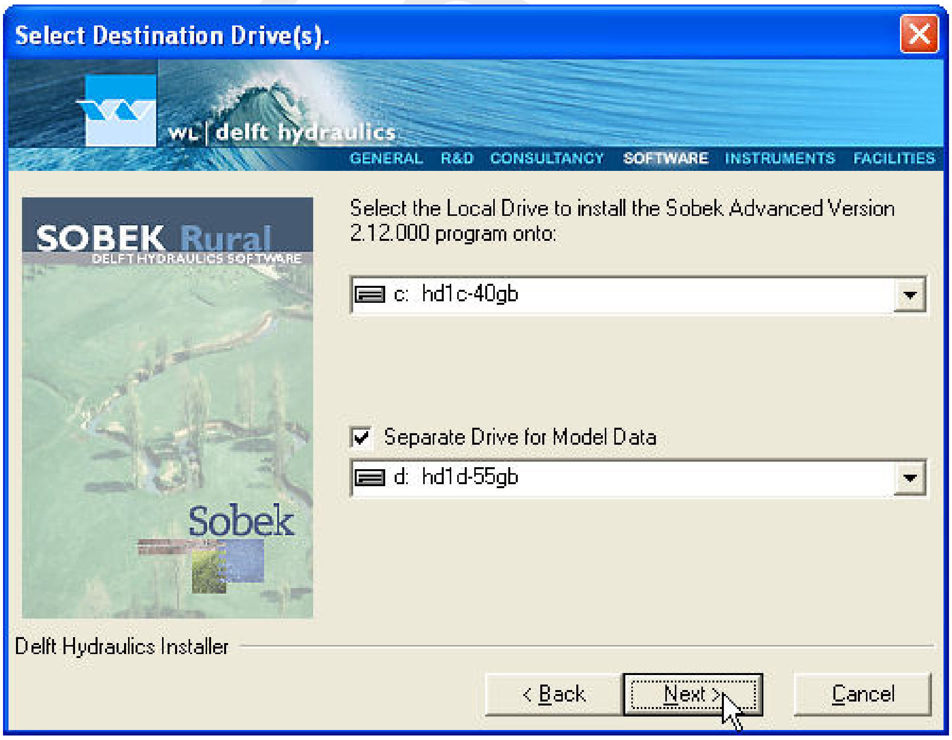


Figure 3.8: *Select destination drive(s)*

◇ Click the *Next* button to got to the Miscellaneous Options dialog.

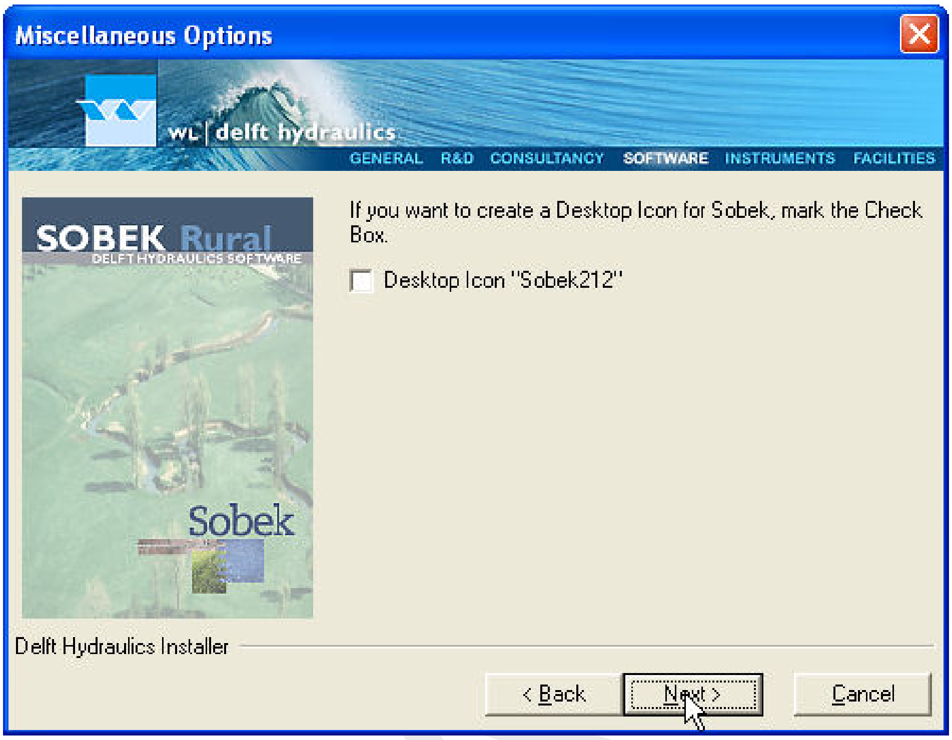


Figure 3.9: *Miscellaneous Options*

At this moment you can specify that you want to create a Desktop Icon to be able to start SOBEK fast. To distinguish the several SOBEK installations, the Installation ID is added to the Icon name. If you want to create the Desktop Icon, mark the checkbox 'Desktop Icon "Sobek..."'.

◇ Click the *Next* button to go to the last pre-install dialog.

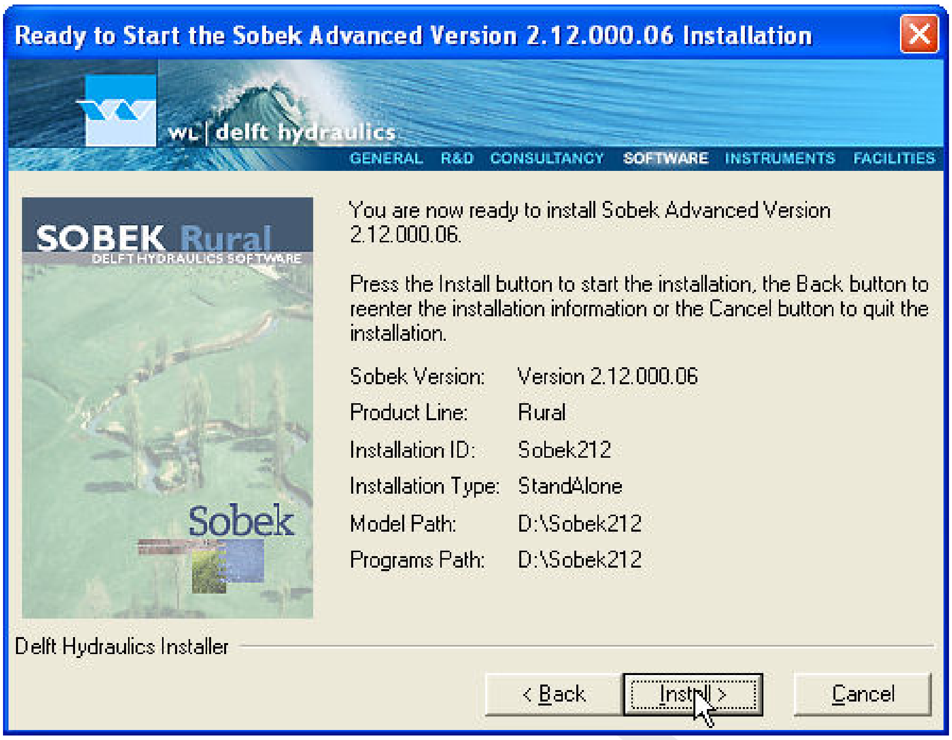


Figure 3.10: *Check installation properties*

The dialog above gives an outline of the options that you selected in the previous steps of the installation. If you find anything incorrect, you can go back by clicking the *Back* button and re-enter the installation information or quit the installation by clicking the *Cancel* button.

If everything looks OK, you can start the installation by clicking the *Install* button. The installation will start now and take some time. The duration depends on the type of installation that you selected and the capacities of your computer. A few minutes is a normal time span.

During the installation the progress can be seen in the window as shown below:

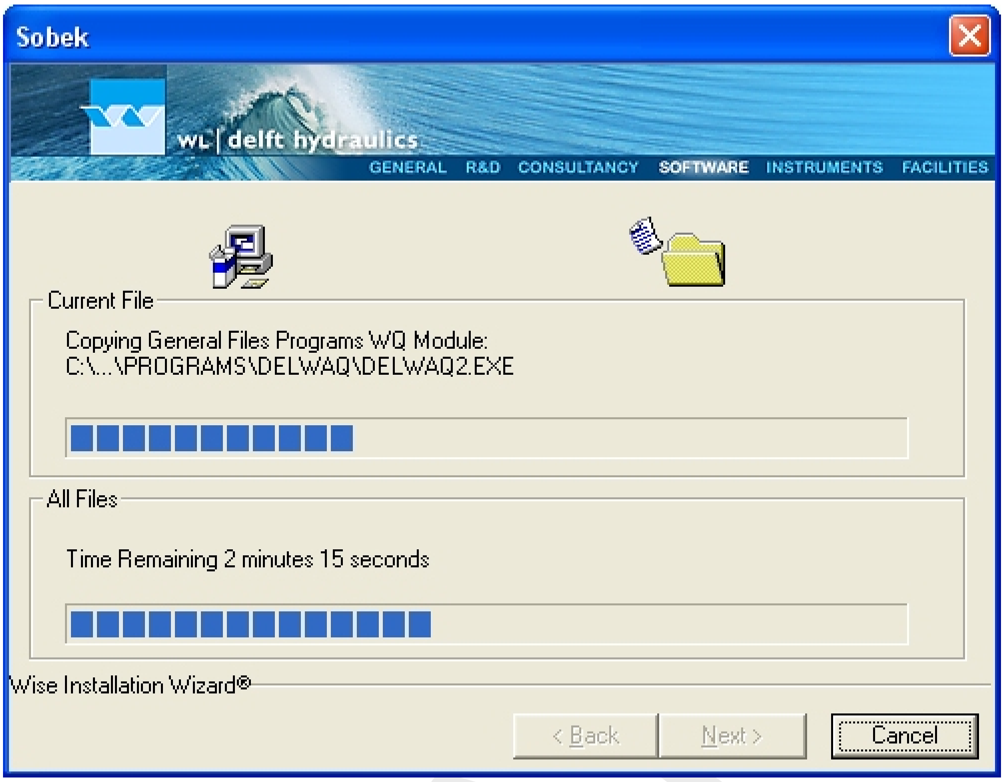


Figure 3.11: *Installation progress bar*

After the installation process has been finished successfully, the following dialog will pop up:

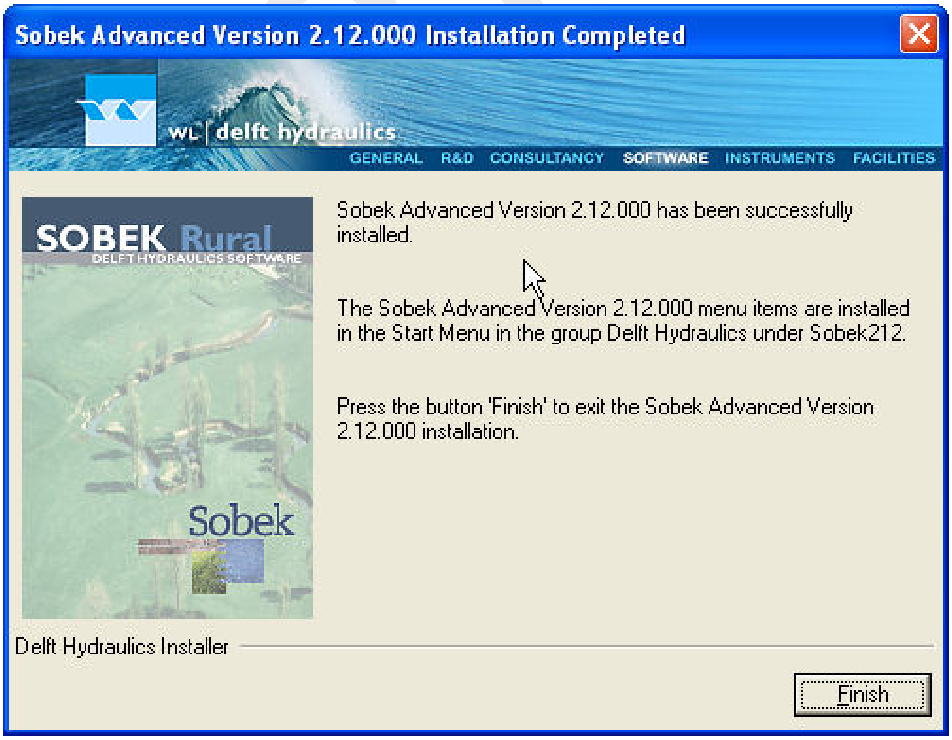


Figure 3.12: *Finish installation window*

◇ Click the <Finish> button to leave the installation program.

You will be asked for a reboot to conclude the installation. It is recommended to do so.

General information
If you have a question about SOBEK for which you cannot find the answer in the on-line manual, you can contact SOBEK Support at Deltares.

You should have the following information ready:

◇ the version number of SOBEK (visible in the upper-left corner of the window after the program is started);
◇ the type of hardware you are using;
◇ the operating system you are using;
◇ the exact wording of any message that appeared on your screen (write it down or take a screenshot);
◇ a description of what happened and what you were doing when the problem occurred;
◇ a description of how you tried to solve the problem;
◇ whether you are able to reproduce the problem by repeating what you did when the problem occurred.

It may also be necessary to send the project data to SOBEK Support. The best way to do this is to close SOBEK, zip the relevant project folder <*.lit> and send it using an e-mail. If the project is too large to be included as an attachment in an e-mail, SOBEK Support can provide the credentials to an ftp account where the data can be uploaded.

3.3.1 Installing the SOBEK using the command line

Below, the various options for installing SOBEK using the command line are detailed. These options are intended for advanced users only. Most of the input validations performed in the user interface version of the SOBEK setup are not performed when using a command line installation.

`/SILENT`	Run silent/unattended.
`/DESKTOP`	If present, a desktop icon will be created.
`/DRIVE=D`	Specifies the drive to install SOBEK on.
`/SBTYP=Rural`	Specifies the SOBEK product line. Default: Rural. Can be Rural, Urban or River.
`/SBID=SOBEK215`	Installation ID and Directory name. E.g. for SOBEK 2.15, use SOBEK215. Because SBID is used as directory name it has to be specified according to the rules for any directory name. Additionally, this ID is limited to a maximum of 8 characters.

For example:
```
SetupSOBEK_2.15.001.exe /SILENT /DESKTOP /DRIVE=D /SBTYP=Rural /SBID=SOBEK215
```

If the switches are specified as above: SOBEK 2.15 Rural will be installed in <D:\SOBEK215> and a desktop icon will be created.

3.4 Starting SOBEK

The installation procedure creates a menu item for SOBEK on the ‘Programs’ menu under the Windows *Start* button. To start SOBEK you must do the following:

- ◇ Click on the Windows *Start* button;
- ◇ Select the ‘Programs’ menu;
- ◇ Select the ‘Delft Hydraulics’ menu;
- ◇ Select the SOBEK menu item;
- ◇ Click on the SOBEK icon.

4 Tutorials

4.1 Tutorial Hydrodynamics in open water (SOBEK-Rural 1DFLOW module)

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.

Getting started

◇ Click the Windows *Start* button.
◇ Select the 'All Programs' or 'All Apps' menu.
◇ Select the 'Delft Hydraulics' menu.
◇ Select the 'SOBEK' menu item (SOBEK215).
◇ 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.
◇ 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.

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.

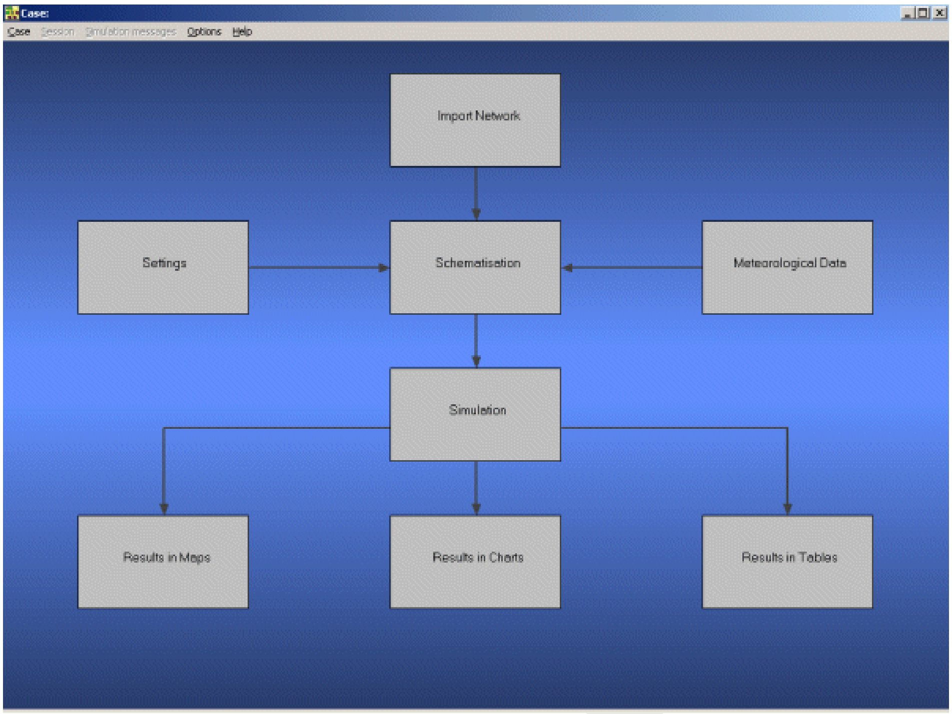


Figure 4.1: *The case manager window.*

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:

1 Administration of cases (which data is 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.);
   ◇ view and check the status of all tasks;
   ◇ view the relation between the various tasks.
   ◇ choose and run predefined tasks (modules);

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:

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 again when the task is finished.

Now, we will discuss each task block.

4.1.1 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:

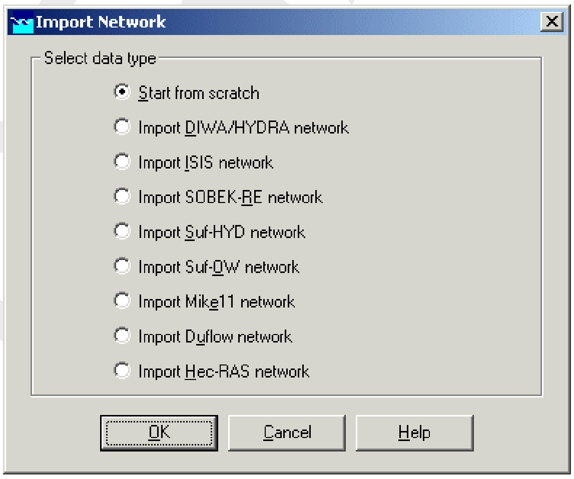


Figure 4.2: *The import network window.*

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.

4.1.2 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.

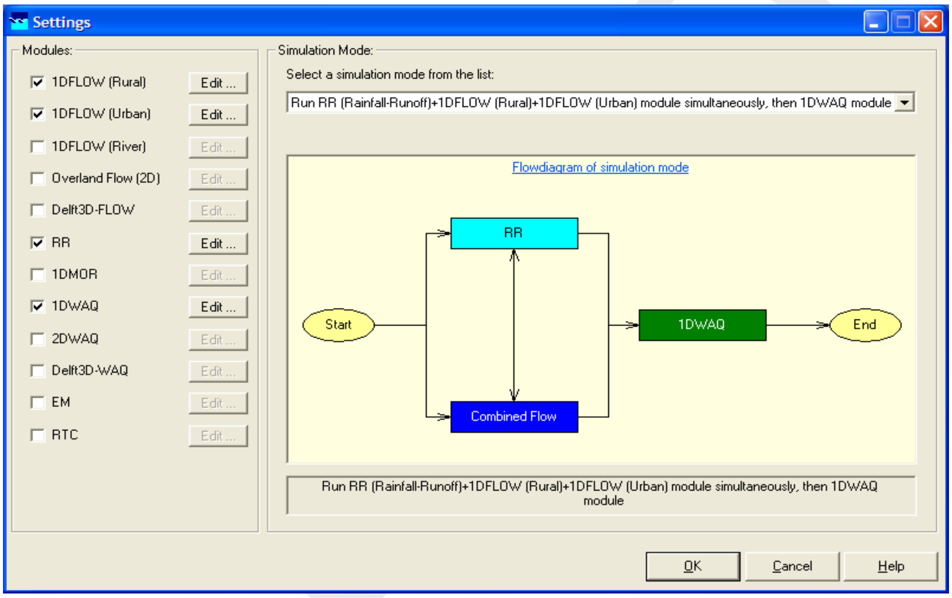


Figure 4.3: *The settings window.*

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'.
- ◇ Set the 'time step in computation' to 10 minutes (type “10” in the 'min' edit box and “0” in the others).
- ◇ 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:

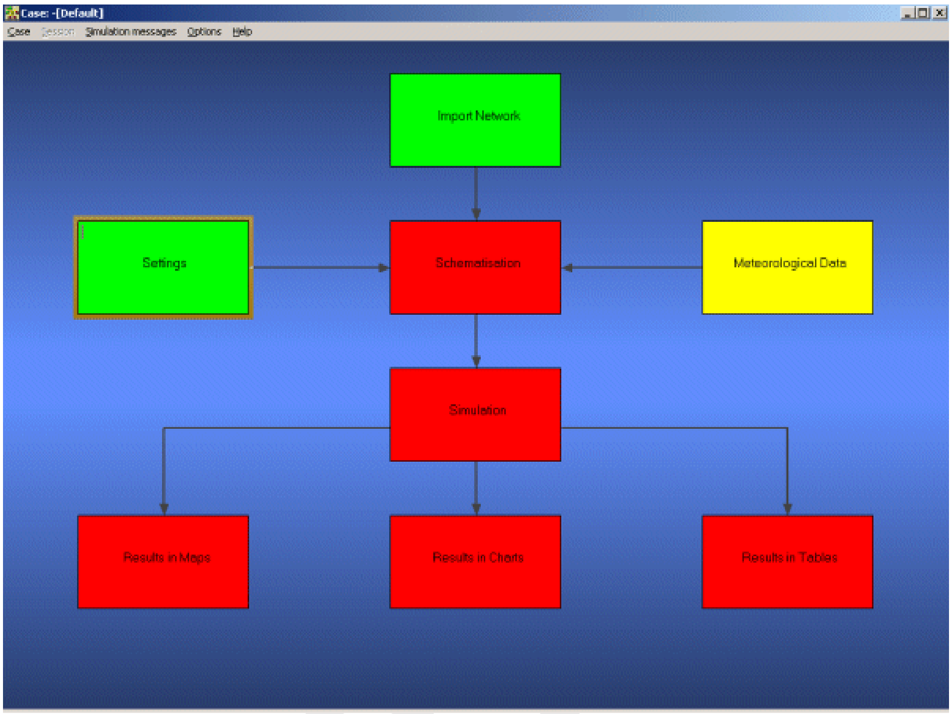


Figure 4.4: *The case manager after completing the 'settings' and 'import network' tasks.*

4.1.3 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:

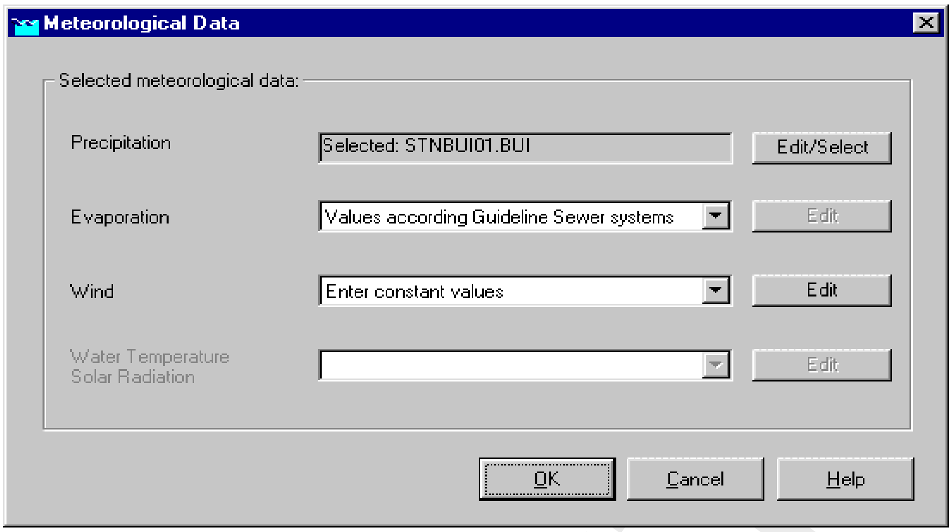


Figure 4.5: *The meteorological data window.*

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!

4.1.4 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.

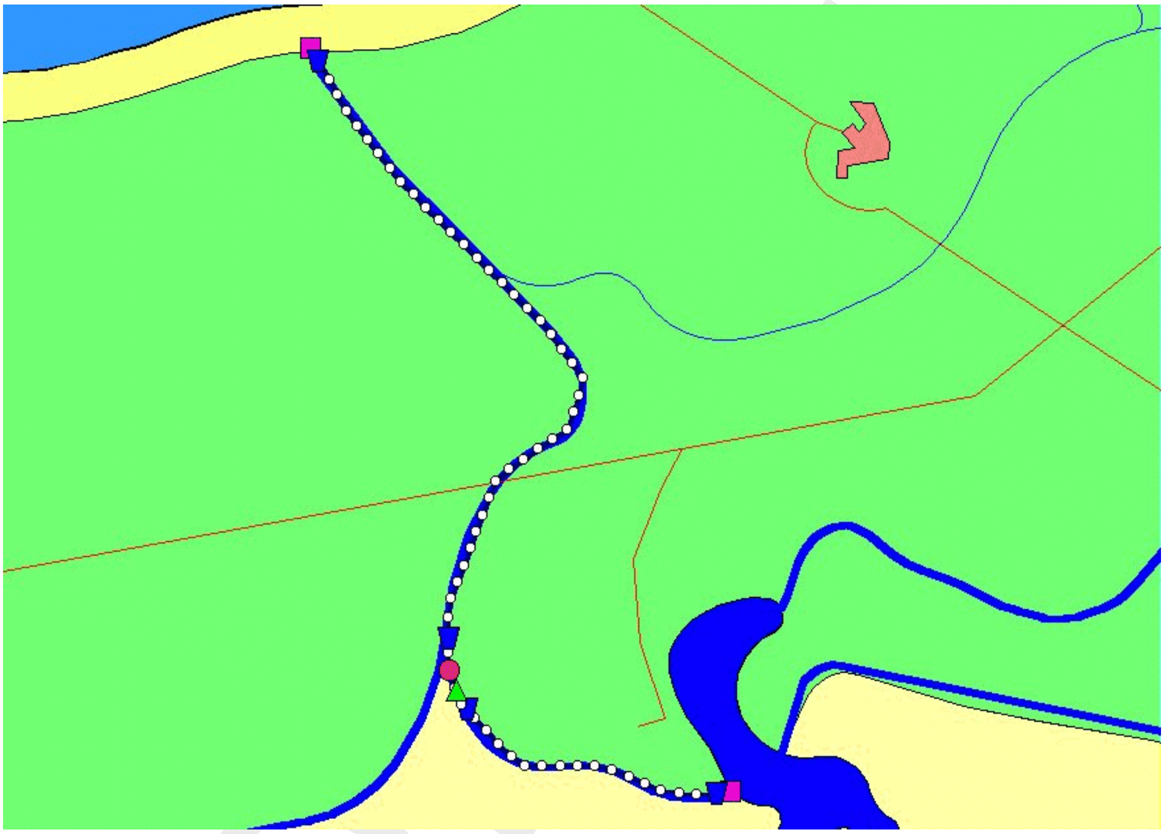


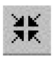
Figure 4.6: *The schematisation to be created in this tutorial.*

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 No edit action 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:

Figure 4.7: *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.

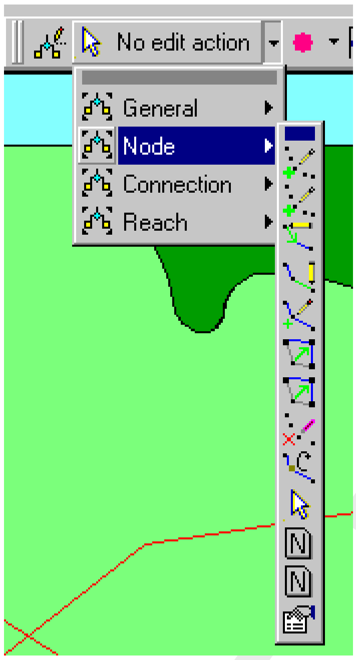


Figure 4.8: *Node functions toolbar*

◇ 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'.
◇ Set the 'ID' and 'Name' to 'automatic'.
◇ 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:

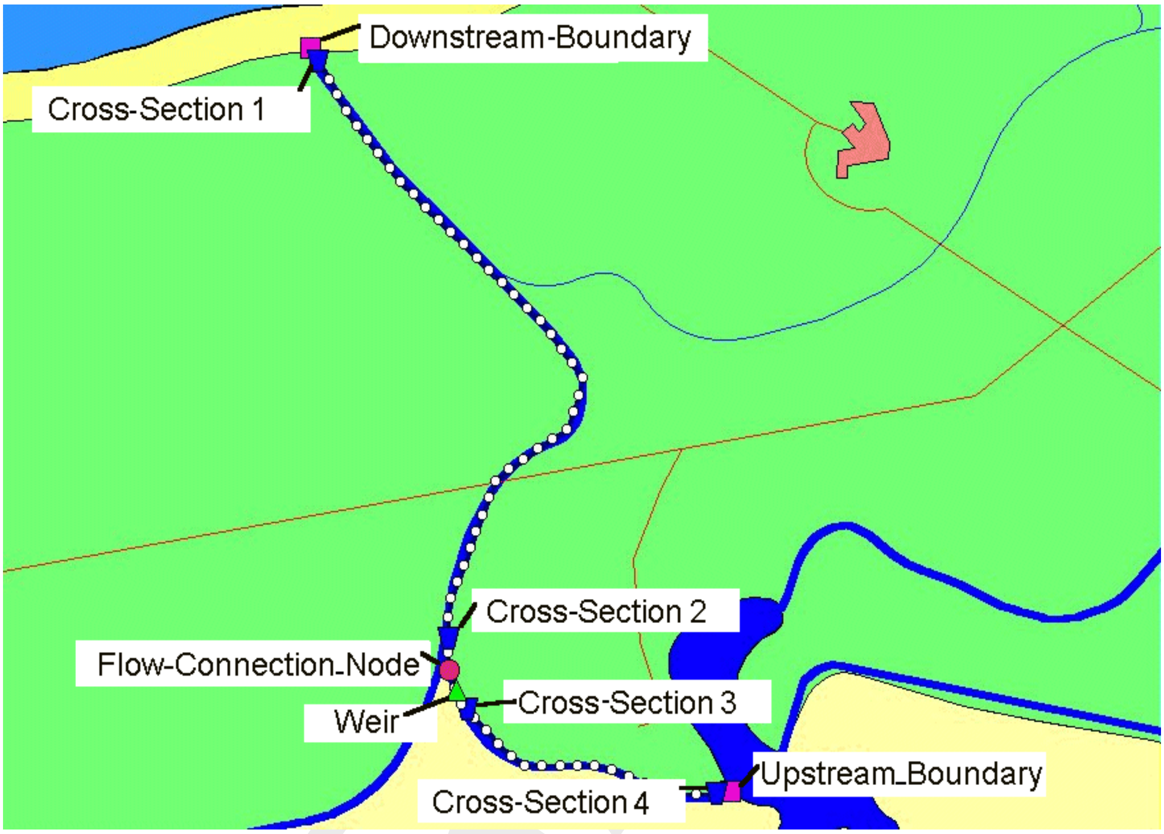
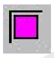


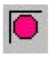
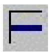
Figure 4.9: *Create a network similar to this one*

- ◇ 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.

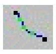
- ◇ Select the  node, Flow-Connection Node.
- ◇ Select the  button again.
- ◇ Enter 'Flow-Connection_Node' in both input fields.
- ◇ Click the *OK* button.
- ◇ Click on the left-mouse button to actually add the Flow-Connection node on your screen.
- ◇ Select the  button, Flow-Channel.
- ◇ Select the  button to select the function 'Connect nodes'.
- ◇ Click with the left mouse button on the upstream boundary node, and drag to the connection node while keeping the button pressed. Release the left mouse button to add the connection.
- ◇ Click with the left mouse button on the connection node and drag to the downstream boundary node while keeping the button pressed. Release the left mouse button.

Now the two boundary nodes and the connection node are connected.

The model will need a calculation grid. We will switch off the automatic generation of names before generating this grid because we do not want to generate calculation points with names for this tutorial.

- ◇ 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.

Now we can view the defined direction of the links on which we just generated a calculation grid. The option 'defined direction' can be used to see the positive defined direction. This option is enabled by default, and can also be manually enabled by following these steps:

- ◇ Select 'Options' - 'Network Data...'.
- ◇ Select the tab 'Link'.
- ◇ In the data group 'Show Direction', select the radio button *Defined*.
- ◇ Click the *OK* button.

To disable titles on links, if enabled:

- ◇ Select 'Options' - 'Network Data...'.
- ◇ Select the tab 'Link'.
- ◇ Under 'Show Titles', Select the radio button *None*.

The vector layer:

To show schematizations with high performance, by default NETTER shows 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 either of the two reaches.
- ◇ 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-Section1" 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 as shown in the example Figure 4.9.
- ◇ 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.

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 on the node with the left mouse button).
◇ 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 (by using the left mouse button) the downstream Flow-Cross section node Cross-Section1.
◇ 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”.
◇ Click the *Save dimensions* 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 'Chézy (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-Section2 (second from downstream)	-1	4	My profile 1	local, Chézy 35	2m water level
Cross-Section3 (third from downstream)	-1	4	My profile 1	local, Chézy 35	3m water depth
Cross-Section4 (fourth from downstream)	5	9	My profile 1	local, Chézy 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-Section2.
◇ Enter "4" as Surface Level [m AD] of Cross-Section2.
◇ Enter "-1" as Reference Level [m AD] of Cross-Section3.
◇ Enter "4" as Surface Level [m AD] of Cross-Section3.
◇ Enter "5" as Reference Level [m AD] of Cross-Section4.
◇ Enter "9" as Surface Level [m AD] of Cross-Section4.

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.

To stop showing Reference levels on the map:

- ◇ Save the newly added data by choosing 'File' - 'Save Data' within the Multiple Data Editor.
- ◇ Close the Multiple Data Editor by selecting 'File' - 'Exit'.
- ◇ Click on 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.

Now we will add the friction data.

- ◇ Select the button, 'Select by rectangle'.
- ◇ Select the whole schematisation again by dragging a rectangle around it.
- ◇ Select 'Model data' - 'Flow Model' to open the multiple data editor.
- ◇ Select 'Flow - Cross Section'.
- ◇ Select 'Friction'.
- ◇ In the remaining reach: Select Friction Type 'Chézy'.
- ◇ Enter Value: "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.
- ◇ Set the 'Discharge m3/s' for this reach to '0'.
- ◇ Select 'Depth' as the type of the initial level for the upstream reach.
- ◇ Enter Value: "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.

4.1.5 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.

4.1.6 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.

4.1.7 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 have selected the item '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 'Link'.
- ◇ 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.

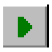
- ◇ Select 'Tools' - 'Output options'.
- ◇ Select 'Flow module' from the 'Module' list.
- ◇ Select 'water level' from the 'Output title' list.
- ◇ Select 'maximum' from the 'Function' list.
- ◇ Press *Add* button.
- ◇ Select 'Flow module' from the list again.
- ◇ Select 'water depth' from the list.
- ◇ 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:

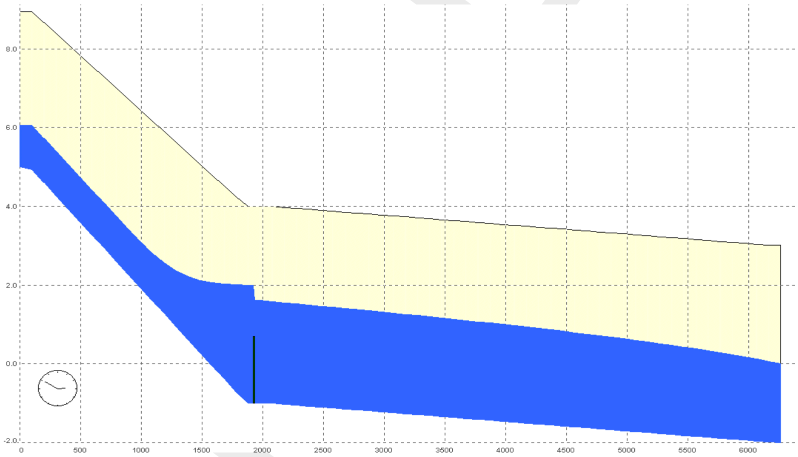


Figure 4.10: *An example of a side view animation.*

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 [m^3/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'.

4.1.8 Task block: Results in Tables

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

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

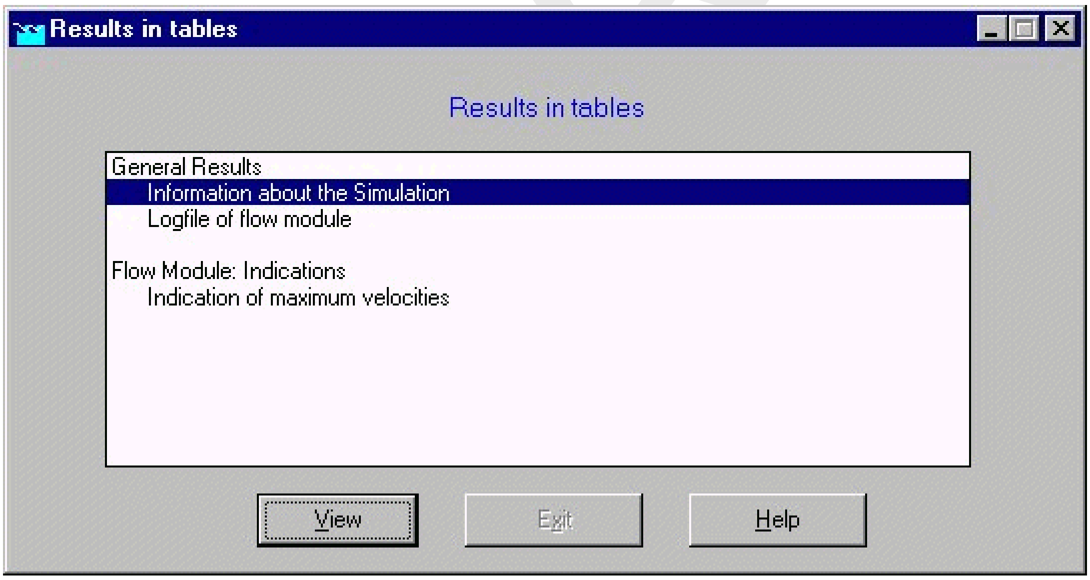


Figure 4.11: *The results in tables window.*

◇ 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.

4.1.9 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 (1 000 m3)'.
- ◇ 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'.

4.1.10 Interpolation over a Connection Node

In order make it possible to easily add branches to a main drainage system, Flow - Connection Nodes offer user-definable interpolation of cross-section bathymetry data. To demonstrate this functionality we will add a torrent to the existing schematisation.

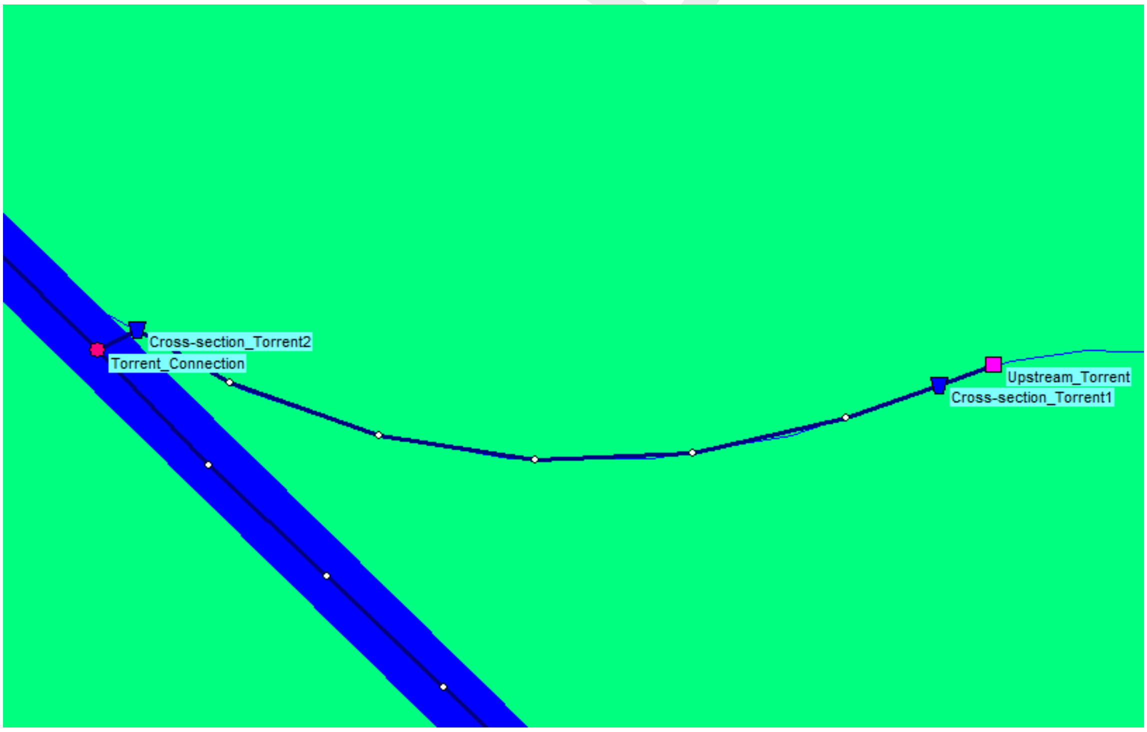
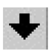


Figure 4.12: *The torrent branch.*

- ◇ 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.

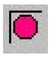
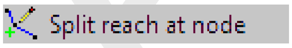
- ◇ Press *OK* button.

Now we will add the Flow - Connection node with which we connect the torrent to the existing schematisation. A connection node can only be added to an existing reach by splitting the reach. We will manually add a h-calculation point and split the reach at its location:

- ◇ Select the  button, Edit Network, to start the edit network mode.
- ◇ Select the  button, Flow - Calculation Point.
- ◇ Select the  button to select the function 'Add node'.
- ◇ Enter "Torrent_connection" in both input fields.
- ◇ Click the *OK* button.
- ◇ Click on the left-mouse button again to actually add the Torrent_connection node on your screen. The node should be placed in the intersection between the main river and the torrent.

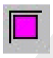
Remark:

- ◇ It is possible that a h-calculation point is already present at the intersection between the main river and the torrent in your tutorial model. In that case, it is also possible to split the reach at the automatically generated calculation node. However, it should be noted that this node will not have a name, and have an automatically generated ID. For the purpose of this tutorial, it is assumed that the 'Torrent_connection' node was manually added as described above.

- ◇ Now select the  button, Flow - Connection node.
- ◇ Select the button 'Split reach at node' 

in the 'Reach' toolbar.
- ◇ Use the left-mouse button on the Torrent_connection node to actually add the Flow - Connection node on your screen.

The h-Calculation Point is now changed to a Connection Node, splitting the reach and providing a location to which we can connect the torrent section of this network.

- ◇ 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 Torrent_connection node and the Flow - Boundary node.

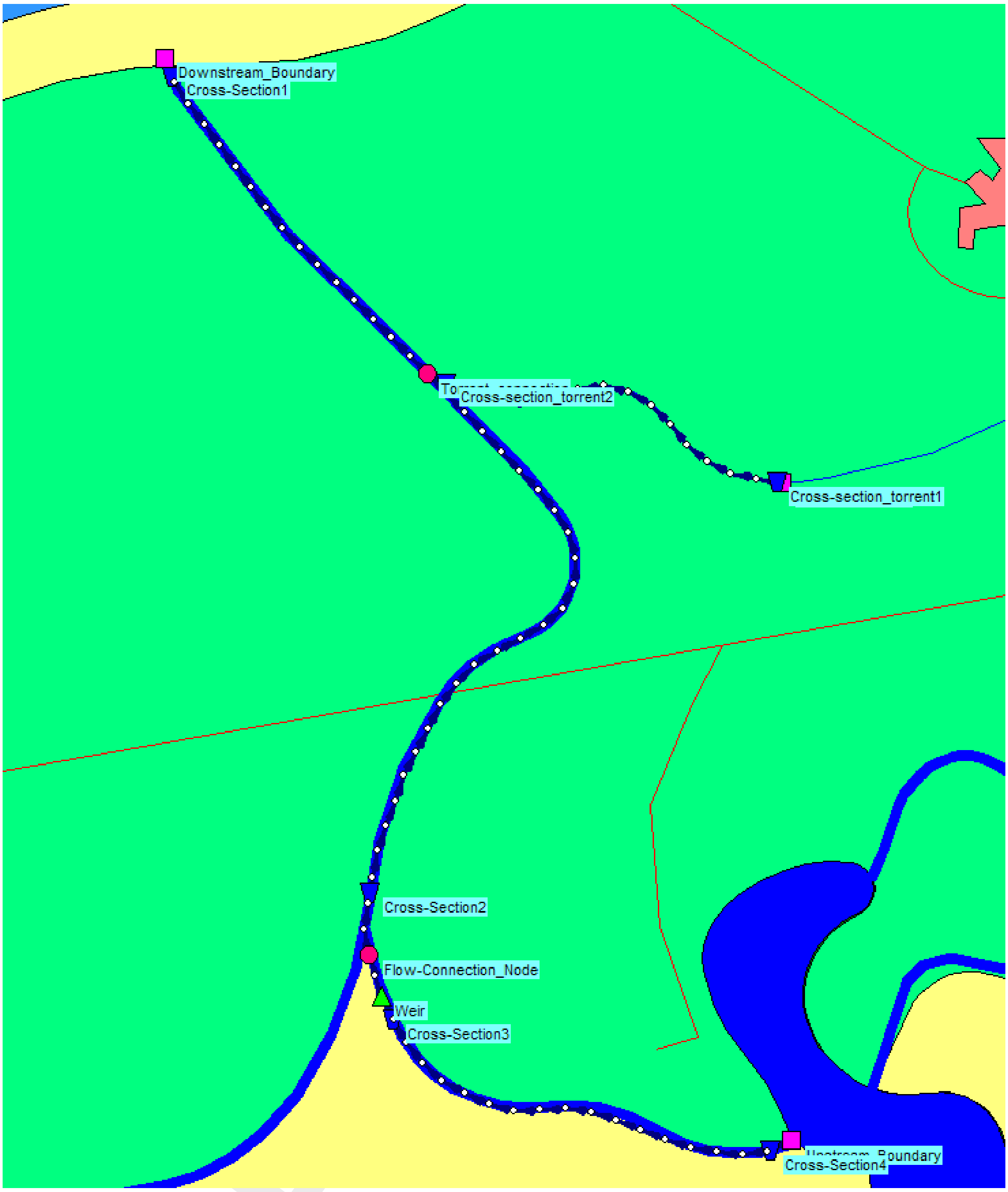


Figure 4.13: *The entire network after adding the torrent branch*

- ◇ 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 Torrent_connection 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.

Now we should enable interpolation along the connection node ’Torrent_connection’. When we split the reach earlier, the upstream and downstream parts of the main river were also split in two. For this tutorial we want the main river to be treated as a single river, as it was before we split the reach.

◇ Select the menu item ’Options’ - ’Network Data...’.
◇ Select the tab ’Link’.
◇ Select the radio button *Reach*.
◇ Press the *OK* button.

The reach ID’s are now shown in the schematisation. This is useful when enabling interpolation along a connection node.

◇ Select the Torrent_connection by left-mouse clicking it
◇ Click with your right mouse button.
◇ Select ’Model data’ - ’Flow Model’.
◇ Left-mouse click the tab ’Interpolation Over node’.
◇ Enable the option ’Interpolate Cross-section bathymetry (only) over Node’ by left-mouse clicking it.
◇ For ’First Reach’, select the upstream reach ’2’.
◇ For ’Second Reach’, select the downstream reach ’3’.
◇ Press the *OK* button.

Interpolation of Cross-section data over the Torrent_connection is now enabled.

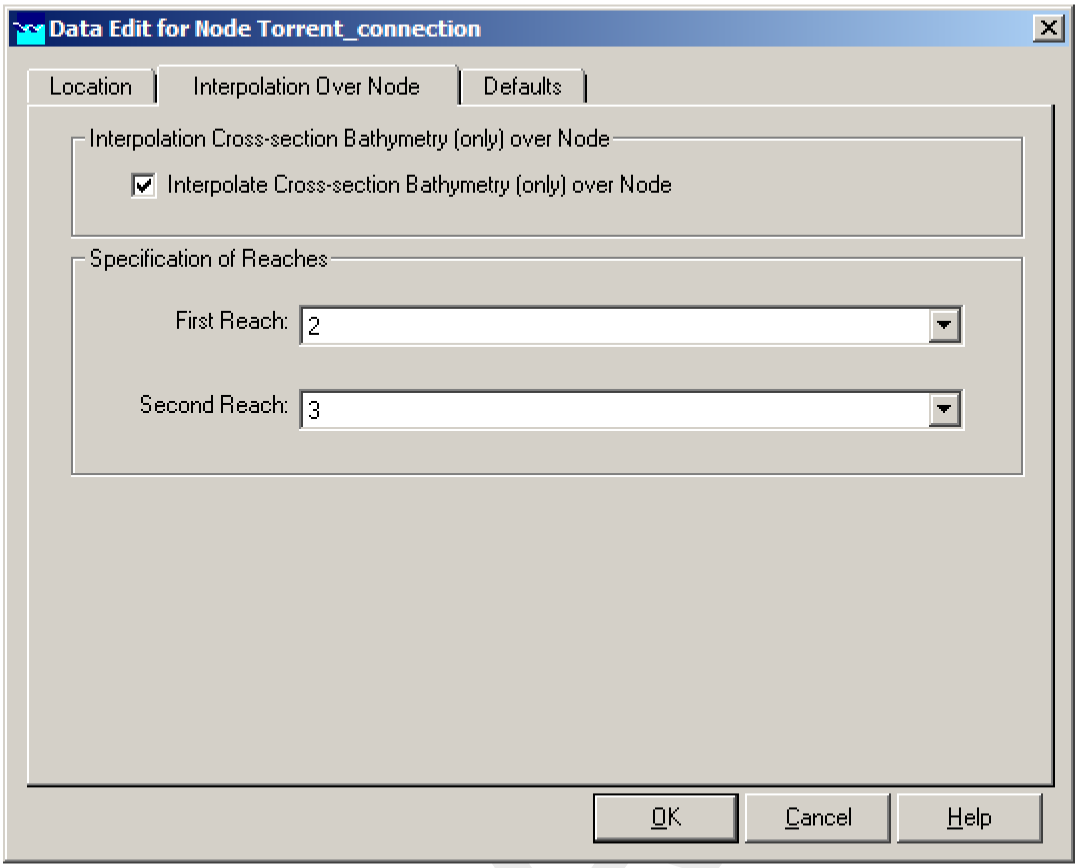


Figure 4.14: *The filled in Data Edit screen for the 'Torrent_connection' node*

We will now disable the showing of reach ID's in NETTER:

- Select the menu item 'Options' - 'Network Data...'.
- Select the tab 'Link'.
- Under 'Show Titles', Select the radio button *None*.
- Press 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 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 the Torrent_connection 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 dragging 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 ’Chézy’.
◇ 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 “0” for ’Discharge’.
◇ Enter “1.25” for "Value".
◇ 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.

4.1.11 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 when using interpolation of cross-section bathymetry data over a Connection Node. SideView is a good tool for this.

Epilogue

In this tutorial the most important aspects of working with SOBEK have been discussed. Extended documentation can also be found in the SOBEK Manual. All PDF files containing documentation can be found in the "Manuals" directory in the Start menu, next to the SOBEK start icon. Since you have now gained experience it will not be that difficult to find out the options and possibilities of SOBEK that not have been discussed here. Good luck!

4.2 Tutorial Hydrodynamics in sewers (SOBEK-Urban 1DFLOW + RR modules)

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.

Getting started

- ◇ Click on the Windows *Start* button.
- ◇ Select the 'All Programs' or 'All Apps' menu.
- ◇ Select the 'Delft Hydraulics' menu.
- ◇ Select the 'SOBEK' menu item (SOBEK215).
- ◇ 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.
- ◇ 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.

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.

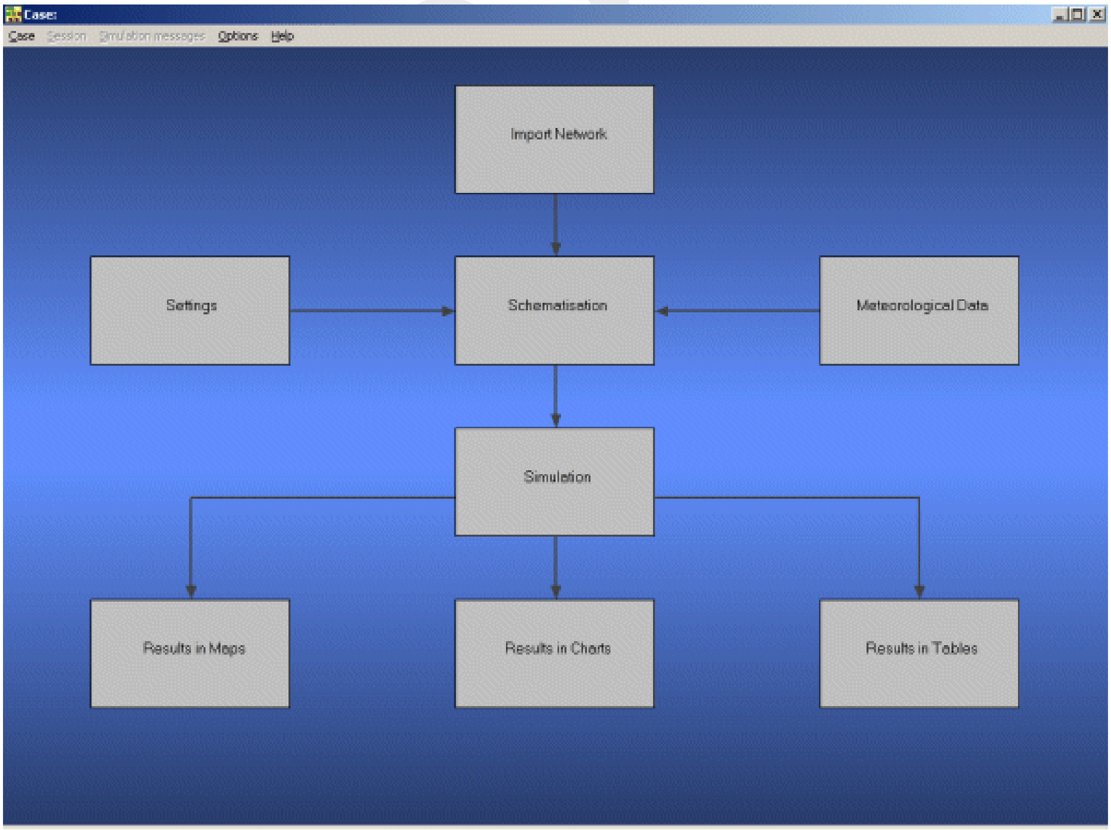


Figure 4.15: *The case manager screen.*

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 is 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.);
    - ◇ view and check the status of all tasks;
    - ◇ view the relation between the various tasks.
    - ◇ choose and run predefined tasks (modules);

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.

4.2.1 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:

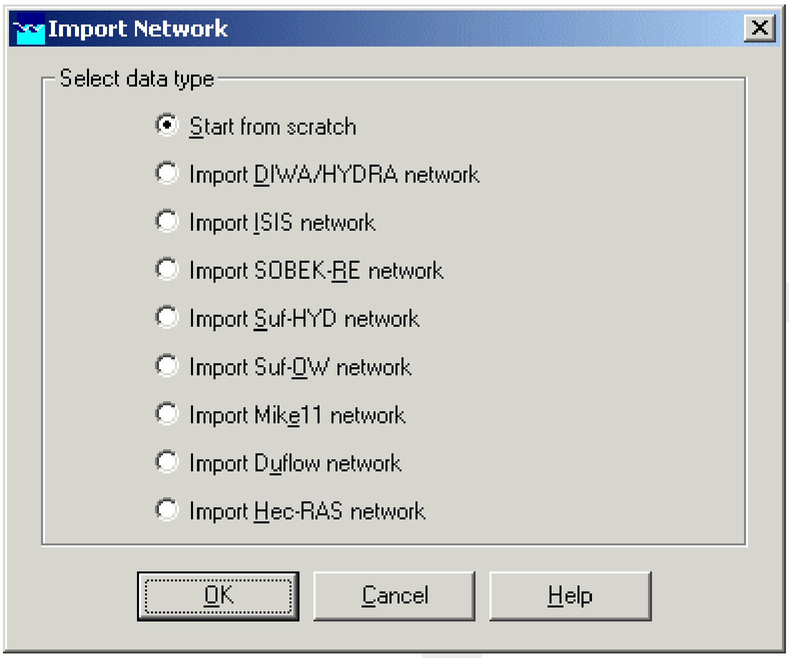


Figure 4.16: *The import network window.*

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.

4.2.2 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.

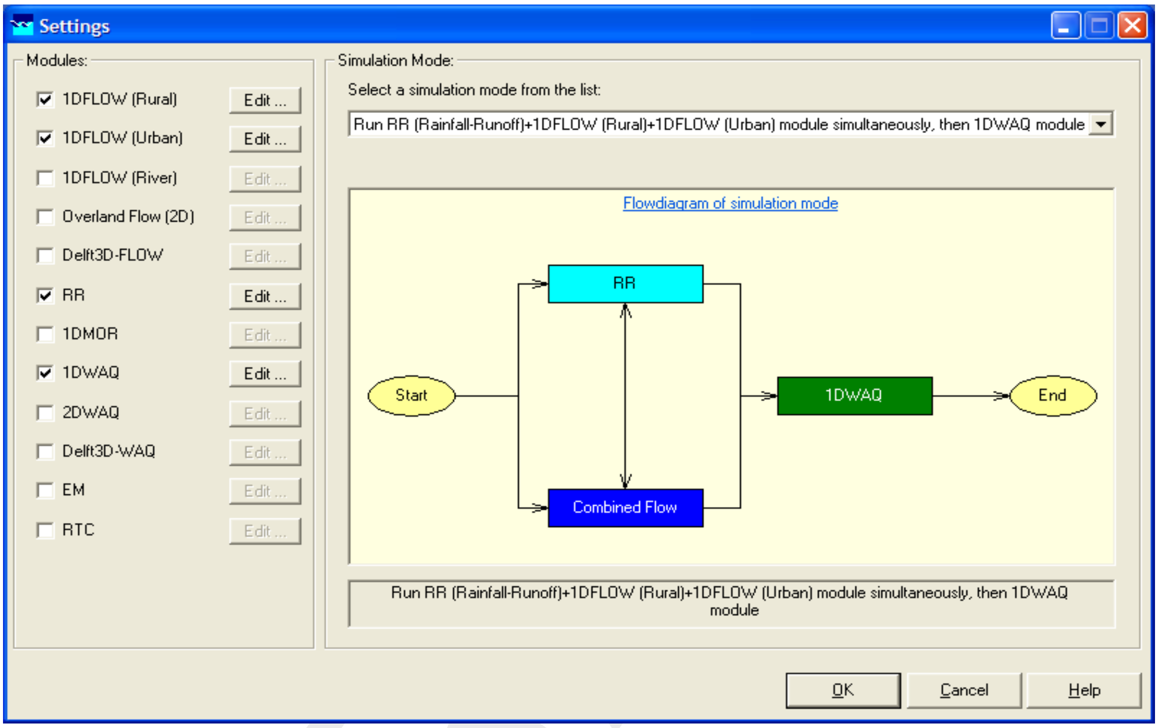


Figure 4.17: *The settings window.*

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 waste water 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'.

- ◇ Double-click the 'Settings' task block.
- ◇ Unselect all the selected modules if any.
- ◇ Select the '1DFLOW (Urban)' module.
- ◇ And select the 'RR' module.
- ◇ 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.
◇ 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*.
◇ Select the 'Output options' tab.
◇ Enter the output time step in computation: “1” minute.
◇ 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.

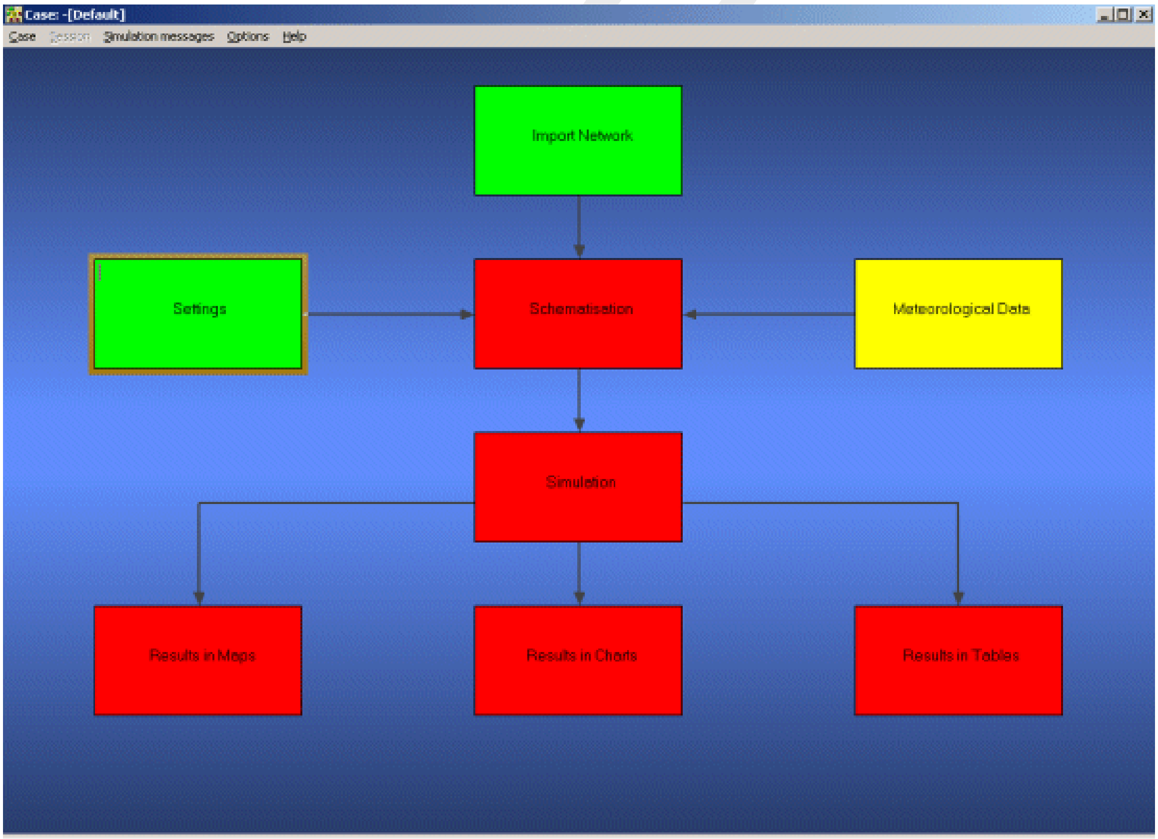


Figure 4.18: *The case manager after completing the 'settings' and 'import network' tasks.*

4.2.3 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:

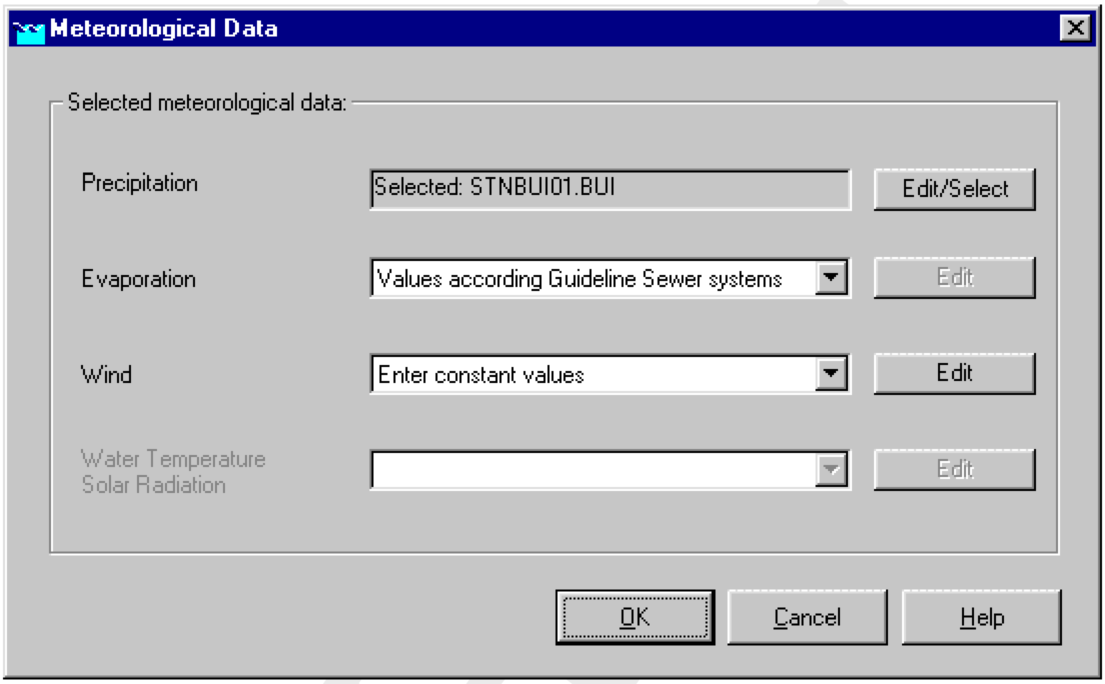


Figure 4.19: *The meteorological data window.*

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

4.2.4 Task block: Schematisation

A schematisation can easily be set up with the help of the network editor.

◇ 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 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 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.

Figure 4.20: *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 dragging 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 node data'.
◇ Press the *OK* button.

The following figure appears:

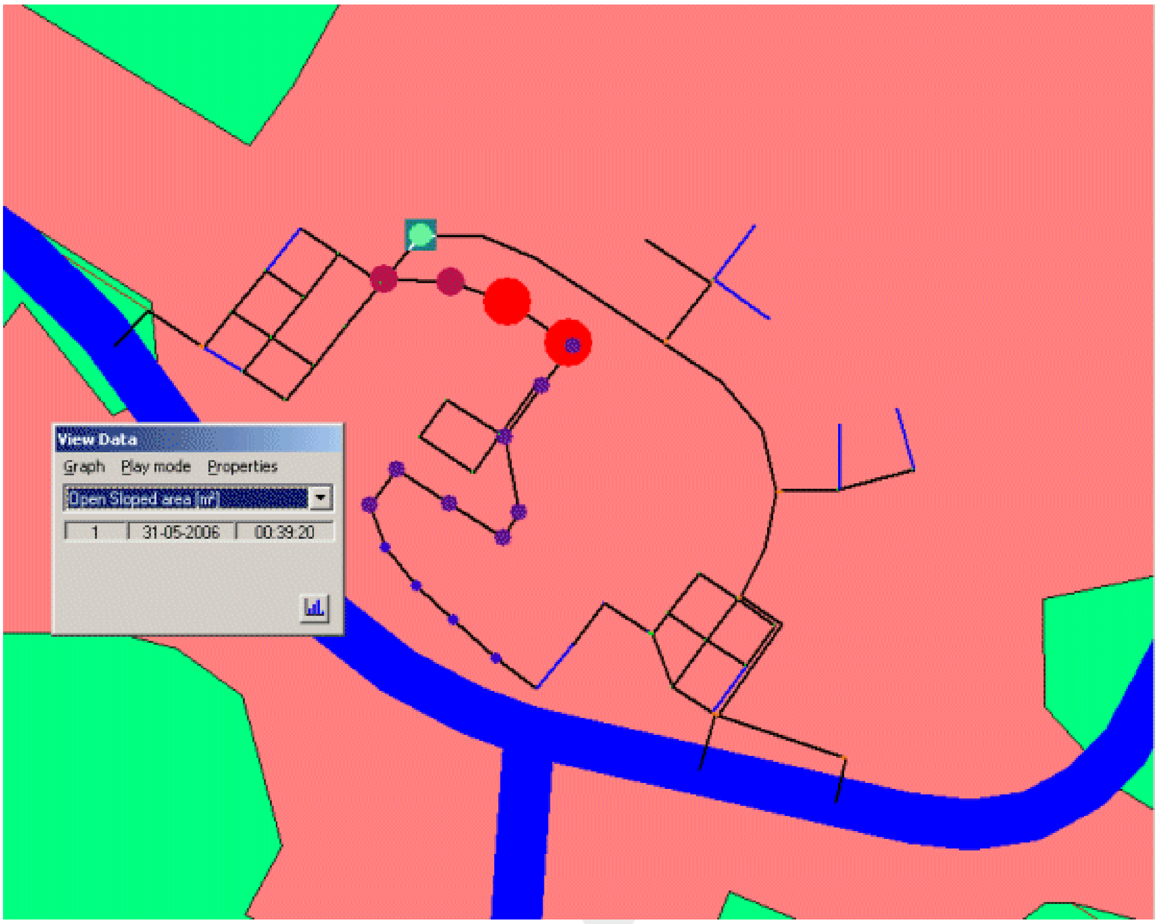


Figure 4.21: *Open sloped area*

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!

Now we will close the Multiple Data Editor and we will return to the default visualisation mode.

- ◇ Click on the  button in the Active Legend.
- ◇ Select the ’All Data’ tab.
- ◇ Unselect ’Width’ of ’Show node data’.
- ◇ Press the *OK* button.
- ◇ Select ’File’ - ’Exit’ in the Multiple Data Editor.

The identifiers of all objects can be viewed on the map, using the following options:

- ◇ Click on the  button in the Active Legend.
- ◇ Select the ’Link’ tab.
- ◇ Select ’ID’.
- ◇ Press the *OK* button.

Now we are going to provide street names to Flow - Pipes:

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

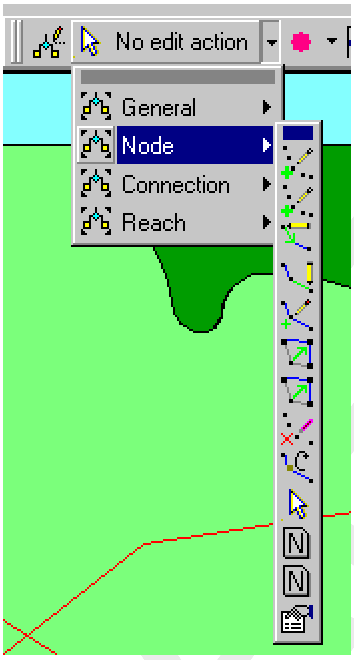


Figure 4.22: *Node functions toolbar*

◇ Select the network editing option 'Properties'.
◇ To search the Flow - Pipe with ID 00-1011-00-1010, press the `Ctrl` key plus the `F` key.
◇ Click 'ID' of 'Link'.
◇ Enter "00-1011-00-1010" (without the quotes).
◇ Press the *OK* button.
◇ Now click on the selected Flow - Pipe on the map using your mouse.
◇ Enter the Name "Market Street".
◇ Press the *OK* button.
◇ Press the `Ctrl` key plus the `F` key.
◇ Enter "00-1010-00-1009".
◇ Click on the selected Flow - Pipe on the map.
◇ Enter the Name "Market Street" again.
◇ Press the *OK* button.
◇ Now rename the Flow - Pipes '00-1009-00-1008', '00-1008-00-1007', '00-1007-00-1006' and '00-1006-00-1005' to 'Market Square'.
◇ And the Flow - Pipes '00-1005-00-1004' and '00-1004-00-1003' to 'Cherry Street'.

The Names can be viewed on the map, using the following options:

◇ Click on the button in the Active Legend.
◇ Select the 'Link' tab.
◇ Select 'Name'.
◇ Press the *OK* button.

Now your model will look as depicted in Figure 4.23.

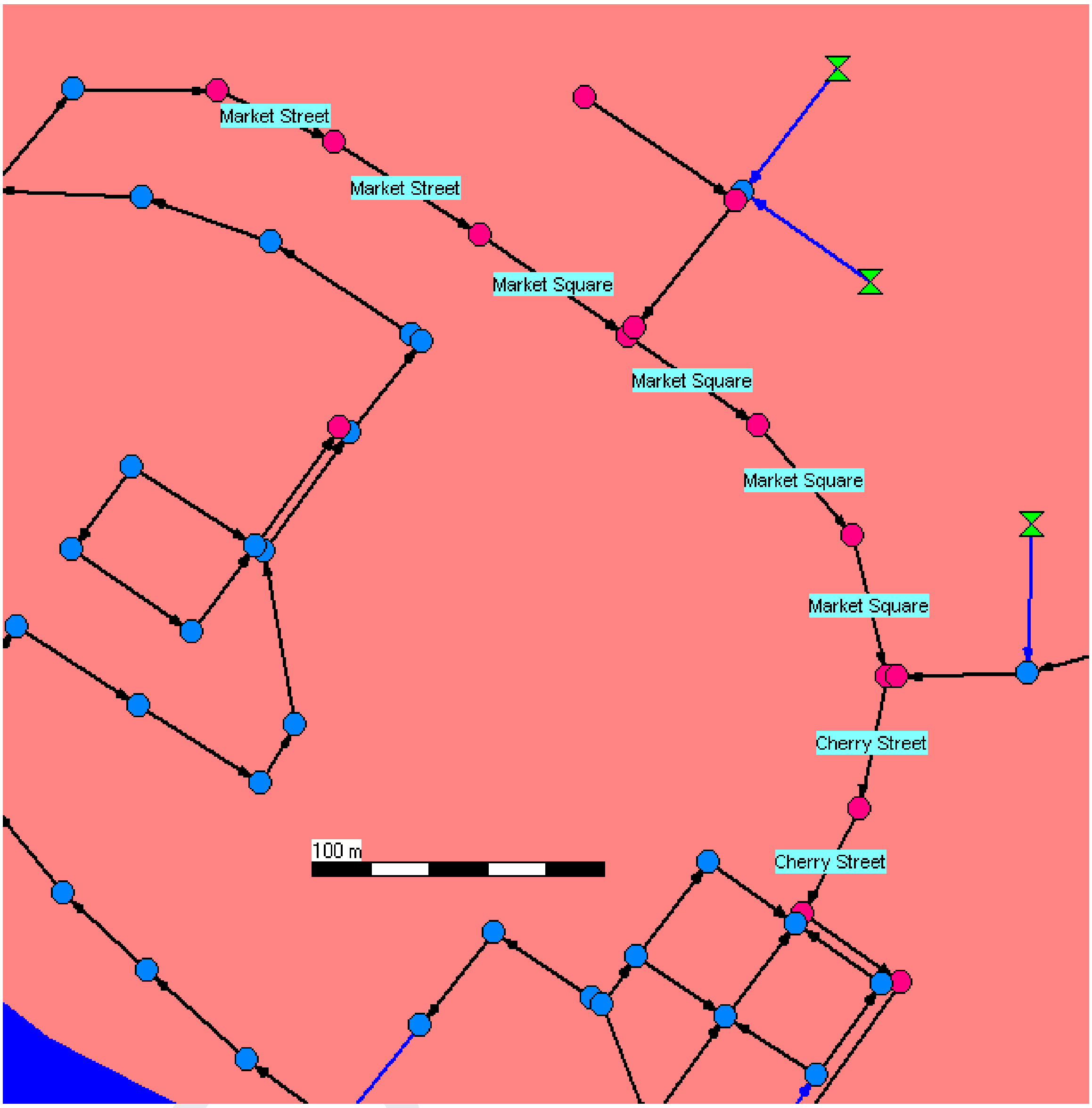


Figure 4.23: *Reach segment names visualised on the map*

Another way to get good insight in the schematisation is make a side view. Note that the street names, used more than once, are displayed only once for the whole street.

- Select the network editing option 'No edit action' to return to using the normal cursor.
- Click on the  button in the Active Legend.
- Select the 'Node' tab.
- Select 'ID'.
- Press the *OK* button.
- To search for the objects with the ID 00-10003 and ID 00-8002, press the `Ctrl` key plus the `F` key
- Click 'ID' of 'Node'.
- Enter "00-10003"
- Press the *OK* button.
- Search for the object '00-8002' too.

◇ Click simultaneously the `Shift` key on the keyboard and select the Flow - External Weir with the ID 00-10003 with the left mouse button. Keep the `Shift` key pressed. Then click the Flow - External Weir with the ID 00-8002. Now, you can release the `Shift` key.
◇ 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.

◇ Select 'Options' - 'Sideview Settings ..'.
◇ Select the tab 'Branch Settings'.
◇ Select the checkbox 'Show branch labels'.
◇ Select the item 'Branch Name'.
◇ Click the *OK* button.

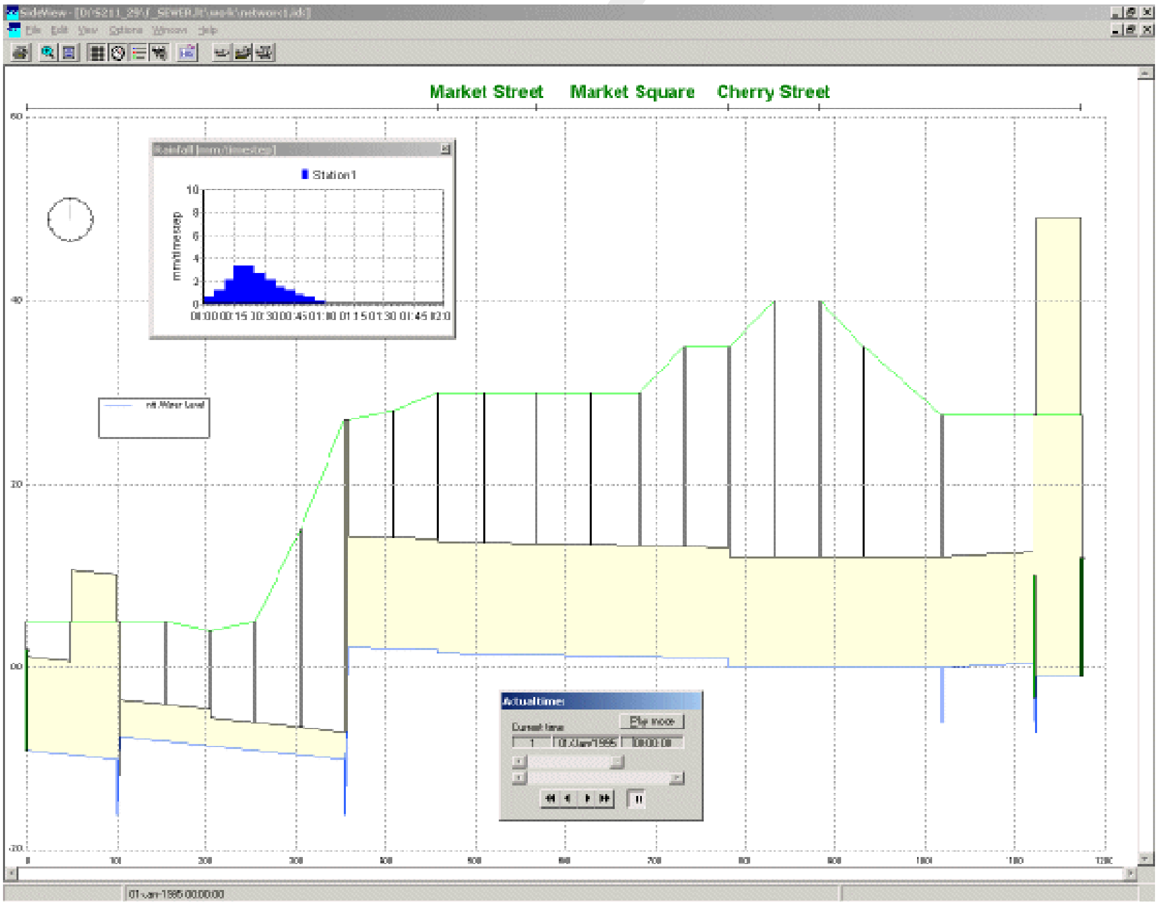


Figure 4.24: *Side view*

You can save this side view path:

◇ Select 'File' - 'Save'.
◇ Enter the file name "Path1.ids".
◇ Press the *Save* button.
◇ Select 'File' - 'Exit'.

We will re-use this path later in this tutorial.

Now we will disable the showing of Link and Node ID's and names:

◇ Click on the  button in the Active Legend.
◇ Select the 'Node' tab.
◇ Select 'None'.
◇ Select the 'Link' 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.

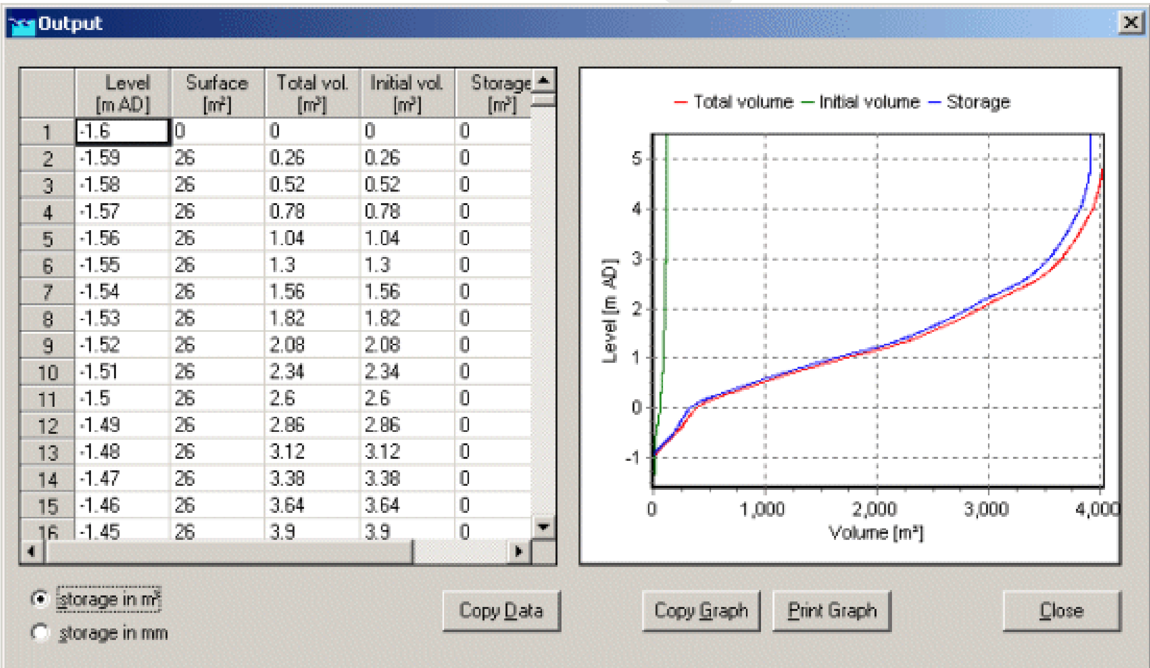


Figure 4.25: *Storage graph*

◇ Click the *Close* button.
◇ Press the *Exit* button.

To start the simulation you will have to leave NETTER.

◇ Select the menu item 'File'-'Exit', to leave NETTER.
◇ Click the *Yes* button to save the name changes.
◇ Click the *OK* button.

◇ Select the menu item 'Case'-'Save As...'
◇ Enter the name "Case_one" to save the case.

◇ Click the *OK* button.

4.2.5 Task block: Simulation

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

◇ Double-click the task block 'Simulation'.

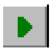
You will see a bar appearing, showing the progression from the simulation. After the simulation has finished, the Results in Maps, Results in Charts and Results in Tables task blocks will become yellow.

4.2.6 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 Figure 4.26.

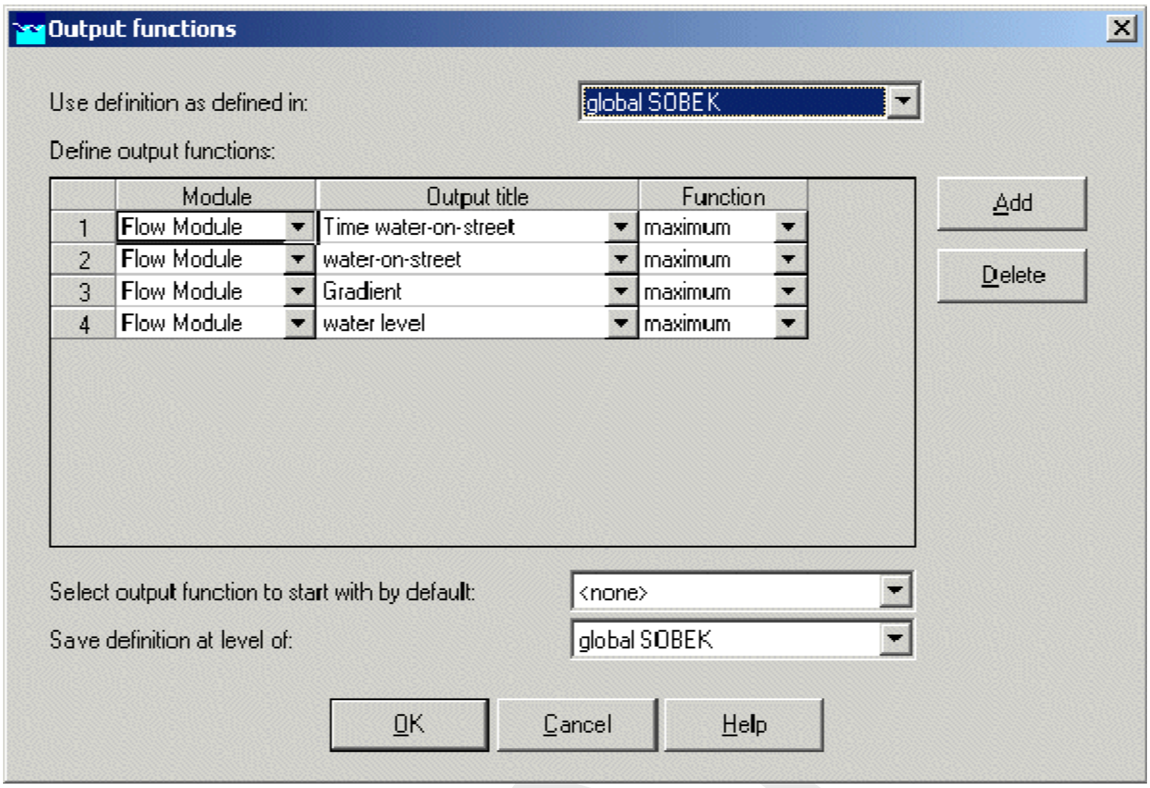


Figure 4.26: *User Defined Output functions*

- ◇ Press the *OK* button.
- ◇ In 'User Defined Output' of the Active Legend, select 'Time water-on-street, maximum'.

The following figure appears:

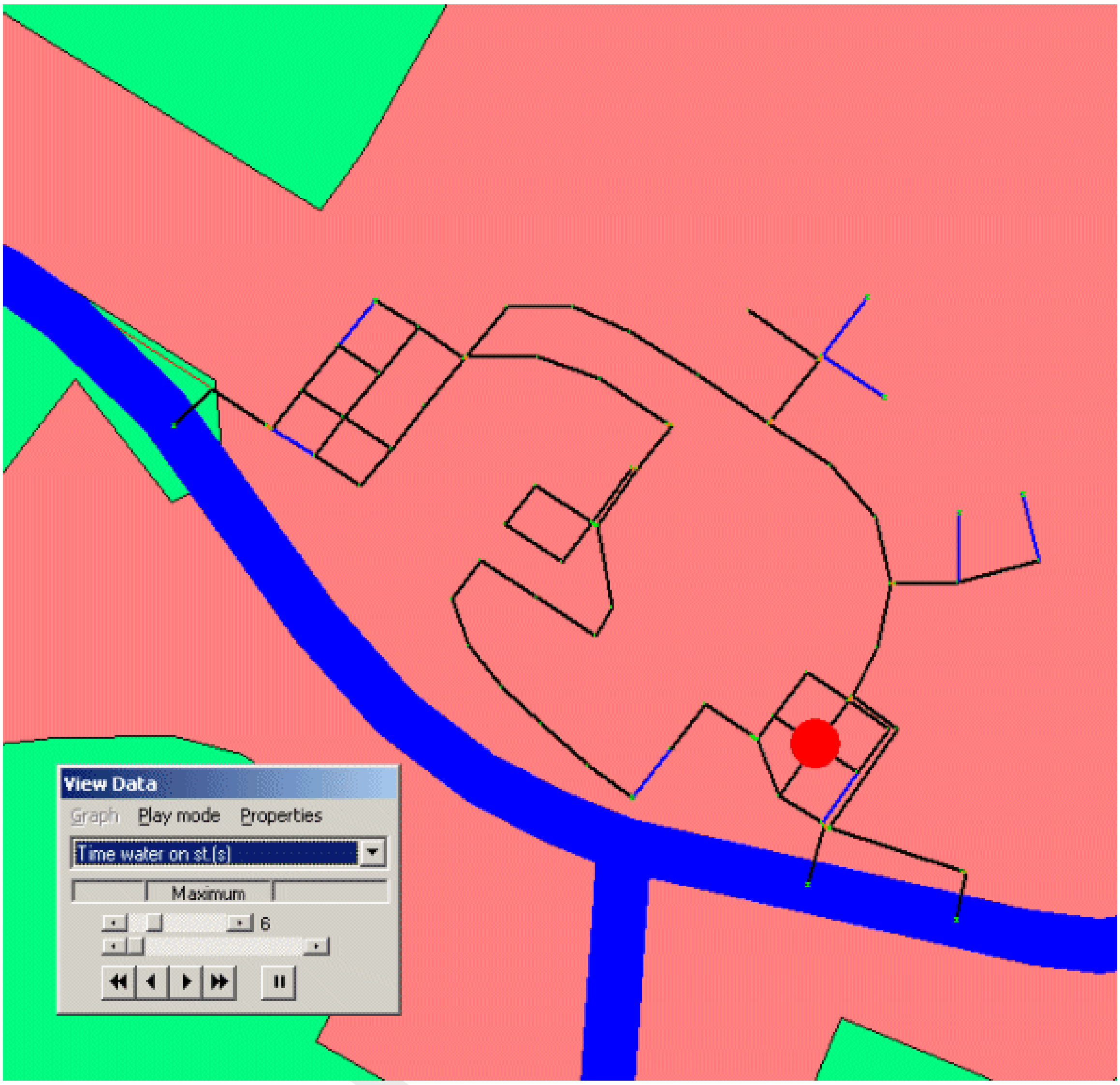


Figure 4.27: *Time water on street, maximum*

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.

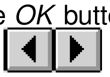
Plotting discharges

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 still have selected the 'Results at reach segments' in the Active Legend.
- ◇ Select 'Options' - 'Network Data...'.
- ◇ Select the 'Link' 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', if it is still turned on.
- ◇ 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.

4.2.7 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.

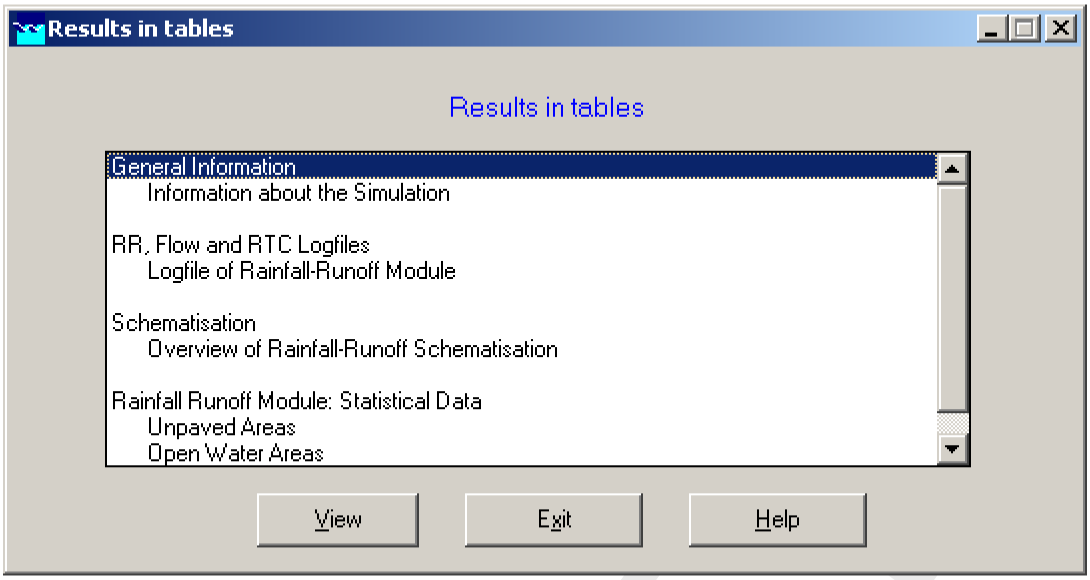


Figure 4.28: *The results in tables window.*

- ◇ 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'.
- ◇ Click the *Exit* button in the 'Results in tables' window.

4.2.8 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.

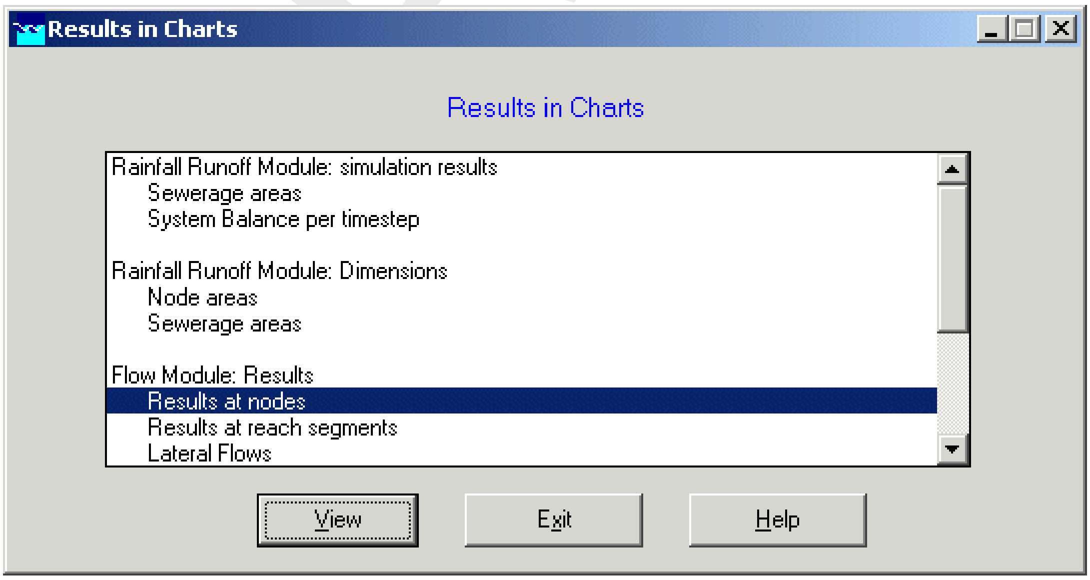


Figure 4.29: *The results in charts window.*

- ◇ Select 'Results at nodes'.
- ◇ Click the *View* button.

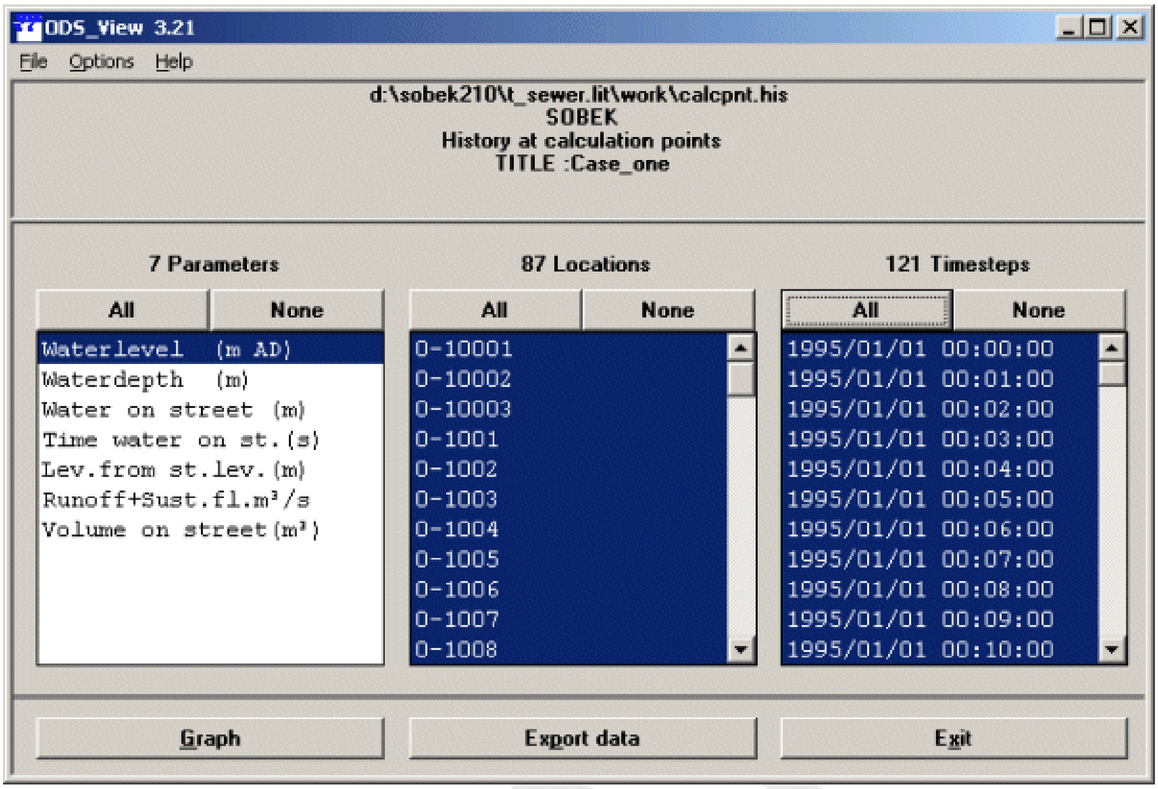


Figure 4.30: *The ODS_VIEW window.*

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.

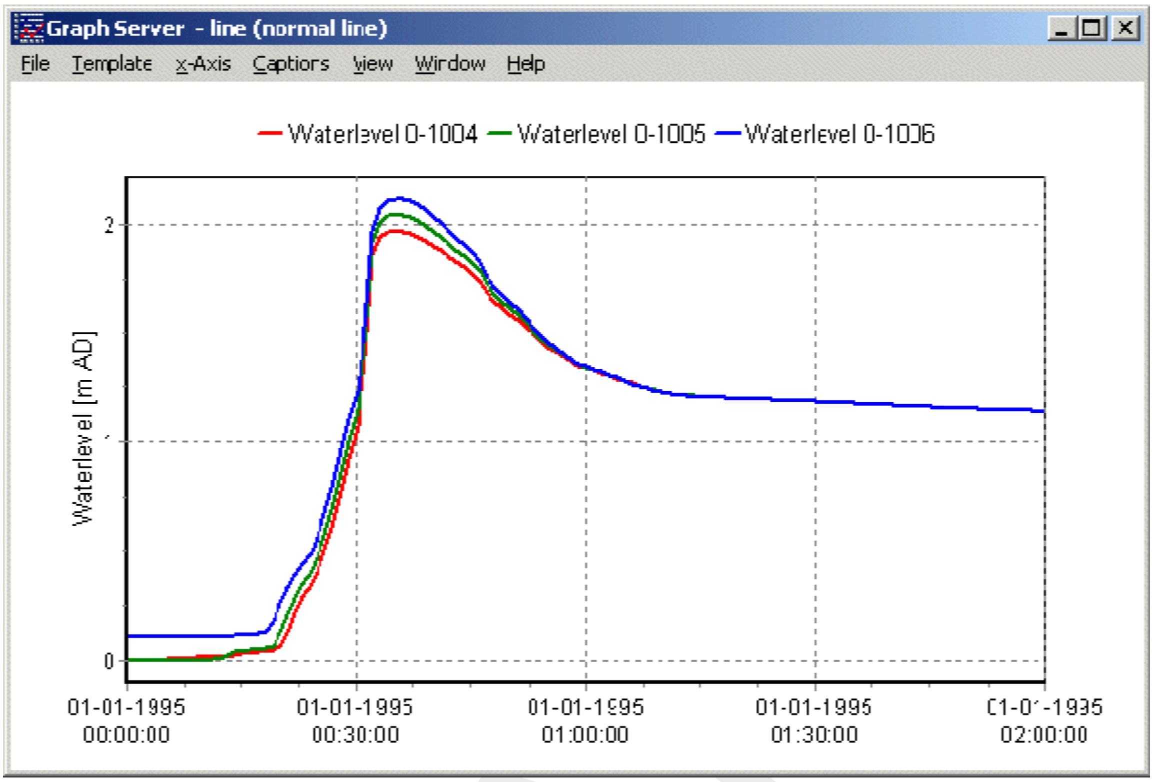


Figure 4.31: *An example of a graph created in the 'Results in Maps' task block.*

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 00-1004, 00-1005 and 00-1006.
◇ Press the *All* button of Timesteps.
◇ Press the *Export data* button.

Feel free to export the data in any of the possible formats.

Now, we will exit the 'Results in Charts' task block:

◇ 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.

4.2.9 Case Analysis Tool

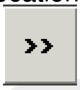
In ’Case_one’ we used the rain event ’STNBUI07.BUI’. The total rainfall in this event is 19.8 mm. To show the impact of a much larger rainfall event we will select the rainfall event ’STNBUI10.BUI. The total rainfall in this event is 35.7 mm.

- ◇ 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’.

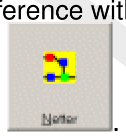
- ◇ Select ’Projects’ - ’Case Analysis Tool’.
- ◇ Select ’T_SEWER’ from the list.
- ◇ Press the *OK* button.

- ◇ 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 .

To easily visualize the maximum difference:

- ◇ Select ’Options’ - ’Data Statistics’ - ’Maximum’.
- ◇ Select ’Options’ - Network Data..’ - ’All Data’ and enable the option ’Width’ of ’Show node data’.

The following figure appears. This figure shows the maximum difference between the two cases.

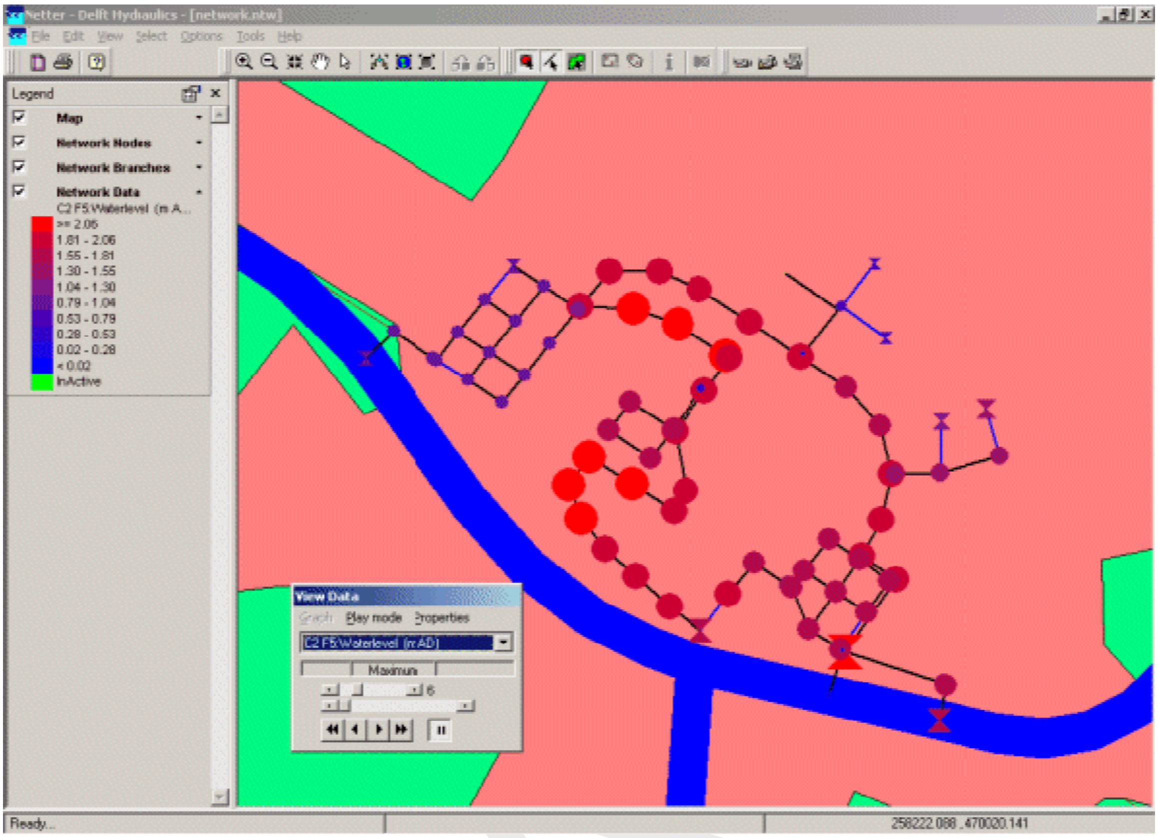


Figure 4.32: *Maximum difference between two cases*

Now reset the network visualisation to normal:

◇ Select 'Options' - Network Data..' - 'All Data' and disable the option 'Width' of 'Show node data'.

◇ Select 'File' - 'Exit'.
◇ Select 'Application' - 'Exit'.
◇ Press the button *No*.

4.2.10 Series simulation based on independent rainfall events

Simulating a long period of time, for example 50 years, 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 to you since it reduces the computation time and focuses on the events that matter. 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 [m³/hr]	Crest Level [m AD]	Width [m]	Heigth [m]	Total Volume [m³]	Volume/Year [m³]	Freq./year [-]	Brut.SpillTime [min.]	Net.SpillTime [in]	Vol.1:1 yr [m³]
1	00-10003	External weir	pos.		0.20	5.00		208.12	208.12	3.00	64	64	126.75
2	00-2013	External weir	pos.		2.90	1.00		3410.60	3410.60	7.00	176	176	1522.79
3	00-4003	External weir	pos.		0.20	3.00		409.91	409.91	6.00	160	138	217.17
4	00-6004	External weir	pos.		1.85	2.00		1224.11	1224.11	10.00	454	454	276.32
5	00-6006	External weir	pos.		1.85	2.00		1223.92	1223.92	10.00	454	454	276.23
6	00-7004	External weir	pos.		3.00	2.00		1663.24	1663.24	10.00	420	418	469.84
7	00-7005	External weir	pos.		3.25	1.00		0.00	0.00	0.00	0	0	0.00
8	00-8002	External weir	pos.		1.20	5.00		14470.93	14470.93	8.00	348	344	4342.72
9	00-9002	External weir	pos.		4.00	5.00		1656.05	1656.05	10.00	732	728	184.30

Figure 4.33: *Window* ***Structure Statistics****.*

◇ Select the cell with ID '00-8002'.
◇ Click the right mouse button.
◇ Select 'Show Graph'.
◇ Scroll to event number 6.

The following figure appears:

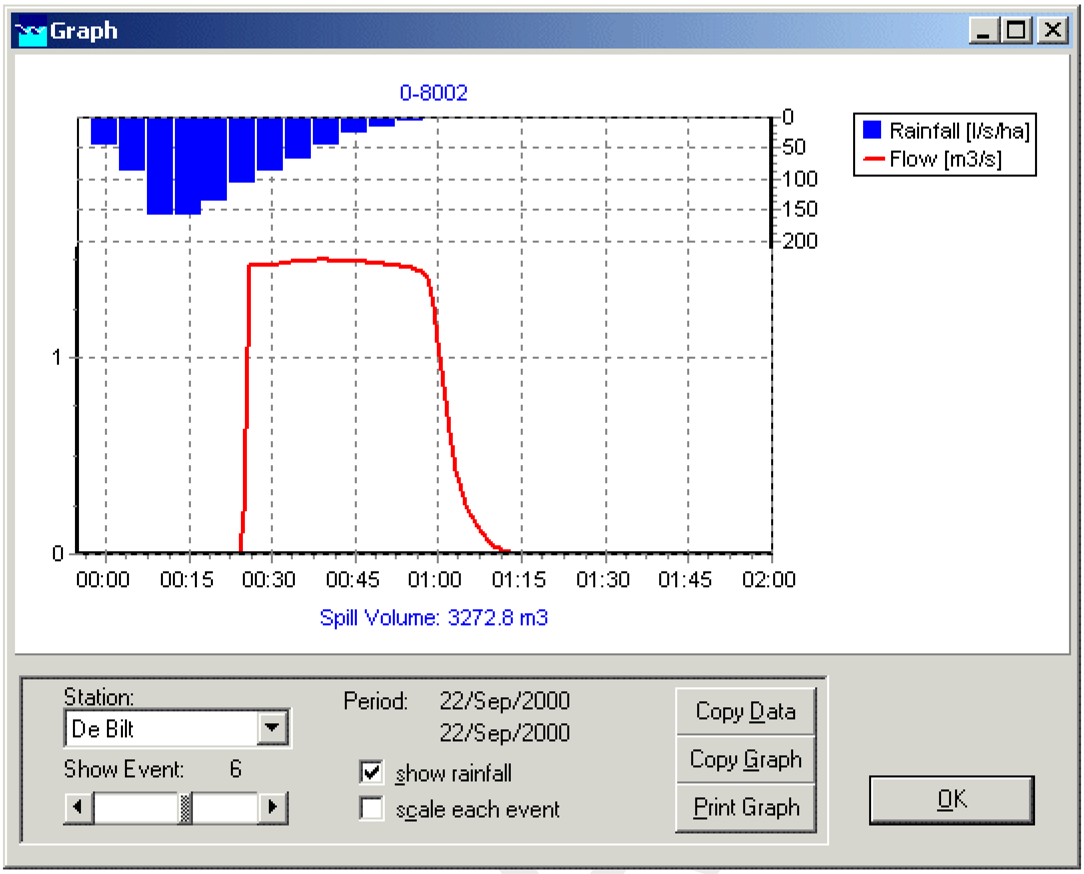


Figure 4.34: ***Structure Statistics*** *graph for external weir 00-8002, event 6*

◇ Press the *OK* button.
◇ Select 'File' - 'Exit'.
◇ Press the *Exit* button.
◇ Select 'File' - 'Exit'.
◇ Select 'Case' - 'Save'.

4.2.11 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, 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 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 and the 'Connection' edit network functions to place the 'Connection' functions toolbar anywhere on your screen.
- ◇ Select the 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 set 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 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*.
- ◇ Select the tab 'Link'
- ◇ Select the radio button *ID*.
- ◇ Press the *OK* button.
- ◇ Select 'Select ' - 'Search...' from the menu bar.
- ◇ Enter "00-7003".
- ◇ Press the *OK* button.
- ◇ Zoom in towards node '00-7003' .

Now we will start drawing the extended schematisation with node '00-7006' and link '00-7006-00-7003'. When finished with this chapter, your extended schematisation will look as follows:

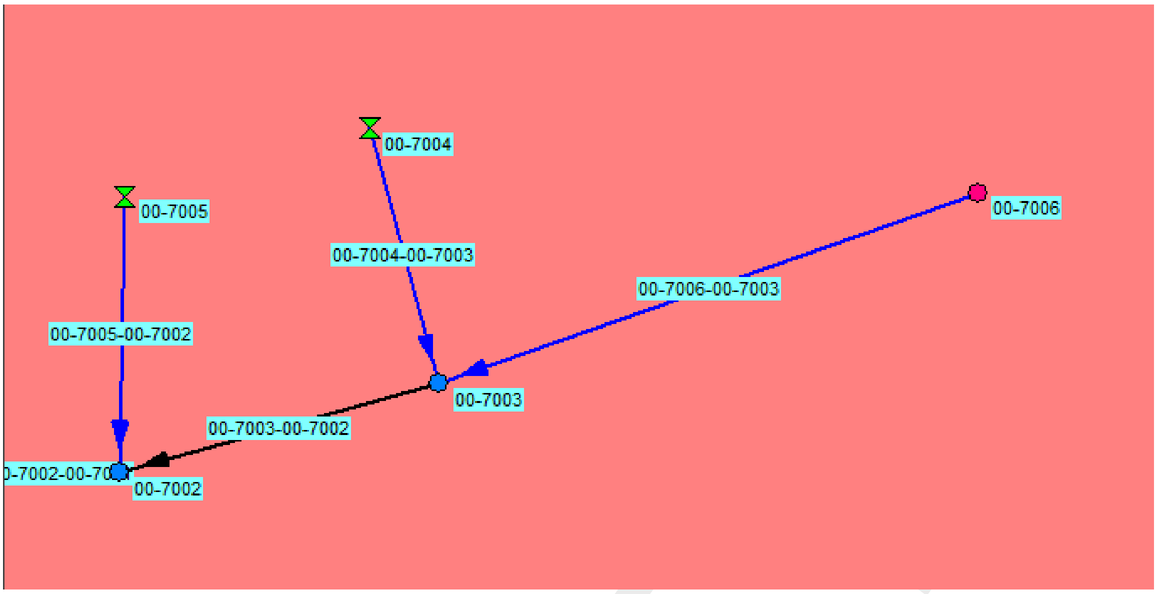
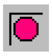
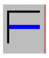


Figure 4.35: *Extended schematisation Urban.*

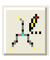
- ◇ Activate the  button, Flow-Manhole, by clicking on it.
- ◇ Activate the  button, Flow-Pipe with Runoff, by clicking on it.
- ◇ Activate the  button, 'Add connect', by clicking on it.
- ◇ Enter “00-7006” as the ID of the Node definition.
- ◇ Enter “00-7006” as the Name of the Node definition.
- ◇ Enter “00-7006-00-7003” as the ID of the Link definition.
- ◇ Enter “00-7006-00-7003” 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 '00-7003' 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...'-'Link'-'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.

For the purpose of this tutorial, either the default straight line or a user-defined vector layer will suffice. The vector layer can be edited as follows:

- ◇ 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.
◇ Unselect the  button, ’Edit Reach Vectors’ to leave the ’vector layer’ mode.
◇ Unselect the  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 ’00-7006’.
◇ Click with your right mouse button.
◇ Select ’Model data’ - ’Flow Model’.
◇ Select the tab ’Storage’.
◇ Enter “1.6” for the Bottom Level.
◇ Enter a constant value of “4” of the storage area at Storage Reservoir.
◇ 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 Water on Street.
◇ Press the *OK* button.

Editing reach (pipe) data
◇ Select the newly added 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 ’00-7006-00-7003’, the Flow - Pipe ’00-7004-00-7003 and the Flow - Pipe ’00-7005-00-7002’ 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  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 *OK* button.
◇ Select 'Case' - 'Save'.

Please start the simulation and analyze the results.

Epilogue

In this tutorial the most important aspects of working with SOBEK have been discussed. Extended documentation can also be found in the SOBEK Manual. All PDF files containing documentation can be found in the "Manuals" directory in the Start menu, next to the SOBEK start icon. Since you have now gained experience it will not be that difficult to find out the options and possibilities of SOBEK that not have been discussed here. Good luck!

4.3 Tutorial Hydrodynamics - 1D2D floodings (SOBEK-Rural 1DFLOW + Overland Flow modules)

General

The Overland Flow (2D) module of SOBEK-Rural is designed to calculate two-dimensional flooding scenarios. The module is fully integrated with the 1DFLOW module for accurate flooding simulation. It is especially designed to simulate dam breaks and dike breaks. The hydrodynamic simulation engine underneath is based upon the complete Saint Venant Equations. It can simulate steep fronts, wetting and drying processes and sub critical and super-critical flow.

This tutorial will guide you through a certain number of basic steps of both the Overland Flow (2D) module and the 1DFLOW module. Some experience in working with the Microsoft® Windows® operating system is required.

Good luck!

Setting up a combined 1DFLOW and Overland Flow (2D) system

◇ Click on the Windows *Start* button.
◇ Select the 'All Programs' or 'All Apps' menu.
◇ Select the 'Delft Hydraulics' menu.
◇ Select the 'SOBEK' menu item (SOBEK215).
◇ Click on the 'SOBEK' icon.
◇ Select the menu item 'Options' - 'SOBEK Options'.
◇ Select the tab 'Background Map'.
◇ Select the file <Tutorial1D2D.map>.
◇ Press *OK* button to save and close SOBEK Options.
◇ Click the 'New Project' button.
◇ Type the name "T_1D2D".
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_1D2D'. You are now asked: do you want to work with this project?

◇ Click the *Yes* button.

◇ Select the menu option 'Case'-'Open'.
◇ Select 'Default' from the list.
◇ Click *OK* button.

◇ Double click the 'Import Network' task block.
◇ Select 'Start from scratch'.
◇ Click *OK* button.

At the 'Settings' task block, you have to activate the two dimensional 'Overland Flow (2D)' module. Notice that you cannot choose the Overland Flow (2D) module alone. It will always work in combination with the '1DFLOW (Rural)' module (see figure):

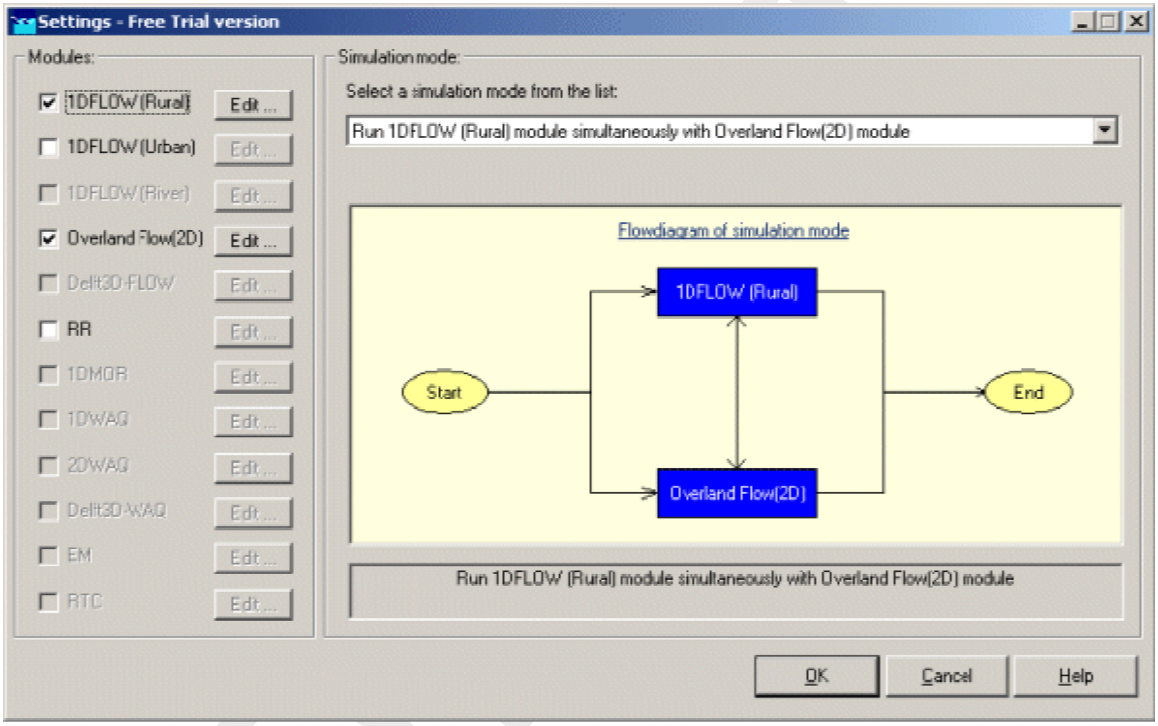


Figure 4.36: *Activating the '1DFLOW (Rural)' and 'Overland Flow (2D)' modules*

◇ Double click the 'Settings' task block.
◇ Unselect all the selected modules if any.
◇ Select the '1DFLOW (Rural)' module.
◇ And select the 'Overland Flow (2D)' module.

Edit the 1DFLOW (Rural) settings:
◇ Press the *Edit* button of the '1DFLOW (Rural)' module.
◇ Select the tab 'Time settings'.
◇ Choose '1 minute' as the 'Time step in computation' (Simulation time step).
◇ Select the option 'Simulation period defined as below' for Simulation period.
◇ Define the 'start of simulation' as '2006/01/01; 00:00:00'.
◇ Define the 'end time' as '2006/01/01 02:30:00'.
◇ Select the tab 'Simulation settings'.
◇ Choose (of course) the 'unsteady calculation'.
◇ Select the 'Initial data' tab.
◇ Choose the initial water depth option 'initial depth in channels [m]'.

◇ Set 1m as the initial water depth.
◇ Select the 'Output options' tab.
◇ Define an output time step of '00:01:00 (hh:mm:ss)'.
◇ Leave the Settings editor for '1DFLOW (Rural)' by clicking the *OK* button.

Edit the Overland Flow settings:
◇ Click the *Edit* button of the 'Overland Flow(2D)' module.
◇ Select the 'Simulation settings' tab.
◇ Choose for the option 'use as height', because the values in the 2D grid that we will use are defined as heights with relation to the reference level.
◇ Select the 'GIS Output options' tab.
◇ Switch off all output parameters.
◇ Select the 'Incremental output' tab, keep the values as they are.
◇ Select the 'history output' tab
◇ Choose Time step output equal to 1 time step.
◇ Click the *OK* button to leave the settings for the 'Overland Flow(2D)' module window.
◇ Click the *OK* button to leave the Settings window.

◇ Double click the 'Meteorological Data' task block. There is no special data input necessary.
◇ Click the *OK* button.

The following section describes the steps that must be followed to create a schematisation and provide data to the model.

4.3.1 Starting with a 2D grid

The 2D grid

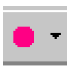
A predefined 2D grid will be used in this tutorial. This 2D grid file is named <2D_OF Tutorial.asc> and is available in the <Drive:\SOBEK Install Directory\map\tutorial2Dflooding\> directory. During this tutorial, we will show you how to add this grid file to your network.

Perform the following actions to open it in your schematisation:

◇ Double click the 'Schematisation' task block.
◇ Click the *Edit model* button.
◇ Enter the 'edit network' mode by clicking the  button.

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 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 *Automatic*.
◇ In the group box 'Name', select the radio button *Automatic*.
◇ Click the *OK* button.

◇ Click on the button from the  section, click on the 'Overland Flow model' option and drag the toolbar that shows its node types to your screen.

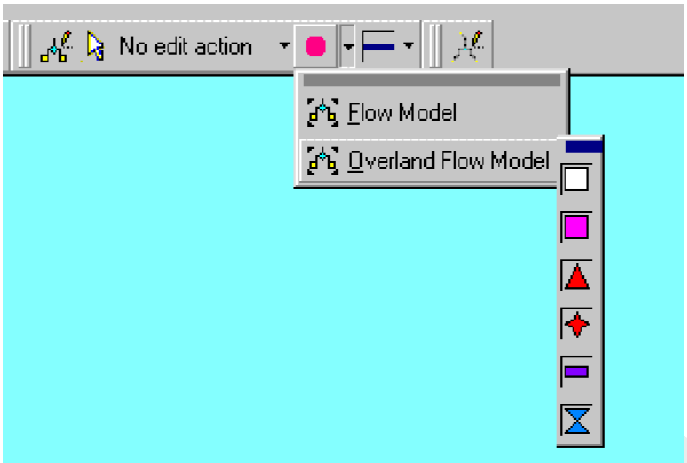


Figure 4.37: *Drag the toolbar containing Overland Flow node types to your screen*

◇ Select '2D Grid tool bar'and drag it to your screen.

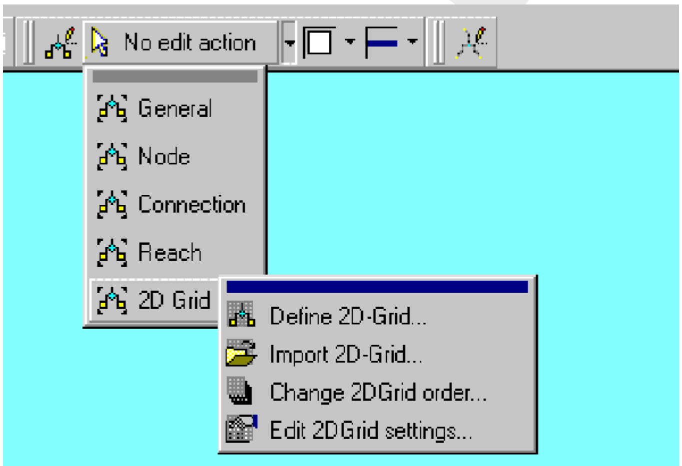


Figure 4.38: *Drag the toolbar containing the 2D-grid edit actions to your screen*

◇ Select '2D - Grid' object from the overland flow node types toolbar (white square).
◇ Now select the 'Import 2D-Grid...' option from the 'Edit 2D Grid' toolbar.
◇ Enter “2D-Grid” in both input fields.
◇ Click the *OK* button.
◇ Browse to the <2D_OF Tutorial.ASC> file (<Drive:\SOBEK215\map\tutorial2Dflooding\>).
◇ Open the <2D_OF Tutorial.ASC> file.
◇ Zoom in on the entire network by clicking the  button (show full network).
◇ Finally, make the grid cells visible by choosing 'Options' - '2D-Grid options...' - 'General'.
◇ In the window that pops up, select 'Solid' from the combo-box under the topic named 'Lines'.
◇ Click the *OK* button.

The grid you imported should now look like this:

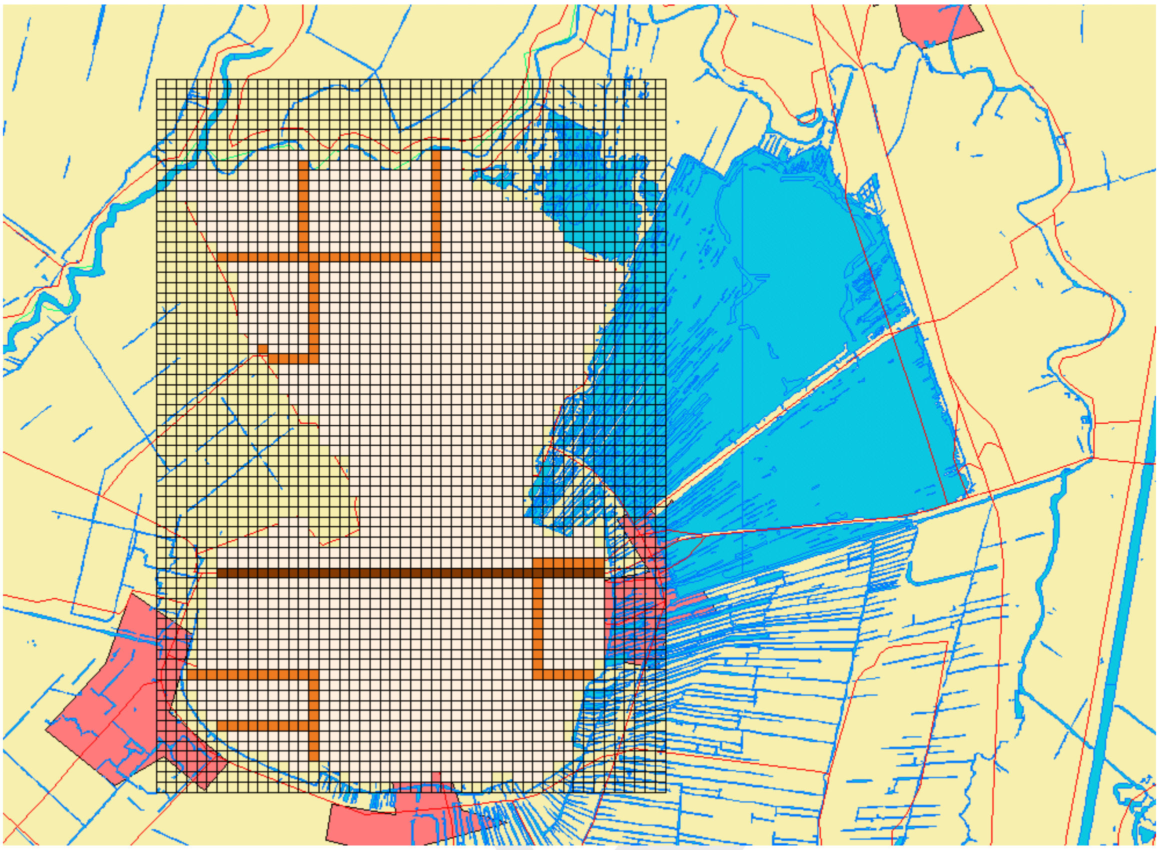


Figure 4.39: *Impression of the model within its GIS environment.*

The model shows a grid of the 'Groot Mijdrecht' polder, located alongside the 'Vinkeveense Plassen' in the Netherlands.

With the menu option 'Select' - '2D grid cell info' and selecting 2D grid cells you can investigate the altitude of the 2D grid at several places. The grid has an altitude of 0 m from above datum, except for some dikes which are 1 or 2 m higher. The dike which splits the polder in two parts is a railway dike.

- ◇ Now change the appearance of the grid under 'Options', followed by '2D Grid options...' - 'Model data'.
- ◇ Click on the *classify* button.
- ◇ Enter at minimum "0" and maximum "6".
- ◇ Click the *OK* button.
- ◇ Click the *OK* button to close the 'Properties 2D - Grid: ' window.

- ◇ In the menu bar, select 'Edit' - 'Model data'.
- ◇ In the 'Model Data' window, select '2D-Grid' in the list box.
- ◇ Press the *Edit* button.
- ◇ Select the 'Friction' tab.
- ◇ Select the option 'Manning (mn)' for friction type.
- ◇ Enter a constant value of "0.02". Keep the 'Friction value (Vertical Obstacle Friction)' at 0.
- ◇ Press the *OK* button.
- ◇ Close the 'Model Data' window.

Adding a simple network

Now we can start the 1D part: the channel flow network. Our schematisation of a ditch will be modeled as shown in Figure 4.40.

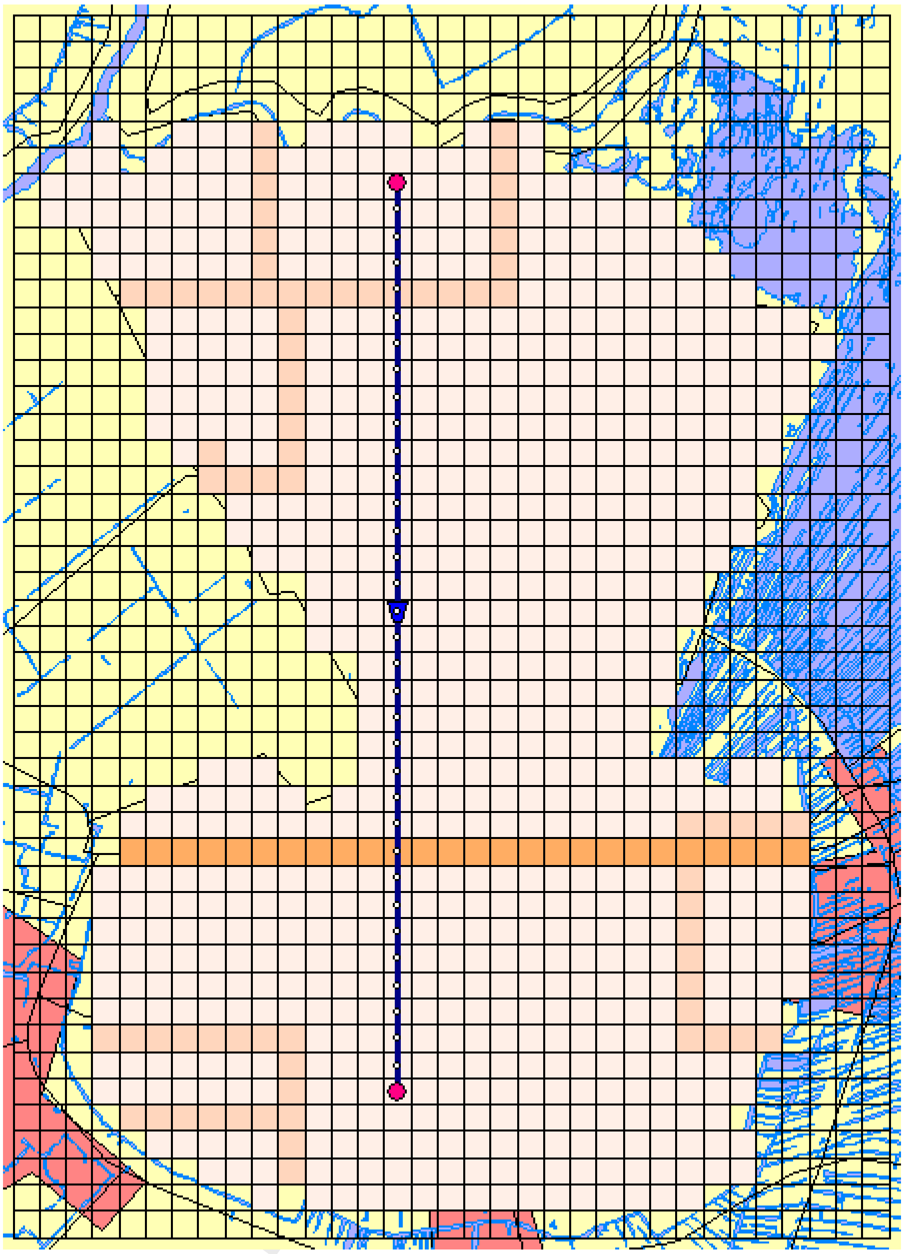
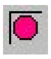
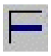


Figure 4.40: *The 1D network of the tutorial case.*

- Select the  button, Flow-Connection Node.
- Select the  button to select the function 'Add node'.
- Enter "North" in both input fields.
- Click the *OK* button.
- Click on the left-mouse button again to actually add the Flow-Connection Node on your screen, in the Northern part.
- Select the  button, 'Add node', again.
- Enter "South" in both input fields.
- Click the *OK* button.

◇ Click on the left-mouse button again to actually add the Flow-Connection Node on your screen, in the Southern part.

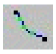
◇ Select the  button, Flow-Channel, by clicking on it.

◇ Select the  button to select the function 'Connect nodes'.

◇ Click with the left mouse button on the node "North" and drag to the node "South". Release the mouse button.

The model will need a calculation grid. Before generating this grid we will switch off the generation of names.

◇ 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 *Automatic*.
◇ In the group box 'Name', select the radio button *No Names*.
◇ Select the tab 'Link'.
◇ In the group box 'ID', select the radio button *Automatic*.
◇ In the group box 'Name', select the radio button *Automatic*.
◇ Press the *OK* button.

◇ 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 "150" in the length edit box to set the calculation grid to a 150 m length.
◇ Click the *OK* button.

Now re-enable the naming of new nodes:

◇ 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.

For a useful way of identifying the important nodes in your network, set the following options:

◇ Click on the  button in the Active Legend or select the menu item 'Options' - 'Network Data...'.
◇ In the tab 'Node', select the radio button *Name*.
◇ Select the tab 'Link'.
◇ Under 'Show Titles', Select the radio button *None*.

SOBEK offers a powerful tool to validate your network.

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

The following message appears: "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  node, Flow - Cross Section node.
◇ Select the  (add node) function.
◇ Enter “Cross-Section_Ditch1” 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.

Now, set the model data for the cross section:

◇ Select the ’Cross-Section_Ditch1’ node.
◇ Click with your right mouse button.
◇ Select ’Model data’ - ’Flow Model’.
◇ Select the ’Location’ tab.
◇ Enter the value “-1” as the bed level. The ditch has a bed level at -1 meters with respect to reference level.
◇ Enter the value “0” as the surface level. The ditch has a surface level at 0 m with respect to reference level.
◇ Select the ’Cross section’ tab.
◇ Select the type ’Trapezium’.
◇ First enter the name “profile” at ’Cross section: ’.
◇ Click the *Define dimensions* button.
◇ Set the slope at ’1’.
◇ Set the bottom width at ’1 m’.
◇ And set the maximum flow width at ’3 m’.
◇ Press the *Save dimensions* button.
◇ Press the *OK* button.
◇ Select ’Friction’ tab.
◇ Select ’Use local value(s) for this cross section’.
◇ Select ’Local value(s)’ in the ’Show’ combo box.
◇ Select ’Chézy (C)’ for type friction (Bed).
◇ Enter “50” for constant value.
◇ Click *OK* to close.

4.3.2 Flooding from the lake

The next step in this tutorial will be to add a 2D boundary node, called ’Lake’ at the edge of the grid. This node will schematise the lake, which has a constant water level of 0 m reference level. For the time being we assume that the water from the lake has free access to the polder (so no dikes or dike breaks in that area).

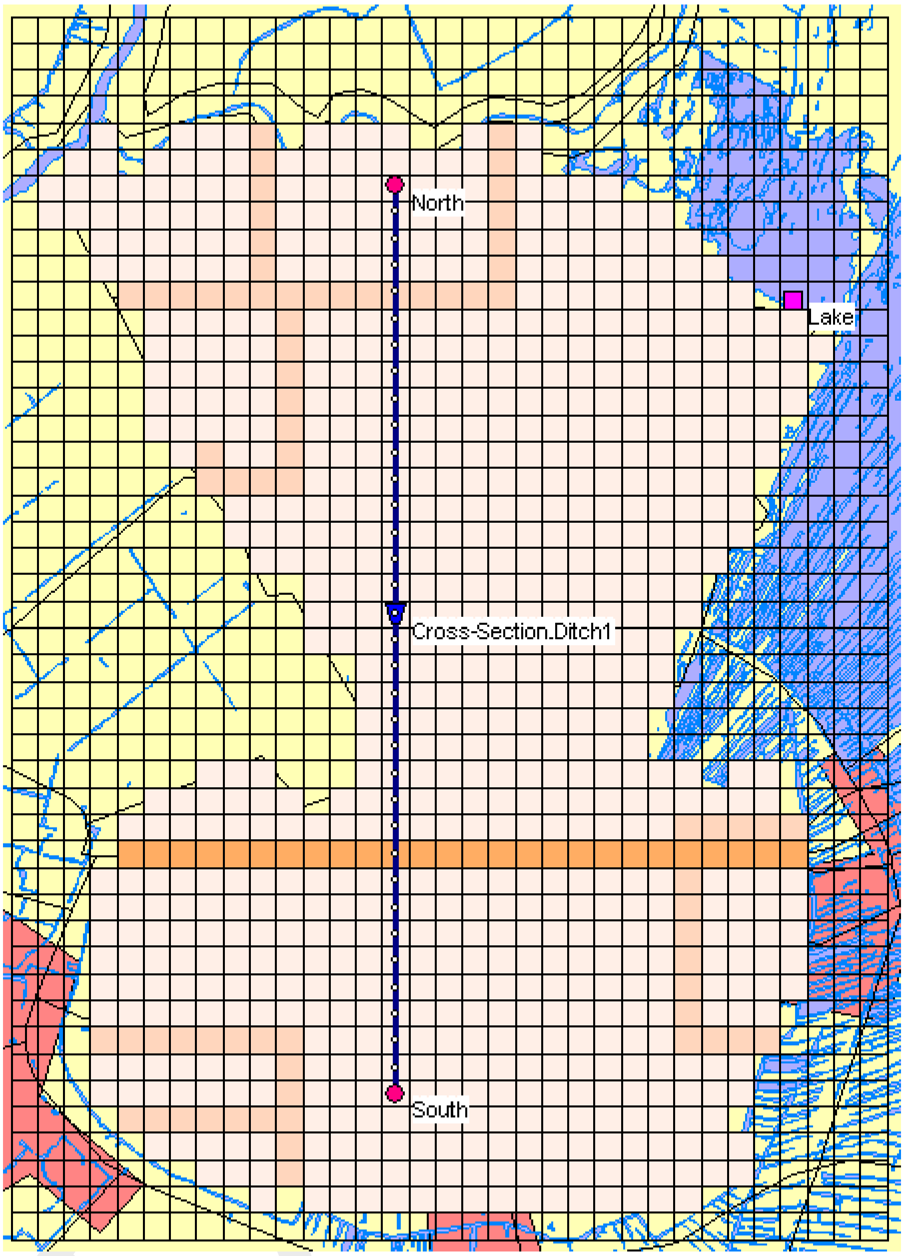
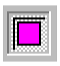
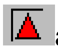


Figure 4.41: *Flooding from the lake.*

◇ In 2D - nodes toolbar, select the  button, **2D - Boundary Node**.
◇ Select the  button to select the function 'Add node'.
◇ Enter "Lake" in both input fields.
◇ Click the *OK* button.
◇ Click on the left-mouse button again to actually add the 2D - Boundary Node on your screen, at column number 30, and row number 36. The row and column numbers are shown in the bottom right corner of the NETTER screen, while you move the mouse pointer over the grid cells. Starting from the right, the first and second number display the column and row number, and the third number displays the (inverted) height of the grid cell.
◇ Select the 'Lake' node.
◇ Click with your right mouse button.
◇ Select 'Model data' - 'Overland Flow Model'.

◇ Select the tab 'Boundary condition'.
◇ Select the type 'water level (h)'.
◇ Enter a constant water level of "3.85" m.
◇ Click the *OK* button.

Next, add some 2D history stations  at the (approximate) locations in the polder, as depicted below.

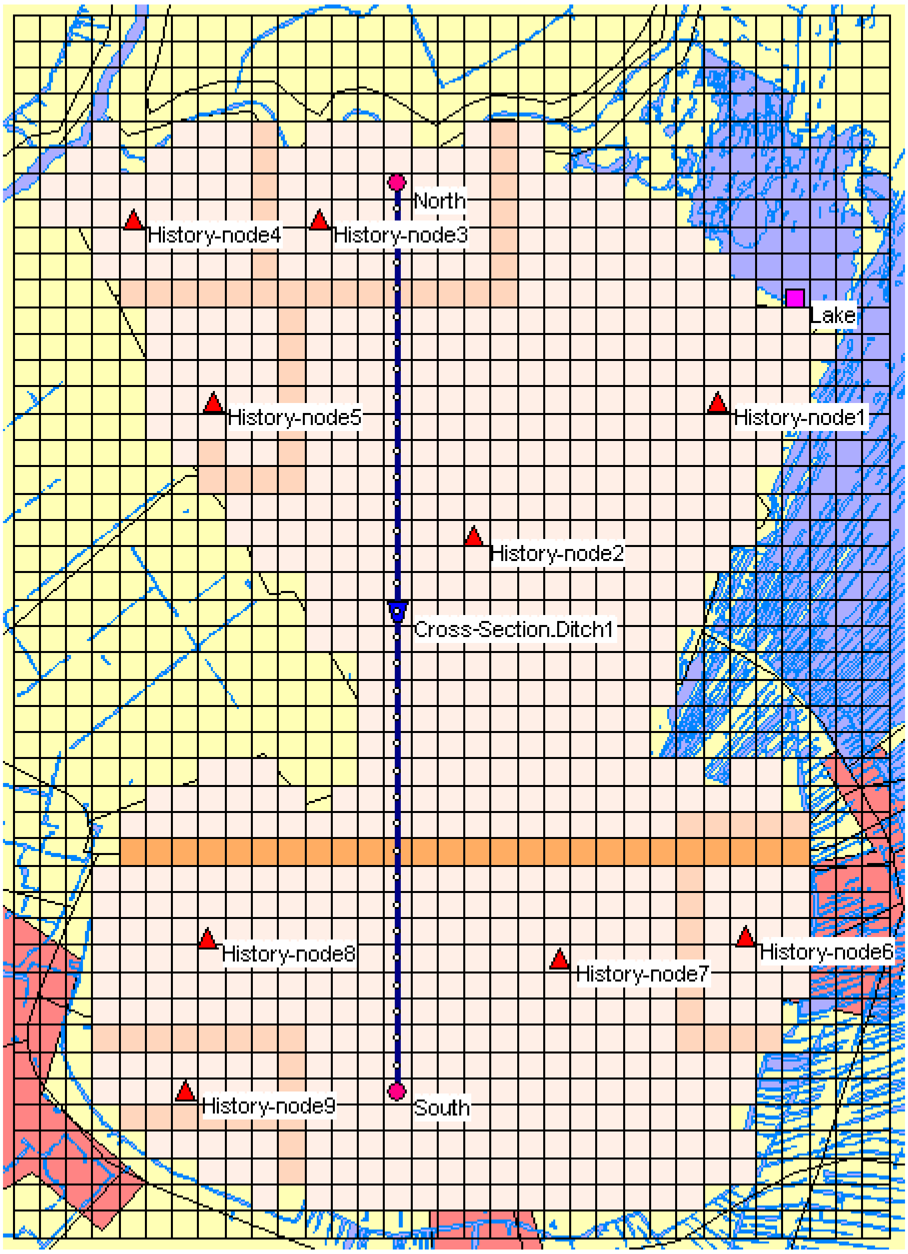


Figure 4.42: *Adding history stations.*

◇ Select the  button, 2D - History Node, by clicking on it.
◇ Select the  button to select the function 'Add node'.
◇ Enter "History-node1" in both input fields.
◇ Click the *OK* button.
◇ Click on the left-mouse button again to actually add the 2D - History Node on your screen,

at column number 27, and row number 32.

◇ Select the button to select the function 'Add node'.
◇ Enter "History-node2" in both input fields.
◇ Click the *OK* button.
◇ Click on the left-mouse button again to actually add the 2D - History Node on your screen, at column number 18, and row number 20.
◇ Add 'History-node3' at column number 12, and row number 39.
◇ Add 'History-node4' at column number 5, and row number 39.
◇ Add 'History-node5' at column number 8, and row number 32.
◇ Add 'History-node6' at column number 28, and row number 12.
◇ Add 'History-node7' at column number 21, and row number 11.
◇ Add 'History-node8' at column number 8, and row number 12.
◇ Add 'History-node9' at column number 7, and row number 6.

History stations will save the calculated results for each time step for water levels at their locations. Place these nodes at 'interesting' places (e.g. behind a breach). These nodes need no extra data input in Model data.

◇ Select button.
◇ Select the menu item 'File'-'Exit', to leave NETTER.
◇ Click the *OK* button to leave the schematisation window.

◇ Select the menu item 'Case'-'Save As'.
◇ Enter the name "Tutorial 1D2D with 2D boundary node" to save the case.
◇ Click the *OK* button.
◇ Proceed with the simulation by double clicking the block 'Simulation' in the case window. Once the simulation is completed you can proceed to view the results in maps and in charts.

The visualisation of 1D results are explained in the Tutorial Hydrodynamics in open water (SOBEK-Rural 1DFLOW module) Together with the visualisation of 1D results, the user can see the 2D results.

◇ Double-click the 'Results in Maps' task block. The Netter GIS application will re-appear.
◇ Select 'Depths (incremental)' from the Active legend.
◇ Click the buttons and keep them pressed to animate the flooding of the area!

If one or several 1D-elements are selected, it is possible to observe time history graphs at those locations for the computed parameters (see figure below). The same can be done for the 2D - History stations.

The figure shows a 2D map water depth output at a certain time step and a graph showing the water depth variation at selected 2D cells. The latter one (made by choosing 'Select'- '2D grid cell info', selecting a 2D cell and clicking the button), is based on incremental information and it shows the moments where the water depth in one cell changes from one class to the next while flooding proceeds.

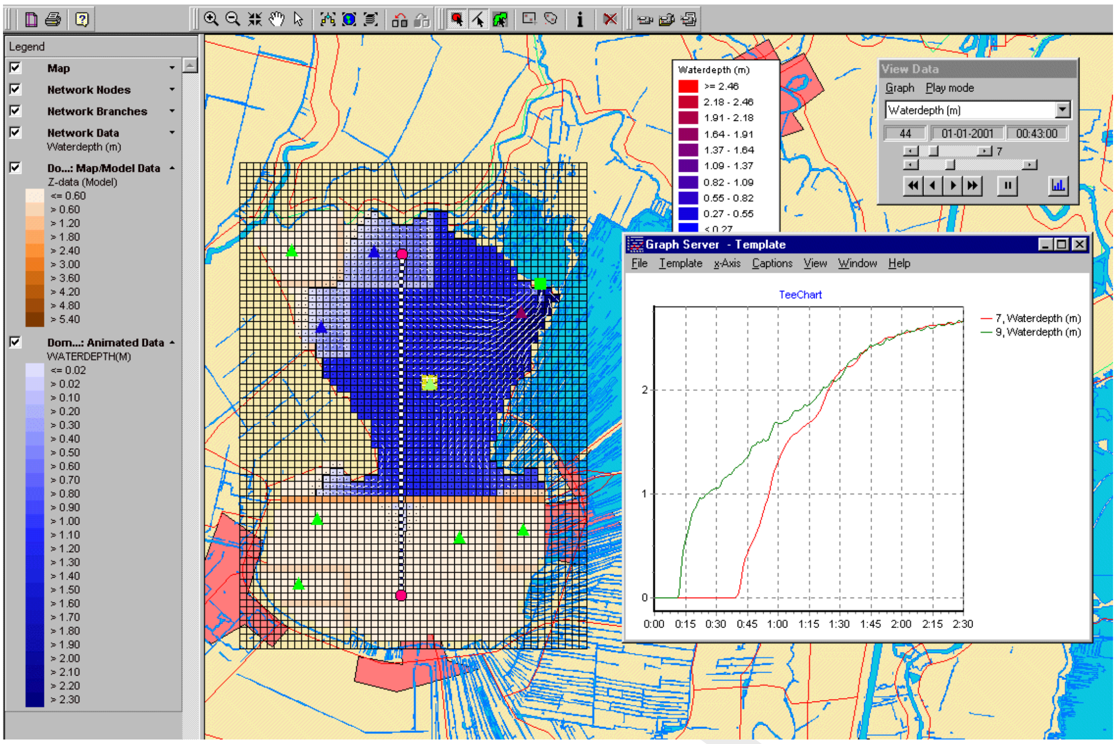


Figure 4.43: *Results of the visualization on maps; 1D + 2D map output for a certain time step and time series graph showing the water level history at two different 2D History stations (which actually are 1D objects)*

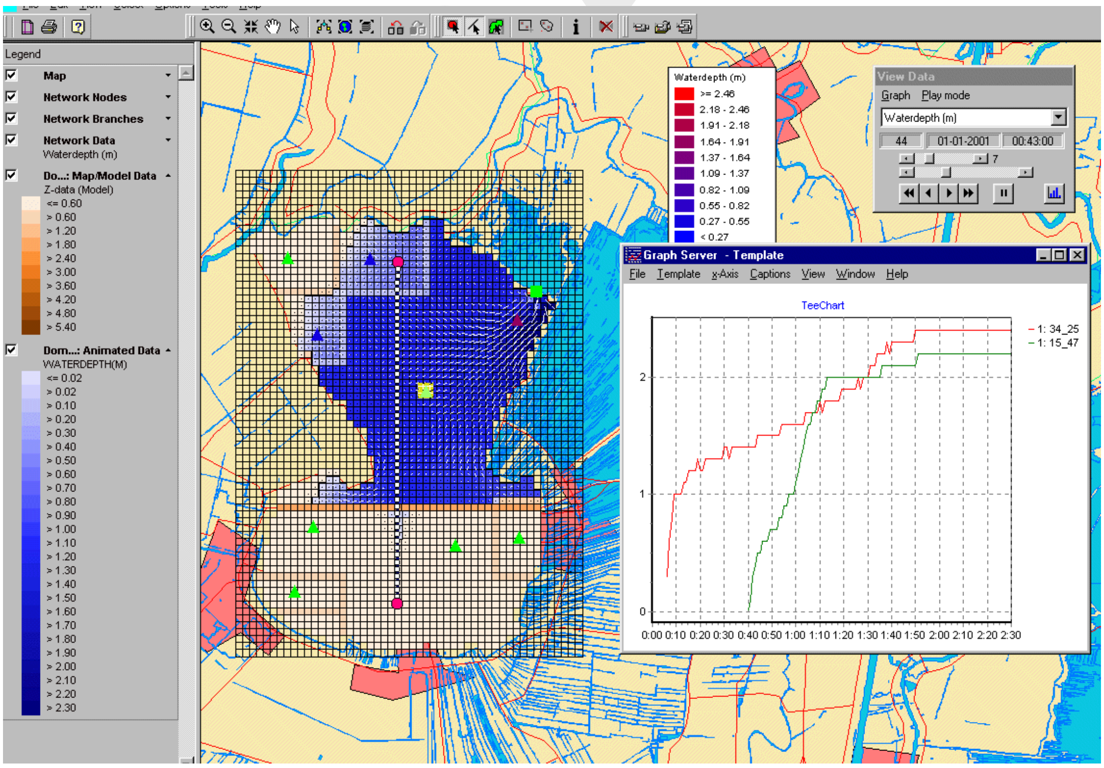


Figure 4.44: *Results of the visualization on maps; 2D map output for a certain time step and graph showing the water depth variation at selected 2D cells based on incremental information (this graph shows the moments when the water depth class changes)*

◇ Select ’File’ - ’Exit’ to leave the Results in Maps task block.
◇ Select ’Case’ - ’Save’.

4.3.3 Flooding from the lake with a 2D dam break

◇ Select ’Case’ - ’Open as new’.
◇ Select the case ’Tutorial 1D2D with 2D boundary node’.
◇ Enter the name “Tutorial 1D2D, flooding from 2D breach growth node”.
◇ Press the *OK* button.
◇ Double-click the ’Schematisation’ task block.
◇ Press the *Edit model* button.

The idea behind this case is to simulate a breach of the dike, followed by a flooding from the lake.

◇ Choose ’Edit’ - ’Model data’.
◇ Select ’2D - Grid’ from the list box.
◇ Select the node ’2D-Grid’.
◇ Click the *Edit* button.
◇ Select the tab ’Grid Cell Bottom Level’.
◇ Enter a value of “8” in the following cells:

Column number	**Row number**
31	34
30	34
29	34
28	34
28	35
28	36

◇ Click the *OK* button.
◇ Enter the file name “2D_DAMBREAK.ASC”.
◇ Save the grid under the directory <T_1D2D.LIT\FIXED>.

◇ Select ’Options’ - ’2D-Grid options’ - ’Model data...’.
◇ Press the *Classify...* button.
◇ Enter the maximum legend scale into 8 (minimum = “0”, maximum = “8”).
◇ Click the *OK* button.
◇ Click the *OK* button.

The result should look like Figure 4.45.

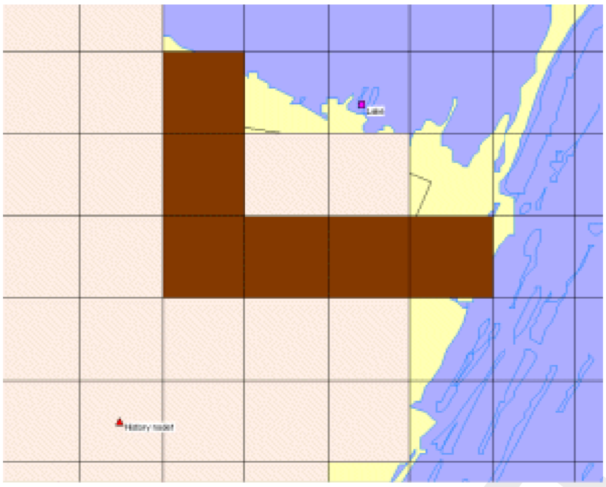


Figure 4.45: *Cells after changing depth from 0 m to 8 m.*

As you can see after saving, the cell beneath the 2D boundary node is raised to 8 m + reference level. Between the lake and the polder is a sort of a dike now, that will break by a breach.

- ◇ Enter the 'edit network' mode by clicking the button.
- ◇ Select the button, 2D - Breaking Dam Node, by clicking on it.
- ◇ Select the button to select the function 'Add node'.
- ◇ Enter "Breaking_Dam" in both input fields.
- ◇ Click the *OK* button.
- ◇ Click on the left-mouse button again to actually add the 2D - Breaking Dam Node on your screen, in the cell with column number 29 and row number 34.

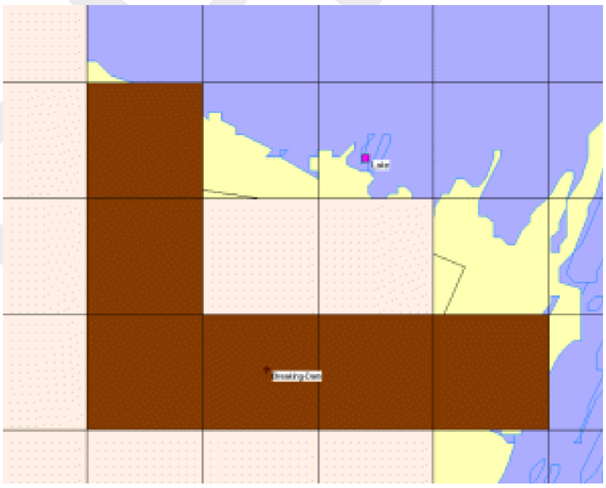


Figure 4.46: *Add a 2D dam break node.*

- ◇ Select the node 'Breaking_Dam'.
- ◇ Click with your right mouse button.
- ◇ Select 'Model data' - 'Overland Flow Model'.
- ◇ Select the tab '2D Breaking Dam'.
- ◇ Press the *Table...* button.
- ◇ Click on *Add Row* button.

◇ Enter “1”.
◇ Press *OK* button.
◇ Fill in the table as shown in Figure 4.47.

	Date [dd/mm/yyyy]	Time [hh:mm:ss]	Lowering w.r.t. Start Level [m]
1	01/01/2006	00:00:00	0
2	01/01/2006	00:30:00	6
3	01/01/2006	00:35:00	6

Figure 4.47: *Growth of the breach depth (m) in time.*

◇ Press the *OK* button to leave the ’Edit table window’.
◇ Press the *OK* button to leave the data edit window.

This data means that the cell will decrease in height from 8 m to 2 m with respect to the reference level in 30 minutes time.

◇ Select  button in NETTER.
◇ Select the menu item ’File’-’Exit’, to leave NETTER.
◇ Press the *OK* button.
◇ Select ’Case’ - ’Save’.

◇ Double click the ’Simulation’ task block.
◇ Double click the ’Results in Maps’ task block.
◇ Select ’Depths (incremental)’ from the Active legend.
◇ Click the  buttons and keep them pressed to animate the flooding of the area.

◇ Select ’File’ - ’Exit’ to leave the Results in Maps task block.
◇ Select ’Case’ - ’Save’.
◇ Select ’Case’ - ’Close’.

4.3.4 Flooding from a 1D channel

In the final case of this tutorial, the flooding will occur from a 1D channel, in stead of from a 2D boundary like in the first two cases. This type of model is used when enough detailed information is available about the river/channel network to incorporate a river/channel into the model (1D2D). So the 1D channel network replaces the 2D - Boundary node.

The flooding itself will occur from one of the surrounding channels of the Dutch ’Amstelland-boezem’, and will happen due to a planned inundation of the polder, and not because of a dike break. This kind of inundations may be necessary to prevent dikes from breaking at some other point in the channel, causing great economic damage and possibly loss of human lives. The inundation will occur by opening a structure next to the channel.

◇ Select ’Case’ - ’Open as new’.
◇ Select the case ’Tutorial 1D2D with 2D boundary node’.
◇ Enter the name of the new case “Flooding from 1D”.
◇ Press the *OK* button.
◇ Double click the ’Settings’ task block.
◇ Press the *Edit...* button of the 1DFLOW (Rural) module.
◇ Select the ’Initial data’ tab.
◇ Select the option ’define local values in *Edit network* option.

- ◇ Press the *OK* button.
- ◇ Press the *OK* button to leave settings.
- ◇ Double click the 'Schematisation' task block.
- ◇ Press the *Edit model* button.
- ◇ Enter the 'edit network' mode by clicking the  button.
- ◇ Select the button , 'Delete node'.
- ◇ Select the 'Lake' node.
- ◇ Press the *Yes* button.
- ◇ Zoom out a little bit, until you have a clear view of the channel branch to the North-West of the polder. Take a look at Figure 4.48 for an example.

Next, you will model a part of a channel branch by defining two 1D boundary nodes and a channel reach between them, by adding two profile to the channel, and by defining a calculation grid of 150 meters between every h-calculation point. Finally, you will change the course of the channel according to the map by using the vector layer mode. The result should look something like Figure 4.48:

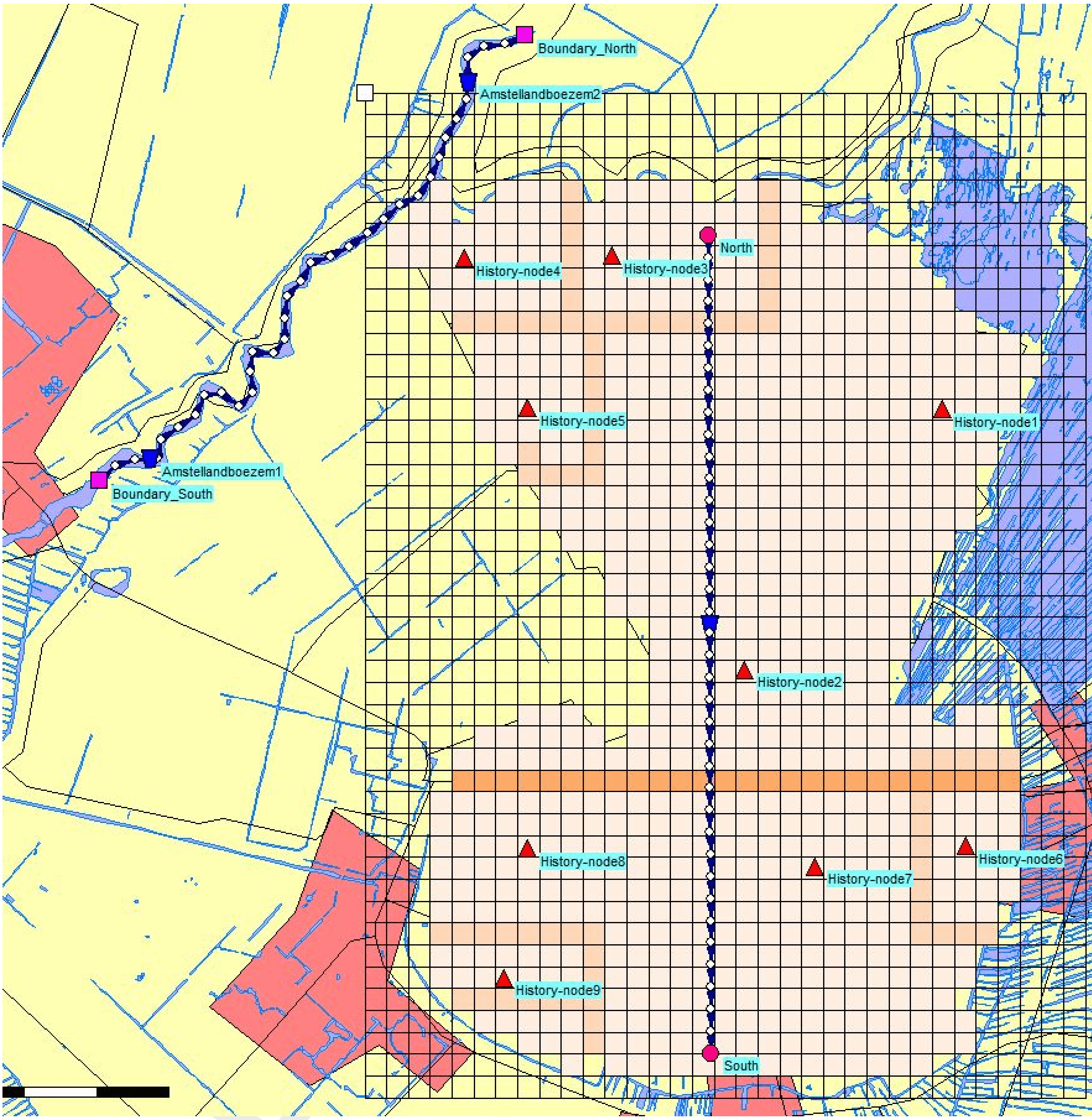
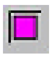


Figure 4.48: *The tutorial 2D grid extended with a 1D channel.*

◇ Select the  node, Flow-Boundary.
◇ Select the  button to select the function ’Add node’.
◇ Enter ’Boundary_North’ in both input fields.
◇ Click the *OK* button.
◇ Click on the left-mouse button again to actually add the ’Boundary_North’ node on your screen.
◇ Select the  button again.
◇ Enter ’Boundary_South’ in both input fields.
◇ Click the *OK* button.

- Click on the left-mouse button again to actually add the 'Boundary_South' node on your screen.
- 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 'Boundary_South' node and drag to the 'Boundary_North' node while keeping the button down. Release the left mouse button.
- 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 'No Names'.
- Press the *OK* button.
- 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, Edit Reach Vectors', to stop editing the reach vector.
- 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 '150' in the length edit box to set the calculation grid to a 150 m length.
- Click the *OK* button.

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 'Amstellandboezem1' 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 'Boundary_South'.
- Select the node, Flow - Cross Section node.
- Select the (add node) function.
- Enter 'Amstellandboezem2' 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 'Boundary_North'.

◇ Select the node ’Amstellandboezem1’.
◇ Click with your right mouse button.
◇ Select ’Model data’ - ’Flow Model’.
◇ Select the ’Location’ tab.
◇ Enter the value 2 as the bed level.
◇ Enter the value 4.5 as the surface level.
◇ Select the ’Cross section’ tab.
◇ First enter the name ’Amstellandboezem’ at ’Cross section: ’.
◇ Click the *Define dimensions* button.
◇ Set the slope at ’2’.
◇ Set the bottom width at ’10 m’.
◇ And set the maximum width at ’20 m’.
◇ Press the *Save dimensions* button.

◇ Select ’Friction’ tab.
◇ Select ’Use local value(s) for this cross section’.
◇ Select ’Local value(s)’ in the ’Show’ combo box.
◇ Select ’Chézy (C)’ for type friction (Bed).
◇ Enter ’50’ for constant value.
◇ Select ’Initial Value’ tab.
◇ Select ’Use local initial value for this reach’.
◇ Select initial ’water level’.
◇ Enter the value 3.85 m.
◇ Enter the value 1 m3/s for the initial flow in positive direction.
◇ Click the *OK* button to close the data edit window.

◇ Select the node ’Amstellandboezem2’.
◇ Click with your right mouse button.
◇ Select ’Model data’ - ’Flow Model’.
◇ Select the ’Location’ tab.
◇ Enter the value 1.9 as the bed level.
◇ Enter the value 4.4 as the surface level.
◇ Select the ’Cross section’ tab.
◇ In the combo box ’Cross section: ’ select: ’Amstellandboezem’.
◇ Select ’Friction’ tab.
◇ Select ’Use local value(s) for this cross section’.
◇ Select ’Local value(s)’ in the ’Show’ combo box.
◇ Select ’Chézy (C)’ for type friction (Bed).
◇ Enter ’50’ for constant value.
◇ Select ’Initial Value’ tab.
◇ Select ’Use local initial value for this reach’.
◇ Select Initial ’water level’.
◇ Enter the value 3.85 m.
◇ Enter the value 1 m3/s for the initial flow in positive direction.
◇ Click the *OK* button to close the data edit window.

◇ Select the node ’Boundary_South’.
◇ Click with your right mouse button.
◇ Select ’Model data’ - ’Flow Model’.
◇ In tab ’Boundary Condition’, select the option ’flow (Q)’.
◇ Select ’Function of time :’.
◇ Press *Table...*.
◇ Press *Add Row*.
◇ Enter 5.
◇ Press the *OK* button.

◇ Fill in the table as shown in Figure 4.49.

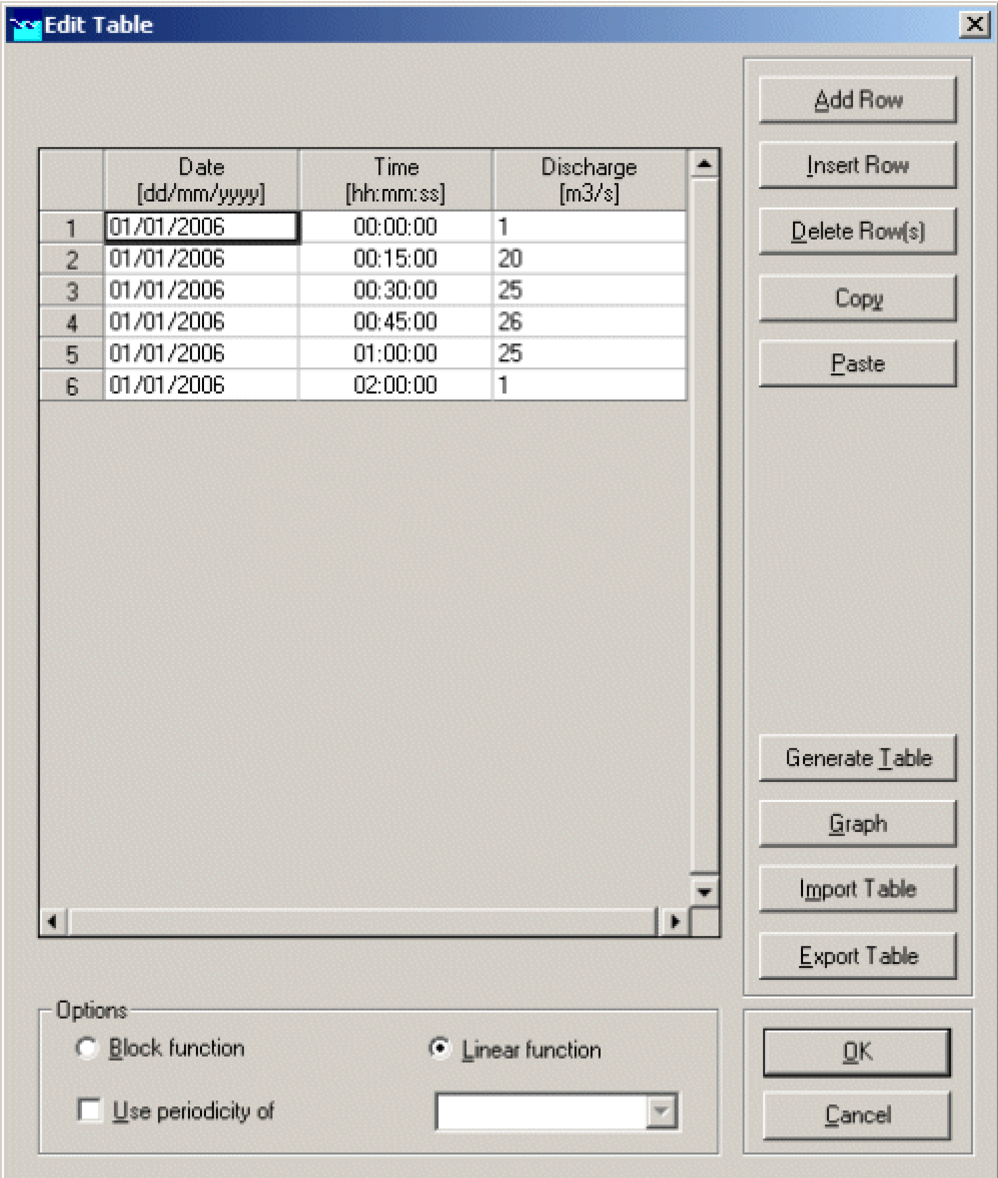


Figure 4.49: *Filling in the 1D Q-boundary table.*

◇ Press the *OK* button.
◇ Press the *OK* button.
◇ Select the node "Boundary_North".
◇ Click with your right mouse button.
◇ Select 'Model data' - 'Flow Model'.
◇ Select the option 'Q-h relation'.
◇ Press the *Table...* button.
◇ Press the *Add Row*.
◇ Enter 4.
◇ Press the *OK* button.
◇ Fill in the table as shown in the figure below.

	Water Level [m ref.lvl]	Flow [m³/s]
1	2.1	-0.1
2	3.85	-1
3	4.5	-5
4	5	-10
5	5.5	-25

◇ Press the *OK* button.
◇ Press the *OK* button.

Now we have defined the channel branch from which the flooding will occur. The next step is to model the 'dummy branch' containing the structure that is used to control the flooding. We will use a Connection Node to connect the dummy branch to the Amstellandboezem at the location shown in Figure 4.50. We will also add a cross-section and a structure, a weir.

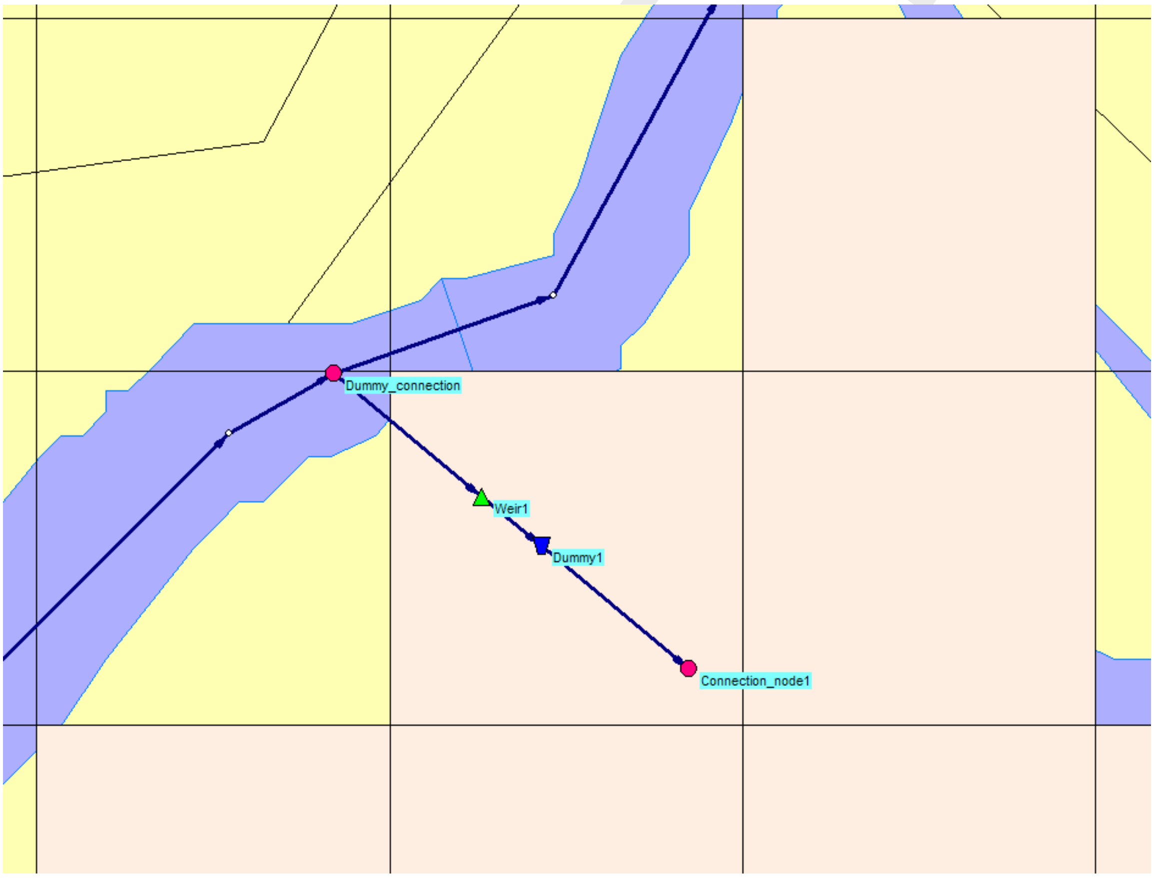


***Figure 4.50:** The dummy branch used to control the flooding of the Amstellandboezem into the 2D grid.*

◇ Select the button, Flow - Calculation Point.
◇ Select the button to select the function 'Add node'.
◇ Enter 'Dummy_connection' in both input fields.
◇ Click the *OK* button.

◇ Click on the left-mouse button again to actually add the Dummy_connection node on your screen. This node should be placed at the location where we want to connect the dummy branch to the existing branch as shown in Figure 4.50).

- ◇ Select the button, Flow-Connection Node, by clicking on it.
- ◇ Select the button *Split reach at node* Split reach at node in the 'Reach' toolbar.
- ◇ Use the left-mouse button on the Dummy_connection node to actually add the Flow - Connection node on your screen.

By following the above steps, we have split the reach at the location of the Dummy_connection h-calculation node and replaced it with a connection node. Next, we will set the Cross-section bathymetry to interpolate along the two reaches:

- ◇ Select the 'Dummy_connection' node by left-mouse clicking it
- ◇ Click with your right mouse button.
- ◇ Select 'Model data' - 'Flow Model'.
- ◇ Left-mouse click the tab 'Interpolation Over node'.
- ◇ Enable the option 'Interpolate Cross-section bathymetry (only) over Node' by left-mouse clicking it.
- ◇ For 'First Reach', select the upstream reach '2'.
- ◇ For 'Second Reach', select the downstream reach '3'.
- ◇ Press the *OK* button.

- ◇ Select the button to select the function 'Add node'.
- ◇ Enter 'Connection_node1' in both input fields.
- ◇ Click the *OK* button.

- ◇ Click on the left-mouse button again 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 node 'Dummy_connection' and drag to the node "Connection_node1" while keeping the button down. Release the left mouse button.
- ◇ Select the node, Flow - Cross Section node.
- ◇ Select the (add node) function.
- ◇ Enter 'Dummy1' 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.
- ◇ Select the node, Flow - Weir node.
- ◇ Select the (add node) function.
- ◇ Enter 'Weir1' in both input fields.
- ◇ Click the *OK* button.

- ◇ Click on the left-mouse button to actually add the Flow-Weir node on your screen.
- ◇ Select the Cross section node 'Dummy1'.
- ◇ Click with your right mouse button.
- ◇ Select 'Model data' - 'Flow Model'.
- ◇ Select the 'Location' tab.
- ◇ Enter the value 2 as the bed level.
- ◇ Enter the value 4 as the surface level.
- ◇ Select the 'Cross section' tab.
- ◇ Select 'Rectangle' as the cross section type.
- ◇ First enter the name 'Dummy1' at 'Cross section: '.

- ◇ Click the *Define dimensions* button.
- ◇ Enter 10 m as width.
- ◇ Enter 2 m as height.
- ◇ Press the *Save dimensions* button.
- ◇ Press the *OK* button.

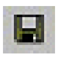
- ◇ Select ’Friction’ tab.
- ◇ Select ’Use local value(s) for this cross section’.
- ◇ Select ’Local value(s)’ in the ’Show’ combo box.
- ◇ Select ’Chézy (C)’ for type friction (Bed).
- ◇ Enter ’50’ for constant value.
- ◇ Select ’Initial Value’ tab.
- ◇ Select ’Use local initial value for this reach’.
- ◇ Select Initial ’water level’.
- ◇ Enter the value 3.85 m.
- ◇ Enter the value $0\,m^3\,s^{-1}$ for the initial flow in positive direction.
- ◇ Click the *OK* button to close.

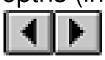
- ◇ Select the Weir node ’Weir1’.
- ◇ Click with your right mouse button.
- ◇ Select ’Model data’ - ’Flow Model’.
- ◇ Enter 2 m as width.
- ◇ Enter 0.7 m as crest level.
- ◇ Select the ’Controller’ tab.
- ◇ Select ’Time’ as type of controller.
- ◇ Enter ’Weir1’ as name.
- ◇ Press the *Table...* button.
- ◇ Press the *Add Row*.
- ◇ Enter 2.
- ◇ Press the *OK* button.
- ◇ Fill in the table as shown in the figure below.

	Date [dd/mm/yyyy]	Time [hh:mm:ss]	Crest Level [m above datum]
1	01/01/2006	00:00:00	4
2	01/01/2006	00:15:00	1.9
3	01/01/2006	02:30:00	1.9

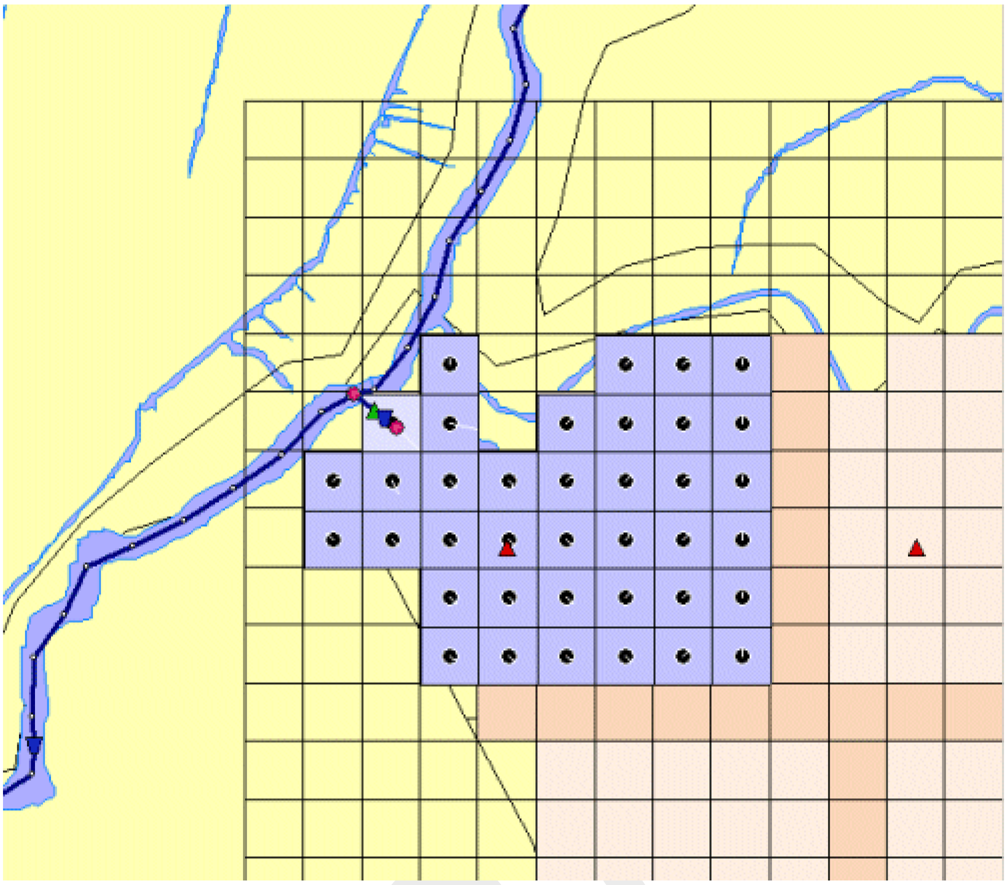
- ◇ Press the *OK* button.
- ◇ Press the *OK* button.

Your new schematisation is now ready.

- ◇ Select  button in NETTER.
- ◇ Select the menu item ’File’ - ’Exit’, to leave NETTER.
- ◇ Press the *OK* button.

- ◇ Select ’Case’ - ’Save’.
- ◇ Double click the ’Simulation’ task block.
- ◇ Double click the ’Results in Maps’ task block.
- ◇ Select ’Depths (incremental)’ from the Active legend.
- ◇ Click the  buttons and keep them pressed to animate the flooding of the area.

The resulting flooding is shown in the figure below:



- ◇ Select 'File' - 'Exit' to leave the Results in Maps task block.
- ◇ Select 'Case' - 'Save'.
- ◇ Select 'Case' - 'Close'.
- ◇ Select 'Case' - 'Exit'.
- ◇ Select 'Files' - 'Exit Sobek'.

4.4 Tutorial Hydrology in polders (SOBEK-Rural RR module)

4.4.1 Introduction

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.

4.4.2 Getting started

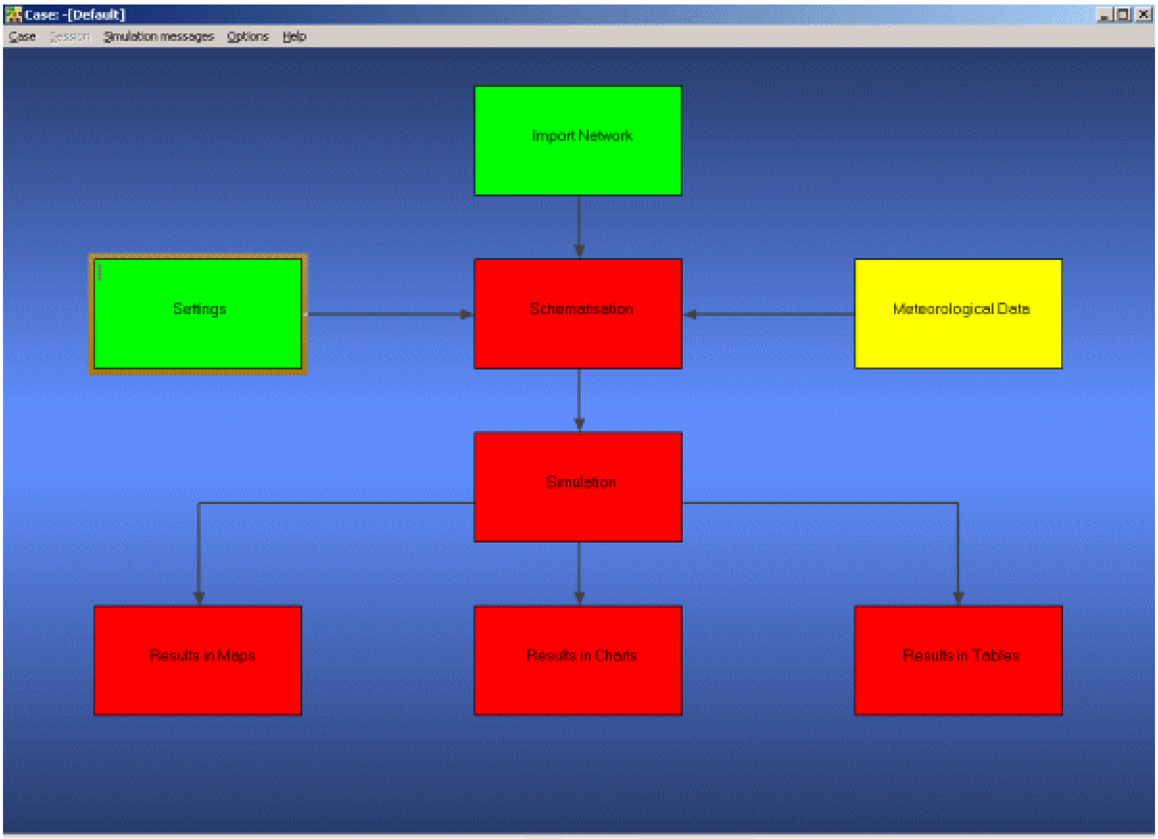
- ◇ Click on the Windows *Start* button.
- ◇ Select the ’Programs’ menu.
- ◇ Select the ’All Programs’ or ’All Apps’ menu.
- ◇ Select the ’Delft Hydraulics’ menu.
- ◇ Select the ’SOBEK’ menu item (SOBEK215).
- ◇ Click on the ’SOBEK’ icon.
- ◇ Select the menu item ’Options’ - ’SOBEK Options’.
- ◇ Select the tab ’Background Map’.
- ◇ Select the file ’Tutorial1.map’.
- ◇ Press *OK* button to save and close SOBEK Options.
- ◇ 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.

4.4.3 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:

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:

1 Administration of cases (which data is 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.);
  ◇ view and check the status of all tasks;

- ◇ view the relation between the various tasks.
- ◇ choose and run predefined tasks (modules);

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:

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 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.

4.4.4 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.

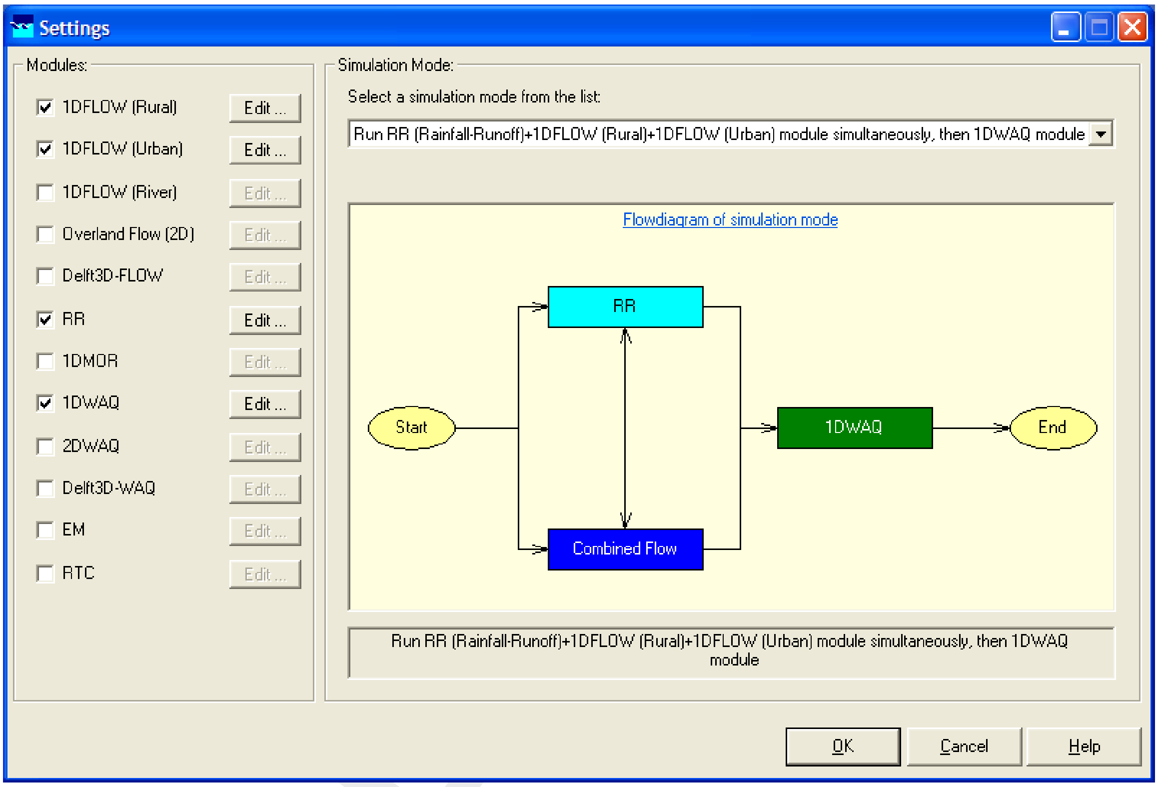
- ◇ Select the radio button 'Start from scratch'.
- ◇ Press the *OK* button.

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

4.4.5 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.

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'.

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

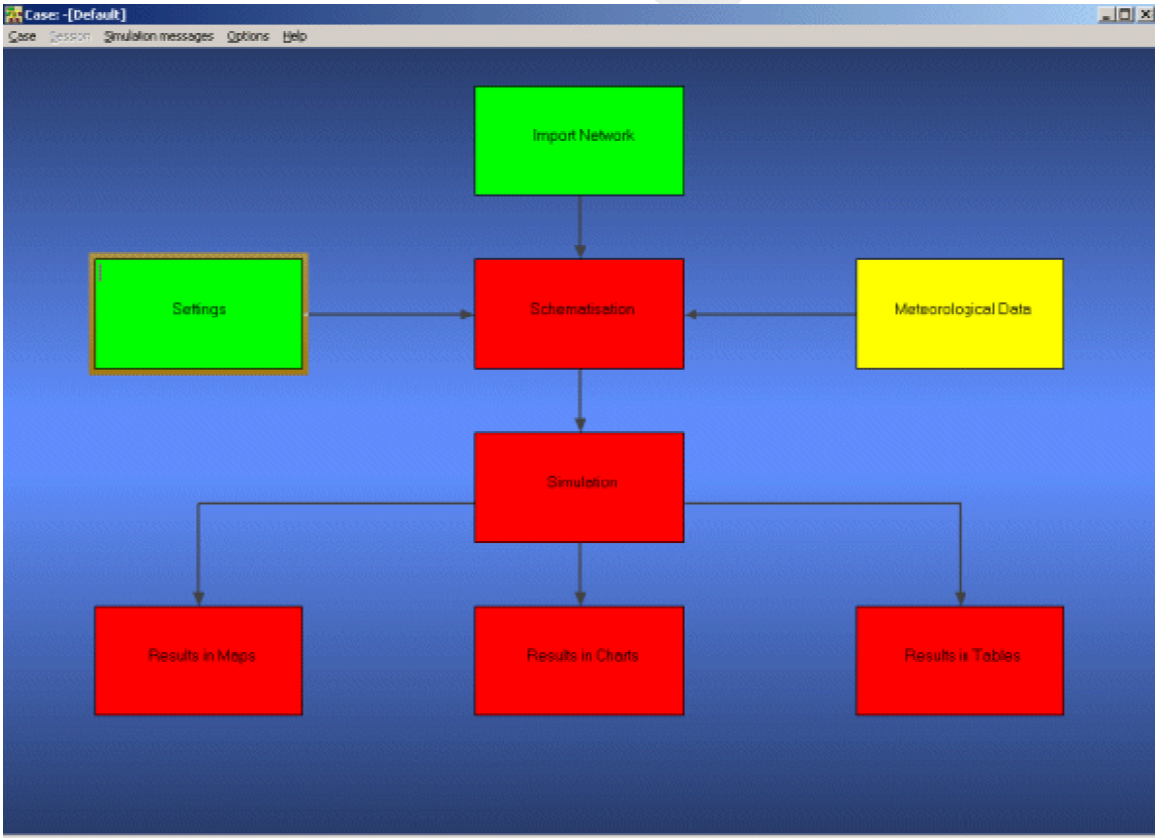
◇ Press the *OK* button.

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.



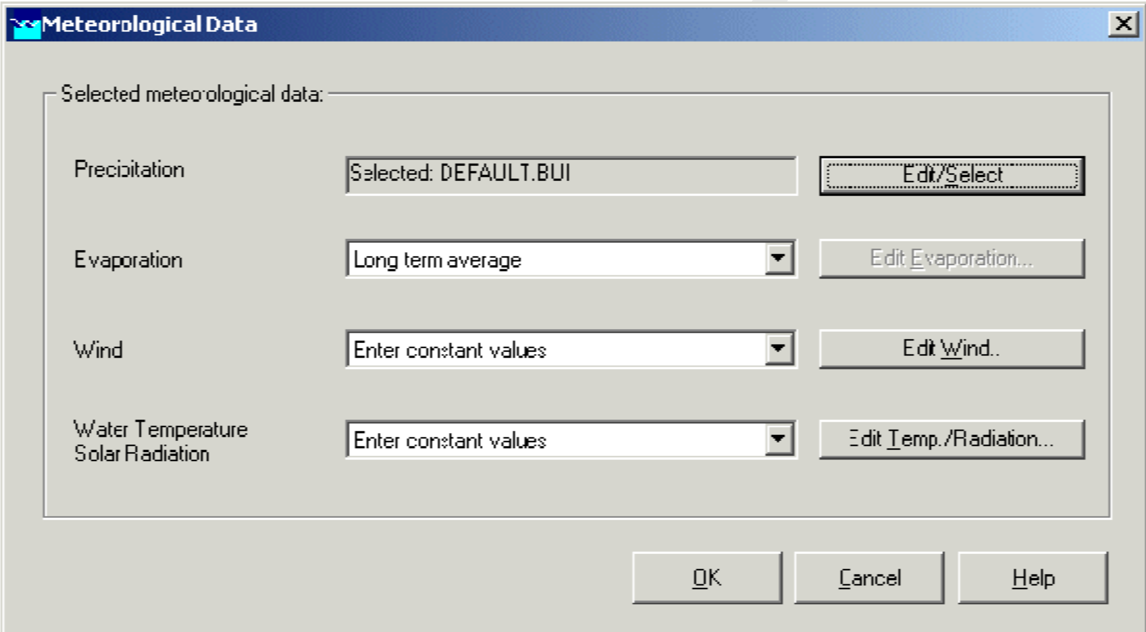
4.4.6 Task block: Meteorological Data

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

4.4.7 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:

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. 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.

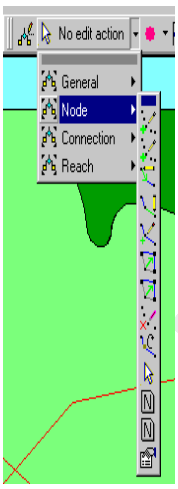
◇ Select button.
◇ Move the mouse pointer to the main window.
◇ Click and hold the left mouse button, make a rectangle by dragging the pointer across the main window. The size of this rectangle determines the magnification.
◇ Release the left mouse button.

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.

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'.
- ◇ Set the 'ID' and 'Name' to 'automatic'.
- ◇ 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.

- ◇ 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.

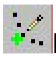
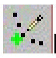
In order to see the identifiers on the map please:

- ◇ Click the button in the Active Legend or select the menu item 'Options' - 'Network Data...'.
- ◇ Select the tab 'Node'.
- ◇ Select the radio button 'Name'.
- ◇ Select the tab 'Link'.
- ◇ Select the radio button 'None'.
- ◇ Press the *OK* 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 Station.
- ◇ 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 Station while keeping the button down. Release the left mouse button.
- ◇ Click with the left mouse button on the RR - Pump Station 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 during this tutorial:

Unpaved area node

1 Area per crop = 280 ha grass;
2 Groundwater area = 400 ha;
3 Surface level = 1 m above reference level (constant value);
4 Soil type = sand_maximum ($\mu = 0.117$ per m);
5 Thickness 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;
6 Initial groundwater level = equal to target level open water or level at boundary node;
7 Storage, Maximum on land = 3 mm;
8 Storage, Initial = 0 mm;
9 Infiltration capacity = 10 mm/hr;
10 Computation option for drainage = De Zeeuw - Hellinga
11 Reaction factor Surface runoff = 100 1/day;
12 Reaction factor Horizontal inflow = 0.05 1/day;
13 Reaction factor Drainage 0 - 1 m alpha = 0.4 1/day;
14 Reaction factor Drainage 1 - 2 m alpha = 0.4 1/day;
15 Reaction factor Drainage 2 - 3 m alpha = 0.4 1/day;

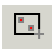
16 Reaction factor Drainage 3 - infinity alpha = 0.4 1/day;
17 Seepage = 0 mm/day.

- 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 groundwater 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 ($\mu = 0.117$ per m)'.
- Type '5' in the input box 'Thickness groundwater layer'.
- Type '1.5' in the input box 'Maximum allowed groundwater 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/d].
- Enter “0.05” as the horizontal inflow reaction factor.
- In the 'Reaction factor [1/d]' 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 – ∞	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 dragging while keeping the button down. Release the left mouse button.
  Alternatively, select both RR - Unpaved nodes by left-clicking the two nodes while keeping the `Ctrl` pressed.
- ◇ 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 Station.
◇ 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.
  ◻ If you changed the default values in this table: Click the *OK* button. ('*Enter the table name*')
◇ In the group box 'High capacity', enter the additional capacity of "0".
◇ Click the *OK* button to leave the 'Data Edit for Pump' window.

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 as well!

◇ Select the menu item 'Case' - 'Save As...'.
◇ Enter the name "Case_one".
◇ Click the *OK* button.

4.4.8 Task block: Simulation

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

◇ Double-click the task block 'Simulation'.

4.4.9 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 graph 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.

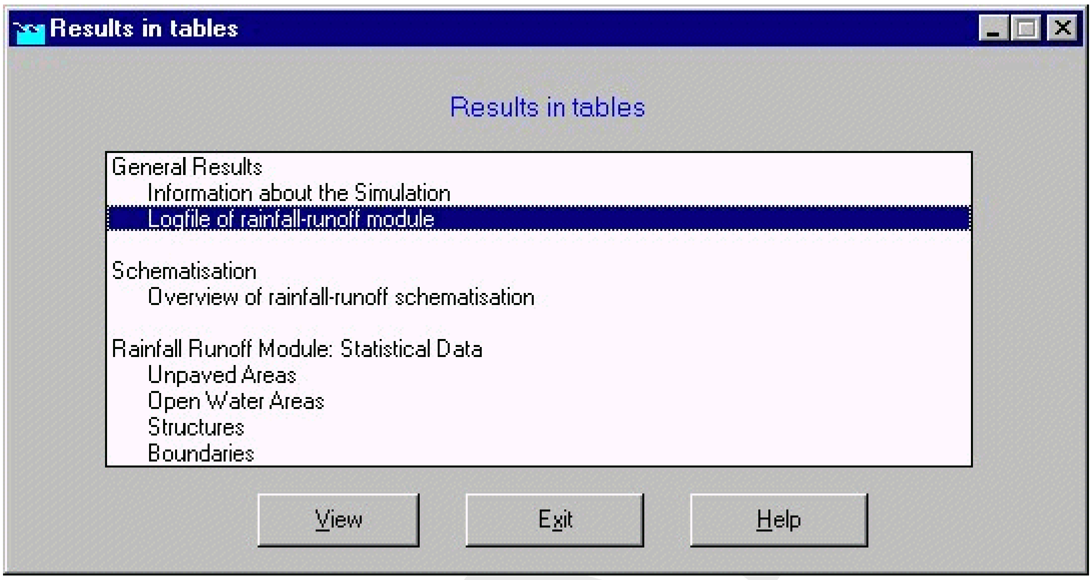
As you see a graph appears containing two variables!

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

4.4.10 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.

4.4.11 Task block: Results in Charts

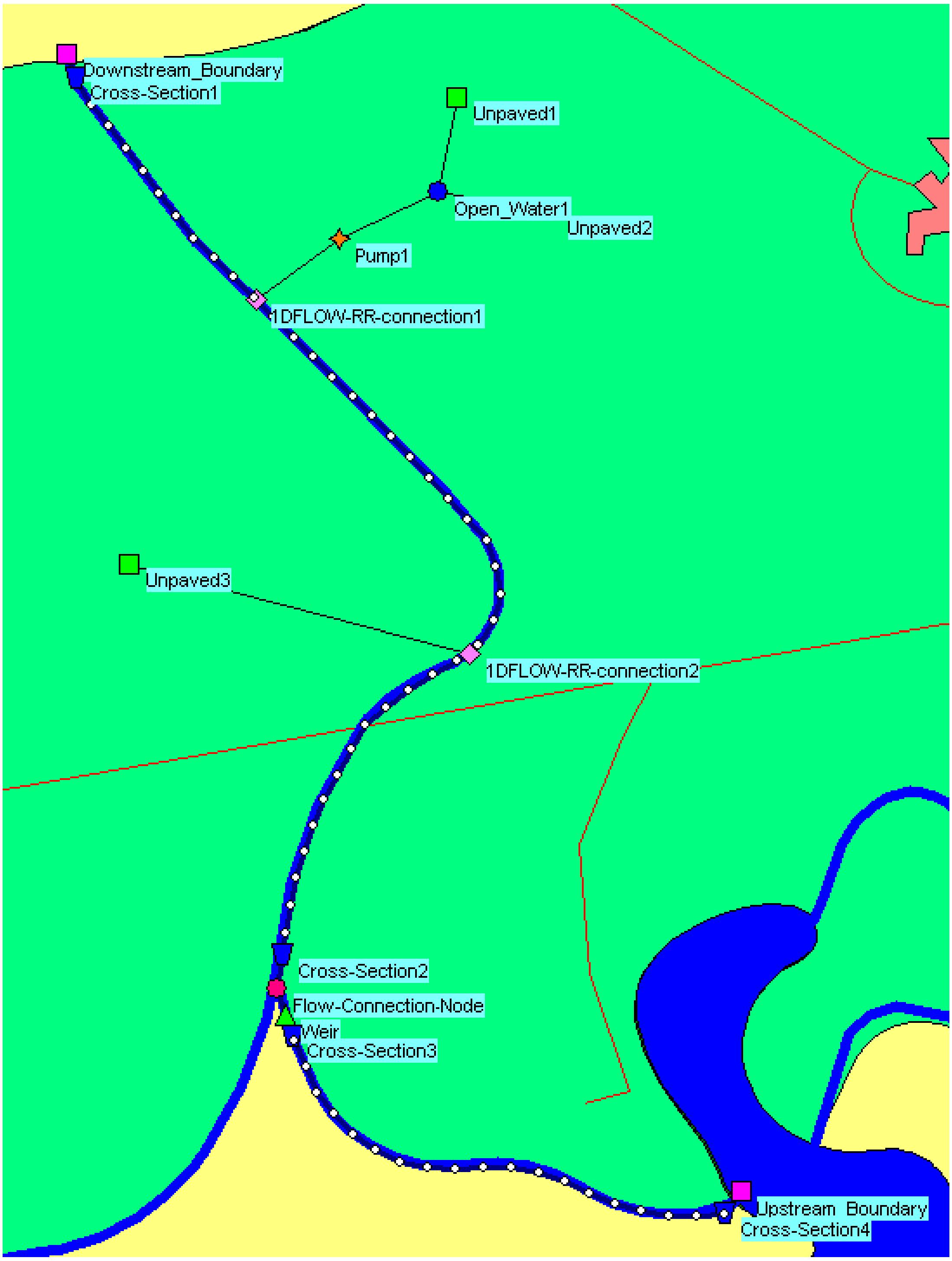
In the task block 'Results in Charts' the user can easily depict result data in one graph.

◇ 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.

4.4.12 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.



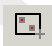
The RR model determines the runoff of various types of areas each with their characteristic influence on the dynamics of the rainfall-runoff process. The different area types can be described by various types of so-called runoff-nodes. The computed runoff is used as input for a one-dimensional hydrodynamic 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 will be 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 schematisation by clicking on the map and dragging 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.

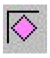
Note:
Please note that for larger models than this tutorial, it is recommended to also run the "Clean Up Utility" on both the models before combining them. This is recommended in order to prevent the IDs of removed or unused network objects for both models conflicting with each other, causing further simulations to stop with an error message "Double ID in file ...". Adding a prefix without cleaning up the models is not enough to guarantee unique IDs. This utility can

be found in the schematisation menu, button "Cleanup 1D/2D Model Data".

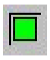
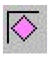
Now we will import the schematisation of the tutorial 'Hydrodynamics in open water'.

◇ 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?*).

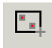
Now we will connect the two schematisations.

◇ 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 Station and drag to the '1DFLOW-RR_connection1' node while keeping the button down. Release the left mouse button.

Note:
Take care not to accidentally connect the pump station to a calculation point rather than the 1DFLOW - RR Connection node!

◇ 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 schematisation by clicking on the map and dragging 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 *OK* button.
- ◇ Select the menu item ’Case’ - ’Save As...’.
- ◇ Enter the name “Extended”.
- ◇ Click the *OK* button.

- ◇ Double-click the task block ’Simulation’.

Note:
This step cannot be completed while in Free Trial mode. The number of nodes in the schematisation now exceeds the amount of nodes allowed in a free trial computation.

- ◇ 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!

4.4.13 Epilogue

In this tutorial some of the most important aspects of working with SOBEK have 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!

5 Graphical User Interface

5.1 Case management

5.1.1 Task block: Import Network

5.1.1.1 Procedure for importing a Duflow model

Duflow models for the computation of the water levels and flows in water systems can be imported into SOBEK-Rural. As SOBEK makes use of a Geographical Information environment, its models need coordinates for the nodes. These coordinates are available in models that are used in Duflow for Windows. For older models the Duflow file <conversion.sid> must be filled with coordinates for all connection nodes according to the format of Duflow for Windows (example):

NOD NOD00000 150912 421819 1
NOD NOD00001 153064 419729 2
NOD NOD00002 141070 417258 6
SCH SCH00000 149755 416906 3
ARE ARE00000 149755 416906 3 1
WEI WEI00000 144207 417130 4 5

If a model is stored in the binary format of Duflow, the model must be converted to the original ASCII-format (.net). This can be done as follows:

- ◇ Open the model (.dms).
- ◇ Select Calculation from the menu.
- ◇ Select Convert network.
- ◇ Select Write Duflow files. Now the .net and .sid files are saved.

The following Duflow functionalities are imported into SOBEK:

- ◇ Network topography (Duflow for Windows format)
- ◇ Profiles (Tabulated profiles containing flow and storage widths)
- ◇ Bed friction (Chézy and Mannings)
- ◇ Boundary conditions. (Constant and time table of discharges and water levels)
- ◇ Structures. (Broad crested weir and round, rectangular and elliptical culverts. Flow factor μ on a culvert is not used in SOBEK. Instead an inlet and outlet loss coefficient must be selected)

Manually the following functionalities have to be added.

- ◇ Initial conditions
- ◇ Simulation time and time step
- ◇ Pump stations
- ◇ Inlet and outlet loss coefficient of culverts.

The import procedure is as follows:

- ◇ Open a project.
- ◇ Open a new case .
- ◇ Click the block <Settings> in the Case Manager, Select the module 'Channel Flow', directly followed by *OK* .

◇ Click the block <Meteorological Data>, directly followed by *OK* .
◇ Click the block <Import Network>.
◇ Select 'Import Duflow network'.
◇ Press *OK*.
◇ Open a Duflow scenario file at any location (other files of the scenario must be available in the same location).

After the reading of the file, the user is asked for a shift, rotation or scaling of the coordinates that are given in the Duflow geometry file <conversion.sid>:

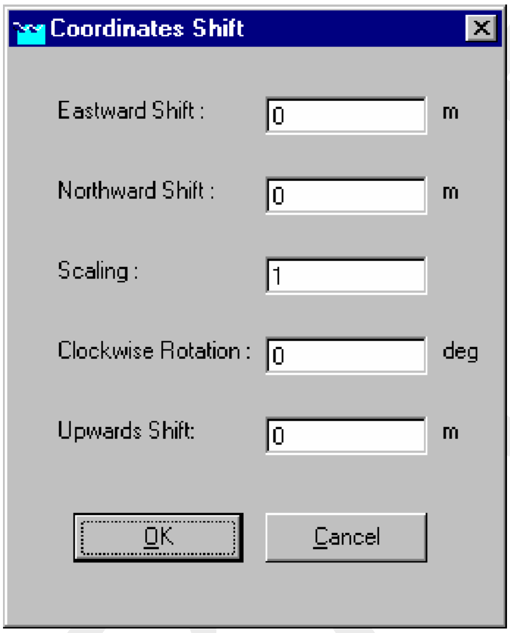


Figure 5.1: *Rotate or shift coordinates during Duflow import.*

◇ Modify the coordinates shift if required and press *OK*. After converting the data the network editor is started and the topology of the network is displayed. If the network is not visible select 'View' from the menu followed by 'Show full network'.
◇ The location of the nodes can be moved with 'Edit', 'Vector Layer'. Also curves in branches can be constructed by adding vector points (See SOBEK manual for details).
◇ If the topology is satisfactory, the network can be saved with 'File', 'Save', 'Network'. In older versions of the network editor the procedure is: 'File', 'Save as', 'Network', change the name from network.sob into network.ntw, 'Save' and replace the existing network.ntw;
◇ Select 'File', 'Exit'.
◇ The log-file is shown.
◇ Close the log-file.
◇ The import procedure is executed successfully and <Schematisation> can now be started to finalize the model manually.

5.1.1.2 Procedure for importing a Mike11 model

Mike11 models for the computation of the water levels and flows in water systems can be imported into SOBEK-Rural. To convert the cross sections, these cross sections should first be exported in Mike11 as raw data. The file name must be identical to the binary cross section, except the extension should be changed to .txt.

The following Mike11 functionalities are imported into SOBEK:

- ◇ Network topography
- ◇ Profiles (first exported as raw data)
- ◇ Locations of boundary conditions and type of boundary condition (water level or flow). The values of the boundary conditions are stored in a binary format and can therefore not be read by the conversion program.
- ◇ Weirs with a fixed width
- ◇ Weirs with a varying width over the height

Manually the following functionalities have to be added.

- ◇ Initial conditions
- ◇ Simulation time and time step
- ◇ Calculation grid
- ◇ Boundary condition values
- ◇ Friction values

The import procedure is as follows:

- ◇ Open a project.
- ◇ Open a new case .
- ◇ Click the block <Settings> in the Case Manager, Select the module 'Channel Flow', directly followed by *OK* .
- ◇ Click the block <Meteorological Data>, directly followed by *OK* .
- ◇ Click the block <Import Network>.
- ◇ Select 'Import Mike11 network'.
- ◇ Press *OK*.
- ◇ Open a Mike11 simulation file (*.sim) at any location (other files of the scenario must be available in the same location).

A shift, rotation of the coordinates can be applied with the window that pops up (scaling is not possible):

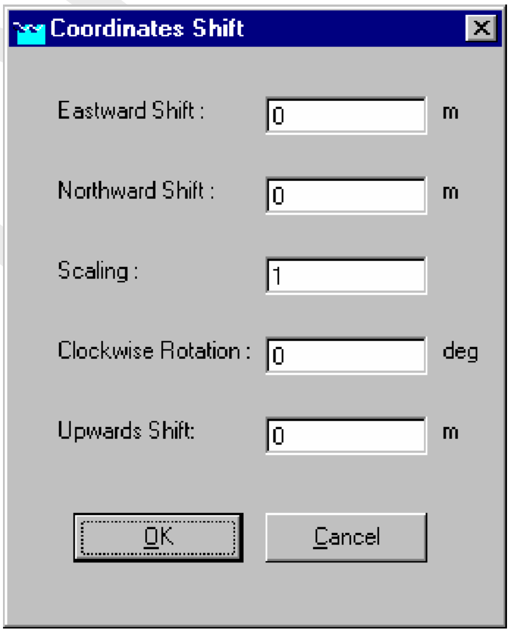


Figure 5.2: *Rotate and shift coordinates during MIKE11 import.*

- ◇ Modify the coordinates shift if required and press *OK*.After converting the data the network editor is started and the topology of the network is displayed. If the network is not visible select 'View' from the menu followed by 'Show full network'
- ◇ The location of the nodes can be moved with 'Edit', 'Vector Layer'. Also curves in branches can be constructed by adding vector points (See SOBEK manual for details). If the topology is satisfactory, the network can be saved with 'File', 'Save', 'Network'. In older versions of the network editor the procedure is: 'File', 'Save as', 'Network', change the name from network.sob into network.ntw, 'Save' and replace the existing network.ntw;
- ◇ Select 'File', 'Exit'.

The log-file is shown.

- ◇ Close the log-file.

The import procedure is executed successfully and <Schematisation> can now be started to finalize the model manually.

5.1.1.3 Import SVK19 files

SVK19 is the standard Scandinavian exchange format for sewer systems, comparable with the SUF-HYD exchange format in the Netherlands. This functionality enables SOBEK to import Mouse schematisations.

Importing Mouse files is available under the Import menu, Import sewerage networks (SUF-HYD), and selecting the file type *.txt.

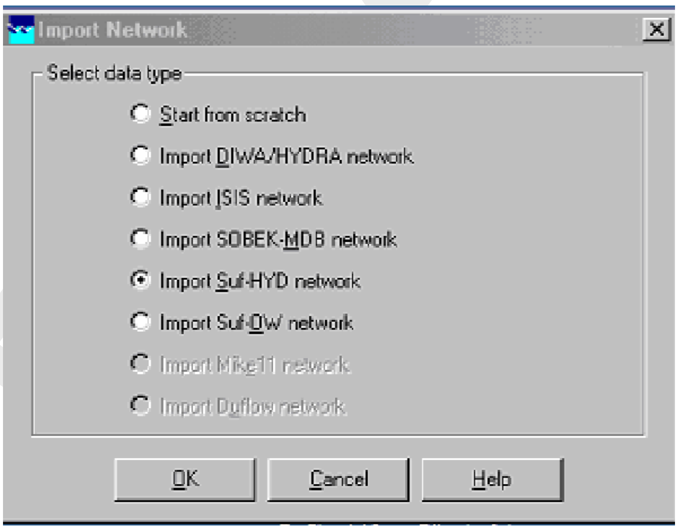


Figure 5.3: *The Import Network window.*

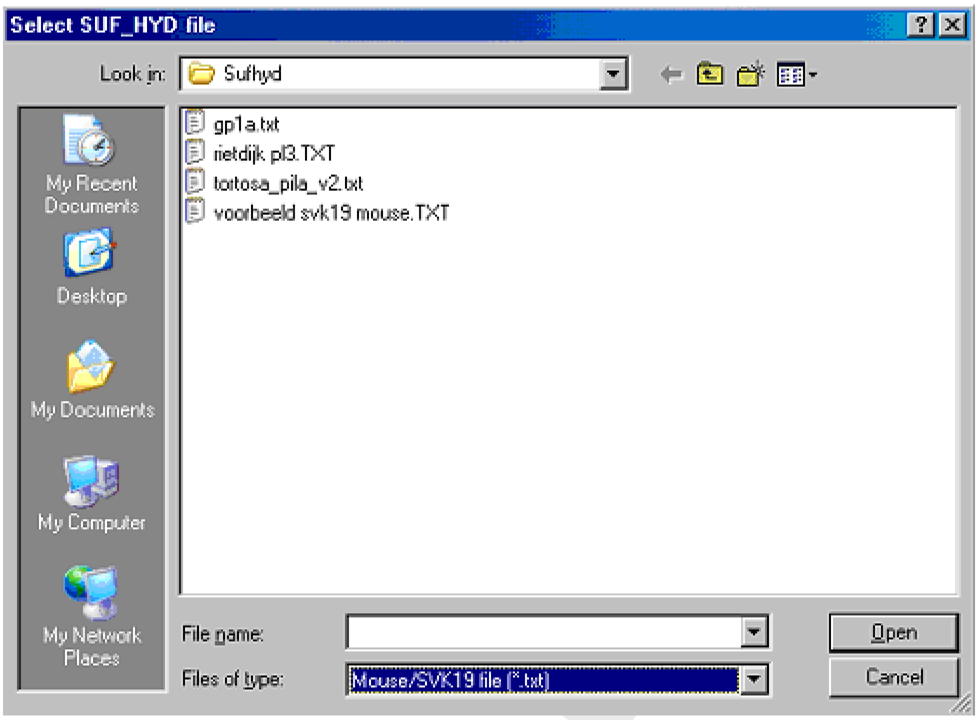


Figure 5.4: *Select SUF-HYD file.*

It is possible to adjust the coordinates from the imported SVK19 file to your own coordinate system by a north/east shift, to be specified in the next screen:

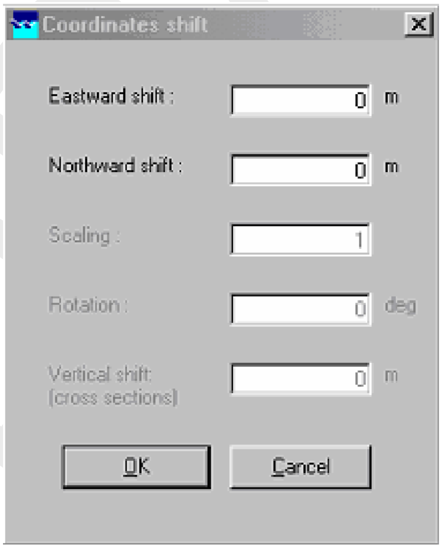


Figure 5.5: *Shift coordinates during SVK19 import.*

When the SVK19 file is imported, it is possible also to import a Mouse runoff file (HGF file).

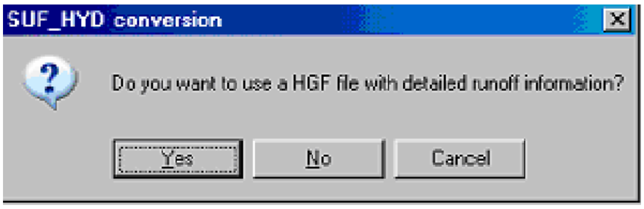


Figure 5.6: *Importing an optional Mouse runoff file.*

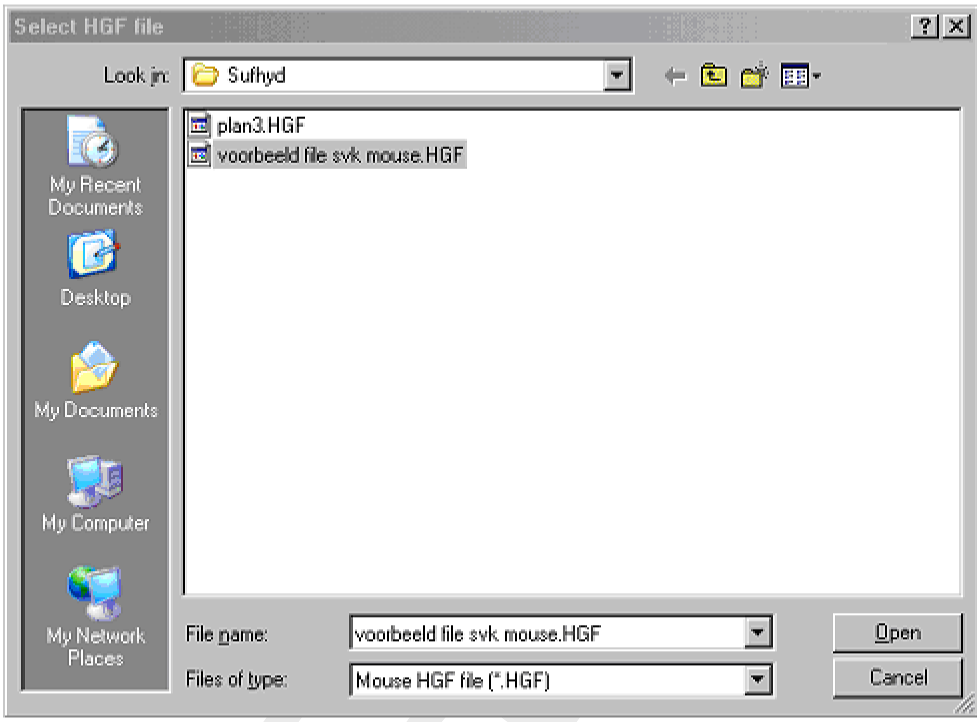


Figure 5.7: *Opening a Mouse runoff file.*

The standard Mouse HGF-runoff catchment types have to be converted to the standard NWRW types of area (combination of surface type and slope).

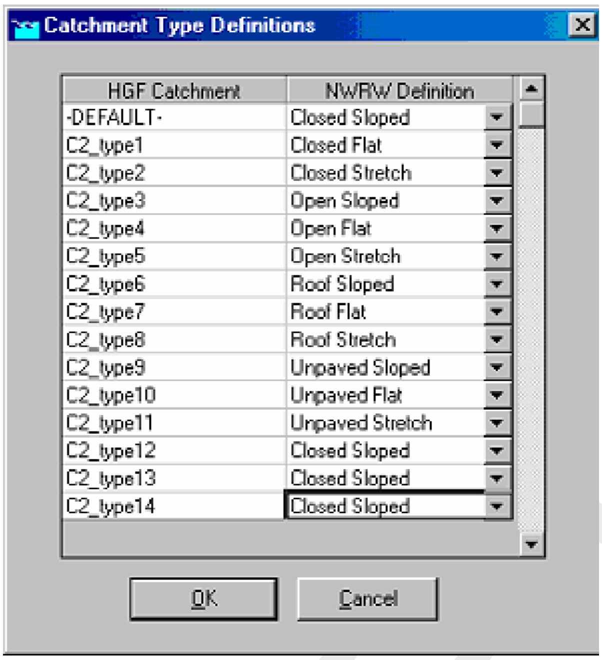


Figure 5.8: *Converting Mouse HGF-runoff catchment types.*

Finally, the SVK19 id's can be prefixed with a district id.

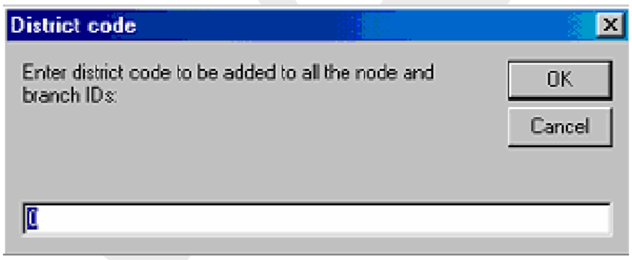


Figure 5.9: *Prefixing SVK19 id's with a district id.*

5.1.2 Task block: Settings

5.1.2.1 Rural

Settings window for Channel Flow
In the **Settings** window, you can define general settings for the module of your choice. Here the section on the Channel Flow module is discussed. The Settings window can be accessed by double-clicking the "Settings" task block in the Case Management Tool window.

One can define:

◇ The simulation period & the computational time step
◇ Initial data, restart data
◇ Output options: which output, and by which time interval
◇ and more

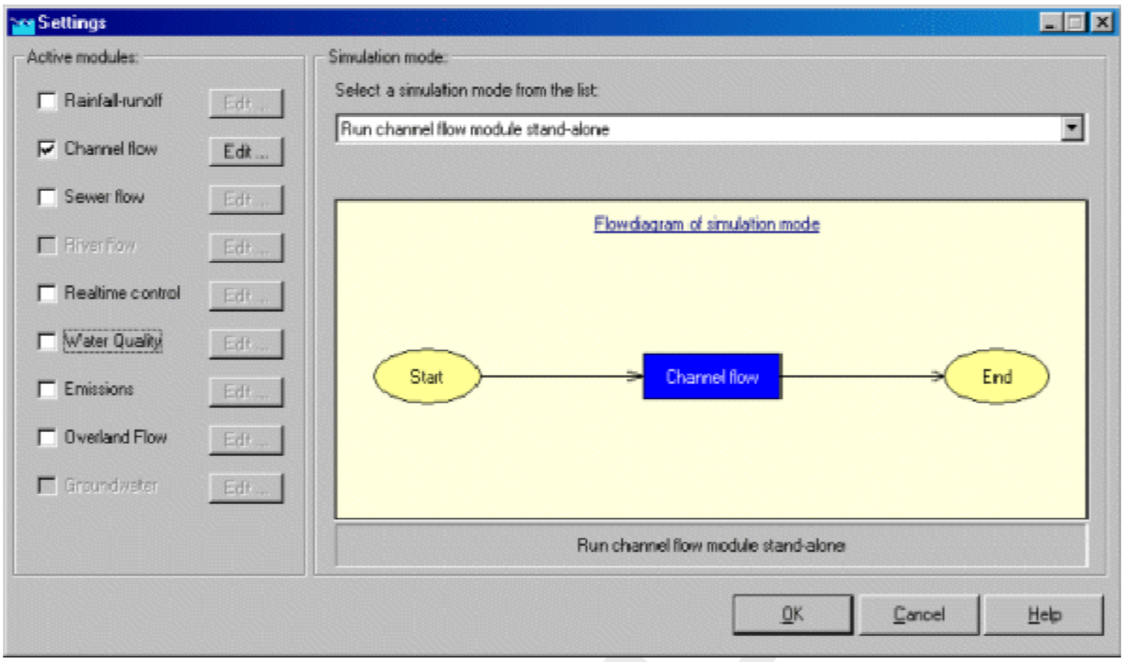


Figure 5.10: *The **Settings** window with only the Channel Flow module activated.*

Clicking the *Edit* button, next to the Channel Flow check box, will open the "Settings for Channel Flow module" window. The options for that module are discussed in the next chapters.

Simulation options

If the Channel Flow module is combined with other modules, the modeller has various simulation options. For example, one can run the Rainfall-Runoff module and the Channel Flow module simultaneously or sequentially:

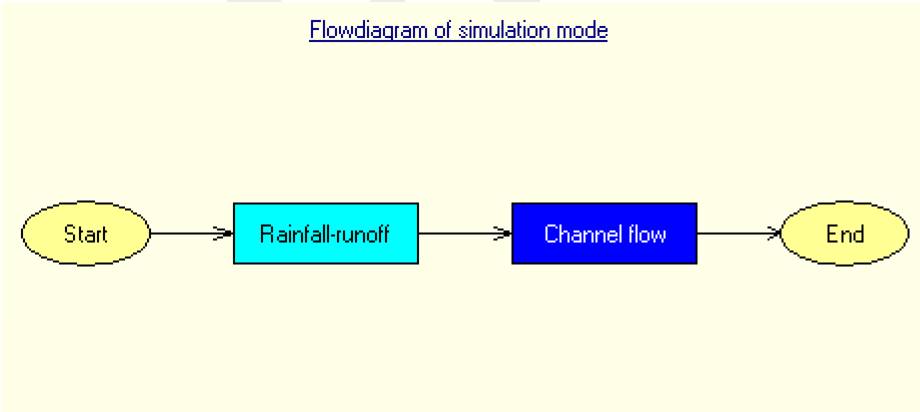


Figure 5.11: *Flowdiagram of running the Rainfall-Runoff module and Channel Flow module sequentially.*

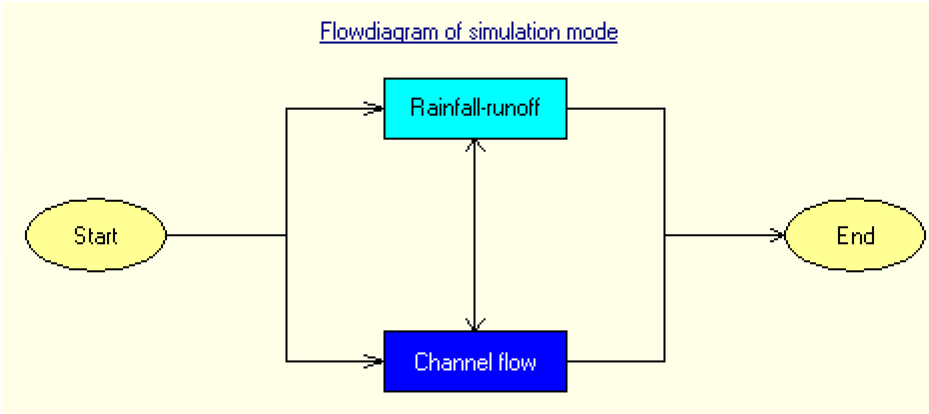


Figure 5.12: *Flowdiagram of running the Rainfall-Runoff module and Channel Flow module simultaneously.*

These options can be selected in the "Simulation mode" section:

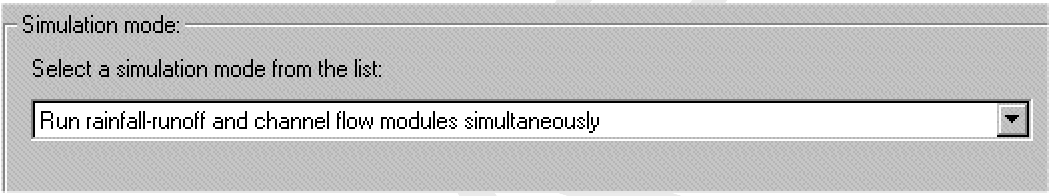


Figure 5.13: *Selecting a simulation mode.*

Tab - Time settings

One of the options when opening "Settings" for Channel Flow is "Time settings". These can be found on the "Time settings" tab. The options are explained below.

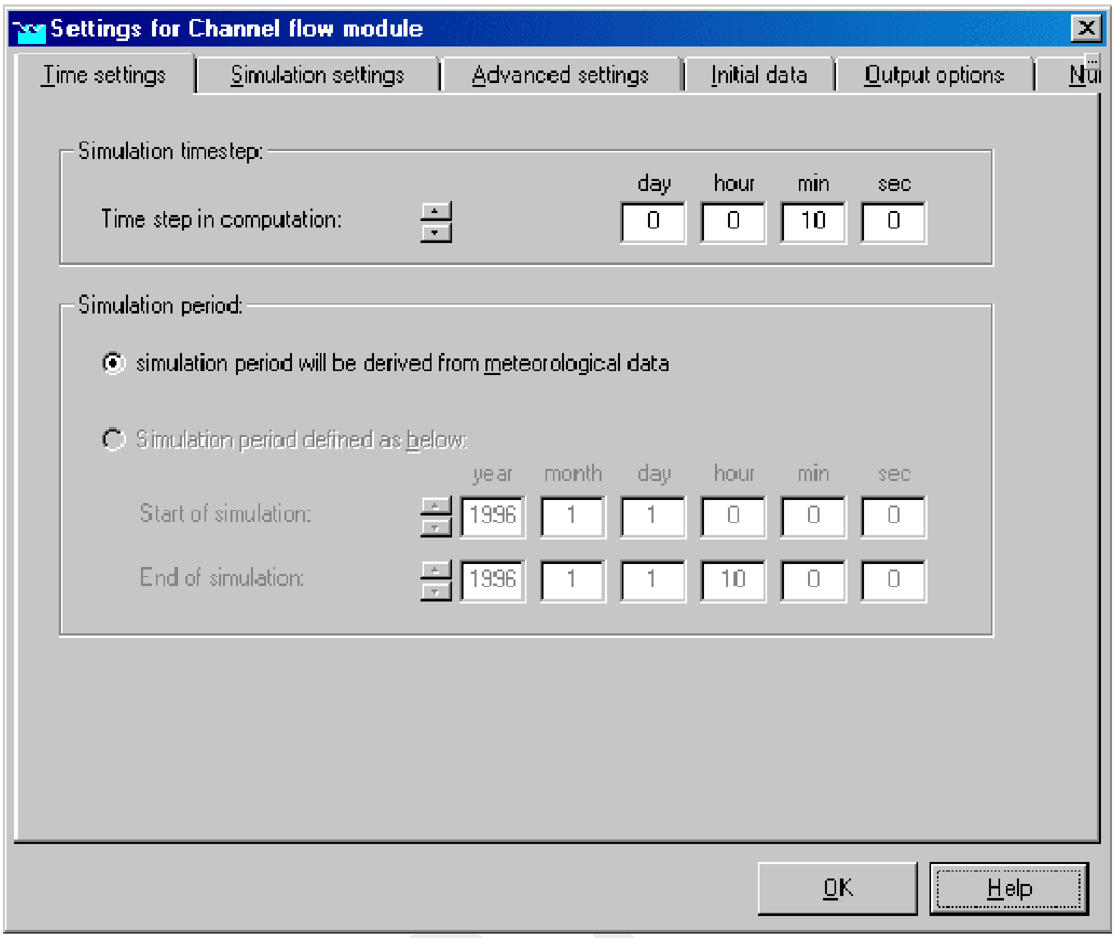


Figure 5.14: *The Time settings tab of the Channel flow module.*

Simulation time step

Here, you can define the computational time step that SOBEK uses in its hydrodynamic computations. Note that the time scale for hydrodynamic processes lies in the order of minutes to even seconds. For most computations, a time step of 10m should be sufficient, but for river systems with a very quick respond-time (such as mountain streams), a smaller time step might be more appropriate.

Note: SOBEK will automatically reduce its computational time step if necessary. Information about whether this has happened during your simulation can be found in the results "simulation info at branch segments".

Simulation period

The simulation period can be defined in two ways: it can be derived from the precipitation event that you have defined (Meteorological data task block), or it can be defined manually.

If the Rainfall-Runoff module is active too, only the first option is available. In the *Edit* section of the Rainfall-Runoff module, you can then define a simulation period manually or refer to a precipitation event.

Tab - Simulation settings

One of the options when opening "Settings" for Channel Flow is "Simulation settings". These can be found on the "simulation settings" tab. The options are explained below.

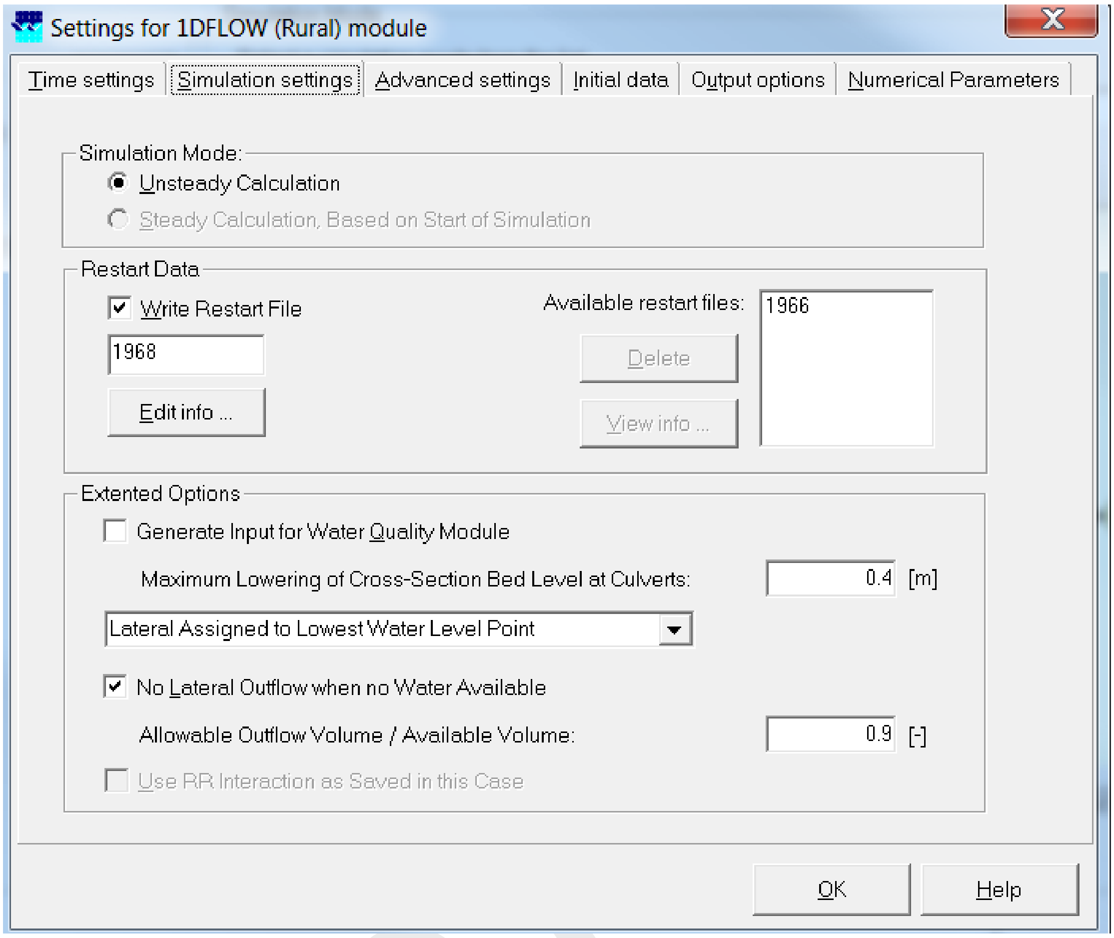


Figure 5.15: *The Simulation settings tab of the Channel flow module.*

Simulation mode

- **Unsteady calculation** means that the computation will be simulating a certain time span, where it will not branch a steady state, but where all conditions within the model may change during time (rainfall, boundary conditions etc.)
- **Steady calculation** means that the simulation will only calculate **one** situation: the equilibrium state. All boundary conditions will be fixed values (non-variable in time) and the model will calculate the situation where an equilibrium is branched.

Restart data

A restart file contains all information of the state of a simulation during its **last** time step. This file can optionally be used to define the initial values for a new simulation.

The option "write restart file" allows you to create a restart file from the last time step of the simulation that you are about to run. You can give this file a name. If you want to use it later on for the initialisation of another simulation, select it on the "Initial data tab".

Note: Restart files are no longer usable if a network has changed since the restart file was written. Additionally, it is recommended to create a new restart file when updating a SOBEK model to a new SOBEK version.

Extended options

- ◇ **Generate output for water quality module.** This option is activated only when the Water Quality module is active. However, if you run the Channel Flow stand alone, and you *consider* to run the WQ module in the future, it is wise to activate this option.If not, you would have to re-calculate the flow computations first to make future water quality computations with this case.
- ◇ **Maximum Lowering of Cross-section Bed Level at Culvert.** If the bed level of a (interpolated) cross-section in front of a culvert is lying more than the value defined for "Maximum Lowering of Cross-section Bed Level at Culvert" above the ground-layer level (= invert level + ground-layer thickness), the bed level of the cross-section is lowered to the ground-layer level. For all (interpolated) cross-sections in front of a culvert, having bed levels lying more than the value defined for "Maximum Lowering of Cross-section Bed Level at Culvert" above the ground-layer level, no changes are made only a warning is given in the SOBEK.Log message file. <u>It is advised to avoid</u> that the bed level of a cross-section in front of a Culvert is above the ground-layer level, since such situation can result in very small computational time-steps (e.g. long required wall-clock times) or even in a termination of the simulation (for more information, see section Culvert, Good modelling practice aspects). The same as discussed for a Culvert also applies for a Siphon and an Inverted Siphon.
- ◇ **List box: Lateral Assigned to Lowest (or Nearest) Water Level Point.** Using this list box, the user can specify if lateral flows are to be assigned to the lowest water level point or that lateral flows are to be assigned to the nearest water level point. For more information on the difference between assigning to the lowest or to the nearest water level point, reference is made to section 6.1.11.2
- ◇ **No Lateral Outflow when no Water Available** By checking the check-box in front of "No Lateral Outflow when no Water Available", SOBEK reduces the user-defined lateral inflow in case insufficient water is available for meeting the user-defined lateral outflow. The advantage of this option is that SOBEK does not apply (extremely) small time-steps, that require a lot of computational effort (wall-clock time). Using this option, even a termination of a simulation might be avoided. The ratio "Allowable Outflow Volume/Available Volume" can be specified. SOBEK provides as output both the user-defined lateral outflow (Defined Lateral) and the actual applied lateral outflow (Lateral Actual).
- ◇ **Use RR interaction as saved in this case** If you don't want to have to re-calculate the Rainfall Runoff computations again for every case where you only change things in the flow module, it is wise to activate this option. Note: in order to click this option, the Rainfall Runoff module should be active.

Tab - Advanced settings

One of the options when opening "Settings" for Channel Flow is "Advanced settings". These can be found on the "Advanced settings" tab. The options are explained below.

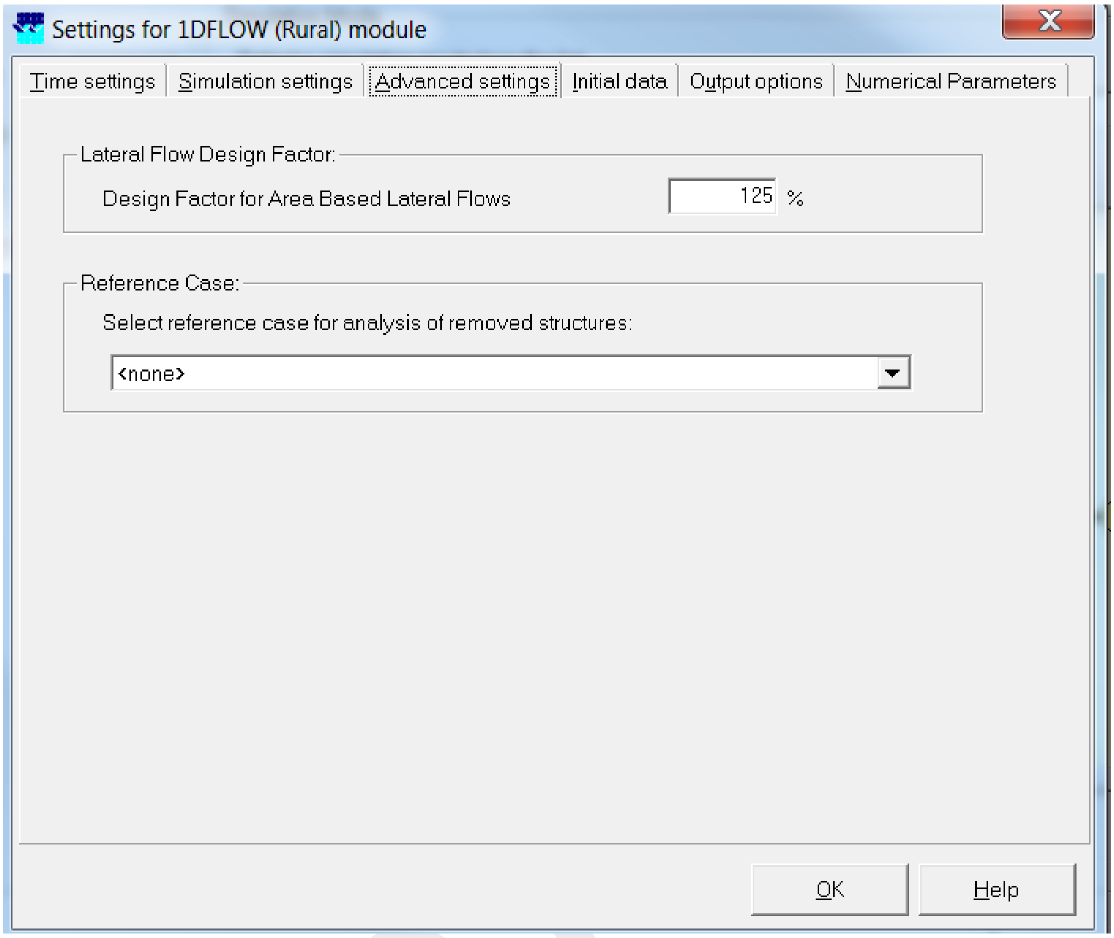


Figure 5.16: *The Advanced settings tab of the Channel flow module.*

Design Factor for Area Based Lateral Flow

The "Design Factor for Area Based Lateral Flows" is a multiplication factor (expressed as percentage) that is applied to only those lateral flows, that are calculated using the so-called "Area Based" method (see Figure 5.100 and Equation (6.14)).

Tab - Initial data

One of the options when opening "Settings" for Channel Flow is "Initial data". These can be found on the "Initial data" tab. The options are explained below.

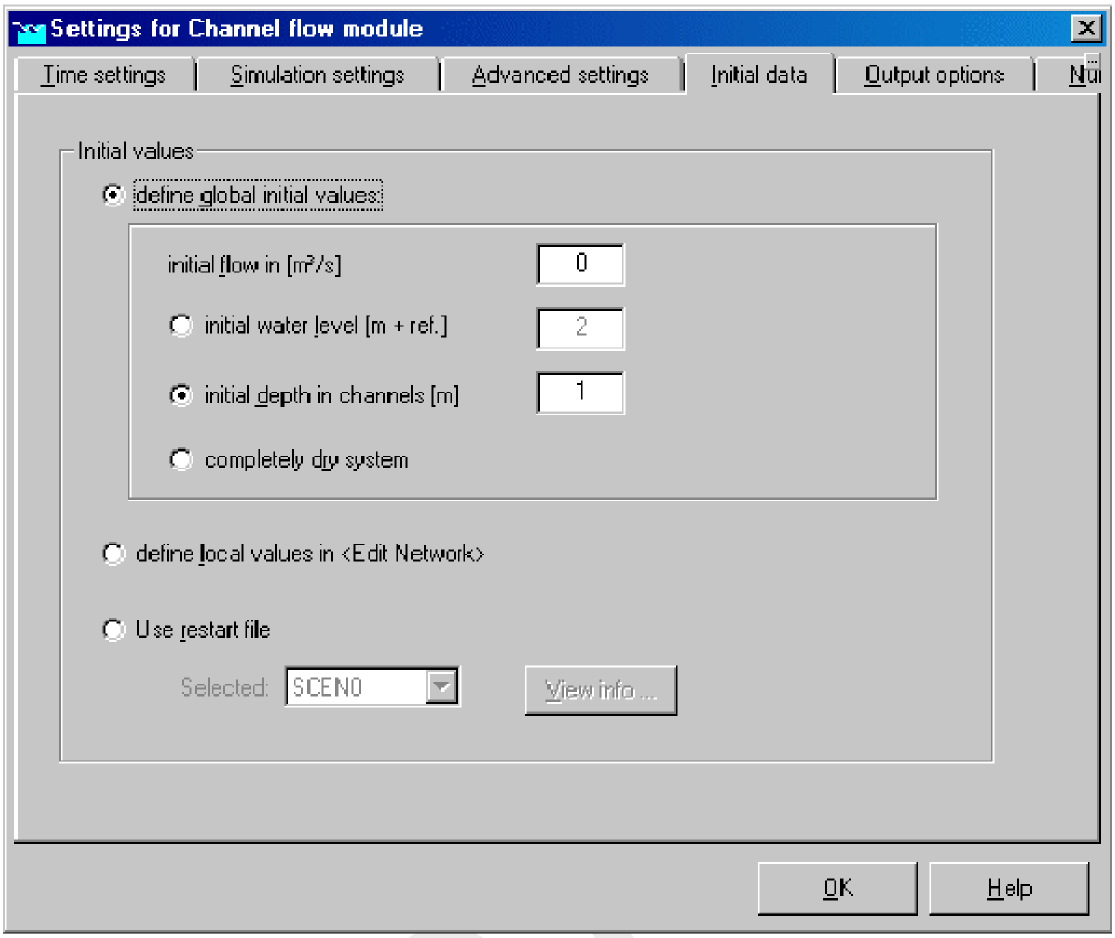


Figure 5.17: The Initial data tab of the Channel flow module.

◇ **Define global initial values** If this option is chosen, the initialisation values that you enter in this section will be applied to the entire model.

◇ **Define local values in <Edit Network>**. This option allows you to define initial water levels and/or depths individually for every branch in the model. For those branches where you do not enter the initial state, the values from the "Define global initial values" section apply.

◇ **Use restart file** Get the initial situation for the model from the results of a previous simulation. In order to do this, there should be a restart file available.You can create a restart file by defining one on the "Simulation settings" tab and running the simulation. After this simulation has run, the results of the last time step will be written to the restart file.

Tab - Output options

One of the options when opening "Settings" for Channel Flow is "Output options". These can be found on the "Output options" tab. The options are explained below.

◇ **Time step Output** For most simulations, it is not useful to have simulation results for every computational time step. That would require enormous disk space and would not give useful extra information. Therefore, an output time step can be defined.You can choose to have current, average or maximum values written to the result file. *For example: if the computational time step is 10 min, and the output time step is 1 hour, there will be 6 computational results available for 1 result to be written. Thus, you can choose to have the actual computed value written for every output time step (current), or you can choose to take the maximum or the average of those 6 results.*

◇ **Output parameters** Here you can define which results should be written to file.

The different Output parameters for Nodes:

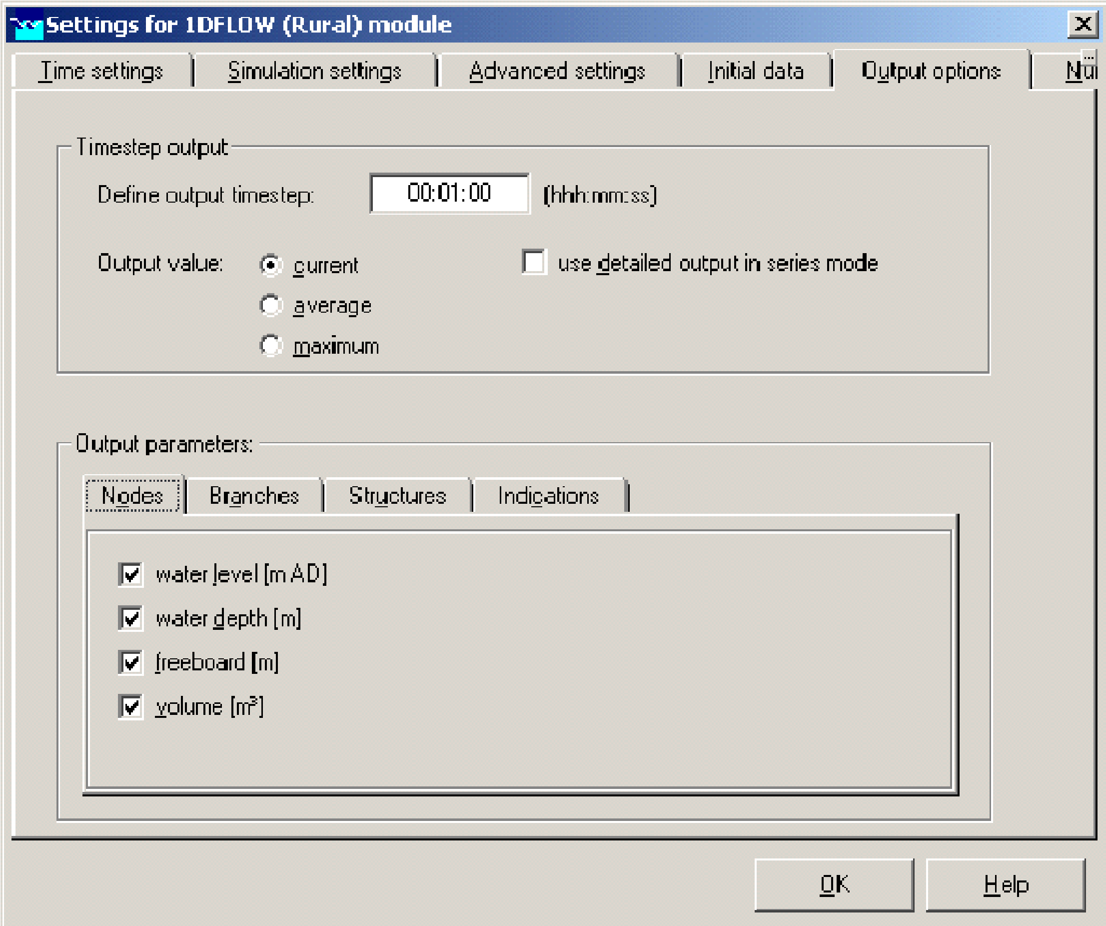


Figure 5.18: *The Nodes tab in the Channel flow Output options.*

It is possible to choose between:

- ◇ water level [m AD]
- ◇ water depth [m]
- ◇ freeboard [m]
- ◇ volume [m^3]
- ◇ The different Output parameters for Branches:

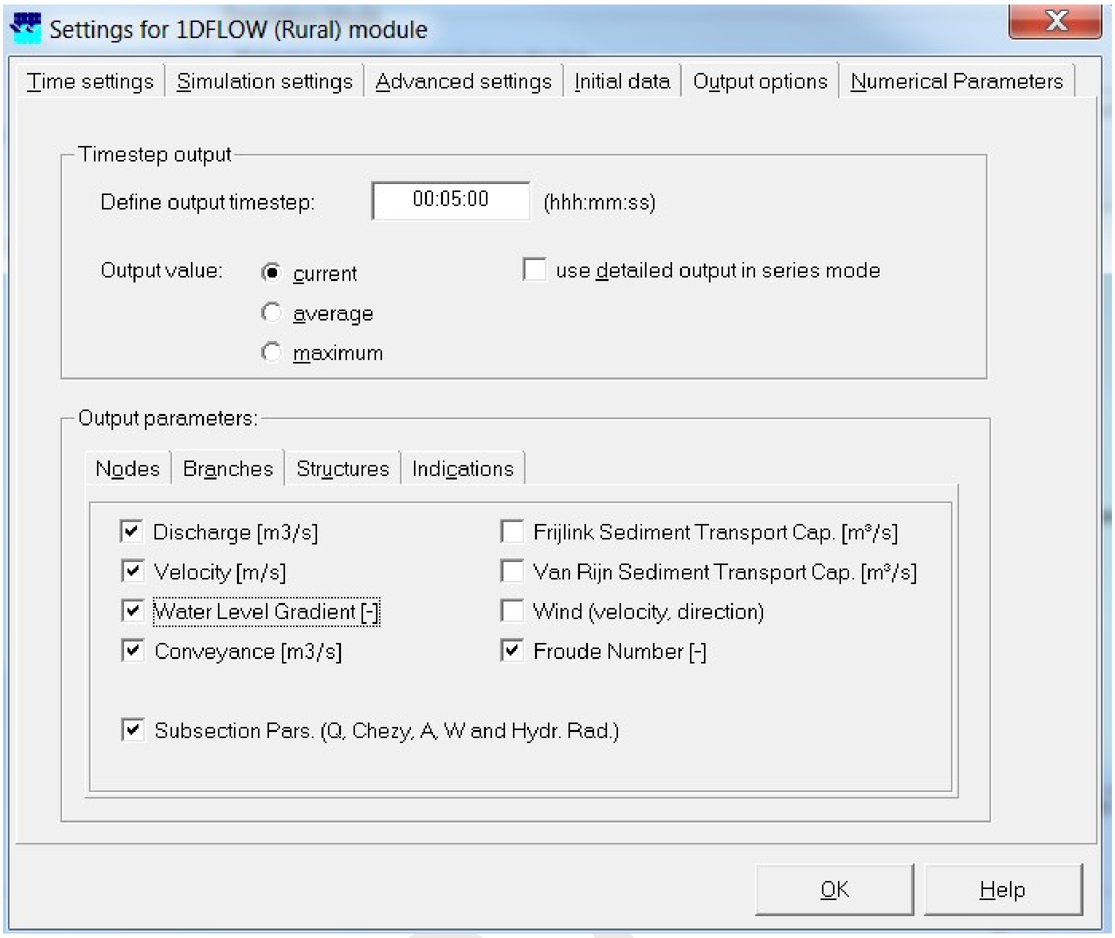


Figure 5.19: *The Branches tab in the Channel flow Output options.*

It is possible to choose between:

- Discharge $[m^3/s]$
- Velocity $[m/s]$
- Water Level Gradient
- Conveyance
- (Subsection) Pars. (Q, Chézy, A, W, and Hydr. Rad.)
- Frijlink Sediment Transport Cap. $[m^3/s]$
- Van Rijn Sediment Transport Cap. $[m^3/s]$
- Wind (velocity, direction)
- Froude Number [-]

The different Output parameters for Structures:

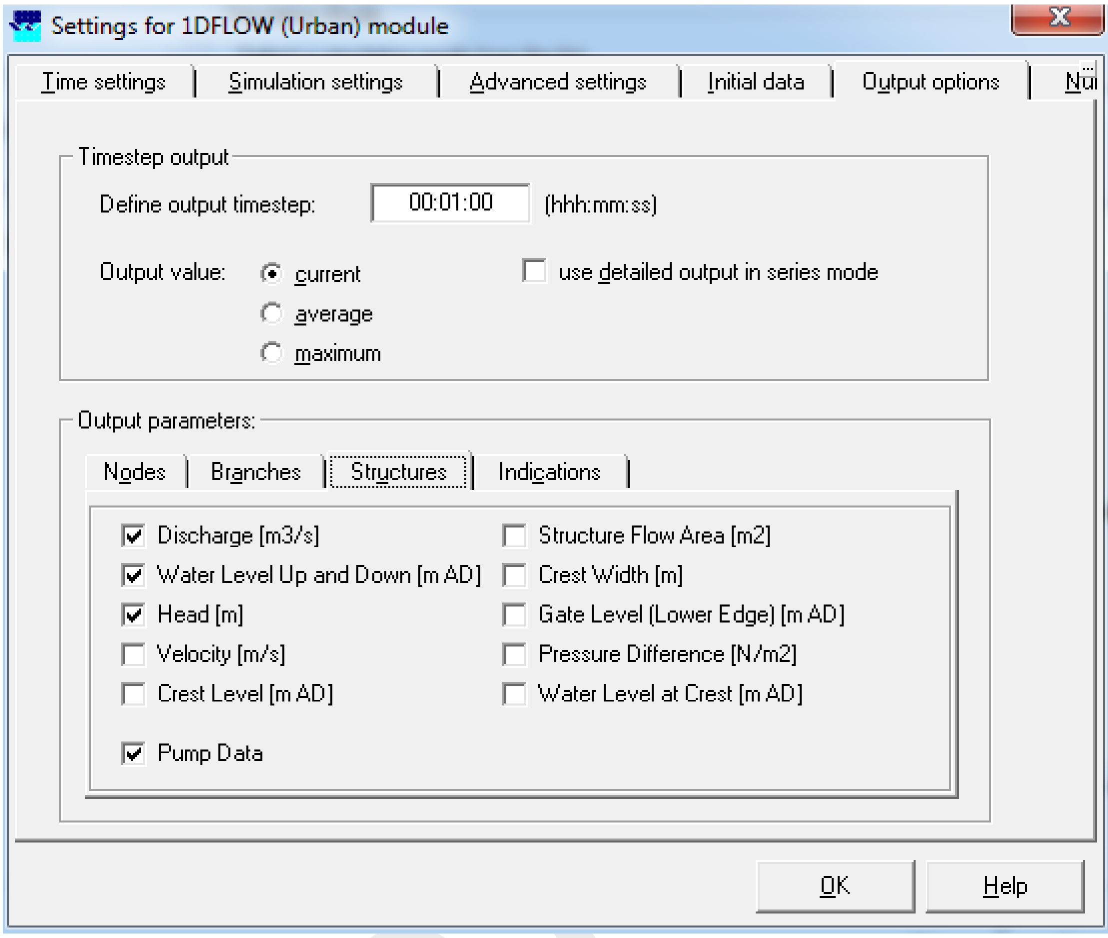


Figure 5.20: *The Structures tab in the Channel flow/Urban flow Output options.*

It is possible to choose between:

- Discharge [m^3/s]
- waterlevel up and down [m AD]
- head [m]
- Velocity [m/s]
- crest level [m AD]
- pump data
- structure flow area [m^2]
- crest width [m]
- gate level (lower edge) [m AD]
- pressure difference [N/m^2]
- water level at crest [m AD]
  - This item is only available for General Structures.

Tab - Numerical Parameters

One of the tabs in settings of Channel Flow is the "Numerical Parameters" tab. The numerical parameters are explained below. **Note:** Modifying these advanced numerical parameters is not supported. Only modify these settings when necessary for specific purposes, and carefully verify results after making changes.

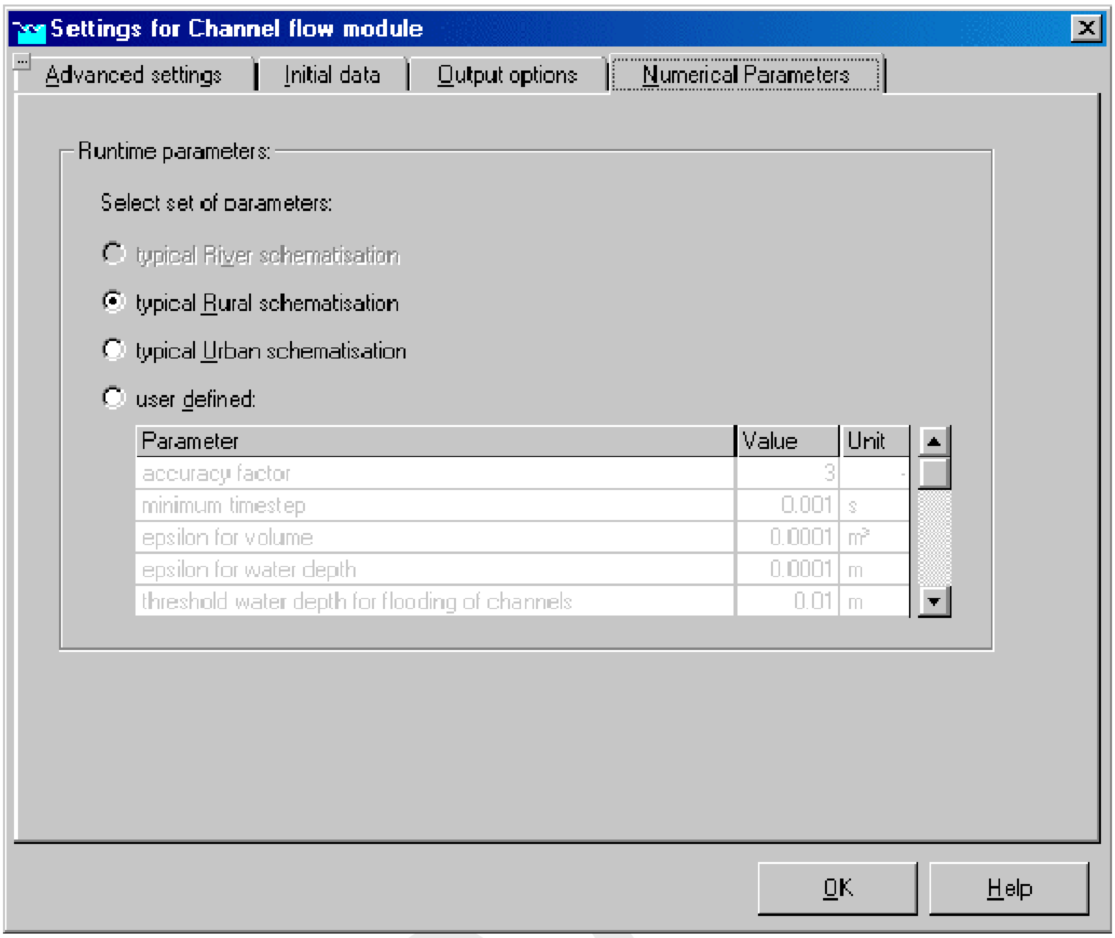


Figure 5.21: The Numerical Parameters tab of the Channel flow module.

Default settings

◇ **Typical Rural schematisation** A default selection of numerical parameters that are most suitable for hydrodynamics in open water systems, mainly channels and small rivers.
◇ **Typical Urban schematisation** A default selection of numerical parameters that are most suitable for hydrodynamics in urban water systems, mainly sewer pipes.

User defined

An option to choose your own numerical settings:

◇ Accuracy factor
◇ minimum time step
◇ epsilon for volume
◇ epsilon for water depth
◇ threshold water depth for flooding of channels
◇ threshold water depth for flooding of land
◇ factor for structure dynamics (Channel Flow & Sewer Flow only)
To the structure equations an inertia term is added. This inertia term acts as a kind of numerical demping. This is done to avoid numerical oscillations/instabilities in case of unsteady flow conditions. The numerical parameter "factor for structure dynamics" is a factor/scalar applied to this inertia term. Default value of 1.0 is suggested for the "factor for structure dynamics" Note: for steady flow conditions the inertia term equals zero.

Hence, the "factor for structure dynamics" is not of influence for the computed steady-state discharge through a structure.

◇ structure inertia dampening factor (River Flow only)
In settings a default value for the so called structure inertia damping factor can be defined, that is applied for the River weir, the Advanced weir, the General structure and the Database structure both as single structure or member of a compound structure. In the schematization Task block, this default value can be overruled for each individual afore mentioned structure type. In the linearization of the concerning structure equation a term $\alpha \partial U / \partial t$ is added, where

α	structure inertia damping factor $[-]$;
U	flow velocity $[m/s]$; and
t	computational time $[s]$.

The structure inertia damping factor can be used for avoiding instabilities during computation.

◇ relaxation factor
◇ theta
◇ minimum length of branch segment
◇ volume based on water level or discharges (0=false; 1=true)

In SOBEK there are two options for computing the volume of a ζ-calculation point (or branch segment) at a specific point-in-time, viz:

1 Volumes based on water level or discharges (0=false; 1=true)

If the option "Volumes based on water levels" is selected, this means that the volume at each ζ-calculation point (or each segment) follows from the computed water level and its corresponding cross-sectional profile.

If the option "Volumes based on discharges" is selected, this means that the volume at each calculation point (or branch segment) is the summation of its volume in the previous time-step and the resulting net inflow during the computational time-step.

In Water Quality computations especially use is made of volumes and discharges. By choosing the option "Volumes computed based on discharges" a more coherent set of volumes and discharges is obtained, than in case the option "Volumes computed based on water levels" is selected.

Note: The selected way in which volume is computed also yields for 2D hydrodynamic computations.

◇ maximum flow (1D) and velocity (2D) Courant number
◇ maximum degree of nodal points to be eliminated by Gaussian elimination
◇ maximum number of iterations
◇ rho (density of fresh water)
◇ gravity acceleration
◇ minimum surface in node
◇ minimum surface on street
◇ extra resistance for general structure
In settings a default value can be defined for the so called extra resistance coefficient of the General structure type, both as a single structure and as a member of a compound structure. In the schematization Task block, this default value can be overruled for each individual General structure type. The so called extra resistance refers to a bed shear stress force, that is accounted for in the impuls balance, that is solved in case of drowned

gate flow or drowned weir flow. The bed shear stress force reads

$$LgW_2\rho_2u_2^2/C^2 \quad \text{or} \quad \lambda\rho_2W_2u_2^2 \tag{5.1}$$

where:

$\lambda = (Lg/C^2)$ extra resistance coefficient;
L length of hydraulic jump behind the structure [m];
g acceleration due to gravity [m^2/s];
C Chézy coefficient [$m^{1/2}/s$];
ρ2 density of water in hydraulic jump [kg/m^3];
U downstream flow velocity [m/s]; and
W_2 downstream structure width [m].

◇ factor on 1D acceleration term dU/dt (can vary between 0 and 1)
0 means that DU/Dt or $\partial U/\partial t + U\partial U/\partial x$ terms in all 1D hydrodynamic momentum equations are neglected, hence waves are computed/considered as diffusive waves.
1 means no reduction on DU/Dt or $\partial U/\partial t + U\partial U/\partial x$ terms in all 1D hydrodynamic momentum equations are neglected, hence waves are computed/considered as dynamic waves.
In case user defines a factor on 1D acceleration term $\partial U/\partial t < 0$, SOBEK applies factor= 0. In case user defines a factor > 1, SOBEK applies factor= 1.
◇ use time step reduction on structure (0= false; 1= true)
In case the user defines a value for "use time step reduction on structure" equal to 1, at the point-in-time of the wetting of the crest of a structure (i.e. for weirs and orifices only) a time step reduction will be applied during a time-span equal to two times the user defined time step. This functionality was implemented for avoiding oscillation in specific Urban schematisations which having sharp inflow hydrographs, it can be applied in Rural schematisations as well. Unneccessary use of this option might result in a longer computational time needed.

5.1.2.2 River/Urban/Rural

General
The settings task block gives the user the following two options (see Figure 5.22):

◇ The combination of SOBEK modules used for the simulation can be selected here, and
◇ For every module selected general settings can be defined.

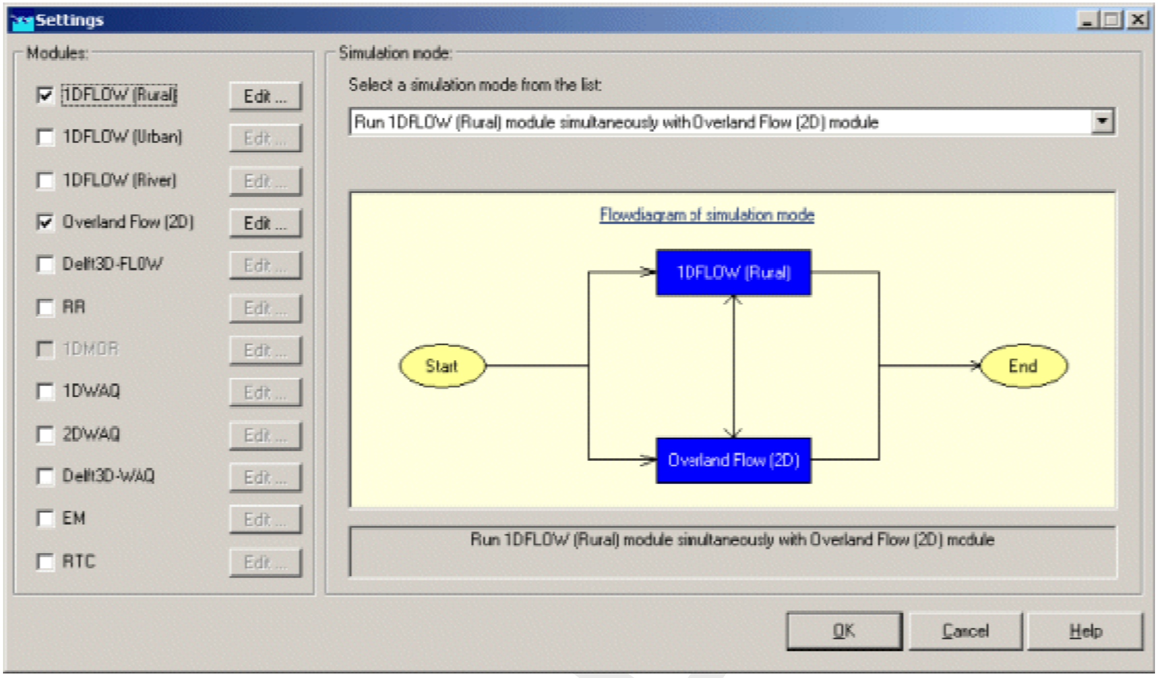


Figure 5.22: *The Settings task block.*

Simulation settings tab

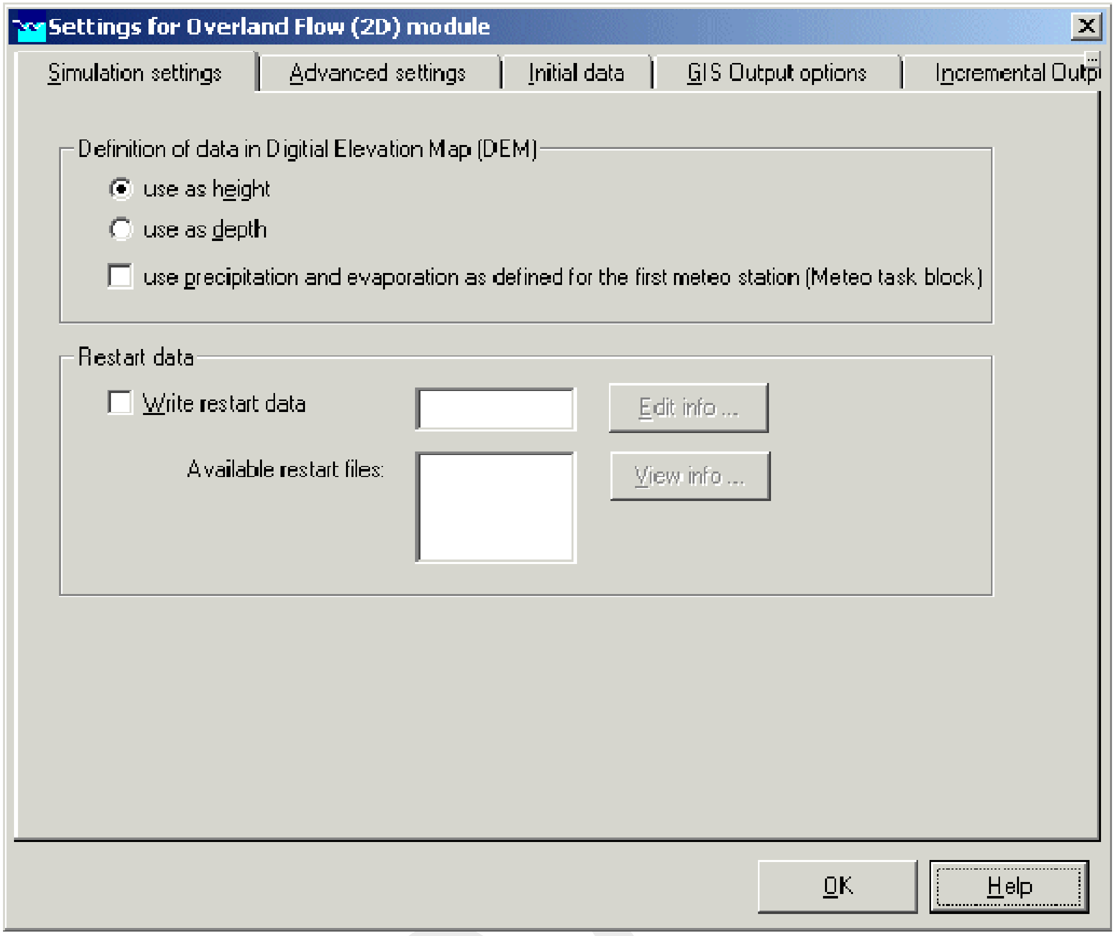


Figure 5.23: The Simulation Settings window for the Overland Flow settings task block.

Definition of data in Digital Elevation Map (DEM)
Here, the user can specify how the data within the DEM to be used is oriented: as heights or as depths.

- ◇ heights: a higher z-value means a higher level
- ◇ depths: a higher z-value means a lower level

With the checkbox "Use precipitation and evaporation as defined for the first meteo station" you can define whether the rain fall and evaporation is to be included on your 2D grid. The precipitation and evaporation from the first rainfall station defined in the "Meteorological data" task block is then applied. If the checkbox is switched off, no rainfall or evaporation will considered on the 2D grid.

Note:

1) Rainfall and evaporation is applied to all active 2D grid cells (i.e. 2D grid cells not having a missing value). At present only spatially uniform distributed rainfall and evaporation can be defined, meaning that each active 2D grid obtains the same amount of rainfall and evaporation. The spatially uniform distributed rainfall and evaporation can be a function of time.

2) Rainfall and evaporation can not be directly entered on 1D open channel network. Using the Rainfall-Runoff module, the outflow of catchment areas as result of nett precipitation can be provided as inflow on a 1D open channel network. The same applies for a sewer

network/system.

Restart data
Restart data allows users to create realistic initial conditions for their calculation. When the "write restart file" option is checked and the model is run, the results for the last time step of the calculation are written to a file with the name you entered. For other calculations the available restart file may then be referred to define the initial conditions.

Advanced Settings tab

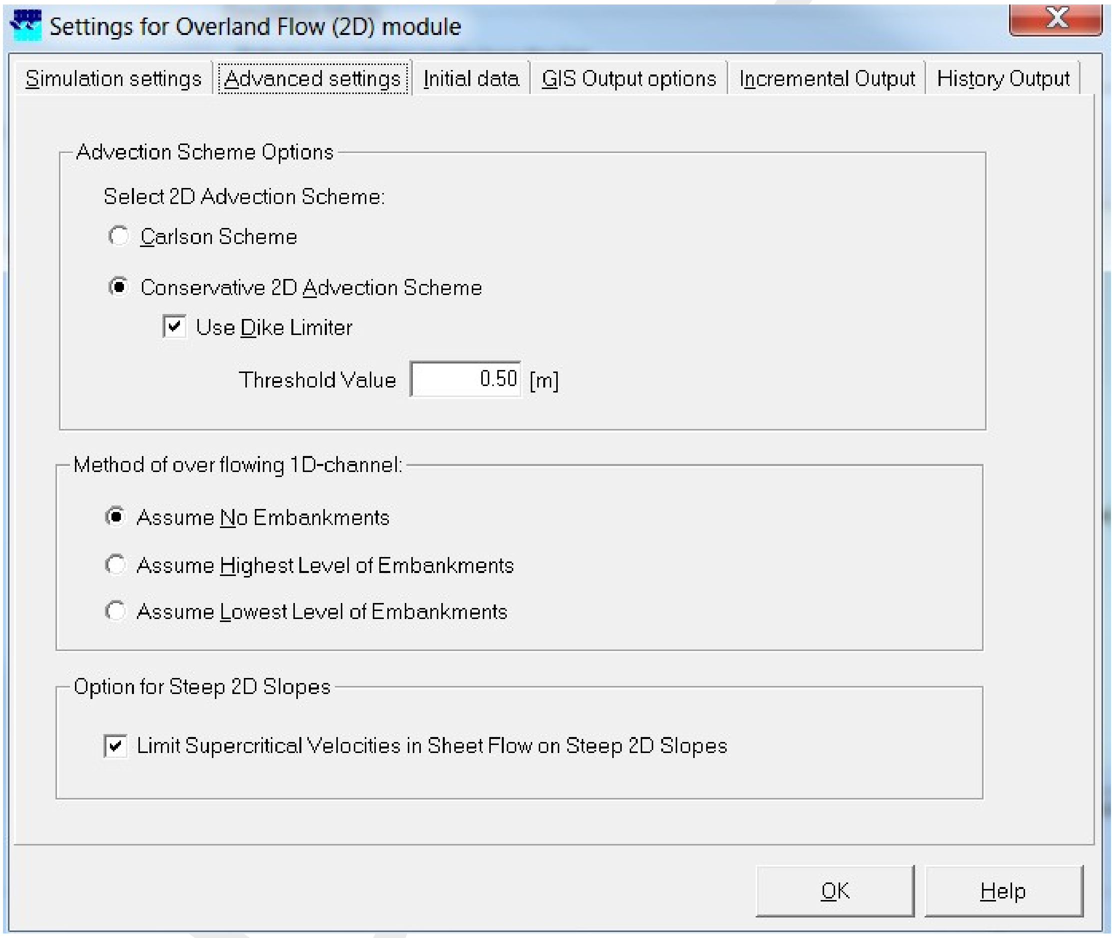


Figure 5.24: *The Advanced Settings window for the Overland Flow settings task block.*

The Conservative 2D Advection scheme allows for a more accurate computation of the propagation speed of flood waves, over initially dry bed or wet bed.

Special care has been taken to ensure that weir flow is computed accurately, even when weirs are represented by only 1 grid cell that is raised above the field level. Weir discharges are reproduced accurately both for the sub critical and the supercritical flow regime. These features are demonstrated and compared to analytical solutions by test computations in several geometries.

The Conservative 2D Advection scheme is based upon ‘A staggered conservative scheme for every Froude number in rapidly varied shallow water flows’ (Stelling and Duinmeijer, 2003).

In order to invoke the Conservative 2D Advection scheme, one can select in ‘Settings, Overland Flow (2D), Advanced settings’, the Conservative 2D Advection scheme. In order to invoke the special treatment for weir flow, select ‘Use Dike Limiter’. Grid cells will be flagged as ‘dikes’ when the height difference with surrounding grid cells exceeds a user specifiable ‘Dike limiter’ treshold value.

Note that the less accurate Carlson scheme is still available for use. The use of the Conservative 2D Advection scheme is recommended.

Limiting super critical flow velocities on a steep 2D flow: If there are steep slopes within the topography of a 2D grid, it is possible that sheet-flow with super critical flow velocities occurs on these steep 2D slopes. In other words that a thin layer of water runs down these steep slopes with very high flow velocities. These very high flow velocities may result in severe reductions of the user-defined time-step and hence increase in the required computational effort. These super critical flow velocities can be limited by checking the option "Limiting super critical velocities in sheet flow on steep 2D slopes" available on the "Advanced settings Tab" of the "Overland Flow (2D) module" in "Settings" (see Figure 5.24).

Method of overflowing 1D channel
Here the user can specify a number of general options for the 2D simulation, starting with the method of overflowing 1D channel. This is a very important option which will greatly influence the results of the simulation, so it is very important that you set this option wisely.

It concerns the connection of the 1D cross-section with the underlying 2D grid cell, and is explained in the following example:

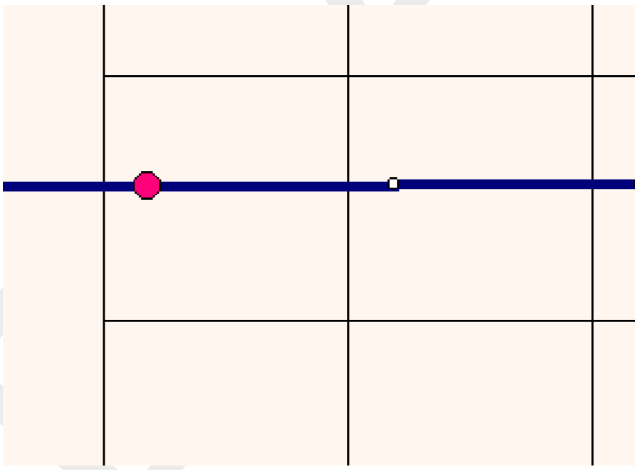


Figure 5.25: *Example of a 1D2D schematisation.*

A 1D channel delimited by dikes passes through a low lying area, say a polder. When the water level inside the 1D channel overtops the dikes, water will start to flow into the underlying 2D grid elements. The dikes of the 1D channel can be modelled in two different ways, as is shown in the following two cross-sections of the schematisation depicted in Figure 5.25.

Options 1: Assume No Embankments.

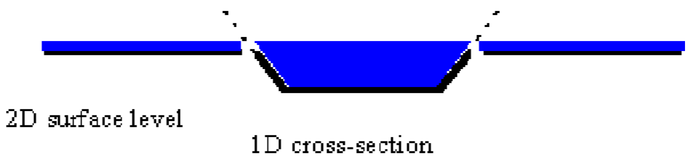


Figure 5.26: *Assume no dikes.*

Notice: At this moment, the option of method of overflowing 1D-channel refers to all 1D cross sections.

In this case, water enters the 2D grid as soon as the channel water level branches the terrain level in the 2D grid, so the part of the dike in the 1D cross-section above this level is neglected.

Option 2: Assume Highest/Lowest Level of Embankments.

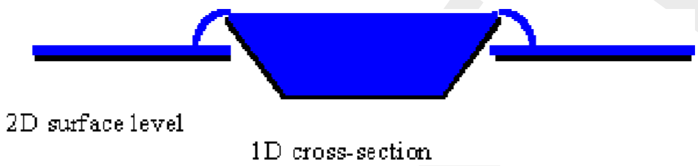


Figure 5.27: *Assume Highest/Lowest Level of Embankments.*

In this case, water enters the 2D grid when the highest level of the 1D profile is overtopped, and not earlier. So, the dikes form a barrier between the channel and the 2D area. Moreover, they form a barrier for water flowing towards this 1D cross section over the 2D grid.

The terms *highest* and *highest* refers to both sides of the cross section. If *Assume Highest Level of Embankments* is chosen, and the left bank's surface level is higher than the right bank's, then the surface level from the left bank is used as the level for overtopping.

If *Assume Lowest Level of Embankments* is chosen, the surface level from the right bank is used.

Note: if overtopping takes place, water will always overtop on **both** sides of the channel, no matter if one side has a higher surface level. If you want to prevent this from happening, create an additional elevation in the grid on one side of the channel.

Initial Data Tab

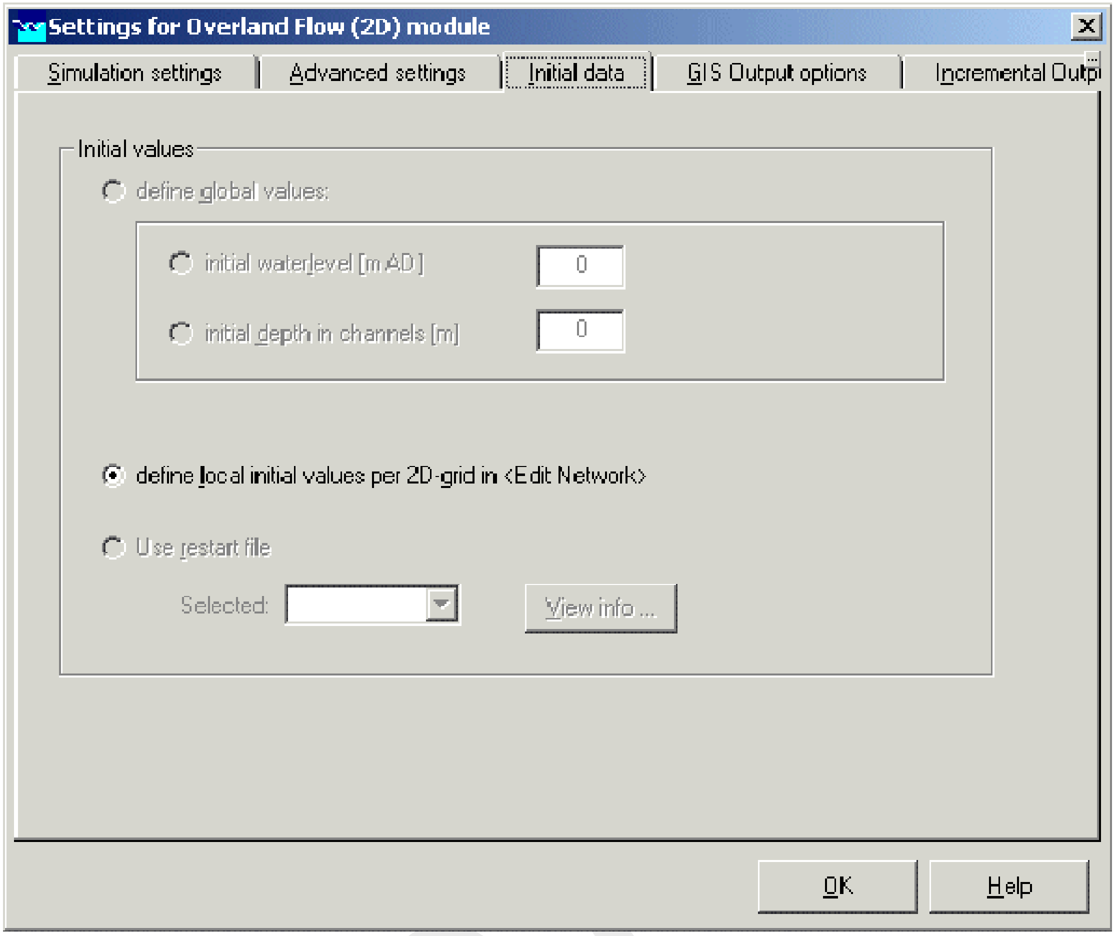


Figure 5.28: *The initial data tab from the Overland Flow SETTINGS task block.*

Note: The first option, *define global values,* cannot be selected at the present.

At this moment, there are either one or two options available here. If you haven't run a simulation before with a restart file as output, you won't be able to select the second option use restart file, so you will need to use the default option, define local initial values per 2D-grid.

There are a number of things you need to remember when using the restart option:

- make sure you select the use restart file option in **both the overland flow module AND the channel flow module** !!
- The restart file can only be used as input for exactly the same schematisation that was used in the simulation that generated the restart file.
- The restart file itself is saved under the project directory; this means that you will only be able to use this restart file as input for a (any) case within the same project, but not for a case in another project.

GIS output options tab

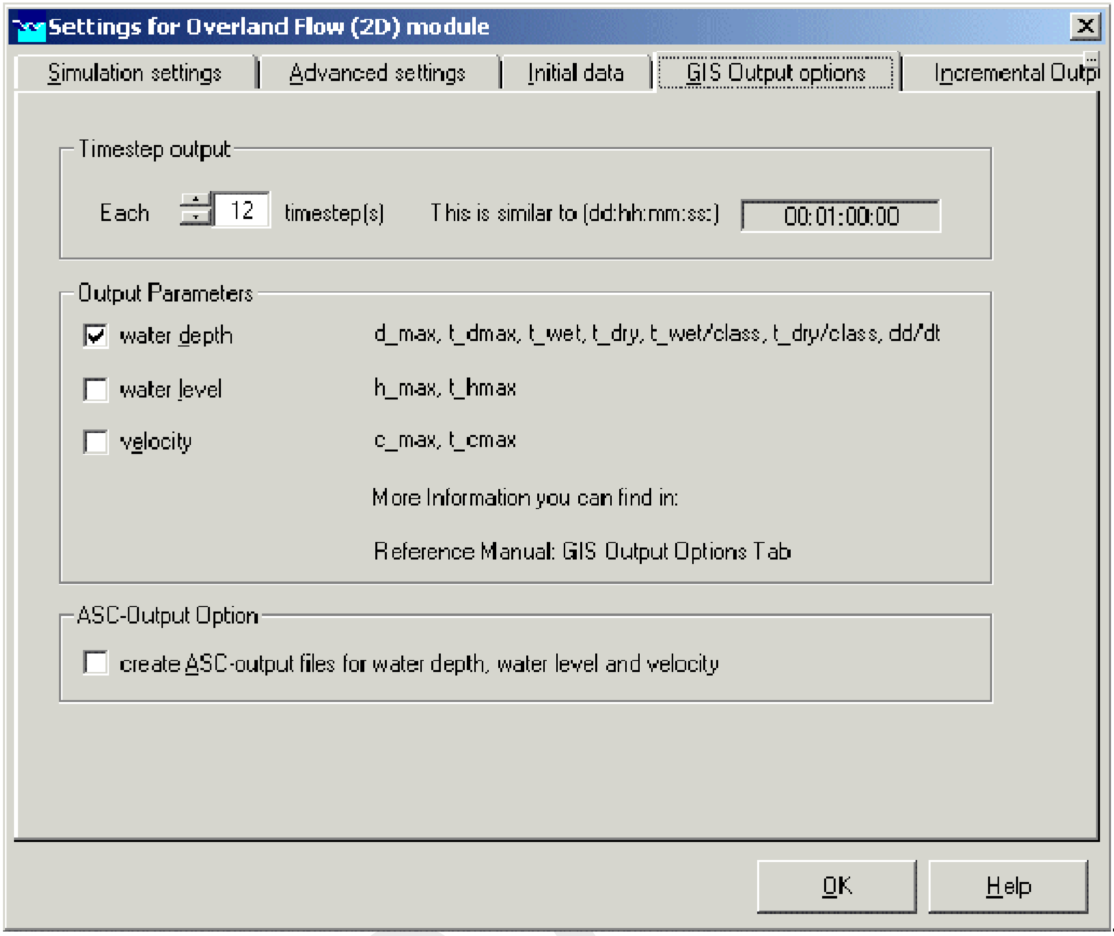


Figure 5.29: *GIS Output tab in the Overland Flow settings task block.*

The output for GIS systems, like ARCView, is one of the three types of 2D-specific output available. In this screen, you can select if you want this output to be generated or not, and for which time steps. If any (combination) of the available variables is selected here, SOBEK will automatically create '.ASC' output files which can be imported into a GIS based program. There are some additional rules to remember, which will be explained further on.

The GIS output data is written to a number of different files. The first type consists of output files in .asc format (see also section 5.1.6.3). SOBEK creates these output files only when the switch 'create ASCII-output for water depth, water level and velocity' is switched on! One of these is generated for every selected output time step and for every selected variable. In general, it is a good idea to wait with generating this output (if you need it at all, of course) until you are pretty sure that the simulation is going to give you the results that you need. Suppose your simulation lasts 24 hours, and you want GIS output for every time step (of, for example, one minute) and for all five variables, you would get 5x24x60 equals 7 200 output files!

Warning: Right now, SOBEK can handle only a certain maximum number of files in the case directory. If you decide to switch on the 'ASCII-Output Option', you might actually branch this maximum number of files (because of all the .asc files generated), which means that you might not be able to save your case properly! In that case you will get the message with saving your case: **"too many files in case directory; Check the configuration of the application; erase unregistered files".** In this case you should reconsider you output choices to decrease the number of output files.

The second type of output files is the binary '.map' file. For every variable chosen, a file is produced. It contains the values of that variable for every time step and every grid element of one particular grid. In *results in maps*, the user can select these .map files and produce .asc files out of them. So actually, the .map file contains the same information as the .asc files described earlier, but in a compressed format. Please note that whether MAP files are written or not, only depends on the *Output Parameters* chosen in the *GIS Output Options* tab, and not on the *ASCII-Output option* switch!

The last type of GIS output files are 'special' .asc files. These files contain the following variables:

- ◇ maximum water depth (d_max)
- ◇ time of maximum water depth (t_dmax)
- ◇ maximum water level (h_max)
- ◇ time of maximum water level (t_hmax)
- ◇ maximum velocity (v_max)
- ◇ time of maximum velocity (t_vmax)
- ◇ time of wetting (t_wet)
- ◇ time of wetting per class (steady state') (t_wet/class)
- ◇ time of drying (t_dry)
- ◇ time of drying per class (steady state') (t_dry/class)
- ◇ rate of change of water depth (dd/dt)

These files are only produced when the corresponding 'basic' variable is selected in the GIS tab, see figure. For example, the maximum depth is generated only when the option 'water depth' is checked. The same goes for other variables like the maximum velocities (linked to 'velocity') or maximum levels (linked to 'water level'). The files giving a time (period) and the rate of change of water depth are all linked to the 'water depth' output option.

Incremental Output Tab

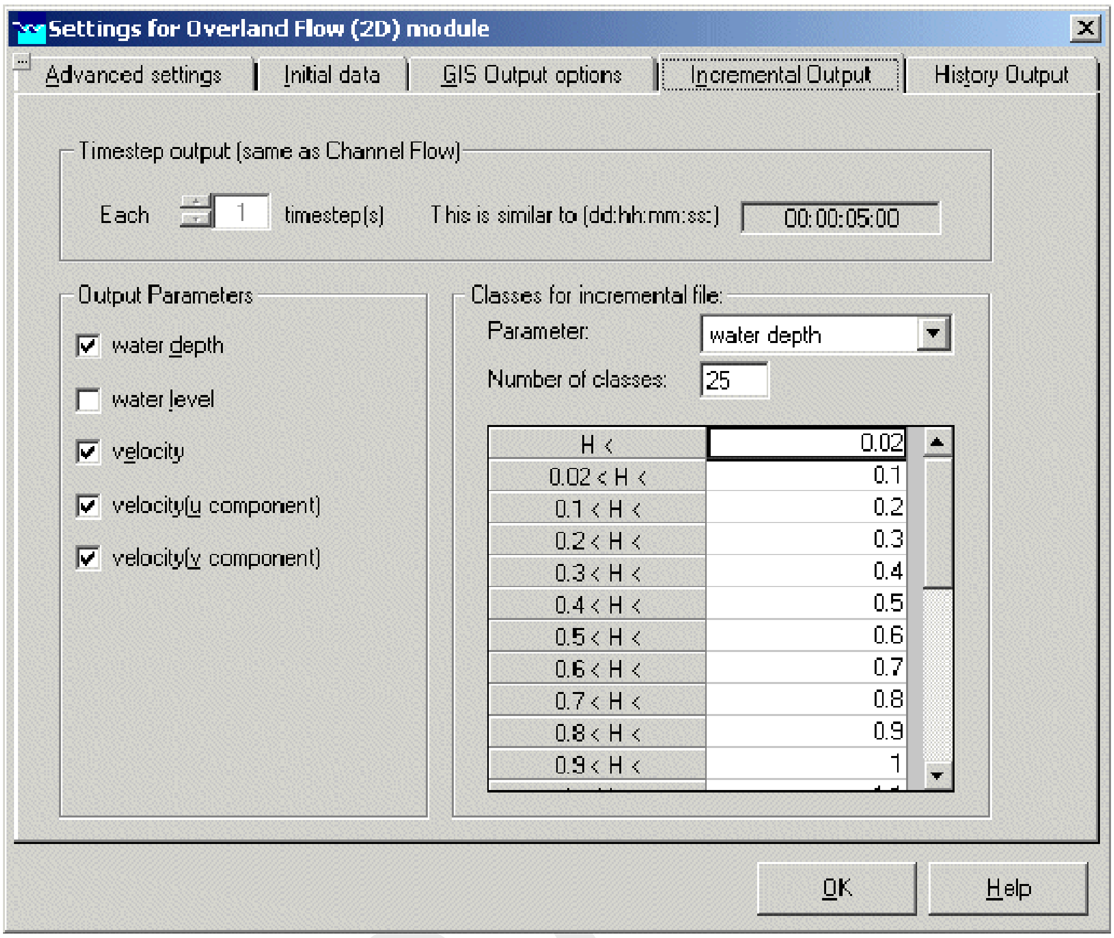


Figure 5.30: *The incremental output tab for the Overland Flow settings task block.*

The second type of 2D output is the so-called 'incremental output'. This is output that is generated specifically for result analysis in SOBEK ('results in maps') itself, so you should normally have at least some of these options turned on. Otherwise, you will not be able to visualize the results in SOBEK.

The incremental file system was developed to reduce the enormous amount of data generated by the 2D simulations. It works by defining a number of classes for every output variable which are then used in the output files instead of the actual data value itself. So, for example, if a water depth in a certain 2d grid cell equals 0.43 m, it would fall in class '0.4 - 0.5' (if you have specified such a class). This is also the result you would get when examining the results for that particular grid cell in 'results in maps'. So, this output is especially useful for getting a quick idea as to how the water flows through the 2d grid system; if you are interested in the exact values for the variables for a certain number of grid cells, there is another option available to the user, which is the *history output,* explained in the next paragraph.

The incremental output always has the same time step as the actual calculation time step chosen in the *channel flow* module.

Concerning the output parameters, it is important to know that the *velocity (u component)* and the *velocity (v component)* are necessary if you are interested in seeing the velocity vectors later on, when viewing the results in *results in maps.*

The number of classes can be selected manually, and influences the detail you will get when viewing the results; ten classes of 10 cm difference show much more information than 2 classes of 50 cm difference, but also require more disk space and computation time. The number of classes will be the same for all variables, so it's not possible to have 5 classes for the water depth and 15 for the velocity. But, if you have defined 15 classes (for all variables), and you need only 5 for one particular variable, you can define the unnecessary classes as no-data values (-999).

A few rules to remember:

◇ The water depth can be only positive, so it's no use to define negative classes here
◇ The 'u' and 'v' velocity components can be both positive and negative. It is important to remember to use exactly the same classification for both variables.
◇ This classification needs to be classified symmetrical around zero for correct determination of the vectors!

History Output Tab
If you are interested in the exact values of all variables on specific locations in the 2D grid, it is possible to use a special kind of node called a History station. For more information, please see the 2D-History node chapter.

2D Bathymetry input as heights (or levels)
Functionality:

2D bathymetry input as Heights (or levels) refers to the fact that the user can define bed elevations of a 2D grid according to a vertical axis, having its positive axis pointing in upwards direction.

Up to now 2D bathymetry input could only be defined as Depths, meaning that bed elevations are defined according to a vertical axis, having its positive direction pointing in downwards direction. All 2D bathymetry input is either defined as Heights or as Depths. Hence it is not possible to define a particular 2D bathymetry input grid as Height while another is defined as Depths.

How to define Bathymetry input as Heights (or levels):

◇ Double click the 'Settings' task block.
◇ In the 'Settings' window, take care that the Overland Flow (2D) check-box is checked and click on the *Edit* button next to it
◇ In the 'Settings for Overland Flow (2D) module' window under Definition of data in Digital Elevation Map (DEM), define whether you want bathymetry input defined as Heights (or levels) or bathymetry input defined as Depths.
◇ Close the 'Settings for Overland Flow (2D) module' window by clicking the *OK* button,
◇ Close the 'Settings' task block by clicking in the 'Settings' window on the *OK* button,
◇ Now start to import your 2D bathymetry input grids under the 'Schematisation' task block.

5.1.2.3 Water Quality

Editing time settings

The simulation period for the Water Quality module can be derived from a selected meteorological event (if the Rainfall-Runoff module is active) or specified by hand. In the latter case, it is necessary that the simulation period for the Water Quality module is the same or lies within the simulation period for the Water Flow module.

The simulation time step may be equal to, smaller than and larger than the time step for the other modules. However, we advise you to select a time step either equal to the time step of the Water Flow module, or equal to the time step of the Water Flow module multiplied by a whole factor (2, 3, 4, etc.) or equal to the time step of the Water Flow module divided by a whole factor (2, 3, 4, etc.).

Furthermore, the selected numerical method may impose a stability limit on the time step (see below). Also, the modelling of water quality processes may impose a stability limit on the time step (see par. 4.2).

◇ Make sure that the "Channel Flow" and "Water Quality" modules are activated in the "Settings" Window.
◇ Click the *Edit* button of the "Water Quality" module.
◇ Adjust the simulation period and/or the time step in the corresponding edit boxes.

Editing simulation settings

The simulation options include the choice between fraction computations and regular water quality computations, as well as the selection of initial conditions. Optionally a user-defined file with boundary conditions can be connected.

◇ Click the "Simulation options" tab form.
◇ Select "Calculate fraction" or "Calculate water quality".
◇ Make sure that the tick box "processes active" is on, if you have selected "Calculate water quality" (unless you want to run a water quality simulation without substance-specific processes).
◇ If you intend to run the current case in batch mode, decide whether or not you want the previous flow results used in batch mode. If you do, check the associated tick box. (See also: the 'Simulation' task).
◇ Select the appropriate option for the initial conditions:
◇ select a file with initial conditions you generated yourself (.RSF file);
◇ select a file with initial conditions generated by earlier SOBEK runs;
◇ start with 100% initial water (fraction computation) or model-wide initial conditions.
◇ To edit the model-wide initial conditions click the *Edit* button.

If you specify a Water Quality computation and you click the *Edit* button belonging to the "start simulation with global initial values" option, the "Initial Values" window appears.

Note: The user-supplied files with initial conditions must have the extension <*.RSF>. They should be located in the <SOBEK\FIXED\> directory (mentioned in the "Simulation Options" tab form).

Note: The option to use files with initial conditions computed by earlier SOBEK runs is only active if such files exist.

Warnings:

◇ If you use hand-made restart files, be sure that the state variables or fractions are in

agreement with the current selection. The Water Quality module does not check this!
- ◇ If you use restart files from an earlier run, be sure that you have not changed the state variables or fractions (or your Water Quality schematisation) since you generated the file. The Water Quality module does not check this!

5.1.3 Task block: Metereological Data

Time-dependent and spatial constant wind-fields
Functionality:

The user can define time-dependent and spatial constant wind-fields acting on 2D flow. No distinction can be made between different 2D grids. Wind velocity and wind direction can vary as function of time. The wind is taken into account in the momentum equations as an additional shear-stress acting on the water surface (see manual). Please note that once the user defines wind, this wind will act both on the 1D Channel flow and on the 2D water movement.

How to define:

For how to define time-dependent and spatial constant wind-fields acting on 2D grids, reference is made to the explanation given in the user manual concerning 1D Channel flow.

Time-dependent and spatial constant rainfall or evaporation
Functionality:

The user can define time-dependent and spatial constant rainfall or evaporation acting on 2D flow only (hence not on 1D flow). No distinction can be made between different 2D grids or different rainfall stations (i.e. only one station can be considered). The rainfall or evaporation is respectively taken care as an external lateral inflow or outflow in the continuity equations.

How to define:

For how to define time-dependent and spatial constant rainfall or evaporation acting on 2D grids, reference is made to the explanation given in the user manual under 'Meteorological Data' task block.

Meteorological Data
After opening or creating a SOBEK project and case, you'll see the Case Manager screen appear, which consists of a number of task blocks. Each block represents a different part of the modelling process. In this chapter, the **Meteorological Data Task Block** is discussed:

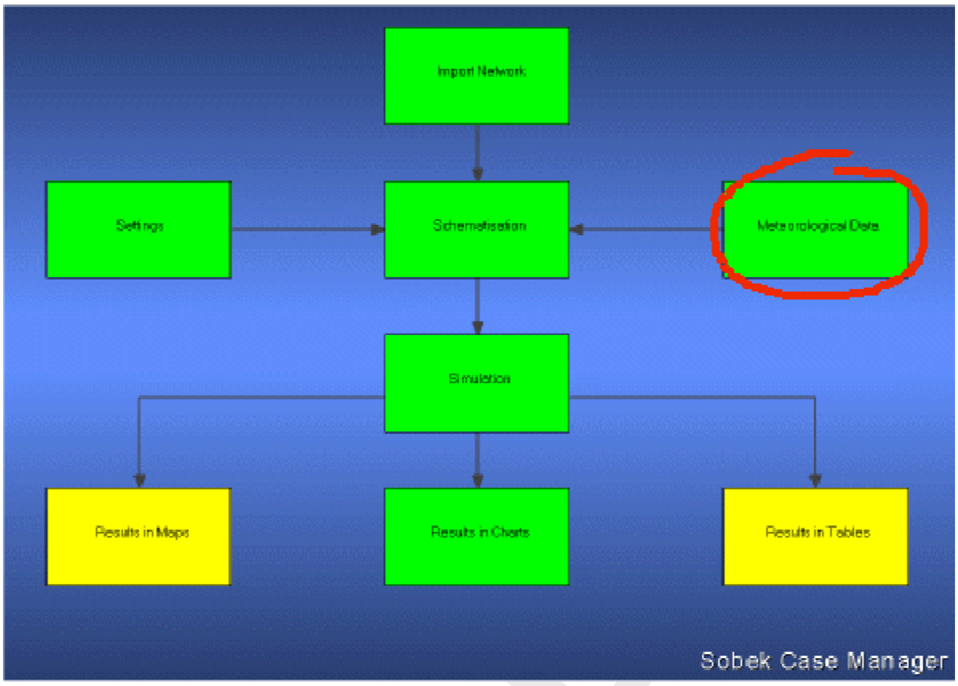


Figure 5.31: *The Meteorological Data task block.*

Under the Meteorological Data task block, you can define the meteorological circumstances that apply to your study.

These are:

- ◇ Precipitation
- ◇ Evaporation
- ◇ Wind data

SOBEK contains a default precipitation event (<DEFAULT.BUI>).

************** IMPORTANT ***

The meteorological data is stored in a separate directory <\SOBEK215\Fixed>, which is accessible for all SOBEK projects. If you make adjustments in a meteorological file that is also used within another project, these adjustments will **also** affect the results of that project (after running its computation again)!

****

Remark:

- ◇ File names of meteo files are limited to a maximum of 8 characters. Files created in the user interface will automatically be limited to 8 characters. When renaming or creating meteo files outside of the user interface, make sure the file name length does not exceed 8 characters. This 8 character limitation holds true for most SOBEK files, including .lit directories.

After double-clicking the task block, the following window will appear:

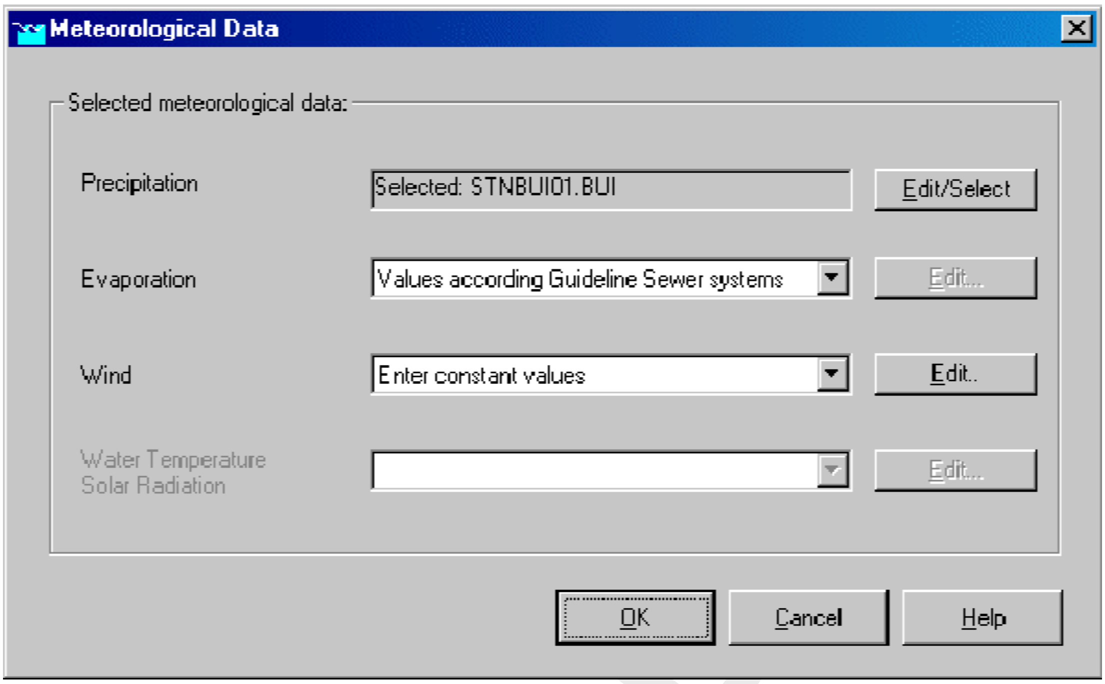


Figure 5.32: *The Meteorological Data window.*

Each of the Meteorological components will be discussed in the chapters hereafter.

Evaporation
During dry periods evaporation plays an important role in the water balance of natural water systems. Especially for Rainfall-Runoff computations.

SOBEK contains two evaporation series: a long term average series and the time series of daily values measured between 1951 and 1994 at the De Bilt , the weather station at the Royal Dutch Meteorological Institute.

Precipitation
A precipitation event is characterised by either consecutive periods with intensive rainfall, or a longer period with continuous precipitation. A precipitation series contains precipitation events separated by periods with very little or no rainfall.

Whether a precipitation event can be used as a design event depends on the problem at issue. For the design of the storage- and pumping capacity, it is mainly the behaviour of the sewer system during extreme precipitation that is of interest. For the evaluation of target levels of pumps and weir settings more moderate events are of interest.

SOBEK-RR-SF meets the Dutch guidelines 'sewer systems computations, hydraulic functioning'. This implies that both storm events and long time series can be simulated.

The guidelines prescribe ten standard events which must be used to analyse the behaviour of sewer systems. The ten standard precipitation events are derived from 15 minutes time series measured at De Bilt, a meteostation of the Royal Dutch Meteorological Institute (KNMI), in the period 1955-1979. Six different return periods and two different shapes are used to determine the events. The shape is characterised by the location of the peak. The criteria used for the selection of the standard events are described in the guidelines. The standard events with characteristic return period and location of the peak are described in Table 5.1.

Table 5.1: *Standard precipitation events*

CODE	RETURN PERIOD (year)	LOCATION OF PEAK
01	0.25	front
02	0.25	back
03	0.50	front
04	0.50	back
05	1.00	front
06	1.00	back
07	2.00	front
08	2.00	back
09	5.00	front
10	10.00	front

The events in Table 5.1 are available within SOBEK-RR-SF.

Figure 5.33 shows the rainfall of standard event 05 and the inflow into the sewer for a flat closed paved area of 1 ha. as determined by the RR-module:

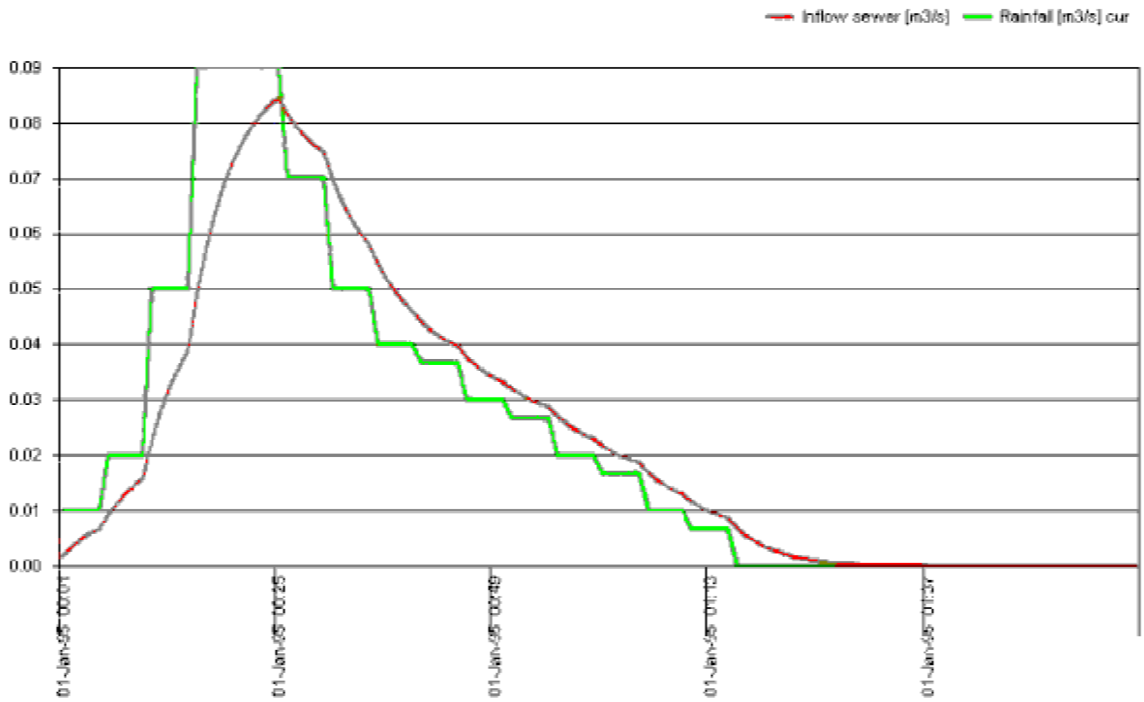


Figure 5.33: *Rainfall and corresponding sewer inflow for a flat closed paved area*

Figure 5.34 shows the rainfall of standard event 05 and the inflow into the sewer for three closed paved areas (with a slope, flat, flat stretched) of 1 ha each:

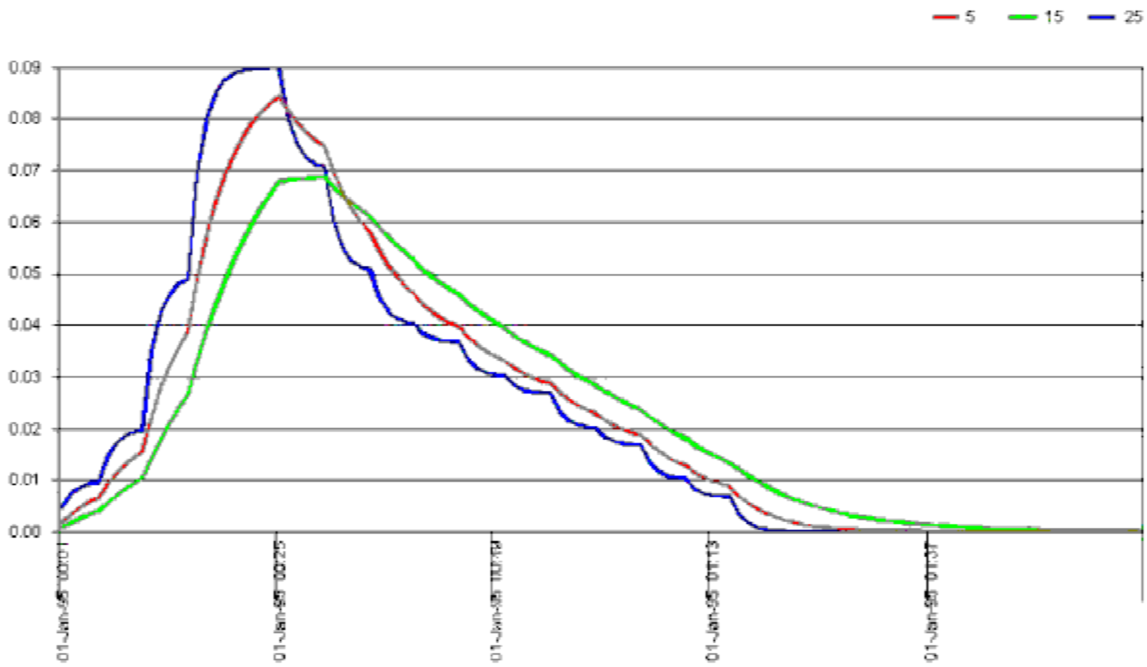


Figure 5.34: *Rainfall and corresponding sewer inflow for three types of closed paved area*

Precipitation options

Under the Meteorological Data task block, you can define rainfall intensities that apply to your schematisation. In this chapter, the precipitation editor is discussed.

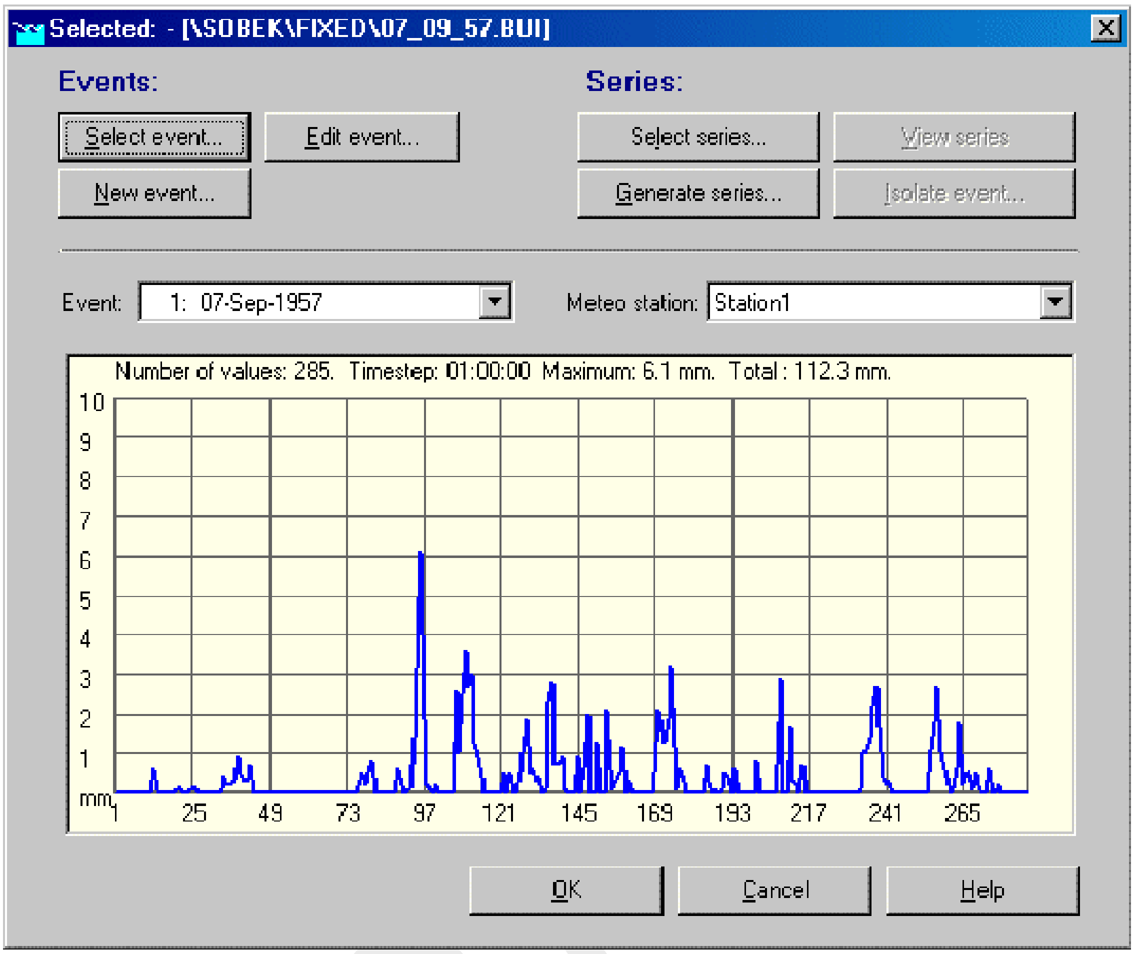


Figure 5.35: *The precipitation window*

Rainfall Events v.s. Rainfall Series

The basis of the precipitation editor is formed by rainfall **events**. These are characterised by either consecutive periods with intensive rainfall, or a longer period with continuous precipitation.

A chain of rainfall **events** may form a rainfall **series**. Rainfall series allow you to model a long time span of precipitation without having to model the dry periods in between.

Rainfall events:

◇ **The <Select event> option:** This option allows you to select a rainfall event from a list of available events.
◇ **The <Edit event> option:** Allows you to edit the currently selected rainfall event. Please be aware that, if this rainfall event is also used within other projects, editing it may also affect the results of those projects after you compute them again.
◇ **The <New event> option:** Allows you to create a new rainfall event. Firstly, you are asked to enter the number of rainfall stations of which you have data for the event. Then, you'll have to enter the names of those rainfall stations. Later, when you create your model, you can refer to any of these stations.

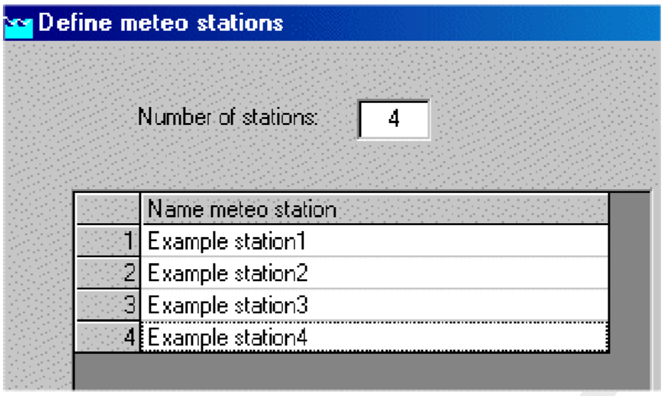


Figure 5.36: *Define meteo stations.*

Next, you are asked to define the name, the starting date, the end date and the timestep of the event:

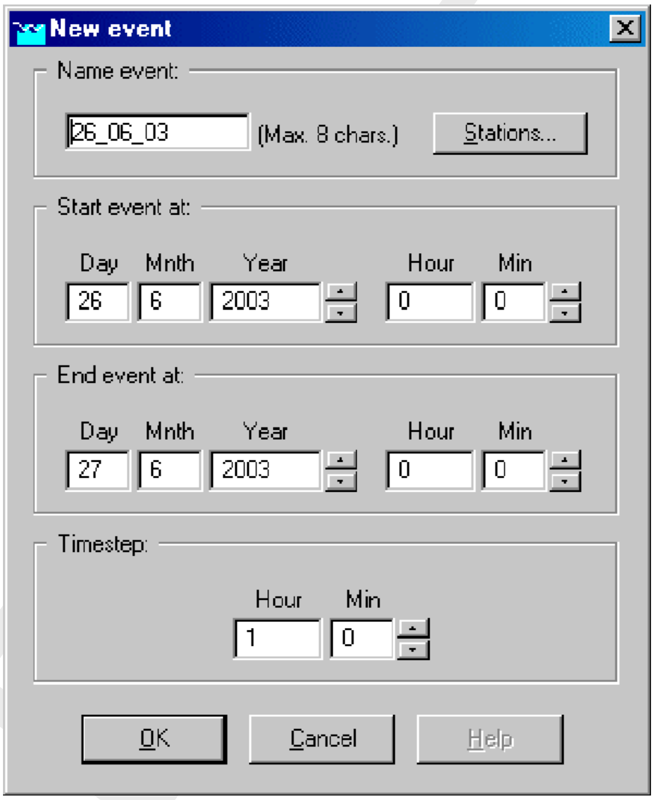


Figure 5.37: *Defining the name, start/end date and timestep of a new event.*

Finally, a window pops up, that contains a spreadsheet-like environment. The user can copy the available data from a spreadsheet, such as Microsoft Excel, into these fields:

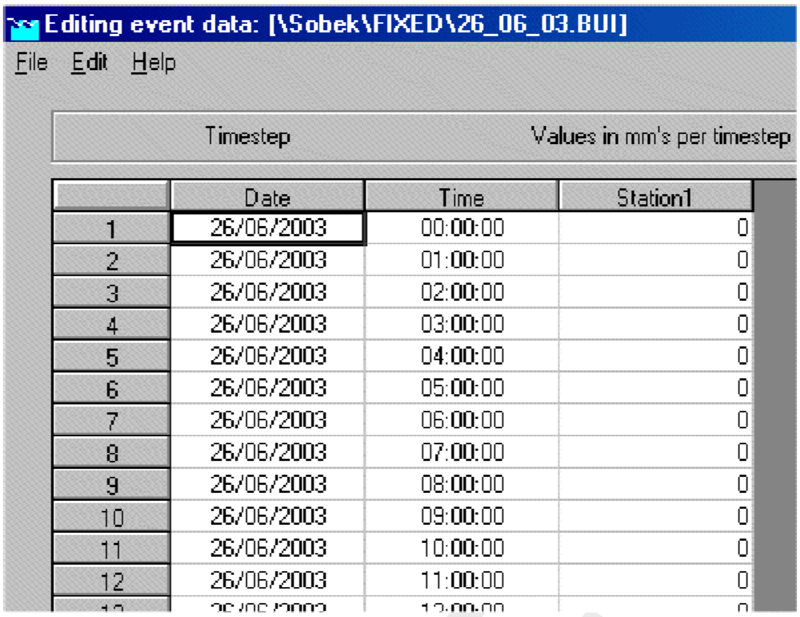


Figure 5.38: *Editing event data.*

Please note: If you want to copy data from a Spreadsheet program into these fields, make sure that the **field format** within the spreadsheet equals that of the fields in SOBEK, thus: dates: dd/mm/yyyy times: hh:mm:ss Then, save the file that you have just created by clicking "File" - "Save" from the menu. In the end, don't forget to actually select the file that you've just created.

Rainfall Series (For the time being only usable for Dutch situations)

As was mentioned above, a rainfall series is a sequence of single rainfall events, which can be run sequentially within one model run. Notice that for every rainfall event within a series, the basic initial conditions apply! It is **not** so that the results of the final time step of event1 provides the initial conditions for event2.

- **The** <**Select Series**> **option:** This option allows you to select an existing rainfall series. Notice that you can view each individual event from that series by choosing it in the <Event> box on the left side of the screen.
- **The** <**Generate Series**> **option:** This option allows you to generate a rainfall series, based on hourly data between 1951 and 1994 from the Dutch De Bilt rainfall station. Based on criteria that you define, the individual rainfall events are extracted from that data. It is not (yet) possible to enter your own long-term rainfall data and generate series from them.

Wind

In channel systems with long stretches of narrow channels and/or large open water surfaces strong winds can cause a local increase in water level. In areas where the storage capacity is small this can cause problems under wet conditions. Water Flow takes the influence of wind into account.

Specifying multiple wind fields

In SOBEK up to version 2.05, the user interface only supported a global wind field.

The computational core of SOBEK 2.07 was extended to support the functionality of specifying multiple wind station data independent of the coupling of SOBEK-branches to wind stations. The user-interface does not support this functionality.

Model data base

First the changes to the model database are described. The changes pertain to the existing wind data file, and the introduction of a new wind location data file.

Wind data file

The wind data file of the meteo data layer contains the wind definitions. This can be one of the following options:

- ◇ a variable or a constant global wind field;
- ◇ a wind definition per branch;
- ◇ using more than one wind.

An example of a global wind field:

GLMT MTEO id ‘1’ ci ‘-1’ wu 1 wv tv 1 TBLE .... tble wd td 1 TBLE .... tble mteo glmt

or

GLMT MTEO id ‘1’ ci ‘-1’ wu 1 wv tv 0 3 wd td 0 270 ... mteo glmt

An example of a wind field per branch:

MTEO id ‘2’ ci ‘1’ wu 1 wv tv 1 TBLE ... tble wd td 1 TBLE .... tble mteo

The exact lay-out of these records has not changed. A more detailed description is available in the on-line help (Model data base). One can also check with existing wind files (*.WND in the <\Fixed> subdirectory of the directory where SOBEK is installed)

Using wind stations the user can specify the wind field table per wind station, and specify the coupling of branches to wind stations separately.

The functionality of a wind field per wind station is made available using the WSTA keyword. The records are very similar to the existing MTEO or GLMT records. An example:

WSTA id ‘Windstation’ wu 1 wv tv 1 TBLE ... tble wd td 1 TBLE .... tble wsta

with

id	wind station id
wu	wind used (as in MTEO records)
wv tv	wind velocity (as in MTEO records)
wd td	wind direction (as in MTEO records)

The appendix contains an example file.

Wind location file
WLOC ci ‘1’ st ‘Windstation1’ wloc
WLOC ci ‘3’ st ‘Windstation1’ wloc with

ci carrier id (branch id)
st wind station id; this id should match with a wind station id in the WSTA records

of the wind data file

If a SOBEK-branch is not coupled explicitly to a wind station, the global wind data will be used by default.

Settings
In Settings, you can specify the desired output. Additional available output is the wind direction and wind speed per branch. Just select the additional item wind (velocity, direction) in the 'Output options' tab form with output for 'Branches'.

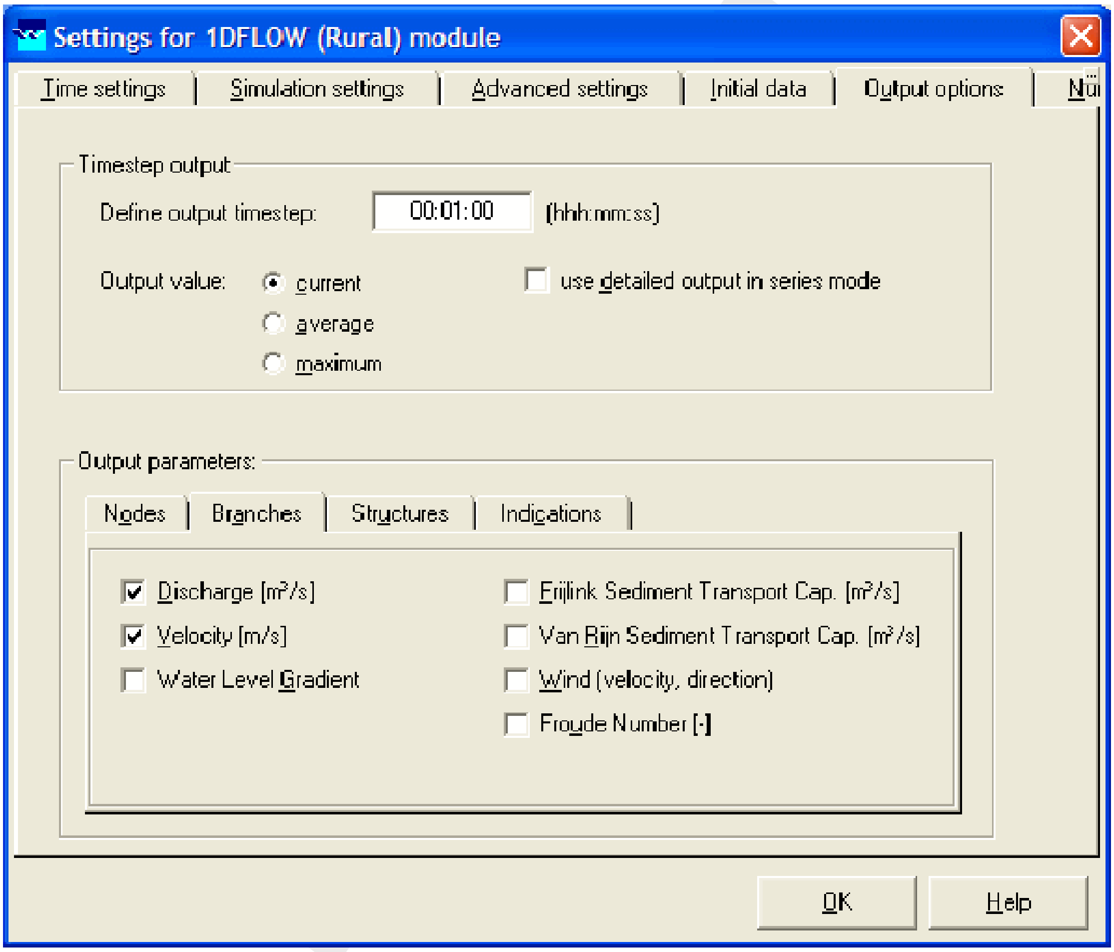


Figure 5.39: *Task block settings, output options.*

Meteo
Use Meteo only to select a wind file. A wind data file containing other data than only global data should **never** be edited in the user interface using Meteo. It should be done 'behind the screens' using a text editor.

Behind the screens

The wind data file and wind location file should be edited using an ASCII text editor.

The wind file is located in <\SOBEK215\Fixed>. The name of the file is taken from the name of the rainfall file. Only the extension is different: the wind file has extension .WDC (constant wind field) of .WND (variable wind field).

The wind location data file Wnd_Loc.Dat can be different for each case. For a fixed set of wind stations, you can investigate the impacts of allocating SOBEK-branches to wind stations without needing to change the wind station data.

The wind location file <Wnd_Loc.Dat> should only be edited in the Work directory.

The simplest and safest way to do this is: Open a case, than double-click on the schematisation task-block, and press *OK* in the next menu. Then you get the question: "Nothing has been changed. Do you want to continue to the <simulation> task?". Answer *Yes*.

After that the wind location data file <WND_LOC.DAT> is in the WORK directory, where it can be edited.

Post-processing
The extra available output per branch (wind velocity and wind direction) can be viewed in the 'Results in charts' and 'Results in maps' task block.

In the 'Result in charts' task block, first select the 'Flows at branch segments' output and then select the wind data you want to see from the next screen.

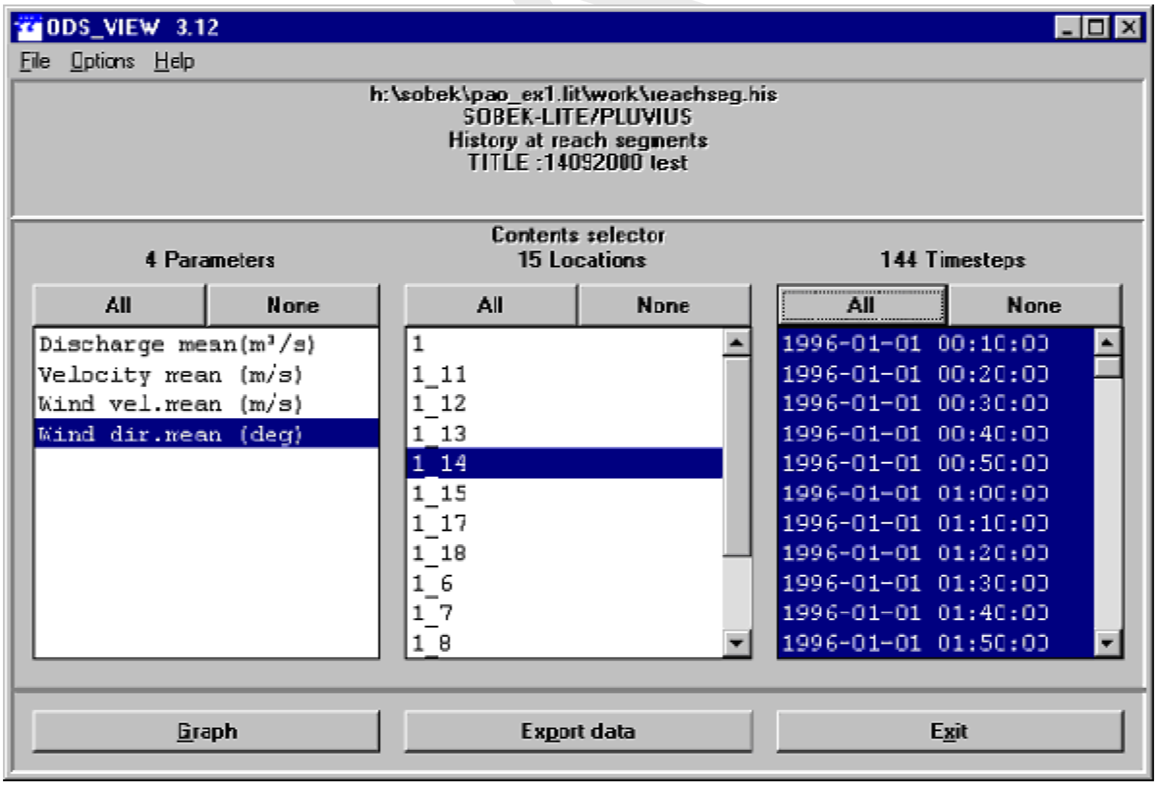


Figure 5.40: *Results in charts, Flows at branch segments.*

In the 'Result in maps' task block, you will get the network schematisation on the screen. After that, open the output data 'Flows at branch segments', select the appropriate output variable (default is the first parameter, i.e. discharge mean [m^3/s], and click on the desired output location.

Example wind file and wind-location file
Wind Location file (Wnd_Loc.Dat)

WLOC ci ’1’ st ’Test1’ wloc
WLOC ci ’3’ st ’Test1’ wloc
WLOC ci ’5’ st ’Test1’ wloc
WLOC ci ’7’ st ’Test1’ wloc
WLOC ci ’389’ st ’Test2’ wloc
WLOC ci ’388’ st ’Test2’ wloc
WLOC ci ’386’ st ’Test2’ wloc

Note:
The branches with id 1, 3, 5 and 7 will be using the wind data from wind station Test1. The branches with id 386, 388 and 389 will use the wind data from station Test2. All other branches in the schematisation will be using the global wind data.

Wind data file (e.g. Sept1994.Wnd)

The example wind file below contains the definition of a global wind field and 2 other wind fields.

GLMT MTEO nm ’(null)’ ss 0 id ’0’ ci ’-1’ lc 9.9999e+009 wu 1

wv tv 1 0 9.9999e+009 ’Wind Velocity’ PDIN 0 0 ” pdin

CLTT ’Time’ ’Velocity’ cltt CLID ’(null)’ ’(null)’ clid

TBLE
’1994/10/23;00:00:00’ 6.000000 <
’1994/10/23;02:00:00’ 6.000000 <
’1994/10/23;04:00:00’ 5.000000 <
’1994/10/23;06:00:00’ 6.500000 <
’1994/10/23;08:00:00’ 6.000000 <
’1994/10/23;10:00:00’ 7.500000 <
’1994/10/23;11:00:00’ 9.500000 <
’1994/10/23;12:00:00’ 9.500000 <
’1994/10/23;13:00:00’ 8.500000 <
’1994/10/23;16:00:00’ 6.000000 <
tble

wd td 1 0 9.9999e+009 ’Wind Direction’ PDIN 0 0 ” pdin

CLTT ’Time’ ’Direction’ cltt CLID ’(null)’ ’(null)’ clid

TBLE
’1994/10/23;00:00:00’ 170.000000 <
’1994/10/23;01:00:00’ 180.000000 <
’1994/10/23;03:00:00’ 190.000000 <
’1994/10/23;06:00:00’ 190.000000 <
’1994/10/23;07:00:00’ 180.000000 <
’1994/10/23;11:00:00’ 190.000000 <
’1994/10/23;13:00:00’ 200.000000 <

'1994/10/23;15:00:00' 190.000000 <
'1994/10/23;16:00:00' 180.000000 <
tble

su 0 sh ts 0 9.9999e+009 9.9999e+009 tu 0 tp tw 0

9.9999e+009 9.9999e+009 au 0 at ta 0 9.9999e+009 9.9999e+009 mteo

glmt

WSTA id 'Test1' nm 'Teststation1' wu 1 wv tv 1 0 9.9999e+009 'Wind Velocity' PDIN 0 0 ” pdin

CLTT 'Time' 'Velocity' cltt CLID '(null)' '(null)' clid

TBLE
'1994/10/23;00:00:00' 16.000000 <
'1994/10/23;02:00:00' 16.000000 <
'1994/10/23;03:00:00' 15.500000 <
'1994/10/23;04:00:00' 15.000000 <
'1994/10/23;06:00:00' 16.500000 <
'1994/10/23;11:00:00' 19.500000 <
'1994/10/23;13:00:00' 18.500000 <
'1994/10/23;14:00:00' 18.500000 <
'1994/10/23;15:00:00' 17.500000 <
'1994/10/23;16:00:00' 16.000000 <
tble

wd td 1 0 9.9999e+009 'Wind Direction' PDIN 0 0 ” pdin CLTT 'Time' 'Direction' cltt CLID '(null)' '(null)' clid

TBLE
'1994/10/23;00:00:00' 279.000000 <
'1994/10/23;01:00:00' 289.000000 <
'1994/10/23;03:00:00' 299.000000 <
'1994/10/23;06:00:00' 299.000000 <
'1994/10/23;07:00:00' 289.000000 <
'1994/10/23;12:00:00' 299.000000 <
'1994/10/23;16:00:00' 289.000000 <
tble

su 0 sh ts 0 9.9999e+009 9.9999e+009 tu 0 tp tw 0

9.9999e+009 9.9999e+009 au 0 at ta 0 9.9999e+009 9.9999e+009

wsta

WSTA id 'Test2' nm 'Teststation2' wu 1 wv tv 1 0 9.9999e+009 'Wind Velocity' PDIN 0 0 ” pdin

CLTT 'Time' 'Velocity' cltt CLID '(null)' '(null)' clid

TBLE
'1994/10/23;00:00:00' 26.000000 <
'1994/10/23;06:00:00' 26.500000 <
'1994/10/23;12:00:00' 29.500000 <

’1994/10/23;16:00:00’ 26.000000 <
tble

wd td 1 0 9.9999e+009 ’Wind Direction’ PDIN 0 0 ” pdin CLTT ’Time’ ’Direction’ cltt CLID ’(null)’ ’(null)’ clid

TBLE
’1994/10/23;00:00:00’ 179.000000 <
’1994/10/23;01:00:00’ 189.000000 <
’1994/10/23;03:00:00’ 199.000000 <
’1994/10/23;07:00:00’ 189.000000 <
’1994/10/23;11:00:00’ 199.000000 <
’1994/10/23;13:00:00’ 209.000000 <
’1994/10/23;15:00:00’ 199.000000 <
’1994/10/23;16:00:00’ 189.000000 <
tble

su 0 sh ts 0 9.9999e+009 9.9999e+009 tu 0 tp tw 0

9.9999e+009 9.9999e+009 au 0 at ta 0 9.9999e+009 9.9999e+009

wsta

5.1.4 Task block: Schematisation

General
Within this task block the actual schematisation is built and verified. As mentioned before, this manual will only deal with the 2D additions to the 1D *channel flow* version of SOBEK. Please refer to the general SOBEK user manuals for more details on the outline of schematisation task block and the use of the 1D channel flow module in specific.

Just like the 1D schematisation, the 2D schematisation is built in two steps using a number of building blocks. In the first step, the *edit network* mode, the grids and the other building blocks, like initial water level nodes, are being placed on the map. In the second step, the *edit model data* mode, the characteristics of all these building blocks are defined. Both steps will be explained in detail in the following two paragraphs.

Edit Network mode
To start creating a network, you have to switch to the ’Edit Network’ mode. To do so, choose "Network" from the "Edit" menu:

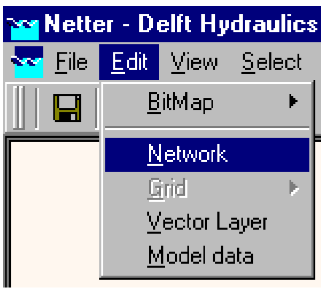


Figure 5.41: *Entering Edit Network mode.*

Or click the  node.

Edit Model Data mode
To start editing the parameters for your model objects, choose "Model Data" from the "Edit" menu.

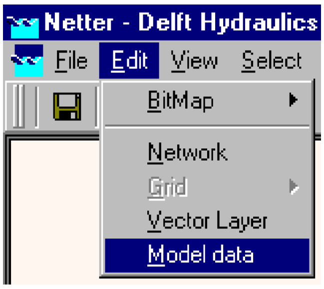


Figure 5.42: *Entering Model Data mode.*

The next step in building the schematisation is specifying the parameters for the different 2D elements defined in the edit network mode. The order in which the 2D elements are handled is the same as in the previous paragraph.

An object needs to be selected before its data can be defined. After the Model Data option has been selected from the edit menu, the model data window appears. This window can be used to browse through all objects and select them for editing. The objects can also be selected by clicking on them in the main Netter screen.

Viewing 2D Grid Info
Functionality:

Using the 2D Grid Info option, information can be obtained on:

◇ individual 2D grid cells, or
◇ on a path of 2D grid cells.

The user can add the information of various individual 2D grid cells to one and the same graph. A path of 2D grid cells can cross at any angle over a 2D grid. At present the path of 2D grid cells can not pass over nested grids (i.e. parent grid and child grid) in such way that only the active parent and child 2D grid cells are being selected.

Further on a distinction is to be made between static values and dynamic values. Static values refers to the values on the underlying 2D grid (for instance a bed elevation) or underlying map file (for instance maximum water depths). Dynamic values refers to time-dependent computation results (for instance incremental water depths).

How to make a 2D path:

Both in the ‘Schematisation’ and ‘Results in Maps’ Task blocks.

◇ In the main menu, click on Select/2D-Grid cell info
◇ Click on the 2D grid cell at which you like to start your 2D path (i.e. starting 2D grid cell).
◇ Note that in the ‘Path2D-Grid Info’ window, the column and row number of the starting 2D grid cell are given in the two boxes in front of the Path check-box.
◇ Click the Path check-box,
◇ While pressing on the Shift key, place the mouse-pointer on the 2D grid cell at which you like to end your 2D path (i.e. the end 2D grid cell). Releasing the Shift key will result in the

selection of the end 2D grid cell. The column- and row number of the end 2D grid cell are now filled in the two boxes behind the Path check-box.

◇ Click on the <show> button in the '2D-Grid info' window to visualise the 2D grid cells contained in your 2D Path.

Remark: In case you have more than 1(one) 2D grid (for instance in case of nested grids), the 2D grid cell of the 2D grid that lies on top will be selected. Using the *Next Grid* button the grid cell on the underlying 2D grid will be selected. Note that in this case the Grid ID will change.

How to retrieve 2D Grid Info:

◇ Under the 'Schematisation' Task block only information on static values of the 2D grid cells are available. *In case the 2D path option is not used*, the static value of the selected 2D grid cell is depicted on the one most last line in the '2D-Grid Info' window (Note that dynamic values are not available under the 'Schematisation Task' block). *In case a 2D path is selected*, the static values of the concerning 2D grid cells can be obtained by clicking the *Graph* button. Note that in the main menu of this graph the x-axis can be changed to either value (distances) or labels (column and row number of the 2D grid).

◇ Under the 'Results in Maps' Task block, information on both static- and dynamic values of the 2D grid cells are available. *In case the 2D path option is not used*, the static- and actual dynamic value of the selected 2D grid cell is depicted on the last two lines in the '2D-Grid Info' window. Please note that static values are displayed in grey. Clicking the most left <Graph> button, provides a graph of dynamic values of each selected 2D grid cell as function of time. Please note that by clicking on another 2D grid cell and sequentially clicking on this <Graph> button again, the user can add the dynamic results of various 2D grid cells in one and the same graph. *In case a 2D path was selected*, clicking on the most left *Graph* button results in a graph of the dynamic values of the starting 2D grid cell of the 2D path. Clicking the most right *Graph* button provides a graph, showing for all 2D grid cells lying on the 2D path, its static value as well as its dynamic values for a particular point-in-time.
Notes:
1) the x-axis of this graph can be plotted as function of value (distance) or labels (column or row number);
2) dynamic values are only available in case they are selected beforehand (Click in main menu on File/Open Data/ Depth Incremental (in the 'Select Item' window));
3) using the 'View Data' window dynamic values can be added to the graph; and
4) using the *forward* and *backward* buttons in the 'View Data' window, the point-in-time for which the dynamic values are shown under the most right *Graph* button can be changed.

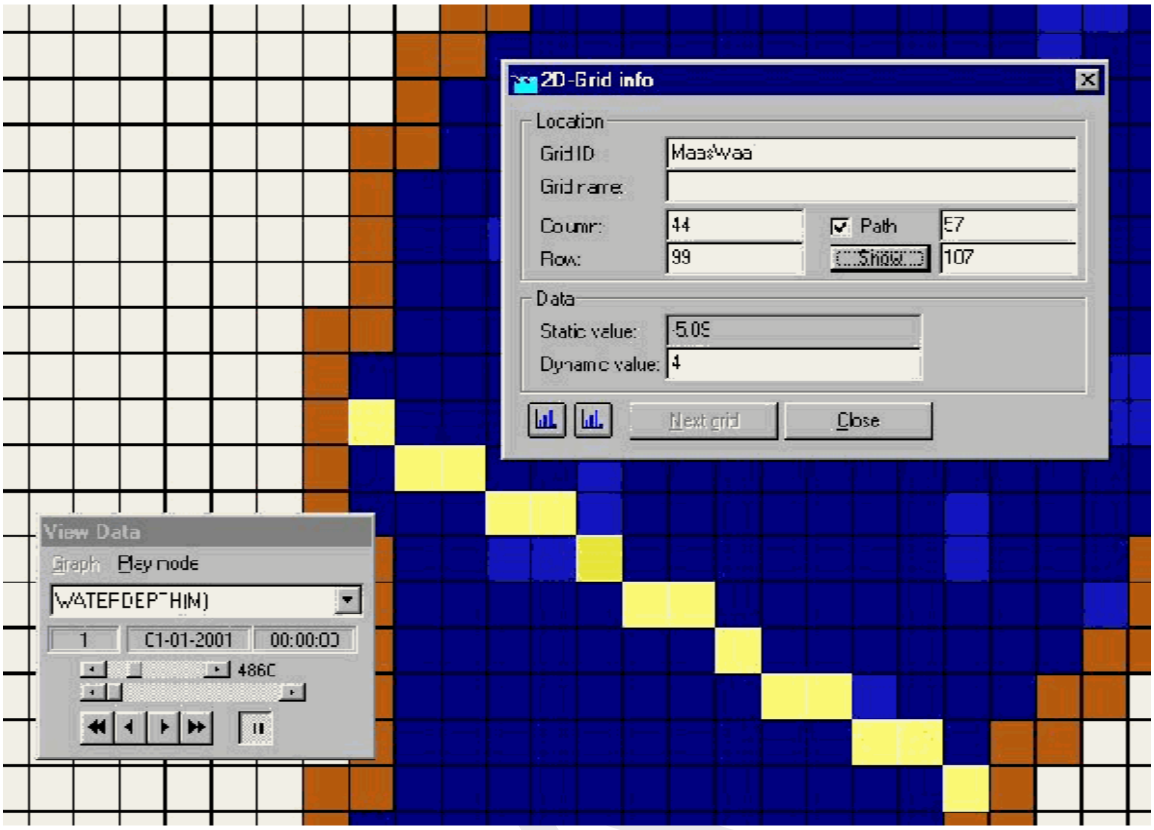


Figure 5.43: *Example of a 2D Path under a particular user defined line of 2D cells.*

5.1.5 Task block: Simulation

5.1.6 Task block: Results in Maps

5.1.6.1 River/Urban/Rural

SideView

In the "Results in Maps" mode, you can create an animation of your hydrodynamic results, plotted alongside a channel, river or sewer pipe. Note: SideView is only available for the SOBEK 1D Flow modules: Channel Flow, Sewer Flow and River Flow.

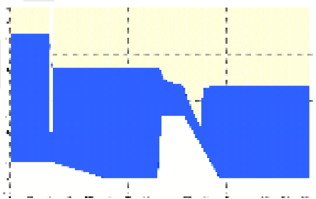


Figure 5.44: *Example of a SideView.*

This chapter explains:

- ◇ How to create a SideView
- ◇ Advanced SideView options

Creating a SideView

◇ Go to the "Results in Maps" task block
◇ Click the starting node (left side of the animation) on the map.
◇ Press the `Shift` key on your keyboard and keep it pressed
◇ Click the end node (right side for the animation) on the map
◇ SideView now automatically selects the shortest connection between both selected points.
◇ Right-click your mouse and select "SideView" from the pop-up menu.

The "Setup Animation" window is then loaded:

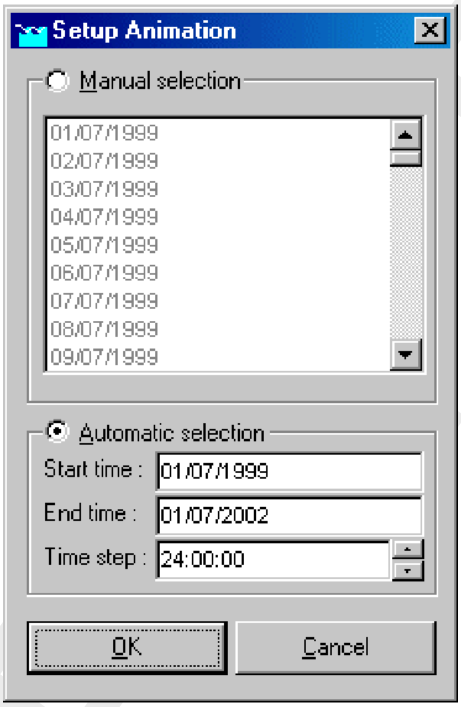


Figure 5.45: *Setup Animation of a SideView.*

Here, you can select the period for which an animation should be shown.

◇ After pressing *OK*, the "SideView" window is opened:

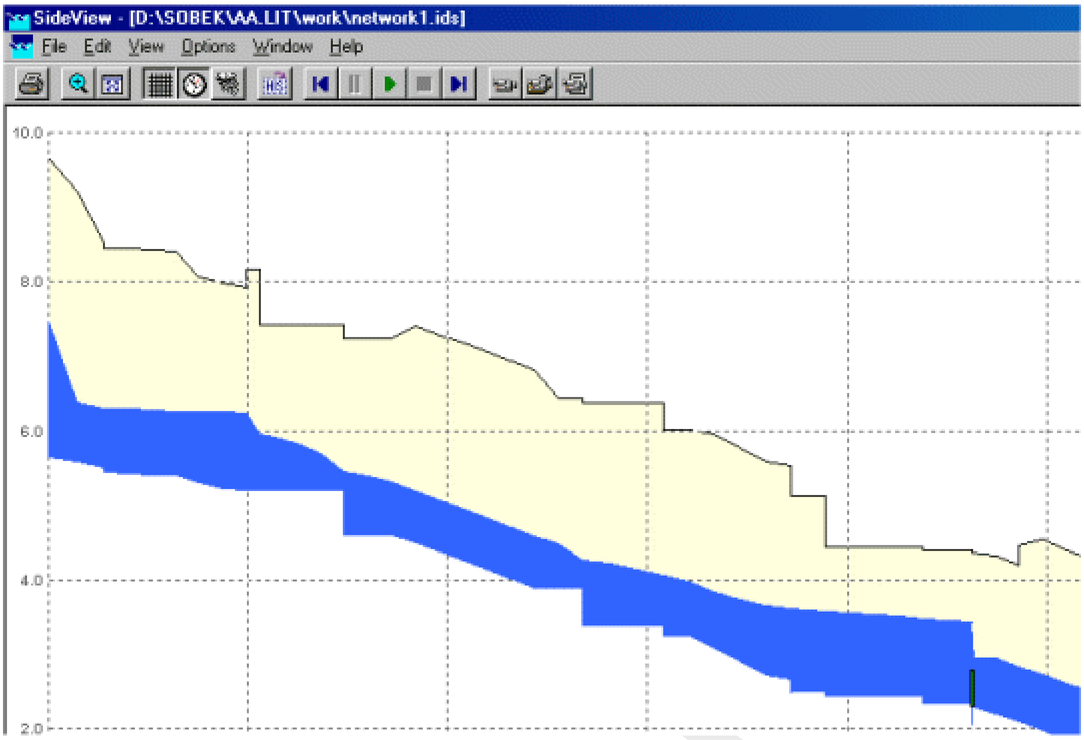
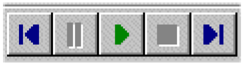


Figure 5.46: The SideView Window.

◇ With the  buttons, you can start, pause and stop the animation.

◇ With the  buttons, you can record the animation and save it as a movie file.

◇ The  button toggles whether you will see the rainfall intensities in a graph during the animation.

Advanced SideView options

◇ If you desire to select another path than the shortest way between the start- and endpoint, you can sequentially add parts of the desired path by keeping the `Shift` Button pressed, thus forcing the path to follow the route that you desire.
Note: if a specific branch cannot be selected with the `Shift` key, because it is never the shortest route, you may release the `Shift` key, and select it (including its surrounding nodes) with the `Ctrl` key.

◇ You can **save the path** for a SideView you have chosen. Choose "File" - "Save" or press `Ctrl`-S to store the path and give it a name.

◇ You can **add the results form another case** to your current SideView. Choose "View" - "User data" - "Get from case" to add them. These results will become visible as red dots in your animation.

◇ You can view **multiple SideViews in one window** by opening previously stored paths in addition to your current selection. Choose "File" - "Open" to do so.

◇ You can **create graphs of the results** at nodes, branch segments and structures from within the SideView window. Click the object in the SideView for which you desire to create a graph, right-click your mouse and select "show graph".

5.1.6.2 Netter

General

The ’results in maps’ task is the most important of the three task blocks when viewing and analyzing results from a simulation. It is usually the first place to go to if you want a quick scan of the results, because the Netter environment provides you with many tools to help you. Once again, the user is expected to have a working knowledge of SOBEK and Netter before reading this manual. This paragraph focuses on explaining the additions to the Netter environment concerning the viewing and exporting of 2D data. It starts with explaining the use of the active legend, which is a great help to quickly understand exactly what results you are viewing and to turn data layers on and off. After that, some more information will be given concerning the three output formats: incremental (only for use within Netter) and GIS (for output to GIS applications). The third output format, History stations (for use in Netter or as output to for example EXCEL), is not explained any further in this document, as it is a standard SOBEK output format which is already explained in the regular SOBEK documentation.

Viewing model and output data in NETTER

Active Legend

The active legend is one of the major additions to this new version of SOBEK/Delft-1D2D. It can be switched on in Netter in both the schematisation task and the results in maps task, by selecting options → active legend. It is a great help with identifying and selecting output data.

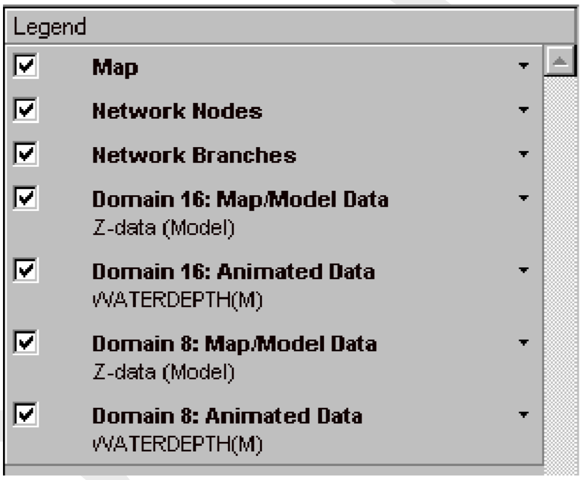


Figure 5.47: *The active legend*

Once the active legend has been switched on, the model and output data become visible as groups in the legend window (Figure 5.47). All groups can be turned on or off using the check-boxes. Selecting a group in the legend window activates a shortcut to the properties/ settings of the group. The triangle-like symbol on the right hand side of the group names turn the group-legends on and off.

Depending on the schematisation and the type of currently selected output data (under file → **open data**), one or more of the following groups become available:

◇ map (model data)
  The currently loaded map-layer.
◇ Network nodes (model data).
  All nodes, both 1D and 2D, in the schematisation
◇ Network branches (model data).
  All branches in the schematisation

◇ Map/model data for every grid (called 'domain')
This data layer can contain either model data OR output data:

  - z-data (model data)
    terrain levels with respect to reference level. Positive means downward. The topography *.asc files are examples of the z-data.
  - depths/levels/velocities (output 'map' data)
    output results for a selected time step. It is not possible to 'scroll' through the simulation time.

◇ Animated (output) data for every grid.
This layer can contain any one of the following output data:

  - Depths
  - Levels
    as opposed to the z-data, the levels are always defined as positive in Upward direction!
  - Velocities
    Should always be positive.

◇ Network data.
This layer can contain the results of a selected 1D-output option (i.e. water levels at nodes) OR the 2D results at history stations.

5.1.6.3 Incremental and GIS output files

Incremental files are generated by SOBEK mainly as a means to visualise animated results in Netter. However, it is also possible to generate an .ASC output file of the current time step and for a selected grid via the **export** option (**file** → **Export** → **2D Grid data** → **Animated data**). Remember that the incremental files contain classes instead of actual values, so the .ASC file generated as output will contain classes as well. In most cases it will be more useful to create .asc files from either a MAP file or use a GIS .asc file directly.

These files are created mainly for the purpose of exporting output data to a GIS-based system such as ARCview.

The GIS files are the .asc output files that the user selected in the settings task. A distinction is made between 'normal' asc files and 'special' asc files. One 'Normal' asc file is generated automatically for every (in Settings) selected GIS output time step and variable. One 'Special' asc file is generated for every special variable, but only one for the whole simulation period! The exception to this rule are the files for the dD/dt variable, which are generated every GIS output time step. After the simulation has completed and the case has been saved, these files are written to the <case name>-directory. Remember that these files contain the actual values and not the classes. Each file represents the actual values for a particular variable and for a particular time.

The following table summarizes the available special .asc output files:

continued on next page

continued from previous page

Parameter	Symbol	Unit	Filename	Explanation

Table 5.2: *Special <*.asc> output files*

Parameter	Symbol	Unit	Filename	Explanation
maximum water depth	d_max	m	dmNMAXDX.ASC/ dNNMAXDX.ASC	in each cell the maximum water depth of all calculated timesteps is written
time of maximum water depth	t_dmax	hr	dmNTMAXD.ASC/ dNNTMAXD.ASC	in each cell is written after what period this maximum water depth has been branched
maximum water level	h_max	m ref. level	dmNMAXHX.ASC/ dNNMAXHX.ASC	in each cell the maximum water level of all calculated timesteps is written
time of maximum water level	t_hmax	hr	dmNTMAXH.ASC/ dNNTMAXH.ASC	in each cell is written after what period this maximum water level has been branched
maximum velocity	c_max	m/s	dmNMAXCX.ASC/ dNNMAXCX.ASC	in each cell the maximum velocity (absolute value) of all calculated timesteps is written
time of maximum velocity	t_cmax	hr	dmNTMAXC.ASC/ dNNTMAXC.ASC	in each cell is written after what period this maximum velocity (absolute value) has been branched
time of wetting	t_wet	hr	dmNTWTXX.ASC/ dNNTWTXX.ASC	in each cell the first timestep is written at which the cell starts to become wet. If the cell doesn't become wet it remains '-999'
time of wetting per class ('steady state')	t_wet/class	hr	dmNWTCII.ASC/ dNNWTCII.ASC	for each incremental class in each cell the first timestep at which this class is exceeded is written. If the class is never exceeded it remains '-999'

continued on next page

continued from previous page

Parameter	Symbol	Unit	Filename	Explanation
time of drying	t_dry	hr	dmNTEMXX.ASC/ dNNTEMXX.ASC	in each cell the last timestep is written at which the cell starts to become dry again after wetting. If the cell doesn't become dry after wetting it remains '-999'
time of drying per class('steady state')	t_dry/class	hr	dmNEMCII.ASC/ dNNEMCII.ASC	for each incremental class in each cell the last timestep at which the water depth is lower than this class is written. If the water depth is always higher than a class, it remains '-999'
rate of change of water depth	dd/dt	m/s	dmNRSSSS.ASC/ dNNRSSSS.ASC	in each cell the difference in water depth is written between two GIS output files divided by the GIS output options timestep Dt

Notice that in the files, where 'hr' is the unit, t = 0 is at the start of the simulation.

Example for the 'rate of change of water depth': if you run 10 hours of simulation period, and you choose in 'GIS output options' Dt = 02:00:00 hh:mm:ss, then this will result in 5 maps.

Filename structure

Most of the .asc filenames generated have the following structure:

dmNVSSSS.asc/dNNVSSSS.asc

General notations used for ASC and MAP files are :

'dm' or 'd' $\rightarrow$ an abbreviated form of 'domain' (another name for 'grid'), dm is used when domain number is between 1 and 9, and d is used for domain numbers between 10 and 99.

'N' $\rightarrow$ domain number, from 1-9. This is an internal number, which is not the same as the domain ID.

'NN' $\rightarrow$ If domain number is from 10 onwards till 99, then the name of the .asc file will begin with 'dNN' i.e. for say domain 19 it will be d19VSSSS.ASC

To check which domain number is related to which domain ID, please refer to file <FLSGIS.HLF> file in your case directory.

'V' $\rightarrow$ type of variable:

d	depth

h	level
c	current
u	velocity in x-direction
v	velocity in y-direction
r	Dd/Dt

'SSSS' $\rightarrow$ represents chosen output time step number (not the time step chosen for the calculation!). In other words, if the time step selected is 15 minutes, then at start of simulation, SSSS = 0000, at 15 minutes after the start of simulation , SSSS=0001 and so on.

'XXXX' or 'XX' or 'X' $\rightarrow$ represents 0 or 1 placed to make the filename 8 character long.

'II' $\rightarrow$ represents the class interval number as given in the Incremental file for a selected parameter.

Similar to the ASC file a MAP file is also created with the name

dmNVXXXX.map/dNNVXXXX.map

The main differences between the ASC and MAP file are that ,

MAP file is a binary file as opposed to ASC file

MAP file contains actual values for all time step, while one ASC file contains actual values for a one time step. Hence no matter how many time steps are defined by the user, there will be only one map file created.

There are a number of other .ASC file created for variables like maximum water depth/ maximum water level/ time at which maximum depth occurs/ time at which maximum water level occurs etc.. These files can be identified from their filenames:

The filenames for such variables are:

Maximum water depth

dmNMAXDX.ASC/dNNMAXDX.ASC

dmNTMAXD.ASC/dNNTMAXD.ASC

The maximum water depth value for the whole simulation run is written in dmNMAXDX.ASC/ dNNMAXDX.ASC while the corresponding timestep when the maximum water depth occurs is written in dmNTMAXD.ASC/ dNNTMAXD.ASC. These files are generated when the option of water depth is checked on in Settings.

Rate of change of water depth

dmNRSSSS.ASC/dNNRSSSS.ASC

The rate of change of depth per selected output time step is written in dmNRSSSS.ASC/dNNRSSSS.ASC. These files are generated when the option of water depth is checked on in Settings. The time step at which these files are written is same as the selected time step for incremental file and thus irrespective of what the output time step is for generation of GIS/MAP files in settings.

Maximum velocity

dmNMAXCX.ASC/dNNMAXCX.ASC

dmNTMAXC.ASC/dNNTMAXC.ASC

The maximum velocity value for the whole simulation run is written in dmNMAXCX.ASC/ dNN-MAXCX.ASC while the corresponding timestep when the maximum velocity occurs is written in dmNTMAXC.ASC/ dNNTMAXC.ASC. These files are generated when the option of velocity is checked on in Settings.

Maximum water level

dmNMAXHX.ASC/dNNMAXHX.ASC

dmNTMAXH.ASC/dNNTMAXH.ASC

The maximum water level value for the whole simulation run is written in dmNMAXHX.ASC/ dNNMAXHX.ASC while the corresponding timestep when the maximum water level occurs is written in dmNTMAXH.ASC/ dNNTMAXH.ASC. These files are generated when the option of water level is checked on in Settings.

Time of wetting/drying

dmNTWTXX.ASC/dNNTWTXX.ASC

dmNTEMXX.ASC/dNNTEMXX.ASC

These two files are special output files. The values in the file dmNTWTXX.ASC/ dNNTWTXX.ASC gives the maximum time span the grid cell remains flooded i.e wetted and dmNTEMXX.ASC/ dNNTEMXX.ASC gives the maximum time span the grid cell remains dry i.e empty. These files are generated when the option of water depth is checked on in Settings.

Time of wetting/drying per class

dmNWTCII.ASC/dNNWTCII.ASC

dmNEMCII.ASC/dNNEMCII.ASC

These two files are special output files. The values in the file dmNWTCII.ASC/ dNNWTCII.ASC gives the time step when a water depth in a grid cell is high enough to be in the particular class. For examples: if there are three classes for depth values i.e. less than 0.1 m, greater than or equal to 0.1 m but less than 1.0 m, and greater than or equal to 1.0 m. So in total three files will be generated for each domain i.e dmNWTC01.ASC/ dmNWTC02.ASC/ dm-NWTC03.ASC. When a grid is just flooded but the water depth is still less then 0.1 m then the time step at which the grid cell is flooded is written in dmNWTC01.ASC, when the water depth in grid cell becomes just higher or equal to 0.1 m but less than 1.0 m that time step is written in dmNWTC02.ASC, and when the water depth becomes just higher or equal to 1.0m that time step is written in dmNWTC03.ASC.

Similarly the values in the file dmNEMCII.ASC/ dNNEMCII.ASC the time step when a water depth in a grid cell is receding and is low enough to be the particular class.

These files are generated when the option of water depth is checked on in Settings.

5.1.7 Task block: Results in Charts

5.1.8 Task block: Results in Tables

Description of the task

- ◇ Double-click the "Results in tables" task box. The **Results in Tables** window appears.
- ◇ Select the appropriate output file and click the *View* button.

The available files are:

- ◇ General Balances Output: cumulative mass balances for the whole model area and the whole simulation period (the <altoys.mes> file).
- ◇ Water Balance for Water Quality network: cumulative water balance for the whole model area and the whole simulation period (the <waterbal.prn> file).

5.2 SOBEK GIS interface (NETTER)

5.2.1 Adjusting the scale settings

When simulation results are opened in NETTER (Task block "Results in Maps"), a scale is automatically created and plotted in the Active Legend. However, you may want to **define your own scale settings and re-use them for other cases.**

- ◇ Activate the "Results in Maps" task block
- ◇ Open a set of simulation results of your choice
- ◇ Open the Color Range window by clicking the name of the results in the Active Legend For example: water level.

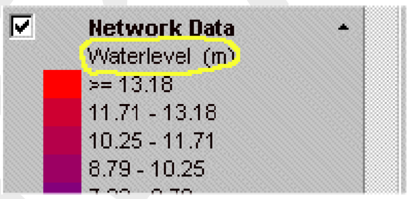


Figure 5.48: *A waterlevel color range in the Active Legend of NETTER.*

Alternatively, you can choose "Options" - "Data value options" from the Netter menu and then click *Scale*.

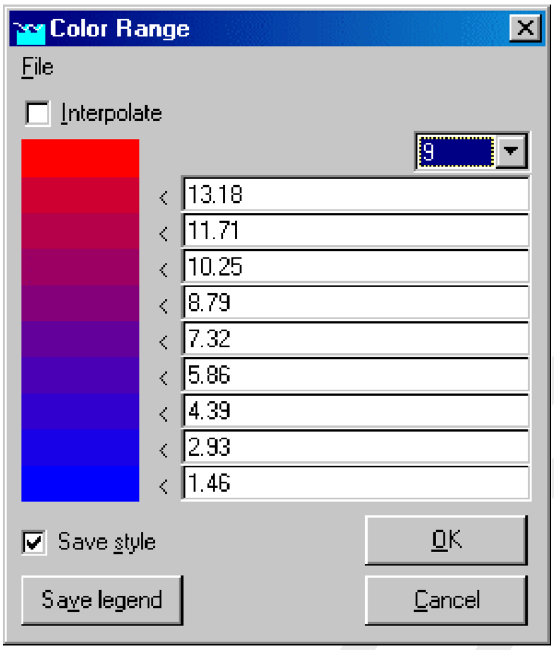
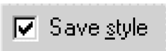


Figure 5.49: *Changing the Active Legend color range and scale.*

◇ Adjust the colours and values until they meet your taste. You can then choose to save these settings by pressing <Save legend>. Use the *Save style* check box if you want to save these settings only for the currently active type of output. Thus:



means: save the current settings only for the currently active results type (i.e. results at nodes)

means: save the legend settings. Depending whether "save style" is switched on, this legend will then apply to all output, or only to the currently active results type. *The legend settings (style) are always stored in the <Netter.leg> file. Additional settings are stored in the <Netter.nls> file.*

◇ Optionally, you can choose to save the legend settings to a file. Choose "File" - "Export style" in order to do so

Important: depending on your SOBEK settings, the customised legends will apply either to **one project only** or to **all SOBEK projects**. See the topic "Saving your custom NETTER settings" to learn more about that.

5.2.2 Curving a branch

The length of the thalway (low water bed) from a river or channel may differ from the distance between two nodes in a straight line. The meanders which cause this can be implemented by switching to the 'vector layer' mode, adding coordinates and dragging them to their natural position on the map. A quicker method is to simply submit a value for the real length. In the vector layer, the branch will then show as a harmonica, forcing itself to adapt the user defined length.

Example:
Suppose there is a river flowing from point A to B:

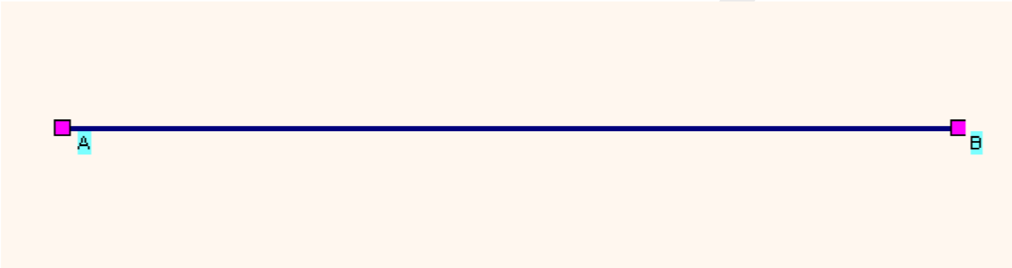
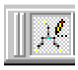


Figure 5.50: *A straight river flowing from A to B.*

The distance in a straight line between A and B is approximately 1 000 m. However, in reality the river meanders, thus making the real length of the river 1 300 meters. Now, how to make SOBEK uses 1 300 meters for the calculation of hydrodynamics?

The simple method to submit the 'real' river length is to:

- ◇ Click the 

button from the "branch" toolbar, click the branch and enter the value of 1300m.
- ◇ See effect this action had in the vector layer by clicking the  button. Notice that in the vector layer the branch has been given a harmonica shape, thus forcing it to be 1 300 m:

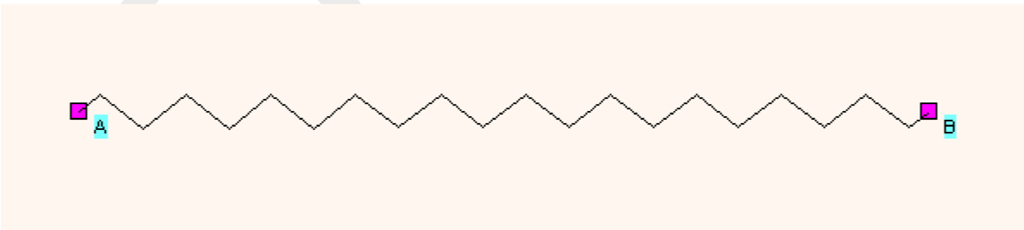
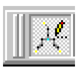


Figure 5.51: *A harmonica-shaped river from point A to B.*

Back in the normal mode you will only see these artificial meanders when you add objects to the branch.

The advanced method to submit the 'real' river length is to:

- ◇ Activate the vector layer by clicking the  button
- ◇ Click the  (show coordinates) button and select the branch.

◇ Click the  button and add coordinates to the branch by clicking and dragging.

Thus you can make the branch follow the real meandering of the river. The result will look like this:

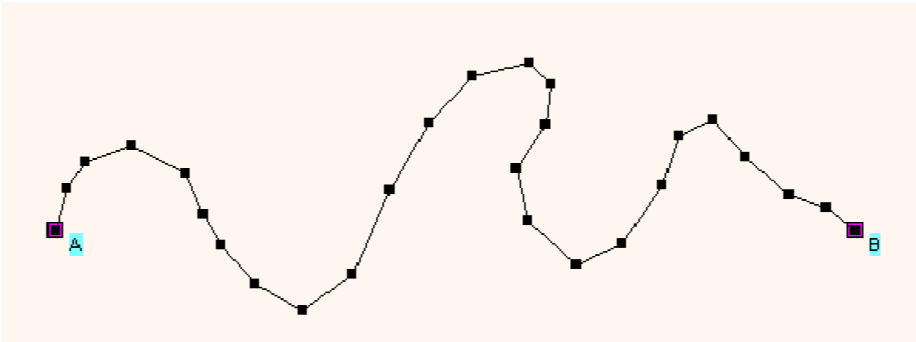


Figure 5.52: *A meandernig river from point A to B.*

Don't be surprised if the meanders disappear again when you leave the "vector layer" mode: the coordinates you just added will only show in the normal mode where there is an object (cross section, calculation point, structure) placed on the branch!

5.2.3 Background map layers

SOBEK offers a library of geographical map layers, such as the map of The Netherlands.

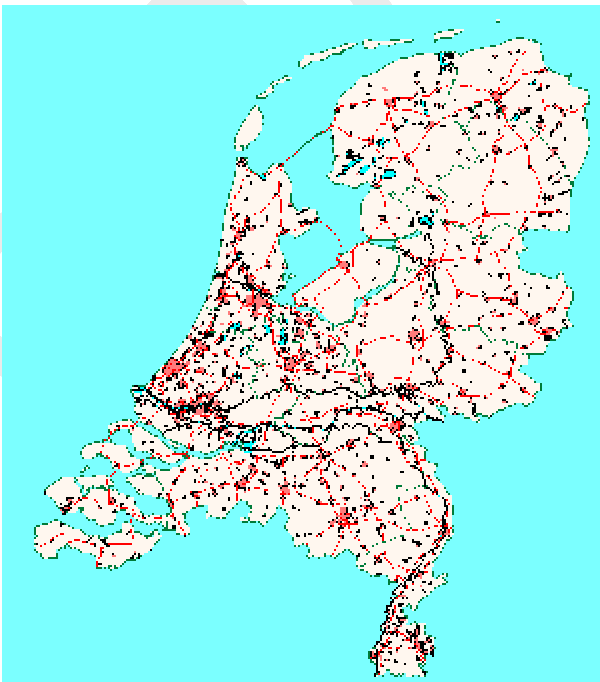


Figure 5.53: *The default map of The Netherlands in SOBEK.*

These maps can be used as the default background for your schematisations. This chapter explains explains how to do this.

◇ Choose ’Options’ - ’SOBEK options’ from the main menu on the SOBEK startup screen. The ’Default Map Settings’ window will pop up:

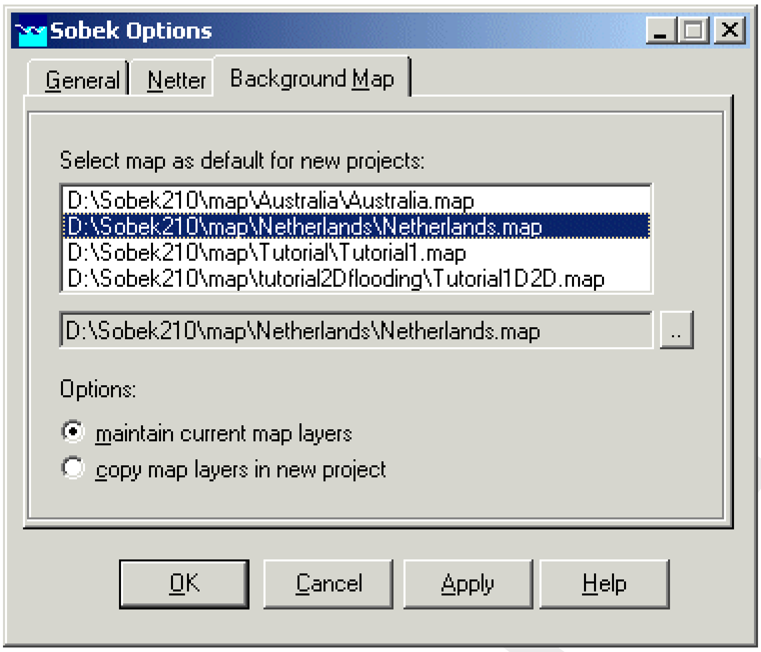


Figure 5.54: *The Default Map Settings window.*

◇ In this window, you can choose which map should be the default for all your new projects. Every time you create a **new** SOBEK project, that map will be used as the default map. The option ’maintain current map layers’ means that your maps will be stored in the specified directory only. This will save disk space, since the maps will not be copied to each new project. However, if you intend to send your project-files to a third-party, it might be handy if the maps are included in the project directory. In that case, choose the ’copy map layers in new project’ option.

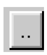
What if your country is not in the list?
If your country does not appear in the list of available maps, you can create one yourself:

◇ Don’t bother about the default map for the time being, but simply start creating a project, based on the existing default map.
Learn how to work with projects first by completing a tutorial!

◇ Once you’ve branched the point where the model schematisation is created (SOBEK’s GIS environment), compose your own map by choosing "Options" - "Map Options" and importing various GIS map layers that you might have. Note: the distance-units of your maps should **always** be meters, as SOBEK interprets them as such! Map layers in lat/lon co-ordinates, yards, inches or feet are not supported!

◇ All the map-based data (including background maps) that you want to use in SOBEK need to be in a UTM-type (WGS84) cartesian projection, with XY coordinates in meters (and not in degrees).

◇ When you’re satisfied with the composition, you can export it. Choose ’File’ - ’Export’ - ’Map’ and give it a proper name.

◇ The next time you start SOBEK, you can go back to the ’Default Map Settings’ option and select your newly created map as the default one! Click the  button to select the location where you stored the map.

5.2.4 Customising NETTER Settings

There are different possibilities to configure NETTER. You can choose to store these configurations on a global level, or to specify them per project:

- ◇ Start SOBEK

Choose "Options" - "Netter options" from the menu in the SOBEK Startup window. The "Netter options" window will appear:

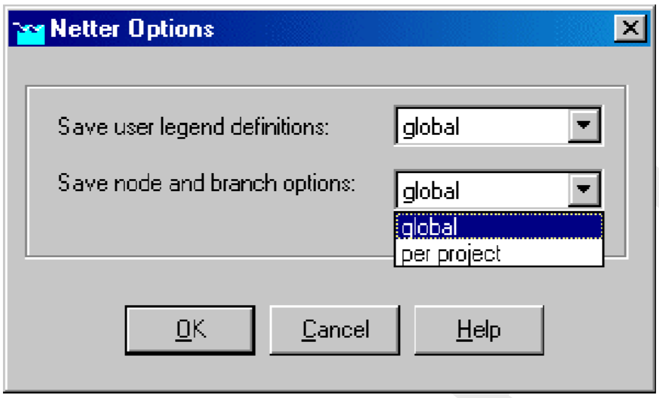


Figure 5.55: *The global Netter options window.*

Here you can choose whether the settings you will define in NETTER (legend and node/branch settings) should be applied globally (to all projects) or per project.

Note: For the ‘node and branch options’, the results are stored in the file <NtrUser.ini>. The Netter Legend options are stored in the <Netter.leg> file. When stored as ’global’, they can be found in <\SOBEK215\Fixed\User>; when stored per project they can be found in the <\Fixed> directory of the project.

5.2.5 Exporting results to a database

It is possible to automatically gather specific results and prepare them for exporting to a Microsoft Access (<*.MDB>) database. This is useful for generating standardised reports.

The export definitions can be stored on three different levels:

- ◇ On a case level - for each case, you will have to create a new definition of the desired output combination
- ◇ On a project level - within every case of the project, the standard export settings will remain the same
- ◇ On a global SOBEK level

For every case, you can select which of the above level should apply.

- ◇ Go to the "Results in Maps" task block
- ◇ Choose ’Tools’ - ’Export Options’. The "SOBEK Export tool" window will appear:

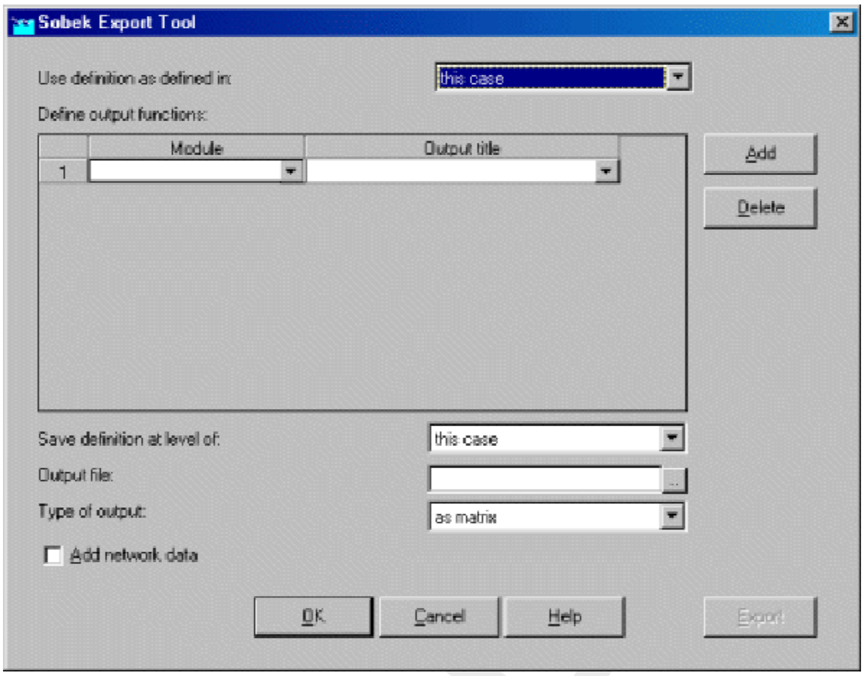


Figure 5.56: *Adding a definition in the SOBEK Export Tool.*

◇ Define for which level your definition should be saved (case, project, global) in the field "save definition at level of:"
◇ Define a name for the output file
◇ Select in the fields "Module" and "Output file"
◇ Select the type of output: as a matrix or as a table
◇ Click *Export* to actually create the MDB file

5.2.6 Model data editor

For many objects in your network, you will have to submit **parameter values**. For example, these can be the *bed level* or *shape* of a river's cross section, the *friction value* of a sewer pipe, the *infiltration capacity* of an unpaved area, and so on.

(If you don't know how to create a network, view the chapter basics of network editing or follow one of the tutorials provided in this document).

There are two ways to define the parameter values (model data) to the schematisation:

◇ Using the single data editor**(editing one object at a time)**
◇ Using the multiple data editor**(editing multiple objects at a time)**

Using the single data editor
To submit parameters to an individual object at a time, the following steps can be followed:

Option 1: accessing the single data editor through the menu

◇ Make sure that you are in the SOBEK model editing environment ('schematisation' task, 'edit model')
◇ Switch to the 'Model data' mode by clicking 'Edit' - 'Model data'

The following window will appear:

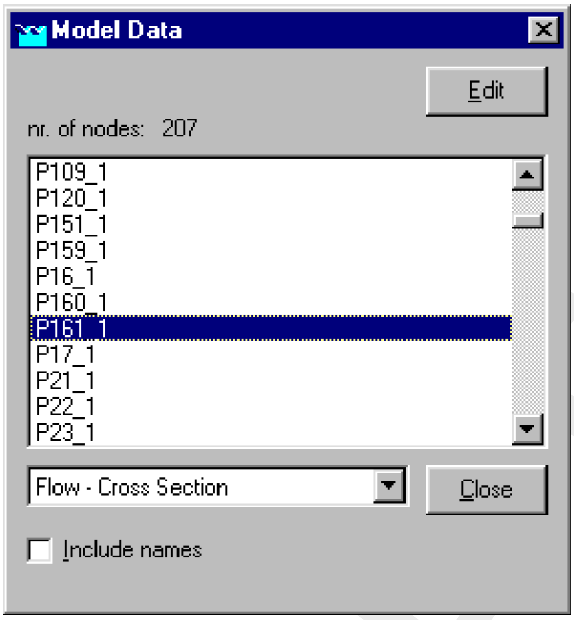


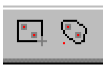
Figure 5.57: *Starting the single data editor from the Model Data menu.*

◇ Select from the combo box and list for which object you want to edit the parameters.
◇ Click the *Edit* button. You'll see an edit-window appearing for the object that you selected.

Option 2: selecting from the map

◇ Make sure that you are in the SOBEK model editing environment ('schematisation' task, 'edit model')
◇ Switch to the 'Model data' mode by clicking 'Edit' - 'Model data'
◇ Click on the  button in the  toolbar to activate the "select" tool.
◇ Click with the **right** mouse button on the object for which you want to edit the data.
◇ Choose 'Model data' from the window that appears.
◇ Notice that an edit-window for the selected object appears.

Using the Multiple Data Editor
To submit parameters to multiple objects at a time, the following steps can be followed:

◇ Choose one of the  buttons to drag an area on the map, covering all the objects you want to edit.
◇ Click with the **right** mouse button
◇ Choose 'Model data' from the window that appears.
◇ Notice that the "multiple data editor" is opened. Here you can edit data for several objects at the same time, in a spreadsheet-like manner.

Some extra options are added to the multiple data editor, allowing quick manipulation of data. For sewer applications, it is useful to have the option to multiply a unit runoff area per meter pipe with the actual pipe length. This option is available by using the right mouse button menu after selecting a column. Also, the option to import data from a database has been added.

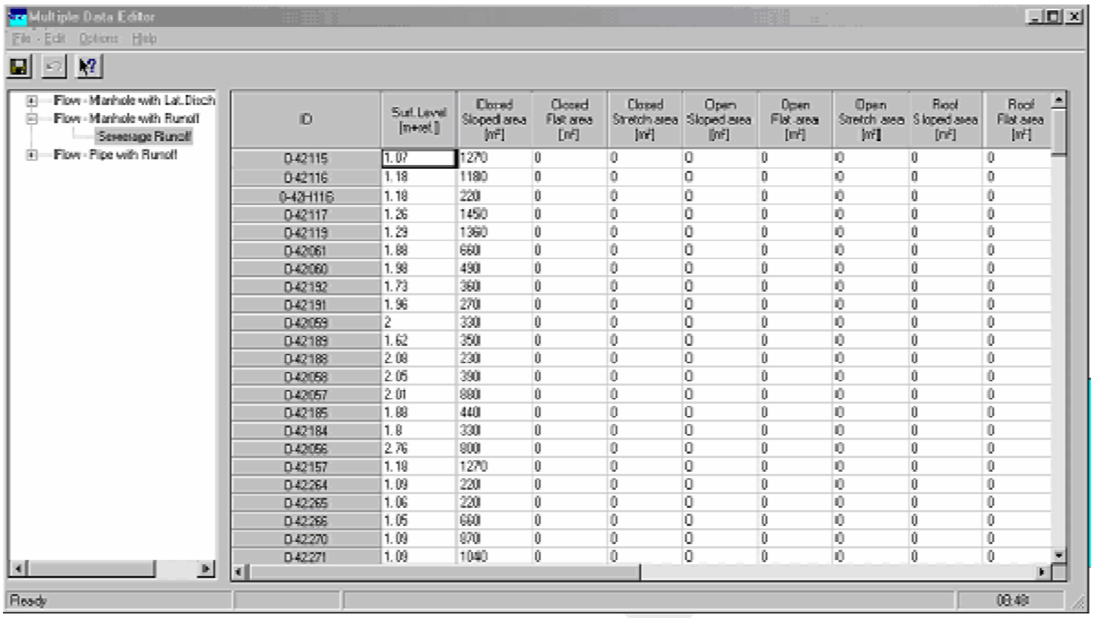


Figure 5.58: The multiple data editor.

The following edit options are possible:

- ◇ Change individually;
- ◇ Select a whole column, use the right mouse button, several options are available(Add, Multiply, Replace);

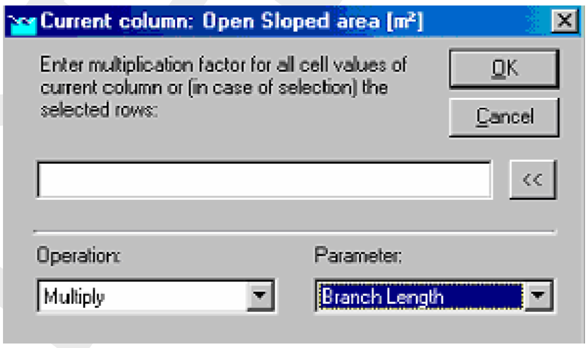


Figure 5.59: Changing a whole column in the multiple data editor.

- ◇ Select a whole column, use the right mouse button, Import option to import a column from a DBF file is available. After selecting the DBF file, the coupling of the id's in the DBF file to the SOBEK id's, and the data column to be imported can be indicated in the following screen. If the DBF contains multiple records for the same id, the data of all records with the same id can be added together using the switch 'Add all occurences together'.

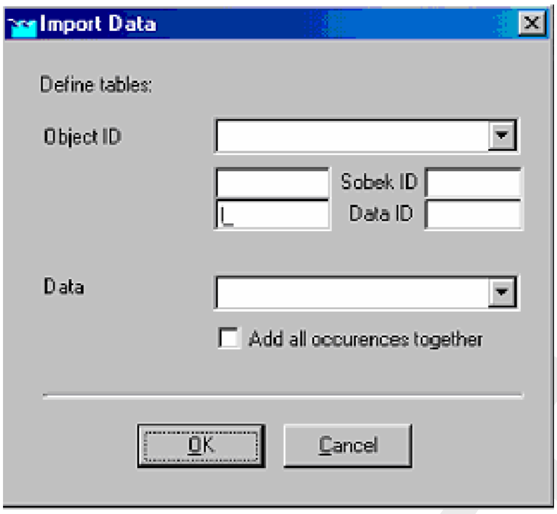


Figure 5.60: *Importing data in the multiple data editor.*

◇ Select a whole column, use the right mouse button, Distribute Catchment option. The specified catchment will be distributed over the runoff area. If it concerns pipes with runoff, the distribution will be according to pipe length. If it concerns manhole, the distribution will be proportionally (all manholes with the same runoff area).
◇ When selecting Flow - pipes, there are also options to interpolate the pipe levels between the first and the last selected row, or to design the pipes with a fixed slope. In order to apply this in a meaningful way, the pipe must be in logical sequence and all in the same direction.
◇ Not every object type is available for editing in the Multiple Data Editor. These objects should be modified with the Single Data Editor.
◇ Although friction for Y-Z and Asymmetrical Trapezium profiles may appear to be editable by using the multiple data editor, this is not recommended. The friction concept that is used in the Multiple Data Editor is the Local (or Branch-wise) friction concept. However, Y-Z and Asymmetrical profiles make use of the so-called Cross-section friction concept. This can cause the (Local type) friction value reported in the Multiple Data Editor to differ from the (Cross-section type) friction value reported in the Single Data Editor for Y-Z and Asymmetrical Trapezium profiles. See the chapter 1D Hydraulic friction concepts for more information.

Basics of network editing
One of the first things to do when you start working with SOBEK is to create a network that represents your water system. Whether it is a river, channel or sewer system, the principles are the same: it will be a GIS-based network or grid. In this chapter the general concepts behind editing your hydraulic model schematisation are explained.

1. Start SOBEK's GIS environment (NETTER)
After you've opened or created a case, you'll see the Case Management Tool (CMT), showing eight different task-blocks that can be executed. Read more about the CMT in the chapter

Managing SOBEK projects and cases. If a task-block is yellow or green, it means that this task is ready to be executed.

◇ Make sure that the 'Schematisation' task-block is ready to be executed;
◇ Double click it;
◇ Click the *Edit model* button on the window that appears. Now SOBEK's GIS-environment named NETTER is started.

2. Switch to the network editing mode
After entering SOBEK's GIS environment under the "Schematisation" task block, one can toggle between three modes:

◇ The data-editing mode
◇ The network-editing mode (discussed in this chapter)
◇ The vector layer mode

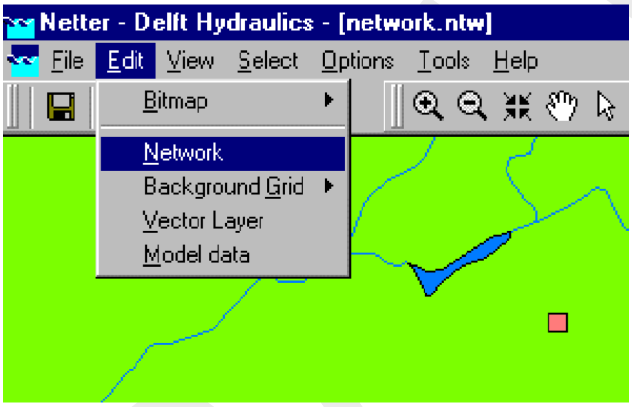


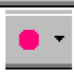
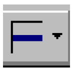
Figure 5.61: *Enter the network editing mode.*

Choose 'Edit' - 'Network' to switch to the mode where you can adjust or create a schematisation.

Switch to the network-editing mode by choosing 'Edit' - 'network' in the main menu or click the  button. You'll see the following toolbar appearing:

3. Network editing toolbars
The toolbar shown above consists of three main sections:

◇ an action section 

◇ a node type section 
◇ a connection type section 

The philosophy behind it is as follows: when creating or editing a 1D hydraulic model, you choose to perform an certain *action* that applies for a certain *node type* and/or a certain *connection type*. So when performing edit actions, make sure you select the right node type (for instance a Flow-cross section) and the right connection type (for instance a Flow-branch with lateral flow).

Each of the sections can be opened by clicking the button next to it.

Notice that all toolbars underneath can be dragged to your screen by clicking their blue line on the top!

An example: you can drag the 'General' toolbar, under the 'action' section to your screen by clicking it's upper part, as shown below.

In a similar way, you can drag the 'node types' toolbar to your screen. When moving with the mouse over the different node types in that toolbar, it will then tell you which types they are.

Helpful hints for the user:

◇ Add labels to the node types shown in the toolbar! If you have difficulty recognising the various node- or branch types in the objects-toolbars, you can have their label shown: 'Choose 'View' - 'Toolbars' - 'Customise' and select 'Icon and caption' for node types and/or branch types.
Note: In recent SOBEK versions, labels are enabled by default.

◇ See what type of node is shown in the toolbar After you dragged the toolbar containing node types to your screen, you can move over it with the mouse. The node type will then be shown:

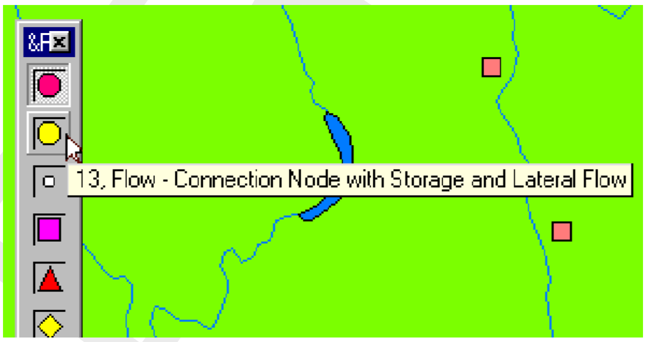


Figure 5.62: *Mouse-over a node to show its label.*

See what node type is shown in the "nodes" toolbar by moving over it (only after the toolbar has been dragged to your screen)

◇ ID's of nodes and branches Object ID's are directly used by the calculation core so users should be very careful when modifying or manually setting such an ID.
It is recommended to let SOBEK pick the ID. Users should preferably use the name field to add additional information about network objects. Although it is possible to use spaces in object ID's, this is not recommended. If the user wishes to manually set an ID, underscores '_' should be used instead of spaces. When using the Water Quality module, ID's should contain no more than 19 characters. See the water quality documentation for more details.

To get started creating a model schematisation, you can learn to: Create a branch, Curve a branch, Place objects on a branch, or do one of the tutorials for Channel Flow, Sewer Flow or Overland Flow

Save unique display settings for a specific type of output
After opening a certain type of results, you may customise the way in which they are plotted, and save these settings for the next time. The procedure is best explained through an example.

Example:
Say, that **every time** we open the *results at branch segments* we want them plotted in such a way that:

- ◇ All nodes are invisible
- ◇ The flow direction will be shown by animated arrows

In order to do so, you will need to do the following

- ◇ Open the *Results at branch segments* by clicking on it in the Active Legend
- ◇ Uncheck the checkbox  left from the category "Nodes" in the Active Legend
- ◇ Click the button , in the right upper corner of the Active Legend to activate the "Settings" menu.
- ◇ Click the "links" tab and activate the options "all data" and after that: "arrow flow"
- ◇ Click *OK* to leave the *Settings* menu.
- ◇ Open the "Scale settings" window by clicking the "Network data" in the Active Legend.
- ◇ Activate the checkbox  left from the option 'save style'. With this option, you will save the output 'style' that belongs to "results at branch segments".
- ◇ Click *OK* to leave the **Scale settings** window.

You can check it by first clicking the "Results at nodes" (which will restore the default settings again) and then clicking the "Results at branch segments" again. The results will again be shown in the way you had previously stored. You can load a series without a saved style/user legend, the last style in use will then still be used.

5.2.7 Shortcuts to various menu options

You can click the **header** of each category to customise its settings. The headers provide a shortcut to the corresponding menu options

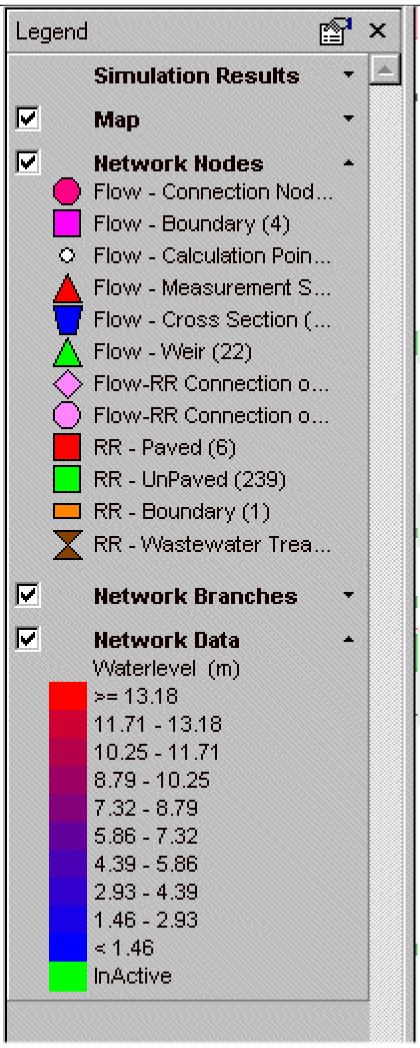
- ◇ Clicking *Map* will open the **Map properties** window. This will allow you to arrange, and adjust the settings for the GIS layers that you use as a background in the schematisation.
- ◇ Clicking *Network Nodes* will open the **Node Settings** window, which will allow you to adjust the way (size, shape, colour) nodes are plotted in the main window. It also allows you to select which node types should be visible, and which not.
- ◇ Clicking *Network Branches* will open the **Link Settings** window, which will allow you to adjust the way (size, colour) branches (branches and links) are plotted in the main window. It also allows you to select which node types should be visible, and which not.
- ◇ Clicking *Network Data* will open the **Scale Settings** window, which will allow you to customise the scale and colours for the data legend.

5.2.8 The Active Legend

The active legend is an interactive legend, showing all objects and results that are available within the GIS interface. The Active Legend can be customised entirely to the user's wishes. If the Active Legend is not yet visible in NETTER, it can be activated with the menu option "Options" - "Active Legend".

Features of the Active Legend:

◇ Gives Shortcuts to various menu options, such as the windows: *Settings, Map Properties, Node Settings, Link Settings and Scale settings*
◇ Allows you to save the display settings for each different type of output. For example: it can remember that you want nodes to be invisible when viewing the *results at branches*. This feature is explained further in this chapter.
◇ Allows you to create a list with user defined output; shortcuts to simulation results that you frequently view.



The following categories can be activated:
Simulation results (select the kind of computational results that should be opened)
Map layers (GIS layers for background plotting)
Network Nodes (shows the used node types in your schematisation)
Network branches (shows the used branches and links in your schematisation)
Network data (showing the legend of the results of a simulation)

General usage of the Active Legend

◇ With the check boxes , each category can be made visible or invisible in the main window.

◇ Clicking the and buttons will respectively show and hide each category's contents.

◇ The button opens the **Settings** window, which allows you to plot labels next you the objects of your choice. This button is a shortcut to the menu "Options" - "Network Data"

5.2.9 Create a list with user defined output

If there's certain output that you often need to watch, you can make it accessible quicker in the following way:

◇ Open the NETTER menu option "Tools" - "Output options"

The **Output functions** window will pop up:

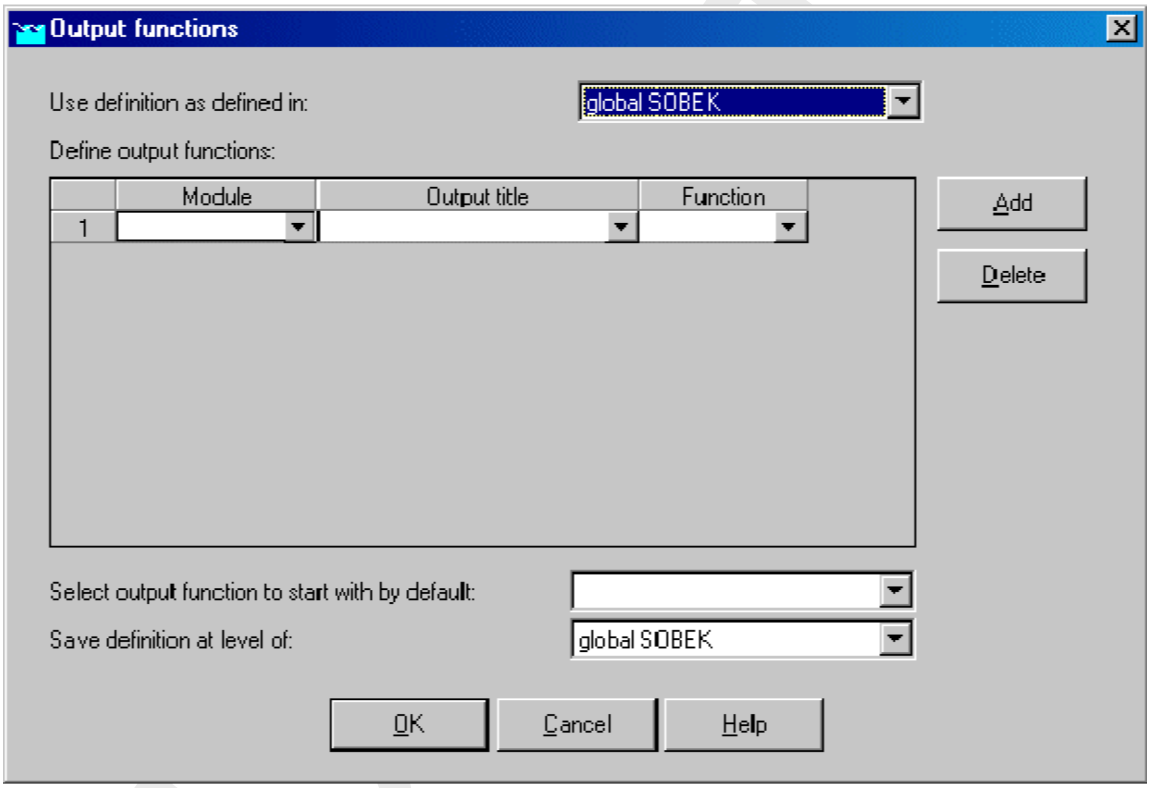


***Figure 5.63: Output functions** window*

◇ In this window, you can add the time series that you desire to access quickly every time after a calculation. Use the fields "module" - "output title" and "function" to define a certain time series.

Example:

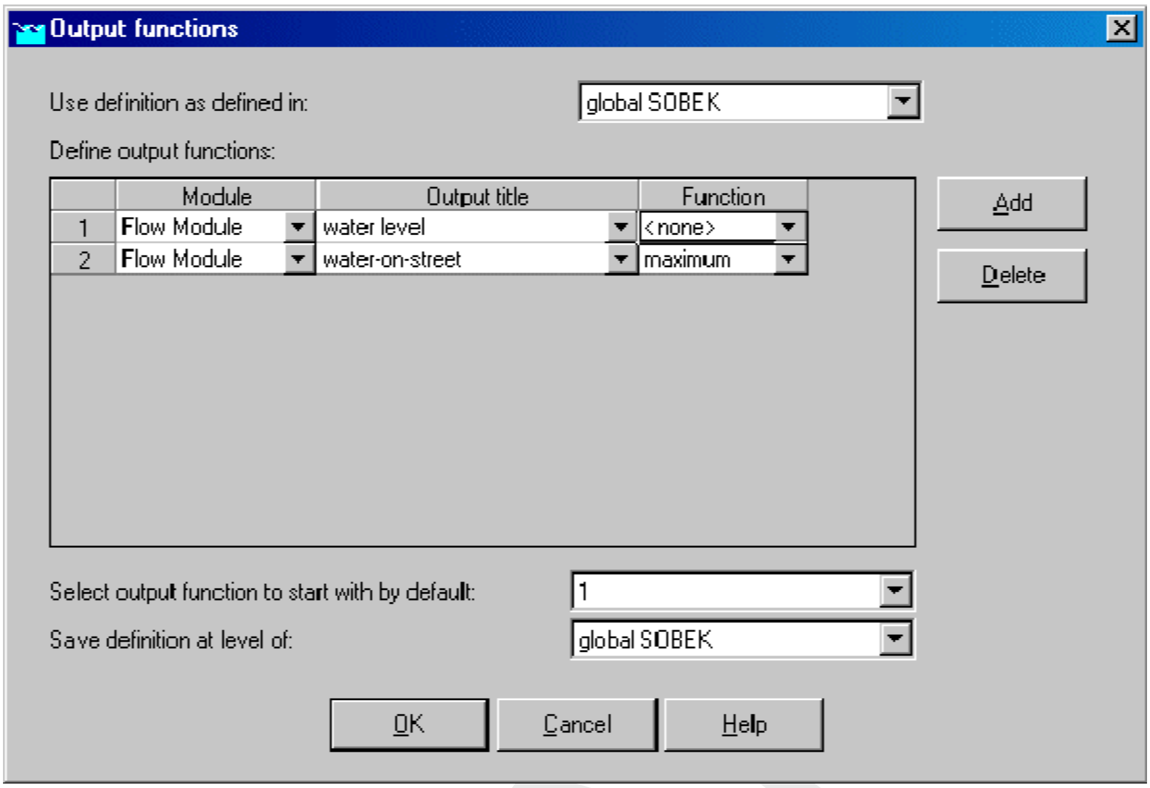


Figure 5.64: *Adding definitions in the* ***Output functions*** *window.*

- **Note:** If you choose <none> in the function field, all values that are available for all time steps from the time series are taken. The other options speak for themselves.
- After clicking *OK* the series that you've selected here, will become available under the header "**user defined output**" in the Active Legend:

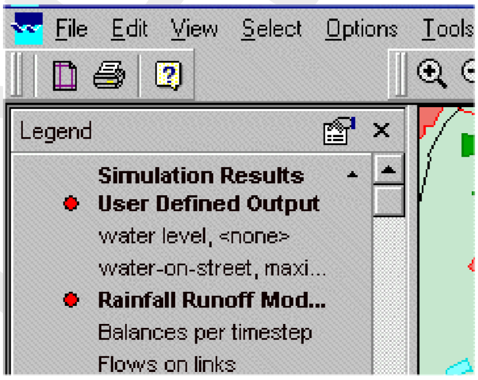


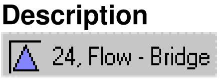
Figure 5.65: *User defined output in the Active Legend.*

- **Note:** Also on these *customised* series, you can save your unique display settings, as described in the chapter above!

5.3 Node description (hydrodynamics)

5.3.1 Flow - Bridge node

Description



In this chapter, the *Flow - Bridge* node type is described.

◇ For a detailed description of this node's input parameters: see the "Flow - Bridge node input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the *Flow - Bridge node topology* section from the Reference Manual
◇ For a detailed description of this node's underlying mathematical equations: see the "Bridge section from the Technical Reference Manual".

With a *Flow - Bridge node* you can simulate four types of bridges:

◇ Pillar bridge
◇ Abutment bridge
◇ Fixed bed bridge
◇ Soil bed bridge

Input screens
When starting the model data editor for an *Flow - Bridge* node type, the following tabs will be available for input:

Bridge:

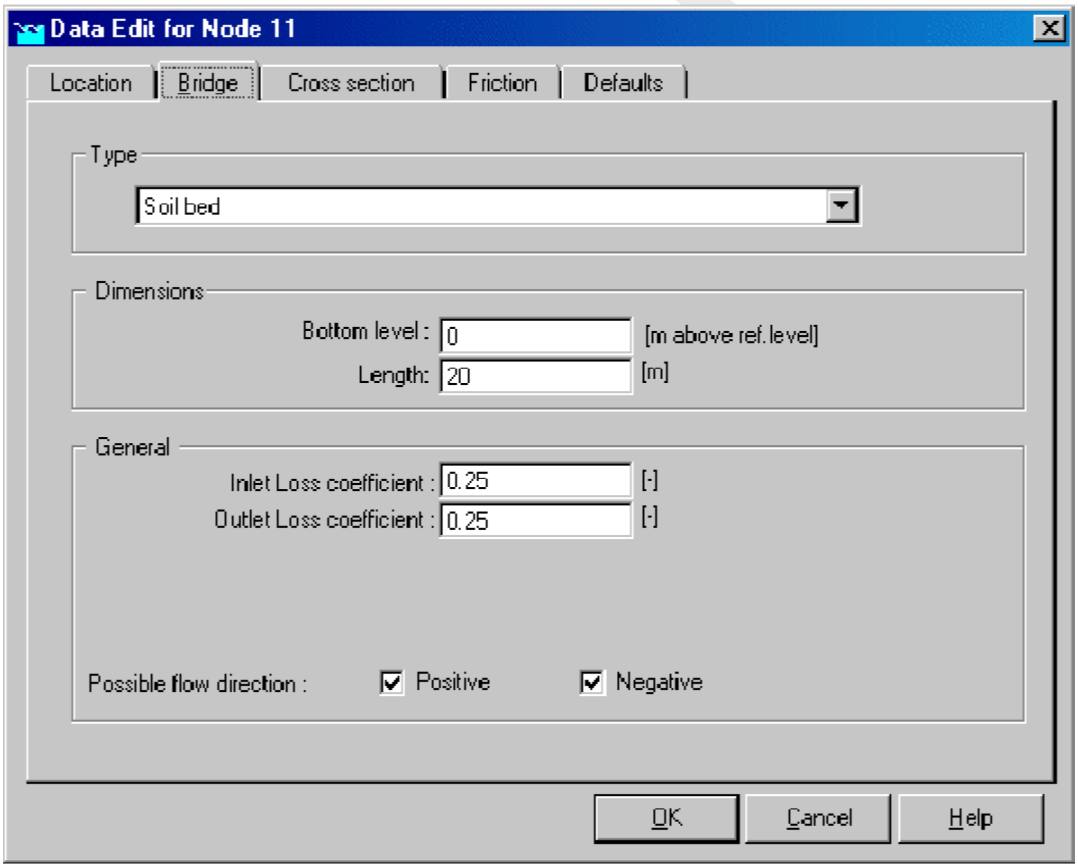


Figure 5.66: *The Bridge tab of a bridge.*

A bridge of the type *Pillar Bridge* requires the following parameters:

◇ Total pillar width (sum of the width of all pillars, perpendicular to the flow direction)

◇ Shape factor
◇ Possible flow direction

See the Technical Reference manual, ‘Bridge Section’ in order to see how these parameters are used in the structure’s equations.

A bridge of the type *Abutment* bridge requires the following parameters:

◇ Bed level
◇ Length
◇ Inlet loss coefficient
◇ Outlet loss coefficient
◇ Possible flow direction

A bridge of the type Fixed Bed Bridge requires the following parameters:

◇ Bed level
◇ Length
◇ Inlet loss coefficient
◇ Outlet loss coefficient
◇ Possible flow direction

A bridge of the type Soil Bed Bridge requires the following parameters:

◇ Bed level
◇ Length
◇ Inlet loss coefficient
◇ Outlet loss coefficient
◇ Possible flow direction
◇ Ground layer depth (note: this parameter is defined on the *Cross Section* tab)

Below, all of the mentioned parameters will be discussed.

◇ **Bed level:** Represents the bottom of the construction, in meters w.r.t. reference level.
◇ **Length:** Represents the length (in the flow direction) of the bridge.
◇ **Inlet loss coefficient:** Represents the energy loss of water that enters the bridge (through contraction). See the "Bridge" section from the Technical Reference Manual to see how this parameter is embedded in the structure’s equations.
◇ **Outlet loss coefficient:** Represents the energy loss of water that leaves the bridge. See the "Bridge" section from the Technical Reference Manual to see how this parameter is embedded in the structure’s equations.
◇ **Possible flow direction:** This parameter defines in which direction(s) water can flow through the orifice. The directions are defined with relation to the defined branch direction. Positive means: in the same direction as the defined branch direction; negative means: in the opposite direction than the defined branch direction. If you don’t know the defined direction of the branch, read the Flow - Branch topology section of the Functional Reference Manual.

Cross Section: (Note: Not available when bridge type "pillar" has been chosen)

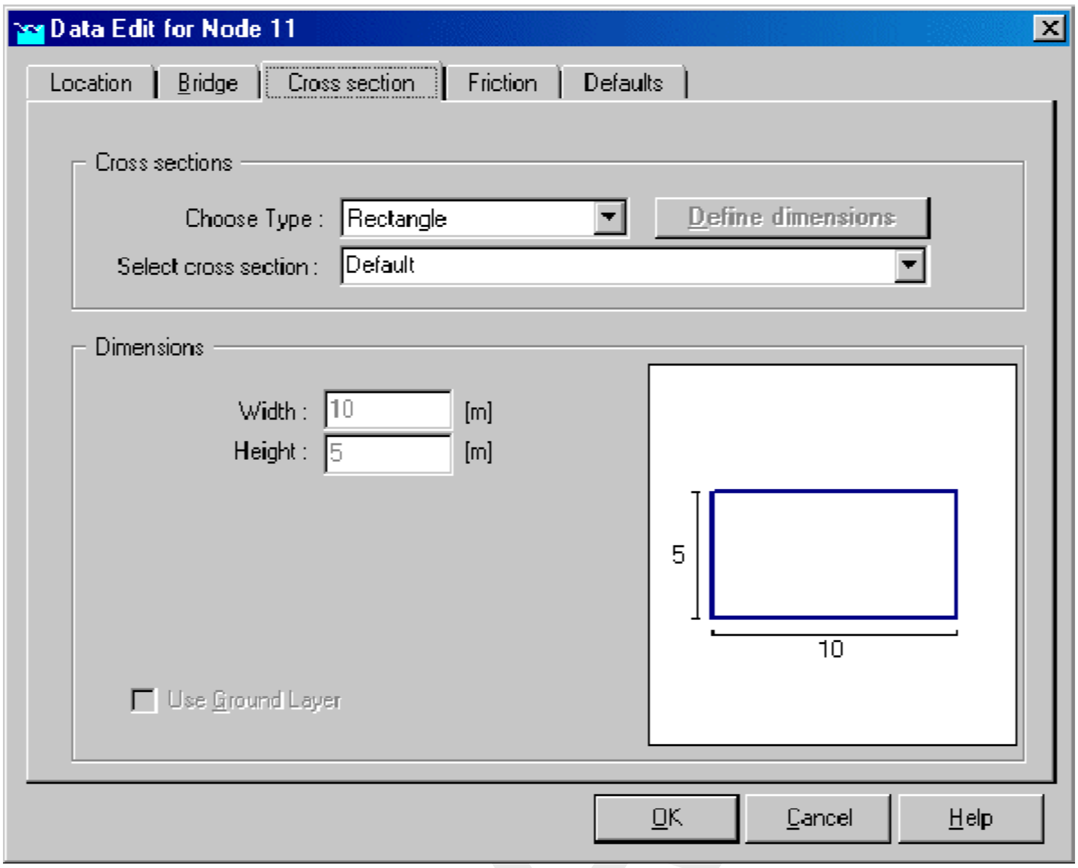


Figure 5.67: *The Cross section tab of a bridge.*

On this tab the cross sectional shape of the culvert should be defined. This is done by means of **cross section definitions**, very similar to the ones you define for the shape of a river bed. The only real difference is that the number of cross section types is limited to two:

- Table-form
- Rectangle

For a detailed description of all available cross section types and their input parameter, see the "Functional Reference Manual" - "Channel Flow module" - "Cross section types" section.

The ground layer option is available only for bridges of the type "Soil layer bridge". For that type of bridges you can enter a ground layer depth, which represents a layer of sediment that's often present in structures. On the "Friction" tab, this layer can be given its own hydraulic roughness value. Note: the option "use ground layer" should be switched on too!

Friction: (Note: Not available when bridge type "pillar" has been chosen)

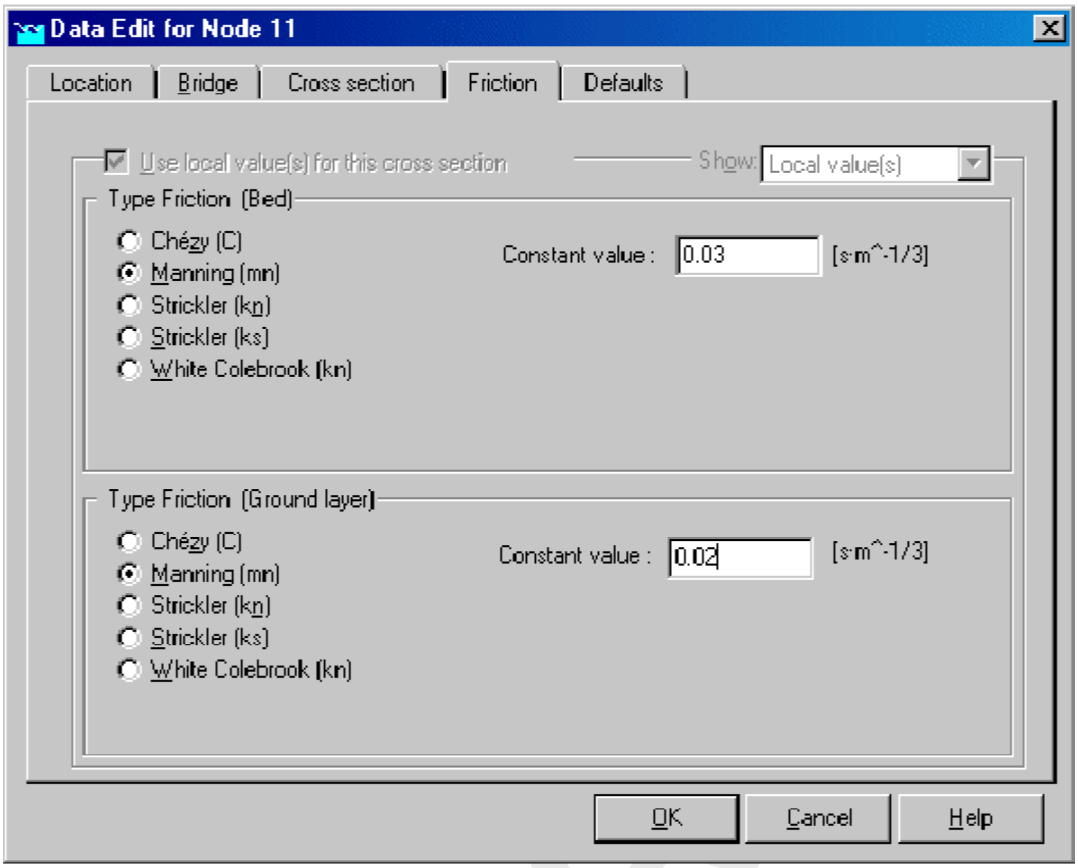


Figure 5.68: *The Friction tab of a bridge.*

On this tab, the friction values for the bridge can be entered:

◇ **Type friction bed:** This friction value represents the roughness of the bridge's walls. Thus, if the bridge is made of concrete, a proper hydraulic roughness value for concrete should be entered.
◇ **Type friction ground layer: (only for bridge type "soil bed bridge")**If the bridge type "soil bed bridge" has been chosen **and** the "use ground layer option has been activated on the "cross section" tab, this option becomes active. It represents the roughness value of the ground layer that lays on the bottom of the bridge.

For more information about the various friction options, see the "Bed Friction" section from the Technical Reference Manual.

Topology

Nodes of the type *Flow - Bridge* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - Bridge* node to your schematisation in the following way:

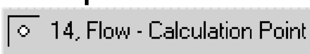
- ◇ Select the appropriate node type (Flow Bridge)
- ◇ Use the "add node" button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- ◇ If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node" button.

Note: Before adding a bridge to your schematisation, you should have at least one of the above Channel types present.

5.3.2 Flow - Calculation point

5.3.2.1 Flow - Calculation point (basic)

Description



This chapter describes the nodes of the type *Flow Calculation Point,* which all together form the *Calculation Grid* or *Computational Grid*. The *Flow - Calculation Point* node type is one of the most important nodes for hydrodynamic modelling as their presence determine the spatial discretisation of the model area.

- ◇ For the input options you have when creating a calculation grid: see the Flow - Calculation grid input screens section from the Reference Manual;
- ◇ For a detailed description of this node's possible network configurations: see the "Flow - Calculation point topology" section from the Reference Manual
- ◇ For a detailed description of this node's underlying mathematical equations: see the "Calculation point section from the Technical Reference Manual".

The SOBEK Water Flow modules solve the model equations on this grid of ζ-calculation points. For good model performance it is therefore important that these points are spaced evenly. Therefore SOBEK has an option to generate the calculation grid automatically. This can be done after the schematisation has been completed.

Flow - Calculation grid input screens
After you have chosen to create a calculation grid, by using the *Set calculation grid branch* or *Set calculation grid all branches* buttons ( ), the following screen will appear:

Tab: General

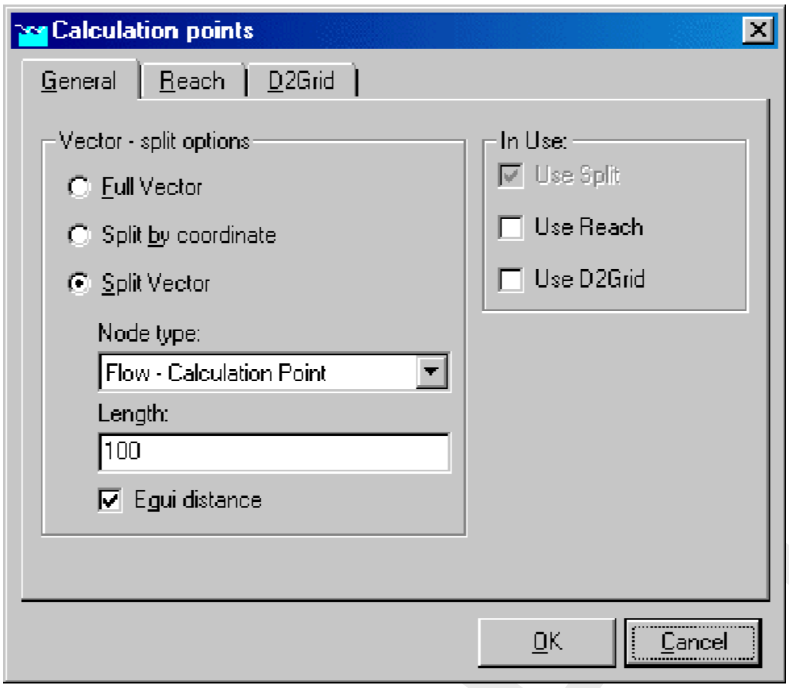


Figure 5.69: *Setting a calculation grid on all branches.*

This tab will probably be the only one you will ever use. Here you can define how the branch(es) should be divided into computational segments. The following options are available:

- **Full Vector:** This option means: the branch will **not** be split up into computational segments. Therefore no calculation points will be added to the branch. You can also use this option to remove calculation points from an existing branch.
- **Split by coordinate:** When using this option, the branch will receive a ζ-calculation point on every vertex. 'Vertex' is a GIS term for all points on a line, that define its curves. For example: a branch may run from node A to node B, but it may have some meanders. These meanders are defined through vertices. These also determine the actual length of the branch. You can manually remove or add such vertices to your SOBEK branch in the Edit Vector Layer mode.
- **Length:** This option will split the branch(es) into computational segments of a certain length. It does so by adding ζ-calculation points.
- **Equidistance:** Gives you a choice of what to do with the remaining lengths while creating the ζ-calculation grid. Suppose that your branch has a length of 1 020 m. Setting a ζ-calculation grid with a length of 100 m: Equidistance = off: this will result in ten branch segments of each 100 m and one of 20 m. Equidistance = on: this will result in ten branch segments of each 102 m.
- **In use:** With these check boxes you can define whether the other tabs will also be used: - Branch- 2DGridMore information about those tabs and their options is given below.

Tab Branch

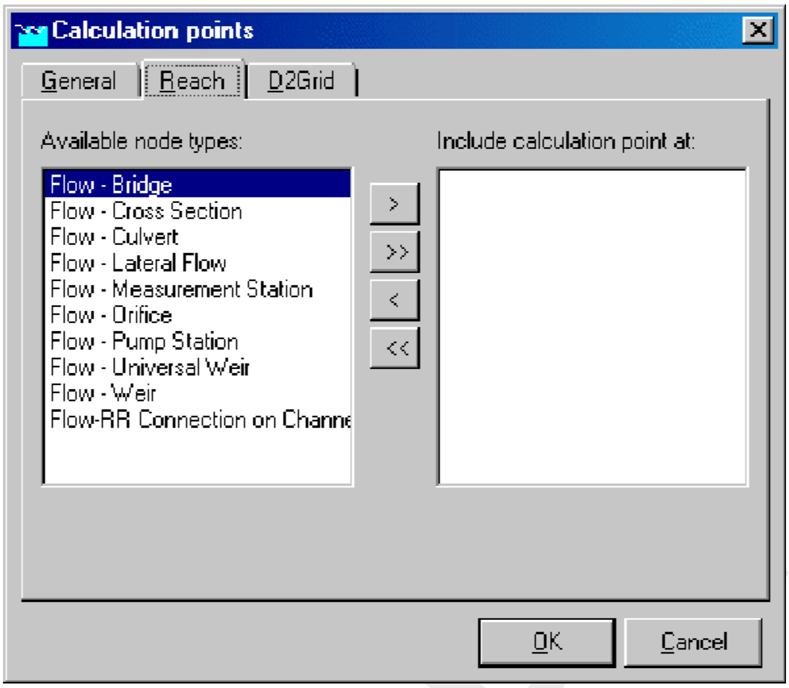


Figure 5.70: *The Branch tab of the Calculation points window.*

On this tab, you can select what type of objects you want to use as a ζ-calculation point (besides their original purpose). When a certain node type has been selected, it will keep its original purpose, but *in addition it will work as a ζ-calculation point and will show water levels in the simulation results.* Select the node types with the $>$ and $\gg$ buttons.

Do not forget to activate the "Branch" checkbox on the "General" tab. Otherwise this option will not be active!

Important Note: If you select a node type that's usually invisible in the "Results in Maps" mode (for instance the *Flow - Cross Section* node type), you'll have to make it visible first in order to view the results on these nodes.

Tab 2D Grid:

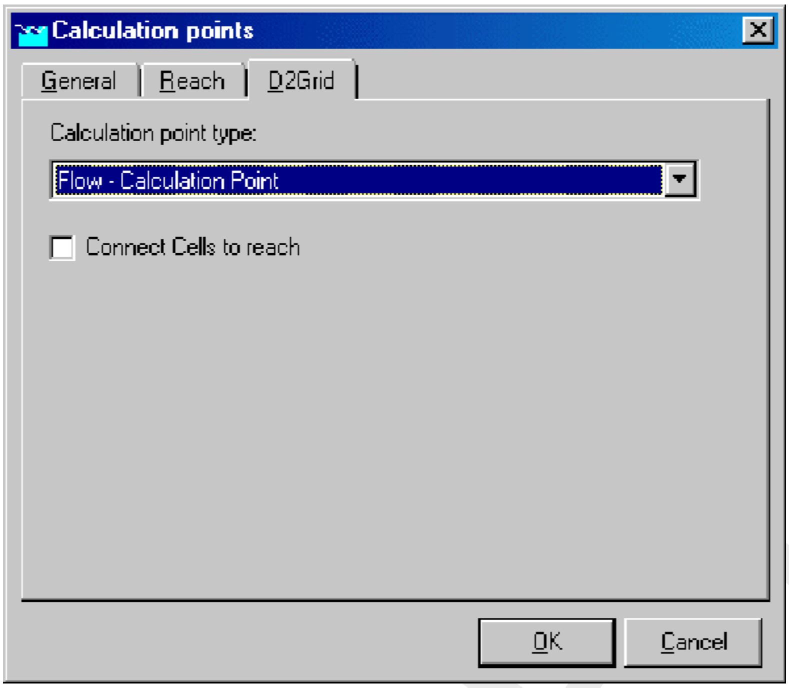


Figure 5.71: *The 2D Grid tab of the Calculation points window.*

This tab is necessary to couple a 1D branch with a 2D grid. When the option "Connect Cells to branch" is activated, for every 2D grid cell that the 1D branch crosses, a ζ-calculation point is created and linked with the ζ-calculation point of that particular cell.

Do not forget to activate the "2DGrid" checkbox on the "General" tab. Otherwise this option will not be active!

Topology

Nodes of the type *Flow - Calculation point* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Unlike structure and cross section nodes, ζ-calculation points are usually not placed in a schematisation one by one. SOBEK provides an option to automatically generate a grid of ζ-calculation points for all branches or for one branch at a time:

◇ Switch to the "edit network" mode in NETTER
◇ Open the "Branch" toolbar and choose one of the following options: 1. click the button <Calculation grid branch> 

and click on the desired branch. 2. click the button <Calculation grid all branches> 

◇ Choose an appropriate grid spacing distance. The best value depends on various things, but to give you an impression: for most channel systems 50 to 100 meters is a good value. For slow rivers that show little changes in bed shape and bed level 500 meters is appropriate. See the chapter: Flow - Calculation Grid input screens from the Functional Reference Manual for a detailed information on the various options.

It is also possible to add calculation points manually. This may be desirable when two structures are located closely to each other. Between two structures or lateral discharges there should **always** be a calculation point.

The image below shows an example of a valid schematisation that contains calculation points. Notice that between the weir and the lateral discharge node at least one calculation point should be present.

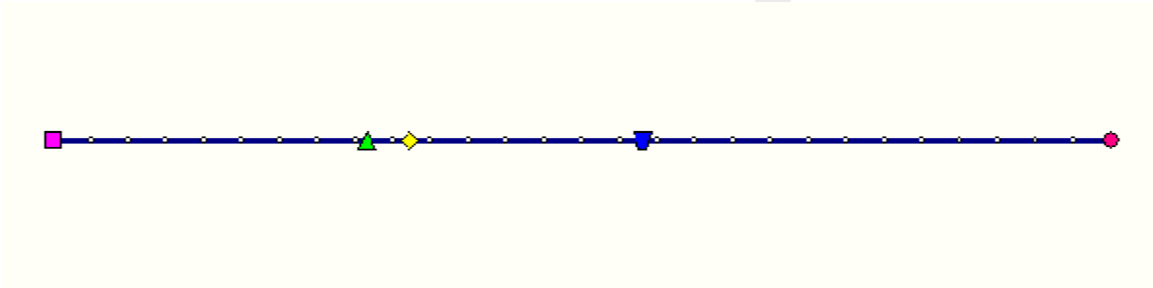


Figure 5.72: *An example of a schematisation that contains calculation points.*

5.3.2.2 Flow - Fixed Calculation point

Description

In this chapter, the *Flow - Fixed Calculation Point* is described.

This type of calculation point is equal to the normal calculation points, except that it will **not** be automatically removed when you decide to re-generate a calculation grid. This might, for example, be a useful feature for locations where you desire to compare measured water levels with computed water levels.

◇ For a detailed description of this node's possible network configurations: see the "topology" section from the Reference Manual

Topology

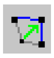
Nodes of the type *Flow - Fixed Calculation Point* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - Fixed Calculation Point* node to your schematisation in the following way:

◇ Select the appropriate node type (Flow - Fixed Calculation Point)

- Use the "add node"  button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node"  button.

Note: Before adding a structure to your schematisation, you should have at least one of the above Channel types present in it.

5.3.3 Flow - Compound Structure

Description

27. Flow - Compound Structure

This chapter describes the Flow - Compound Structure node type. A compound structure can combine two or more different type of structures into one node. It considers these structures to lay parallel to each-other within the same water course.

- For a detailed description of this node's input parameters: see the "Flow - Compound Structure node input screens" section from the Technical Reference Manual;
- For a detailed description of this node's possible network configurations: see the "Flow - Compound Structure node topology" section from the Technical Reference Manual

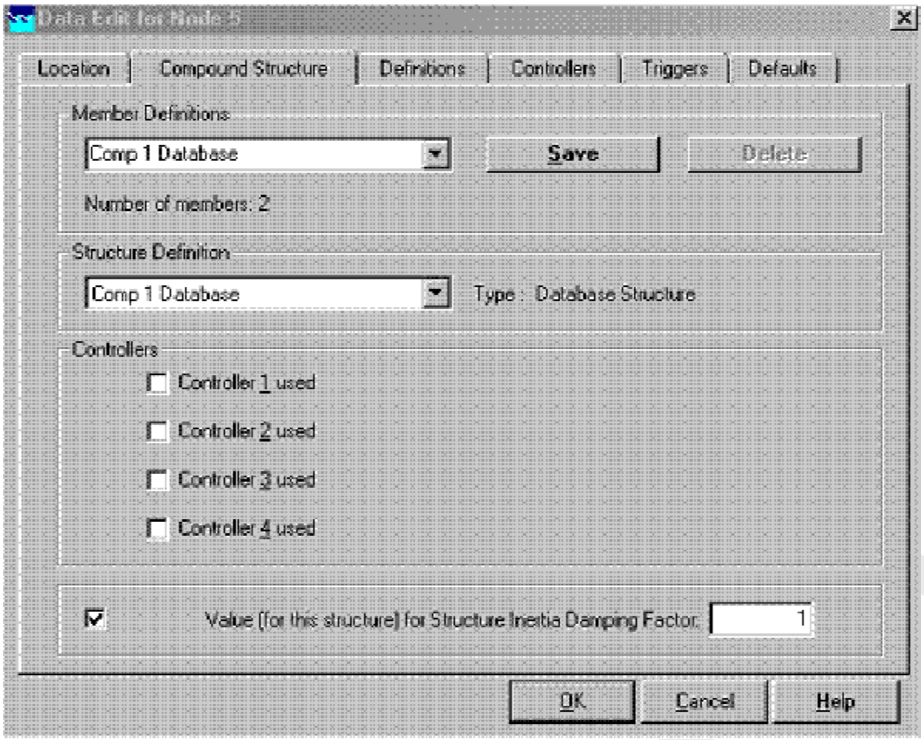
Structure types that can join in a Compound Structure node as a compound member are:

- River Weir (discharge computed on basis of energy-levels)
- Advanced weir (discharge computed on basis of energy levels)
- General Structure (discharge computed on basis of energy-levels)
- River Pump (discharge computed on basis of water levels)
- Database Structure (discharge computed on basis of water levels)

Please note that the above structure types are also available as single structures (respectively: 29. Flow - River Weir, 30. Flow - Advanced Weir, 27. Flow - General Structure, 31. Flow - River Pump; and 28. Flow - Database Structure.

Input screens
When starting the model data editor for an *Flow - Compound Structure* node type, the following tabs will be available for input:

Compound Structure Tab:



- ◇ You can use the default value for structure inertia damping factor as defined in Setting, Tab – numerical parameters, or you can define a value for structure inertia damping factor for each compound structure member separately. For meaning of structure inertia damping factor see Numerical parameter.
- ◇ Under the "Member definitions" section, one can define the structures that become part of the Compound Structure. Type a name of a new member in the field, and click on the <Define> button. For explanation on the other sections of this tab, see the Functional Reference Manual for the coresponding member as single structure.

All other tabs are identical to the tabs as for the coresponding member as single structure. See that chapter in the Functional Reference Manual for a detailed explanation.

Definitions tab:

The definition tab will look differently, depending on the type of structure you are editing. Please refer to the chapters named "input screens" in the "Reference Manual' for the specific structure types that you can edit there:

- ◇ River Weir
- ◇ Advanced weir
- ◇ General Structure
- ◇ River Pump
- ◇ Database Structure

Controllers & Triggers tabs:

see section 5.8.

Topology

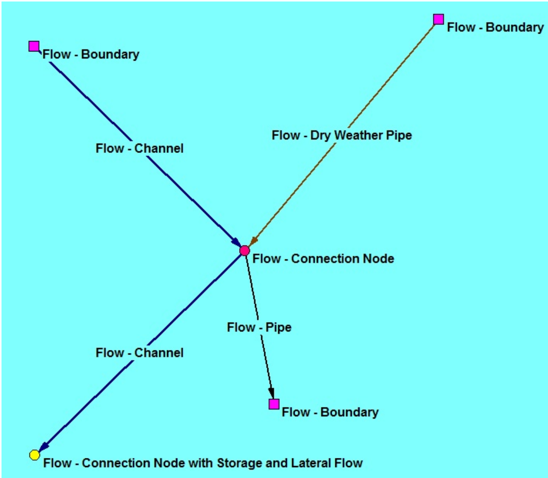


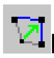
Figure 5.73: *Branches connected to Flow connection nodes*

Nodes of the type *Flow - Compound Structure* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - Compound Structure* node to your schematisation in the following way:

- ◇ Select the appropriate node type (Flow - Compound Structure)
- ◇ Use the "add node"  button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- ◇ If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node"  button.

Note:
1. Before adding a compound structure to your schematisation, you should have at least one of the above Channel types present in it.
2. SOBEK by default places a computational point 0.5 m upstream and 0.5 m downstream of the compound structure location. Hence the compound structure is default located on a branch having a length of 1 m.

5.3.4 Different types of Flow - Connection nodes

The following types of Flow - Connection nodes are available:

- ◇ Flow - Connection node
- ◇ Flow - Connection node with Lateral Flow
- ◇ Flow - Connection node with Storage and Lateral Flow

For each type of connection node yields that one or more branches can be connected to each other (see Figure 5.73). A connected branch might either a pipe or a Flow - Channel.

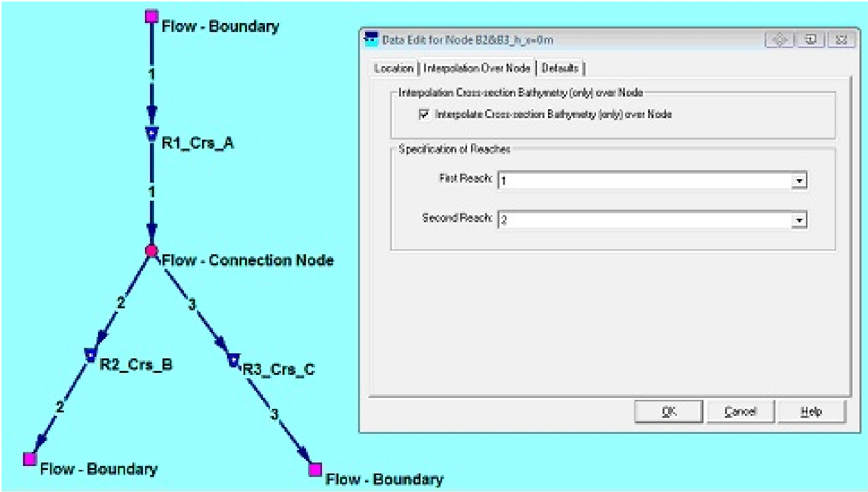


__Figure 5.74:__ Interpolating data over Branch 1 and 2 at a Flow - Connection Node

5.3.4.1 Flow - Connection node

At a *Flow - Connection node* it is possible to interpolate data (see definition and restrictions hereafter) between two adjacent Flow – Channels. For these two Flow - Channels should yield that one ends at (i.e. the so-called First Branch), while the one starts from (i.e. the so-called Second Branch) the concerning "Flow - Connection node" (see Figure 5.74).

A left mouse click on a *Flow - Connection Node* opens the **Data Edit for Node Form**. In a scroll box you can define here the branches for which data is to be interpolated over the Flow - Connection Node.

Definition of data interpolation over a Flow - Connection Node

For the open channel part from cross-section R1_Crs_A to cross-section R2_Crs_B (see Figure 5.74), interpolating data over the Flow - Connection Node means:

- ◇ Bathymetrical data:
  The bathymetry of the cross-section at the *Flow – Connection Node* is obtained by linear interpolation between the nearest cross-section on the "First Branch" (e.g. R1_Crs_A and the nearest cross-section on "Second Branch" (e.g. R2_Crs_B). In hydraulic calculations, this interpolated cross-section is respectively placed at the end of the "First Branch" and at the beginning of the "Second Branch".
- ◇ Friction data
  - ◻ Interpolation of friction data for "Y-Z" and "Asymmetrical Trapezium" profiles ***only***. For Y-Z and Asymmetrical Trapezium profiles yields that friction is defined per cross-section. Firstly, for each cross-section a conveyance table is constructed on basis of its defined roughness and cross-sectional profile. For each h- and u-point lying in between two cross-sections, a tabulated type of cross-sectional profile is obtained by linear interpolation. For each u-point, conveyance tables are obtained by linear interpolating between the conveyance tables available at its adjacent cross-sections.
  - ◻ ***No*** interpolation of friction data for profiles that are not Y-Z and Asymmetrical Trapez-

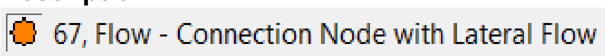
ium profiles. For profiles other than Y-Z and Asymmetrical Trapezium profiles, the friction applied for the interpolated cross-section located at the “First Branch” and the “Second Branch” are respectively obtained from the user-defined friction at the “First Branch” and the “Second Branch”.

◇ Diffusive lateral discharges:
Data with respect to diffusive lateral inflows is ***not*** interpolated over a “Flow - Connection node”.

Restrictions for data interpolation over a Flow - Connection Node:
Table 5.4 in section 5.6.1 provides an overview of the various available different types of cross-sections. Cross-section types that may lay on the same branch are given in Table 5.3. With respect to data interpolation over a Flow - Connection Node yields that interpolation is only allowed between cross-sections that may lay on the same channel branch (see Table 5.3).

5.3.4.2 Flow - Connection node with Lateral Flow

Description



Definition and Restrictions for data interpolation over a *Flow - Connection Node with Lateral Flow*:
A *Flow - Connection Node with Lateral Flow* has the same functionality with respect to interpolating cross-sections as a *Flow - Connection Node*. For more information reference is made to section 5.3.4.1.

A *Flow - Connection Node with Lateral Flow* has the same lateral flow functionalities as a *Flow - Connection Node with Storage and Lateral Flow*. No storage is, however, availabe at a *Flow - Connection Node with Lateral Flow*. For the flow functionalities of a *Flow - Connection Node with Lateral Flow* reference is made to section 5.3.4.3.

5.3.4.3 Flow - Connection node with Storage and Lateral Flow

Description

In this chapter, the *Flow - Connection node with storage and lateral flow* is described.

◇ For a detailed description of this node’s input parameters: see the "Flow - Connection node with Storage and Lateral Flow node input screens" section from the Reference Manual;
◇ For a detailed description of this node’s possible network configurations: see the "Flow - Connection node topology" section from the Reference Manual

Definition and Restrictions for data interpolation over a *Flow - Connection Node with Storage and Lateral Flow*:
A *Flow - Connection Node with Storage and Lateral Flow* has the same functionality with respect to interpolating cross-sections as a *Flow - Connection Node*. For more information reference is made to section 5.3.4.1.

This type of node is very similar to the normal *Flow - Connection* node, apart from the fact that it has some extra features:

◇ It gives the opportunity to add lateral discharge. This option is often used to submit known discharges from industry or smaller rivers that have not been included in the schematisation.
◇ It can bear additional storage area. This option is often used to include the area that is represented by small water courses which have not been included as Flow - Branches in the schematisation. For non-stationary simulations, the storage capacity of such smaller watercourses may be of importance.

Location of Lateral Flow:
The location of the lateral inflow or lateral outflow coincides with the location of the Flow - Connection Node with Storage and Lateral Flow.

Input screens
When starting the model data editor for a *Flow - Connection node with Storage and Lateral Flow* type, the following tabs will be available for input:

Bottom Level:

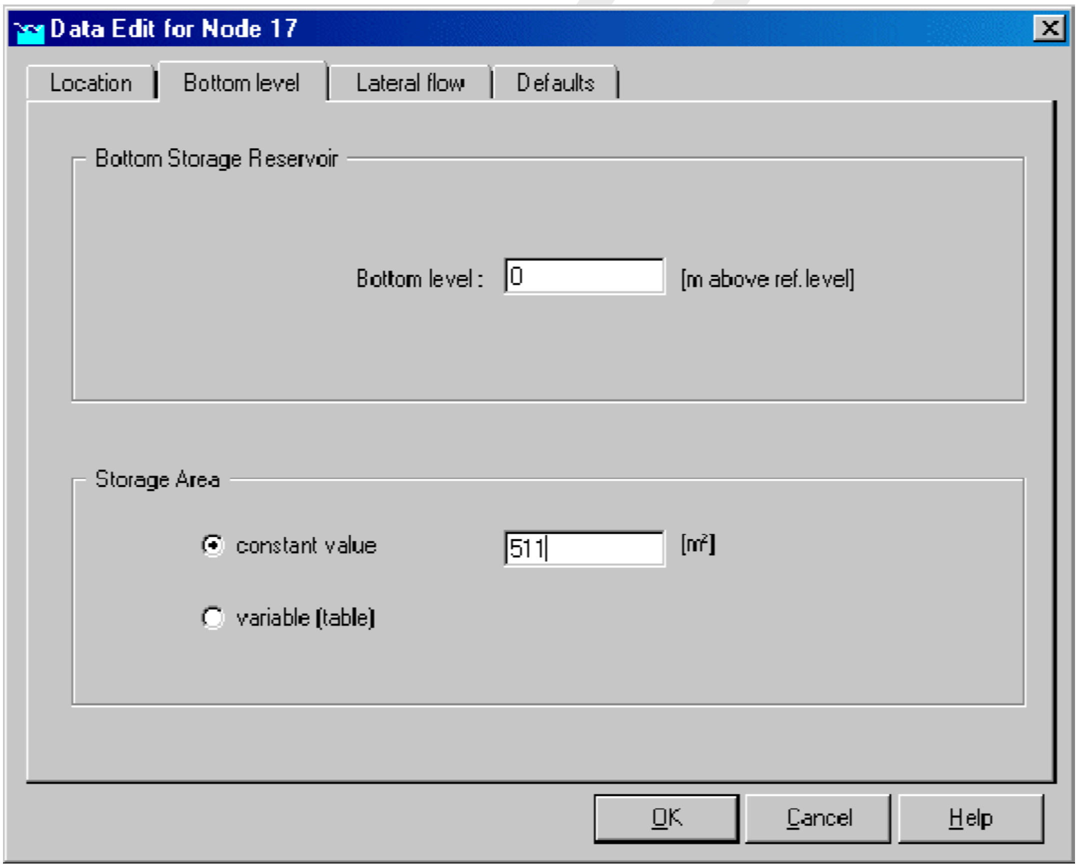


Figure 5.75: *The Bottom level tab of a Flow - Connection node with Storage and Lateral Flow.*

On this tab, the storage capacity of the node should be entered.

◇ **Bed Level:** Represents the level where the additional storage area starts. This may be higher than the bed level of the channel itself. For example: if the storage on this node should represent small watercourses that are connected to the main channel, the bed level of those watercourses should be entered here: not the bed level of the main channel.

◇ **Storage Area:** Here, the storage area for this node should be entered. For example: if the storage capacity of this node should represent a secondary ditch of 200 m long and 2 m wide that has not been schematised as a branch, the area should be 400 m^2.Optionally, the storage area can be entered as a function of level. Use the option "variable" to do so.

Lateral Flow:

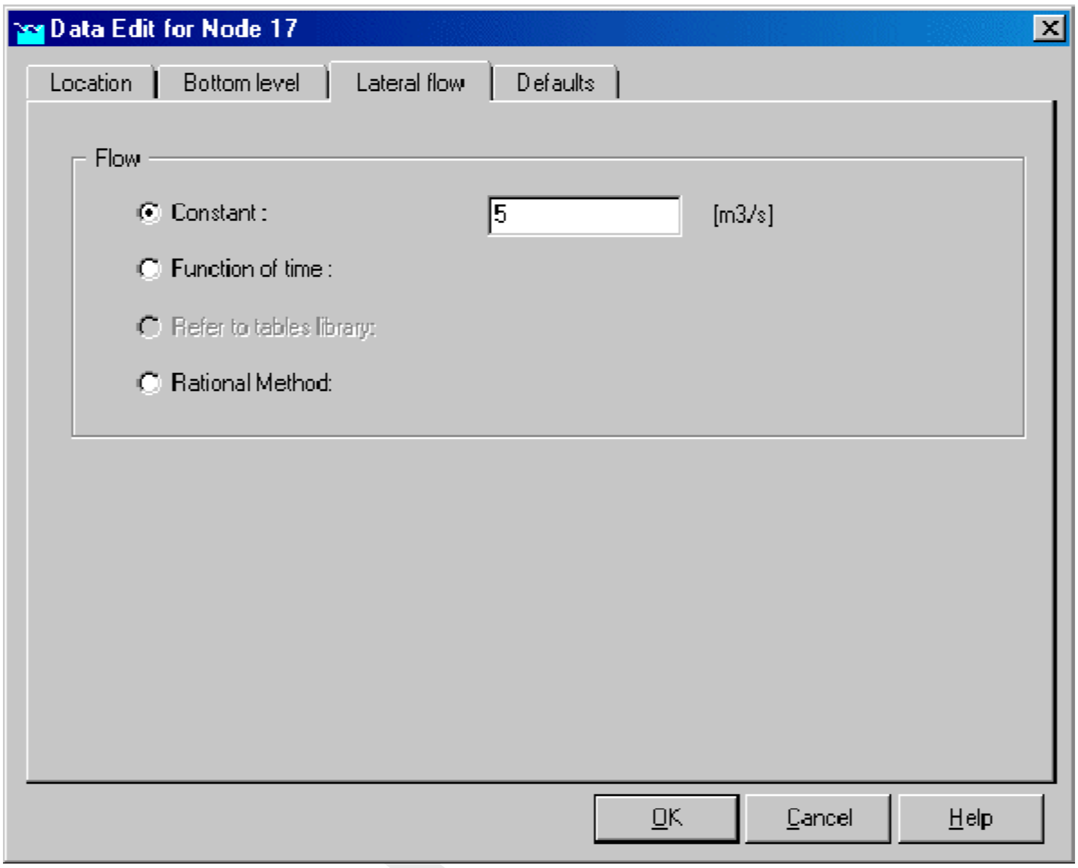


Figure 5.76: *The Lateral flow tab of a Flow - Connection node with Storage and Lateral Flow.*

On this tab, the lateral inflow can be entered. It gives three options:

◇ **Constant:** Use this option if a constant amount of water flows into or out of the model. (plus = in, minus = out)

5.3.5 Flow - Cross Section

Description

In this chapter, the *Flow - Cross Section node* is described. This is one of the most important node types when creating a hydrodynamic model. It defines the dimensions of the channels/rivers that you want to model.

◇ For a detailed description of this node's input parameters: see the "Flow - Cross Section node input screens" section from the Reference Manual;

◇ For a detailed description of this node's possible network configurations: see the "Flow - Cross section node topology" section from the Reference Manual
◇ For a detailed description of this node's underlying mathematical equations and the applied interpolation methods, see the "Cross Section" section from the Technical Reference Manual".

The discharges through a river or channel are greatly determined by the shape of the river bed. To obtain a proper schematisation, it is therefore important to supply it with a sufficient amount of cross section data. This is done by adding nodes of the Flow - Cross Section node type to the schematisation.

From the cross section data, the bed levels and hydraulic radiuses are interpolated towards the calculation points, for which every time step water levels are calculated.

Input screens
When starting the model data editor for an *Flow - Cross Section* node type, the following tabs will be available for input:

Location:

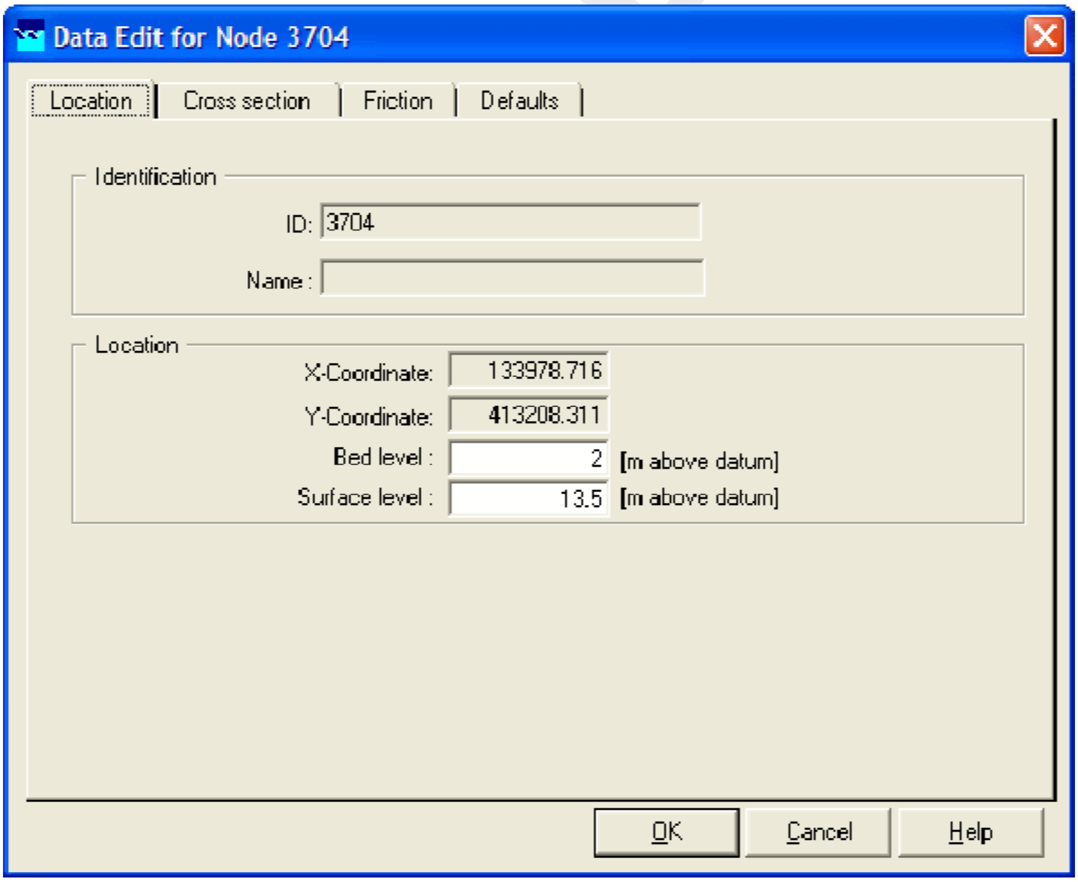


Figure 5.77: *The Location tab of a Flow - Cross Section node type.*

On this tab, you can define the bed level and surface level for the particular cross section.

◇ **Bed level or cross-section level shift:** For some cross-sections a bed level, or lowest point of the cross-section, can be entered. Examples of cross-sections with a bed level are the Trapezium, Round and Egg-Shape types.For other cross-sections, a cross-section level shift can be entered. The defined cross-section level shift is added to all the levels defined in the cross-section definition. Examples of cross-sections with a cross-section level shift are the Y-Z, River Profile and Tabulated types.

◇ **Surface level:** The value you enter here represents the level of the embankments at this cross section. This value will become visible in the results as follows: — in Sideview as the green line — in results at nodes it will be used to calculate the freeboard. **Note:** In the hydrodynamics calculations the value that you fill in for surface level will **not** be used.

Cross Section:

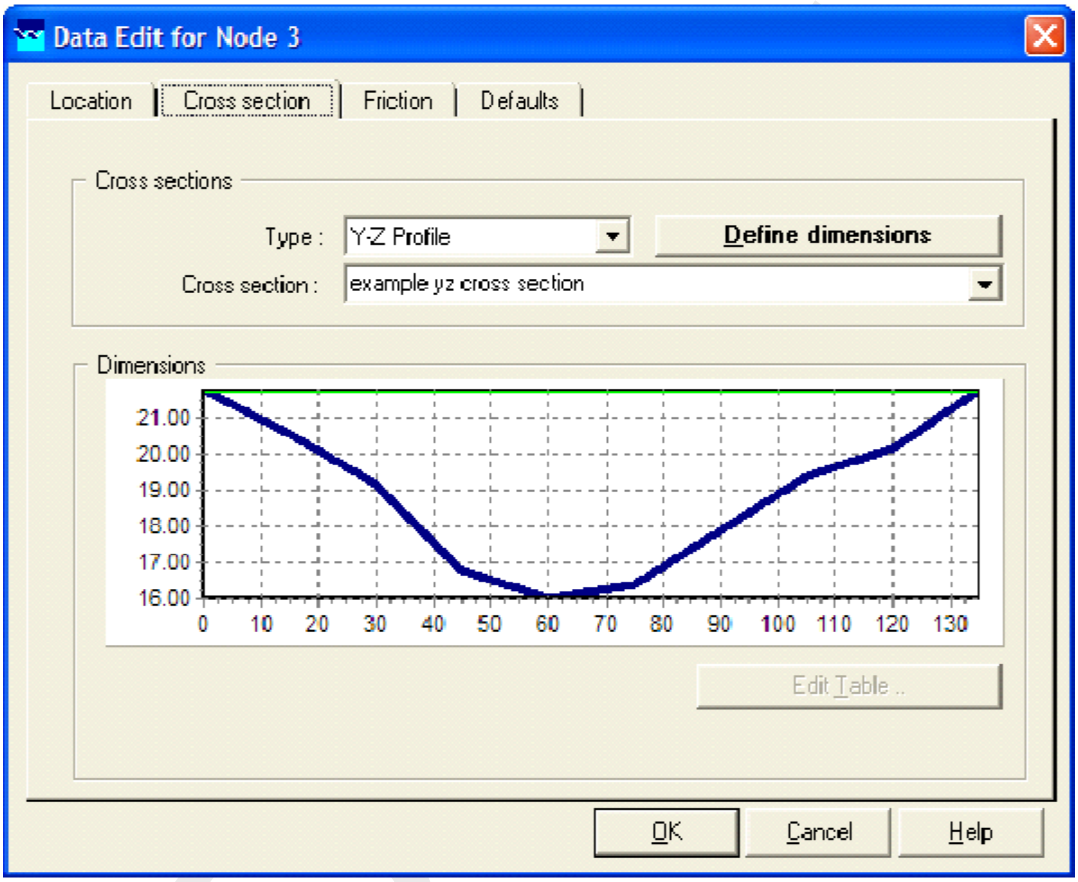


Figure 5.78: *The Cross section tab of a Flow - Cross Section node of the Y-Z Profile type.*

On this tab, you should define the shape of the cross section. Notice you should first type a name for the cross section shape that you want to define. Then later this shape can be **re-used** on other *Flow - Cross Section* nodes! This prevents double work.

◇ **Select Cross Section:** If you had already created several cross section definitions for the chosen type, you can choose one here. If not, you can define a cross section shape by typing a name in this field and clicking the <Define Dimensions> button.Dependent on the cross section type you chose, various buttons will become active where you can define the shape of the cross section. For each type of cross section, a separate chapter is available in the "Functional Reference Manual" - "Channel Flow module" - "Flow - Cross section node" section: - Trapezium type- Round type- Egg-shape type- Tabulated type- Rectangle type- Y-Z profile type- Trapezium type- Asymmetrical trapezium type- Elliptical type- Arch type- Cunette type- Steel cunette type

Friction:
Three different friction input screens are available when using the model data editor.

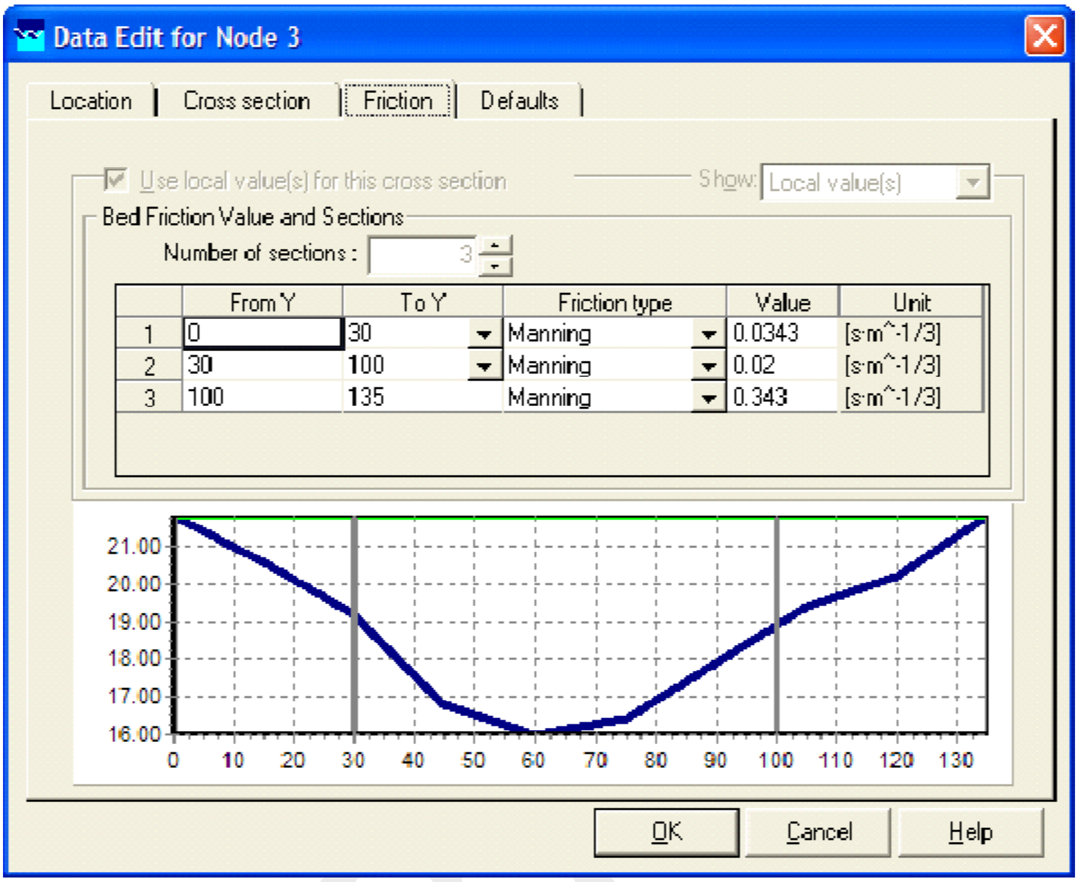


Figure 5.79: Y-Z and Asymmetrical Trapezium friction input screen.

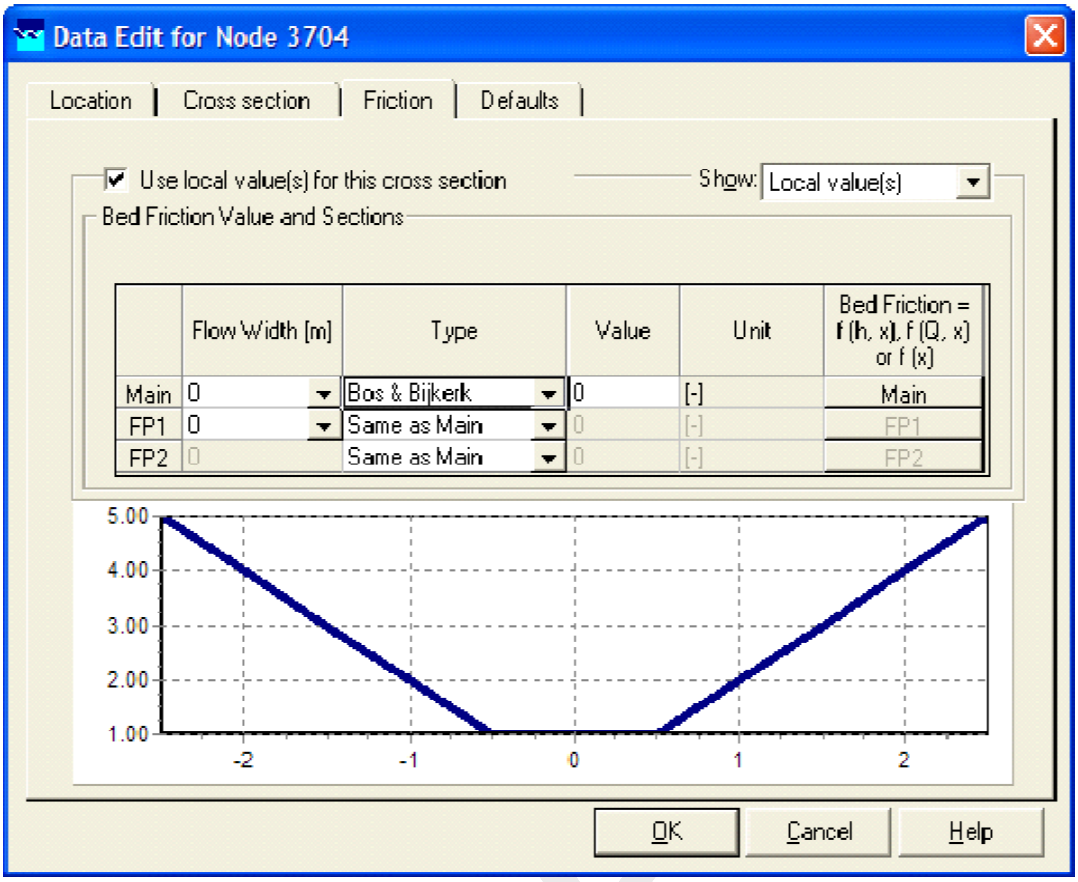


__Figure 5.80:__ River profile friction input screen.

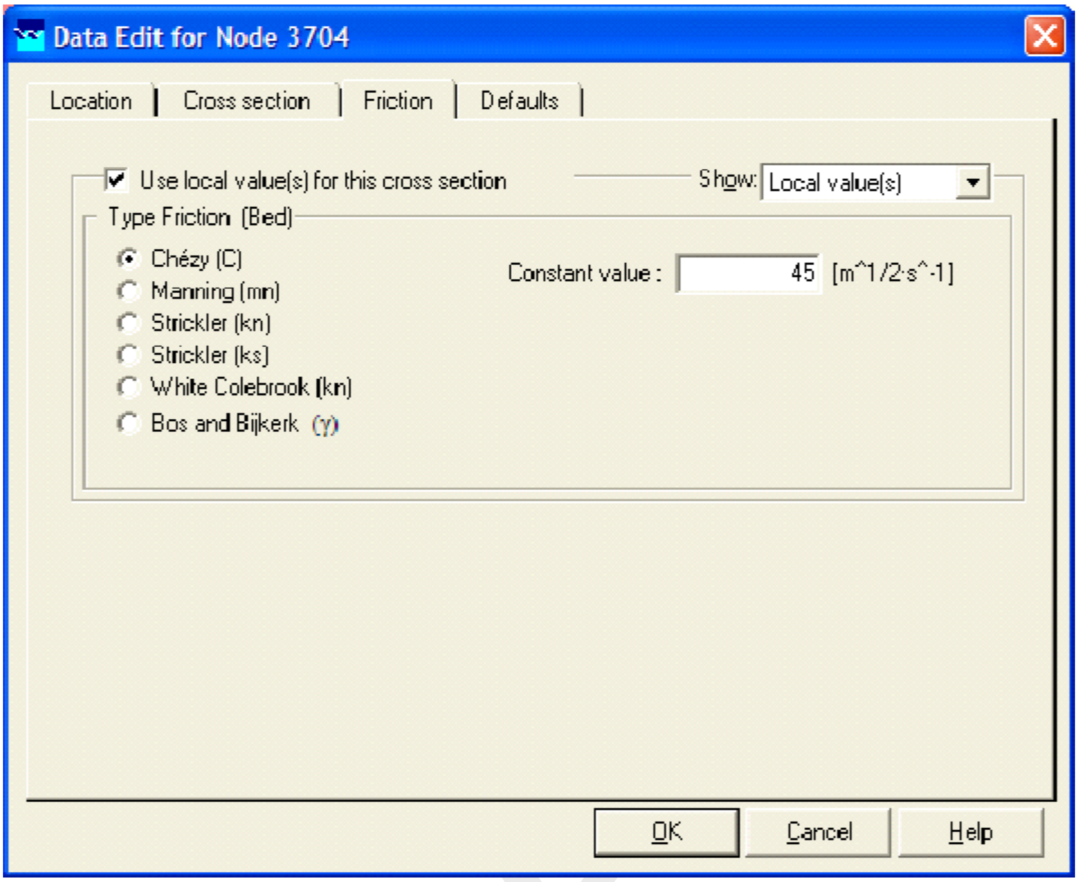


Figure 5.81: *Friction input screen for the remaining profile types.*

For all cross-sections except Y-Z and Asymmetrical Trapezium profiles, the friction value that you define will count for the entire branch on which it is located. For every branch, therefore only the friction value of **one** of its cross sections needs to be edited.

For the Y-Z and Asymmetrical Trapezium profiles it is possible to define different friction values per cross-section lying on a branch. For river profiles it is possible to define friction as $f(x)$, $f(h,x)$ and $f(Q,x)$. More information is available in the chapter 1D hydraulic friction concepts.

- **Use local value for this cross section** You have the opportunity to choose between a **local** friction value or a **global** friction value for the cross section. Global means that you have a global (or model-wide)friction value that will apply to all branches for which you did *not* define "use local friction values for this cross-section". To edit and apply the **global** value, do the following:

  - switch **off** the "use local value" checkbox
  - select in the "Show:" combo box "Global value(s)".
  - Change the friction value according to your wishes.

  Note that if you change the global value, the friction value for all branches that refer to this value is changed too!To edit and apply the **local** value, do the following:

  - switch **on** the "use local value" checkbox
  - select in the "Show" combo box "Local value(s)"
  - Change the friction value according to your wishes.

- **Type friction:** Various friction formulas are available. Choose the one of your wishes.

More information about the various friction types is available in the "Bed Friction" section from the Technical Reference Manual.

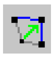
Topology

Nodes of the type *Flow - Cross Section* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - Cross Section* node to your schematisation in the following way:

- Select the appropriate node type (Flow Cross Section)
- Use the "add node"  button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node"  button.

Note: Before adding a cross section to your schematisation, you should have at least one of the above Channel types present in it.

Example of a valid network configuration containing a cross section:

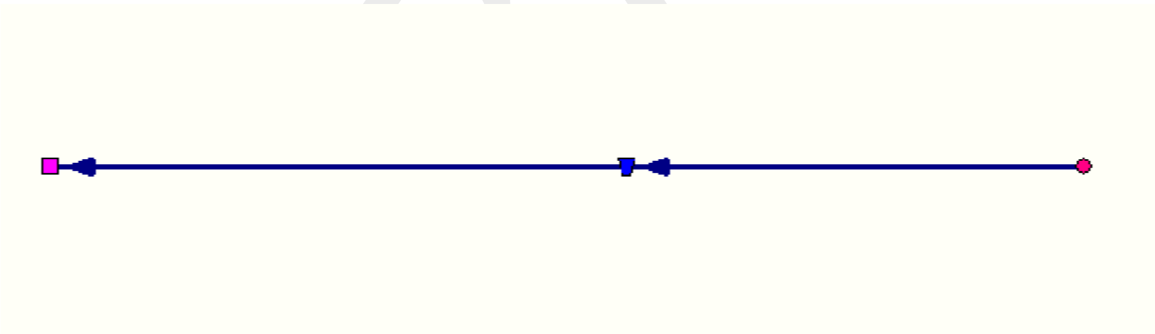


Figure 5.82: *Example of a valid network configuration containing a cross section.*

5.3.6 Flow - Culvert node

Description

In this chapter, the *Flow - Culvert node* is described.

- ◇ For a detailed description of this node's input parameters: see the "Flow - Culvert node input screens" section from the Reference Manual;
- ◇ For a detailed description of this node's possible network configurations: see the "Flow - Culvert node topology" section from the Reference Manual
- ◇ For a detailed description of this node's underlying mathematical equations: see the "Culvert section from the Technical Reference Manual".

Definition of a culvert:
A culvert is a sleeve-shaped structure (with usually a freely moving water level) which connects **two** water courses with each other (Dutch Hydrological Society, 2002).

In SOBEK, a culvert can be modelled by a *Flow - Culvert* node. For this node, a cross section, bed levels on both sides, length and some other parameters should be defined.

Note: It is best to use this node type only for rather short culverts (up to several tens of meters). If you would model a long culvert with this node type, the amount of storage in your water system would be over-estimated. The reason is that adding a *Flow - Culvert* node does not automatically reduce the storage capacity on the branch according to the culvert's length. The branch will still assume that its cross sections remain valid over the entire branch; which is not the case because you just added a long culvert. So in case of a long culvert, where the storage capacity of the channel/river is an important factor, it is best to simulate it by creating a branch of its length and applying cross section to it that's equal to the culvert's dimensions.

Input screens
When starting the model data editor for an *Flow - Culvert* node type, the following tabs will be available for input:

Culvert:

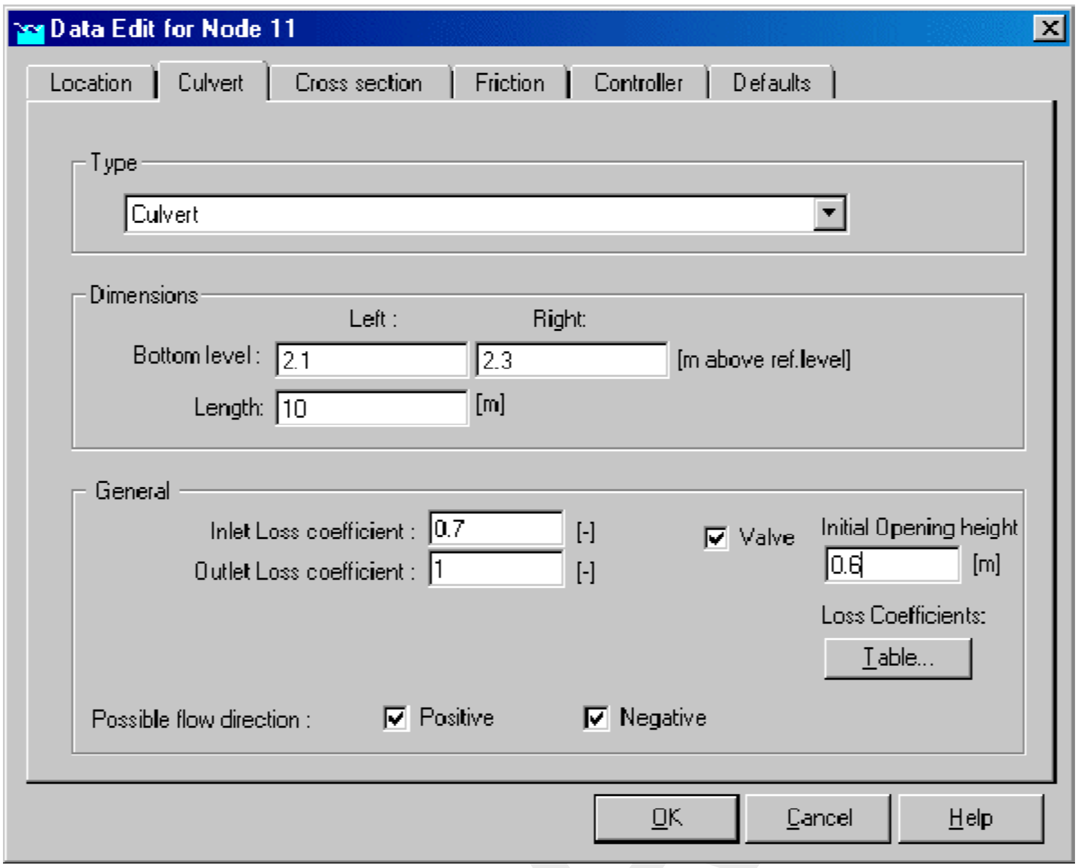


__Figure 5.83:__ The Culvert tab of a Flow - Culvert node.

On this tab, the general dimensions of the culvert can be defined:

- **Type:** Here you can choose what kind of culvert it will be:- Culvert- Syphon- Inverted Syphon
- **Bed level:** These parameters define the bed levels of both sides (upstream and downstream) of the culvert.Left means: the "upstream" side with relation to the defined branch direction.Right means: the "downstream" side with relation to the defined branch direction.
- **Length:** This parameter defines the length of the culvert. See the "Culvert" section from the Technical Reference Manual for more details on the application of this parameter within the structure equations.
- **Inlet loss coefficient:** This parameter represents the energy loss due to the contraction of the water when it has to enter the culvert. See the "Culvert" section from the Technical Reference Manual for more details on the application of this parameter within the structure equations.
- **Outlet loss coefficient:** This parameter represents the energy loss of water when it leaves the culvert. See the "Culvert" section from the Technical Reference Manual for more details on the application of this parameter within the structure equations.
- **Valve:** If the culvert also includes an internal valve, this option should be switched on. Then also the tab "Controller" will become active. The valve should have an initial opening height for the first time step of the simulation. For every next time step, the defined controller will operate the valve. In the table for "Loss coefficients", an energy loss factor has to be defined for various opening heights.
- **Possible flow direction:** This parameter defines in which direction(s) water can flow through the orifice. The directions are defined with relation to the defined branch direction. Positive means: in the same direction as the defined branch direction; negative

means: in the opposite direction than the defined branch direction. If you don't know the defined direction of the branch, read the Flow - Branch topology section of the Functional Reference Manual.

Cross Section:

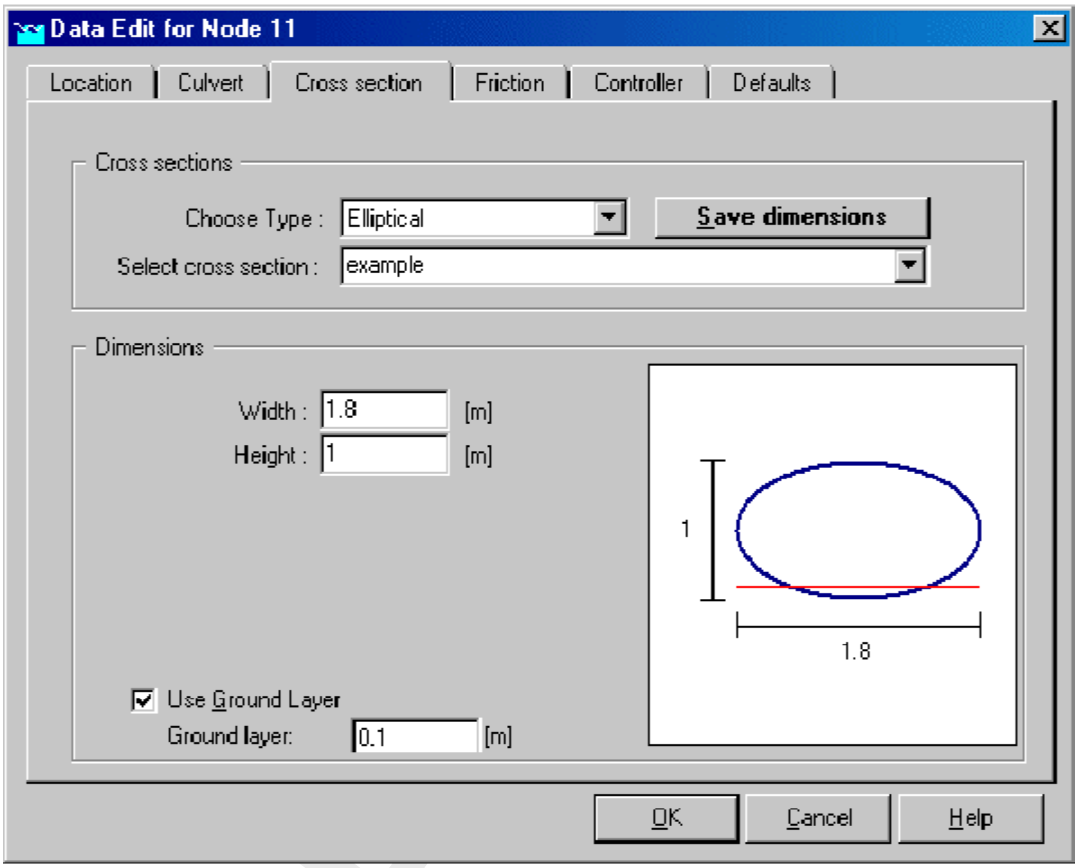


Figure 5.84: *The Cross section tab of a Flow - Culvert node.*

On this tab the cross sectional shape of the culvert should be defined. This is done by means of **cross section definitions**, very similar to the ones you define for the shape of a river bed. The only real difference is that the number of cross section types is limited to eight:

- Round
- Egg-shape
- Table-form
- Rectangle
- Elliptical
- Arch
- Cunette
- Steel cunette

For a detailed description of all available cross section types and their input parameter, see the "Functional Reference Manual" - "Channel Flow module" - "Cross section types" section.

Friction:

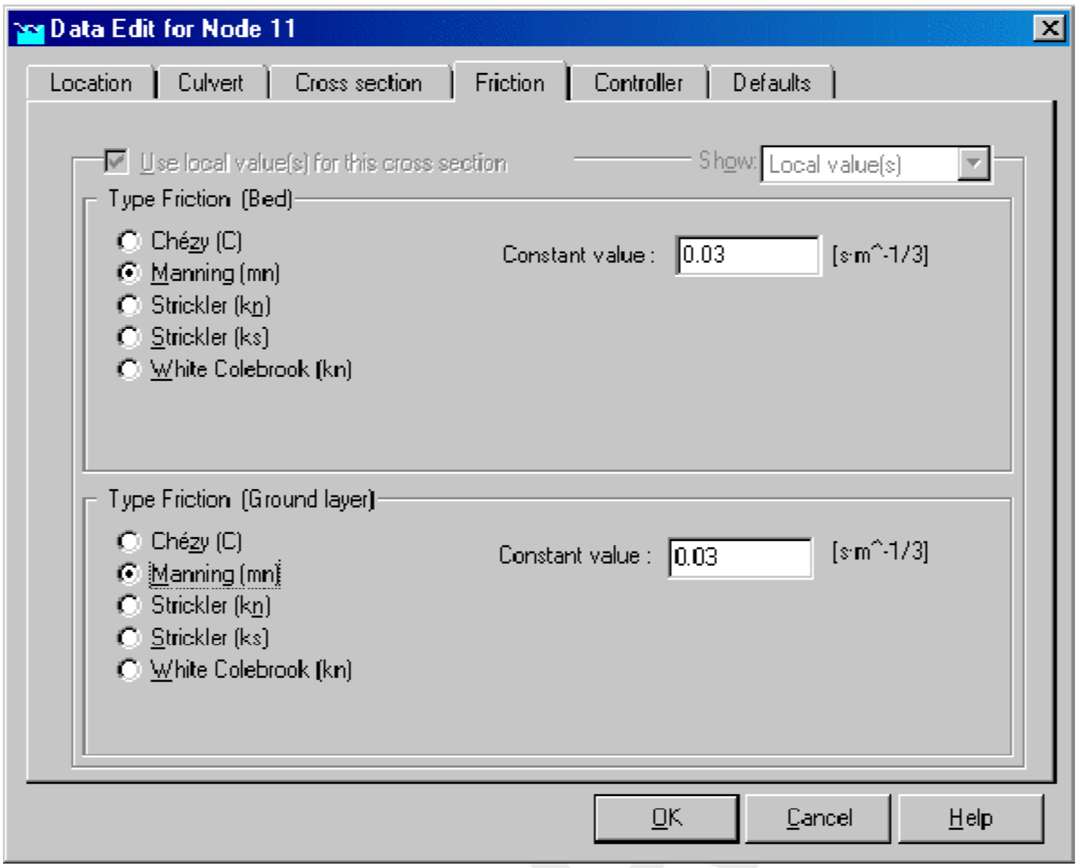


Figure 5.85: *The Friction tab of a Flow - Culvert node.*

On this tab, the friction values for the culvert can be entered:

- ◇ **Type friction (bed):**This friction value represents the roughness of the culvert's walls. Thus, if the culvert is made of concrete, a proper hydraulic roughness value for concrete should be entered.
- ◇ **Type friction (ground layer):**If the option "use ground layer" had been activated on the "cross section" tab, this option becomes active. It represents the roughness value of the ground layer that lays on the bottom of the culvert.

For more information about the various friction options, see the "Bed Friction" section from the Technical Reference Manual.

Controller:

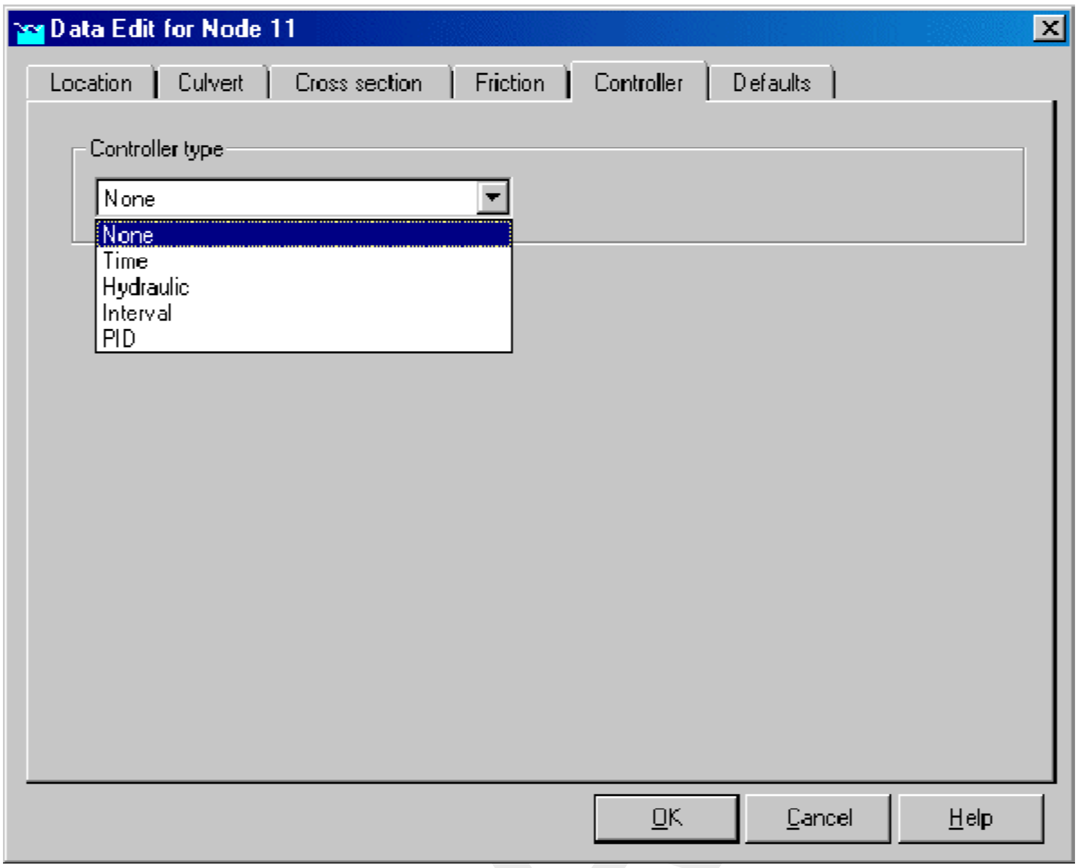
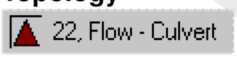


Figure 5.86: *The Controller tab of a Flow - Culvert node.*

This option is only activated when the "valve" option on the "Culvert" tab has been switched on. The valve can be operated by a controller. A controller will overrule your general settings for the valve's **opening height** and apply the chosen controlling rules to it. Read more about them in the Controller section of the Technical Reference Manual.

Note that three of the four available controller options only become available after you have defined a Flow - Measurement station within the schematisation and saved that schematisation.

Topology



Nodes of the type *Flow - Culvert* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - Culvert* node to your schematisation in the following way:

- ◇ Select the appropriate node type (Flow Culvert)
- ◇ Use the "add node"  button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.

◇ If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node" button.

Note: Before adding a culvert to your schematisation, you should have at least one of the above Channel types present.

5.3.7 Flow - Database structure

Description

World-wide various different types of weirs have been constructed in rivers and channels. Although the most common types of weirs are available in SOBEK, definitely not all type of weirs (e.g. crump weir) are available in SOBEK. For this reason the Flow-Database structure was implemented in SOBEK.

Crest levels of a *Flow-Database structure* cannot vary in height along the weir. The discharge over any structure can be defined as a relationship between upstream and downstream water level. The Database structure allows you to define this relationship is in a tabulated form (e.g. database). For more information on how discharges are computed from a user-defined database, reference is made to section 6.1.16.5.

Although not advised (see section 6.1.16.5), the crest level of a Database structure can be varied by a controller. A structure inertia damping factor (see Numerical parameters) can be defined for each Database structure.

Input screens

When starting the model data editor for a *Flow - Database structure node* type, the following tabs will be available for input:

Database structure tab:

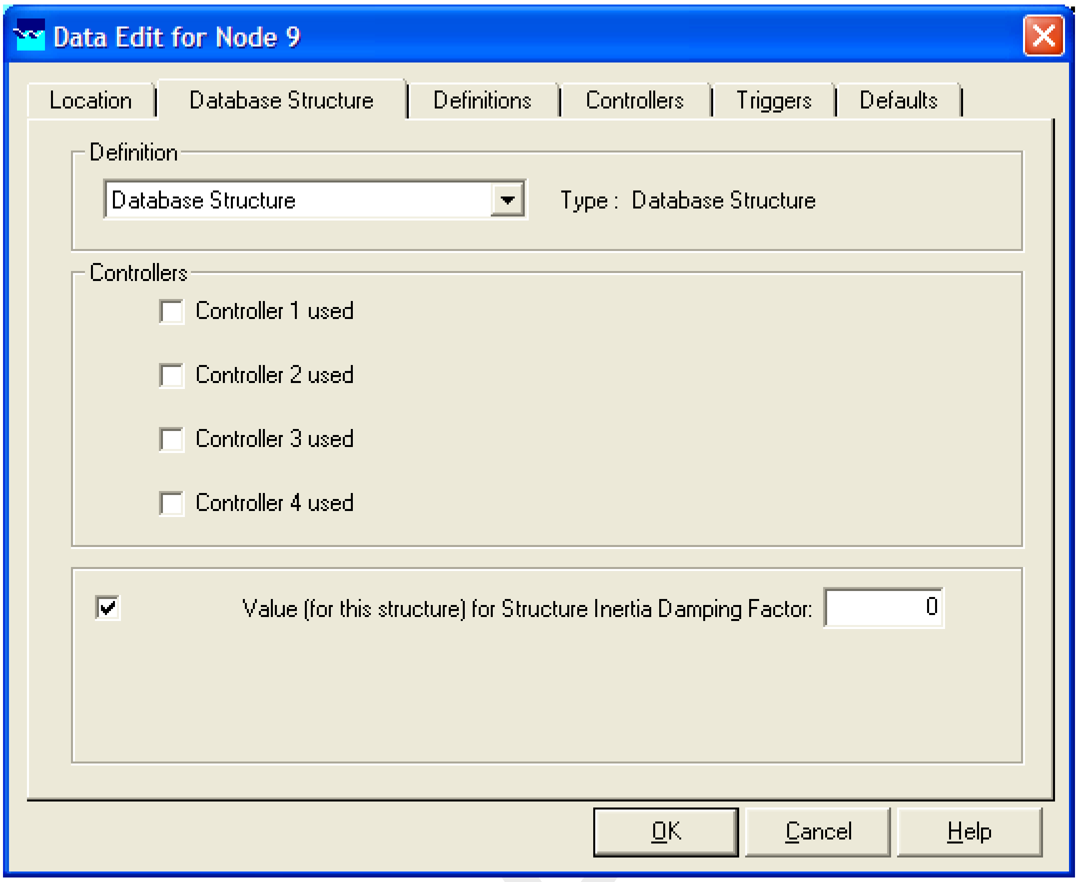


Figure 5.87: *The Database Structure tab of a Flow - Database structure node.*

On this tab, you can select a previously defined *structure definition*. If there is no definition available yet, go to the "definitions" tab first, to create one.

Also, you can select the controllers that apply to this structure. If there are no controllers available to be selected yet, go to the "controller" tab first to create them.

You can use the default value for structure inertia damping factor as defined in Setting, Tab – numerical parameters, or you can define a value for structure inertia damping factor for this particular Database structure. For meaning of structure inertia damping factor see Numerical parameters.

Definitions tab:

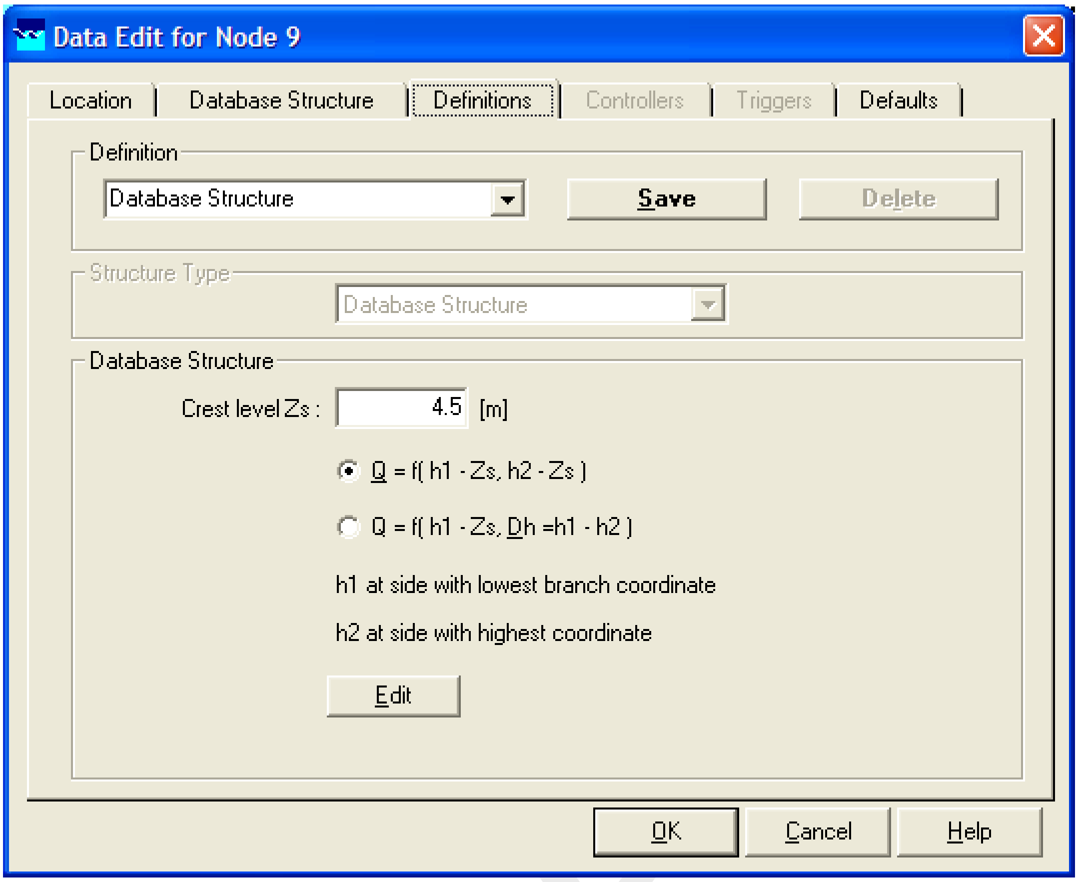


Figure 5.88: *The Definitions tab of a Flow - Database structure node.*

On this tab, you can define the properties of your database structure. Select an existing definition from the drop-down box or create one by typing an appropriate name in the "definition" field and then clicking <Define>.

Then you can:

- ◇ Define the crest level of the structure
- ◇ Choose whether the discharge is a function of:
  - difference between upstream water level & crest level and- difference between downstream water level & crest level or
  - difference between upstream water level & crest level and- difference between upstream & downstream water level
- ◇ Create a table which contains the data for the type of function you chose. Click the *Edit* button to open the "Structure database window"

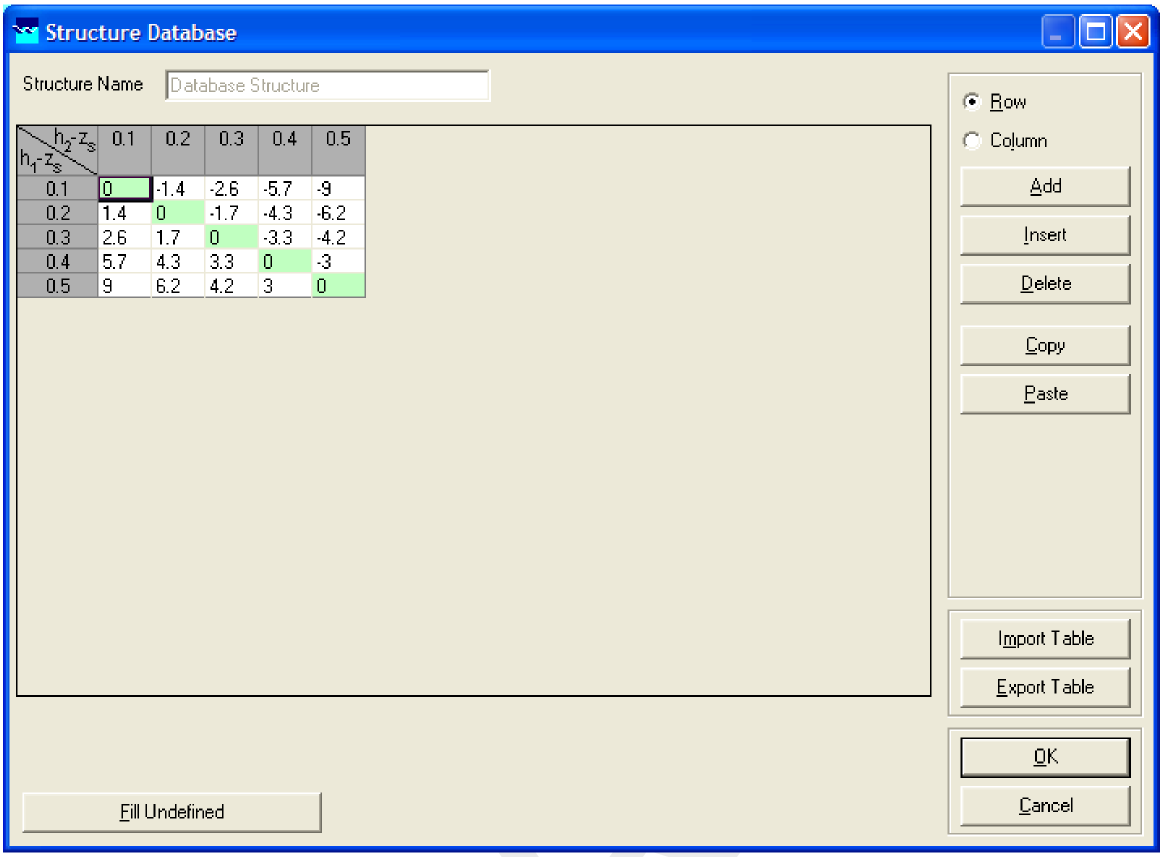


Figure 5.89: *The Structure Database window of a Flow - Database structure node.*

You can add rows or columns by activating the "Row" or "Column" radio button respectively and subsequently clicking $<$Add$>$. First you will be asked how many rows/columns you want to add, then you will be asked to enter the row/column titles (values) one by one.Note that the title (value) of the first row and column should be equal!The table can be checked by clicking the $<$Fill Undefined$>$ button. If everything is fine, the *OK* button will become active.

Controllers & Triggers tabs: see section 5.8.

5.3.8 Flow - Extra Resistance

Description

Placing a *Flow - Extra Resistance Node* on a branch segment, means that an additional term (describing the influence of the Extra Resistance) is added to the momentum equation, which is solved for this specific branch segment (see section 6.1.1.2). More precisely, it means that an additional water level difference is invoked over this branch segment equal to $\xi Q|Q|$, where ξ is the Extra Resistance coefficient and Q the discharge flowing through the branch segment. The additional water level difference is added to the upstream located water level point.

Note: In case of an Extra Resistance Node an additional term is added to the momentum equation. For a structure located on a branch segment yields that the momentum equation is replaced by the structure equation. Hence, an Extra Resistance can not be considered to be a structure. Nevertheless, in the task blocks "Results in Maps" and "Results in Charts", the output of an Extra Resistance Node is provided under "Results at Structures and Extra Resistances"'.

Following restrictions yields for *Flow - Extra Resistance Nodes*:

- *Flow - Extra Resistance Nodes* are only allowed on a "Flow - Channel" and a "Flow - Channel with lateral discharge".
- Only 1 (one) *Flow - Extra Resistance Node* is allowed per branch segment
- A *Flow - Extra Resistance Node* may not be located on a "Flow - Calculation point", a "Flow - Fixed Calculation Point", a "Flow - Connection Node" and a "Flow - Boundary Node.
- A *Flow - Extra Resistance Node* may not be located on a branch segment adjacing a "Flow - Boundary Node".
- A *Flow - Extra Resistance Node* may not be located on a branch segment on which a structure is located".

Input screens

Selecting a *Flow - Extra Resistance Node* and left mouse click opens the Extra Resistance input screen (see Figure 5.90). Clicking on the <Edit KSI Table> button opens a table, where Extra resistance coefficients (ξ) can be defined as function of water level (see Figure 5.91)

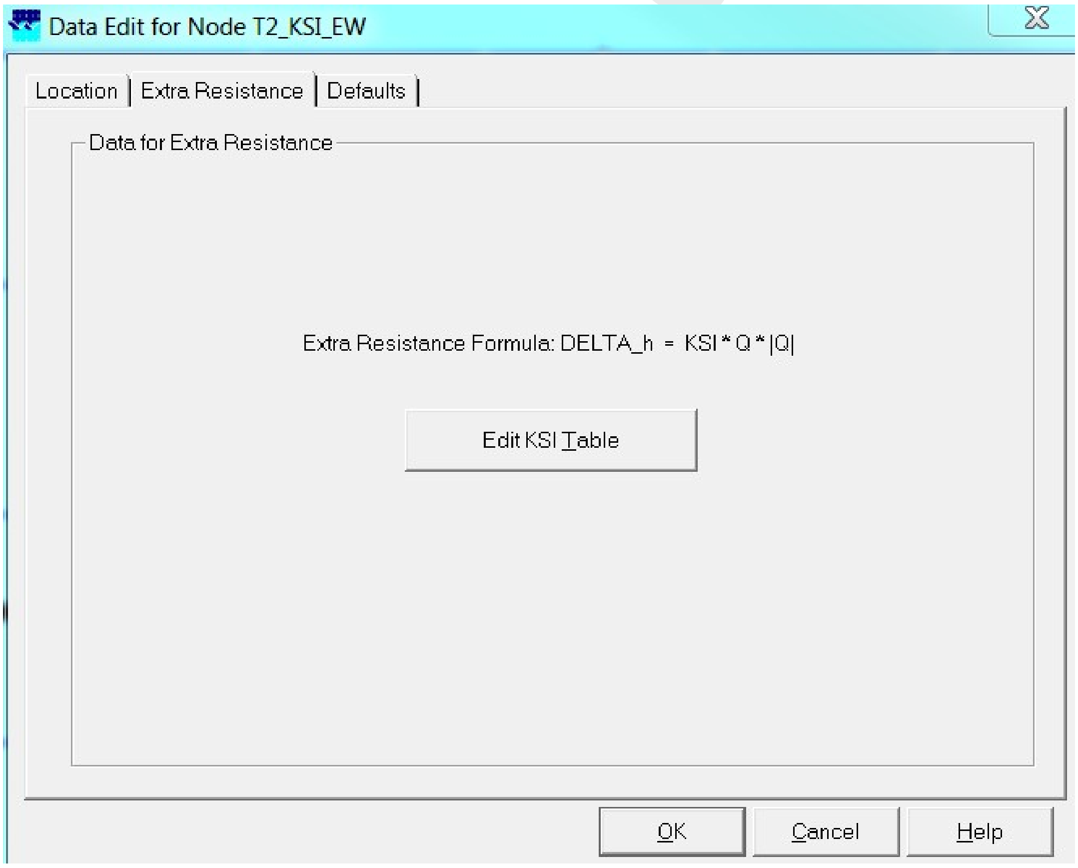
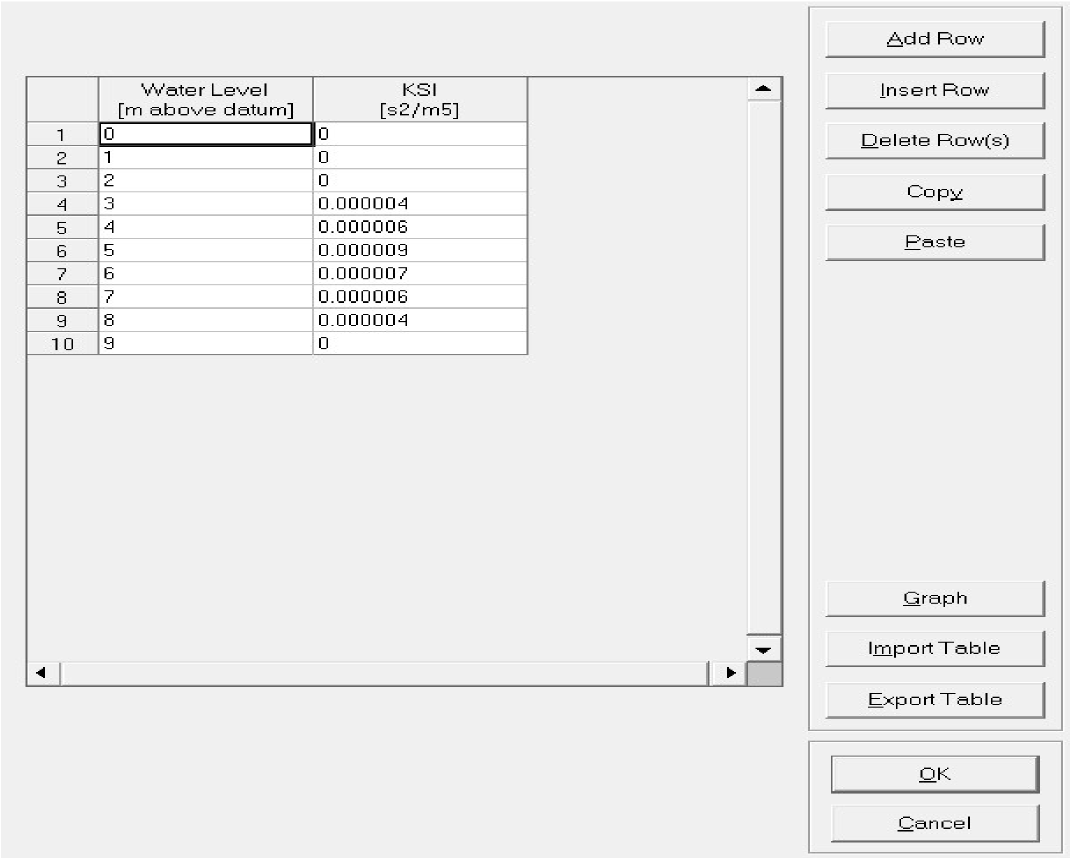


Figure 5.90: *The Extra Resistance tab of a Flow - Extra Resistance node.*



	Water Level [m above datum]	KSI [s2/m5]
1	0	0
2	1	0
3	2	0
4	3	0.000004
5	4	0.000006
6	5	0.000009
7	6	0.000007
8	7	0.000006
9	8	0.000004
10	9	0

Figure 5.91: *Input Table for Extra Resistance coefficients of a Flow - Extra Resistance node.*

5.3.9 Flow - Boundary

Description

In this chapter, the *Flow - Boundary node* is described.

◇ For a detailed description of this node's input parameters: see the "Flow - Boundary node input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the "Flow - Boundary node topology" section from the Reference Manual
◇ For a detailed description of this node's underlying mathematical equations: see the "Boundary section from the Technical Reference Manual".

Nodes of this type mark the outer edges of a hydraulic schematisation. In short, these nodes represent the geographical locations where a model has been *cut out of the real world*. Therefore their interaction with the world outside the model should be applied in terms of hydraulic boundary conditions. The following options are available:

◇ Flow boundary condition: the user submits a constant or alternating discharge that flows into or out of the model (positive values mean in, negative mean out). Example: the upstream edge of a river model where the model receives water from the upstream river branches should be given such a condition;
◇ Water Level boundary condition: the user submits a constant or alternating water level at

the edge of the model's area. Example: a river's mouth, where tidal movement forms the downstream boundary condition for the model;

◇ A Q-H boundary condition: the user submits a relationship between water level and discharge. This type of condition is commonly used on the downstream edge of a river model.

Note:

1 one should always place his or her boundary nodes at a sufficient distance from the area that is to be modelled. The flow conditions that you apply to artificial boundary conditions should never be allowed to influence the hydraulic results in the area of interest!

2 In case more than one branch is connected to a 1D h-boundary node, only a water level boundary can be defined at this particular boundary condition. Further on the user-defined water level boundary condition is applied to each and every branch that is connected to this particular water level boundary.

3 Commonly, nodes of the *Flow - Connection node* type are also used as boundary nodes for hydraulic schematisation. This too is allowed, as these nodes are similar to Flow - Boundary nodes with a Q = 0 boundary condition.

Input screens

When starting the model data editor for a *Flow - Boundary node*, the following tabs will be available for input:

The Boundary condition tab:

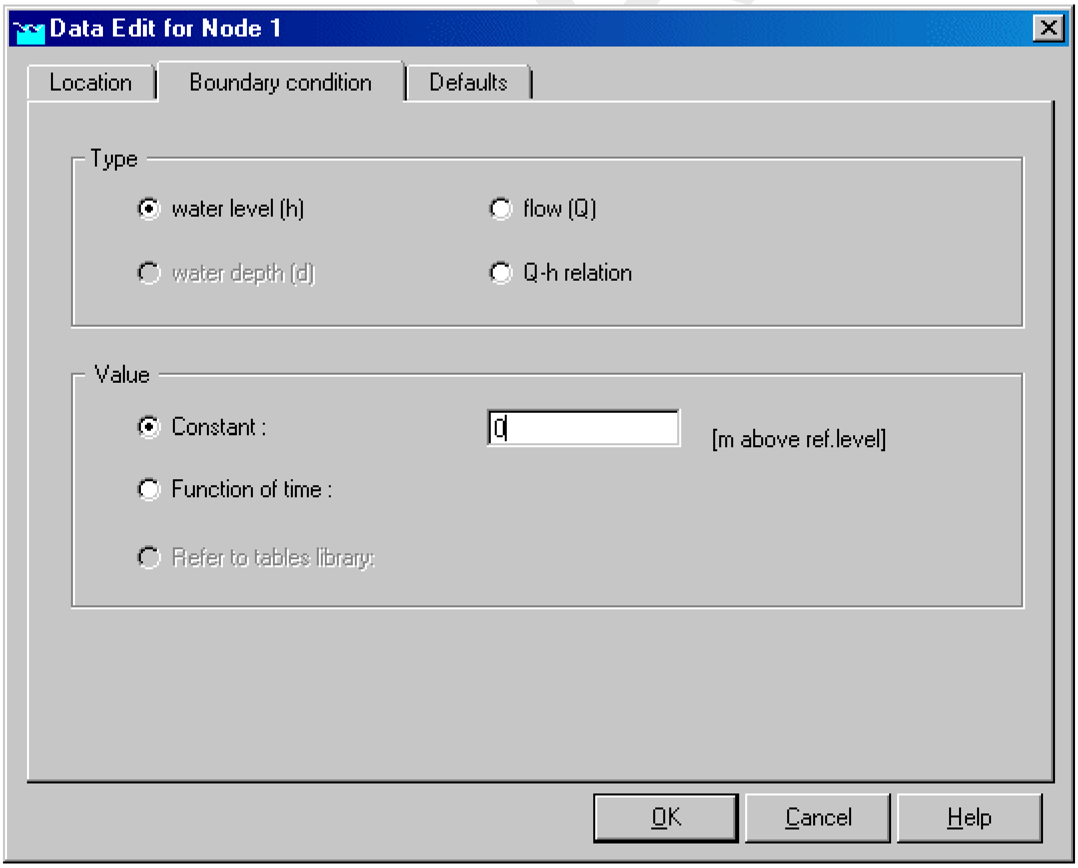


***Figure 5.92:** The Boundary condition tab of a Flow - Boundary node.*

On this tab, the following data can be entered:

◇ **A fixed or alternating water level as boundary condition** By choosing the type "water level (h)", the user can apply a fixed or alternating water level as a hydraulic boundary condition to this node. This type of boundary condition is commonly used on the *downstream* edges of river models that end in a lake or sea. The option "Function of time" can be used when the water levels at the downstream boundary alter in time (i.e. tidal waves) More information about the tables for alternating boundary conditions can be found in the chapter section 5.13.4..

◇ **A fixed or alternating discharge as boundary condition** By choosing the type "flow (Q)" the user can apply a fixed or alternating discharge as a hydraulic boundary condition to this node. This type of boundary condition is commonly used on *upstream* edges of river models, as these receive a certain discharge from upstream branches.
Note: although it is also possible to apply a discharge-boundary to a *downstream* node, we advice you **NOT** to do so.
The reason is rather simple: in reality the discharge through a downstream boundary is affected by the water levels in the area of interest. By enforcing a downstream discharge, this relationship is discarded, thus leading to fake results. On downstream nodes therefore we recommend the usage of "water level" or "Q-H" boundary conditions.
More information about the tables for alternating boundary conditions can be found in the chapter section 5.13.4..

◇ **A Q-H relation as boundary condition** By choosing the type "Q-H relation", the user can define how the water levels and discharge at the boundary are related. This type of boundary condition is commonly used for downstream boundaries in river models, when the downstream boundary node lies somewhere in the middle of a river.
The Q-H relationship consists of a table that relates water levels and discharges to eachother. It is not easy to determine a proper Q-H relation for a river. Globally, there are two ways to determine it: — measure it in the river — estimate it by calculating the hydraulic radius at various water levels and applying the Chézy formula.

Topology

Nodes of the *Flow - Boundary* node type serve as the end- or starting point for a *Flow - Branch* or a *Sewer Pipe*. Do not place nodes of this type just as loose objects in the schematisation or on top of an existing branch. Flow Boundary nodes *should always be part of a branch themselves*. A correct working method is:

◇ Switch to the "edit network" mode in NETTER
◇ Select the node type Flow - Boundary
◇ Select the "add node" option 
◇ Place the node in the schematisation by clicking
◇ Select the appropriate branch type (Flow Branch or Sewer Pipe)
◇ Select the "connect nodes" option 
◇ Connect the boundary node that you just added to another node by clicking and dragging.

Some examples of valid methods to apply *Flow-Boundary* nodes:

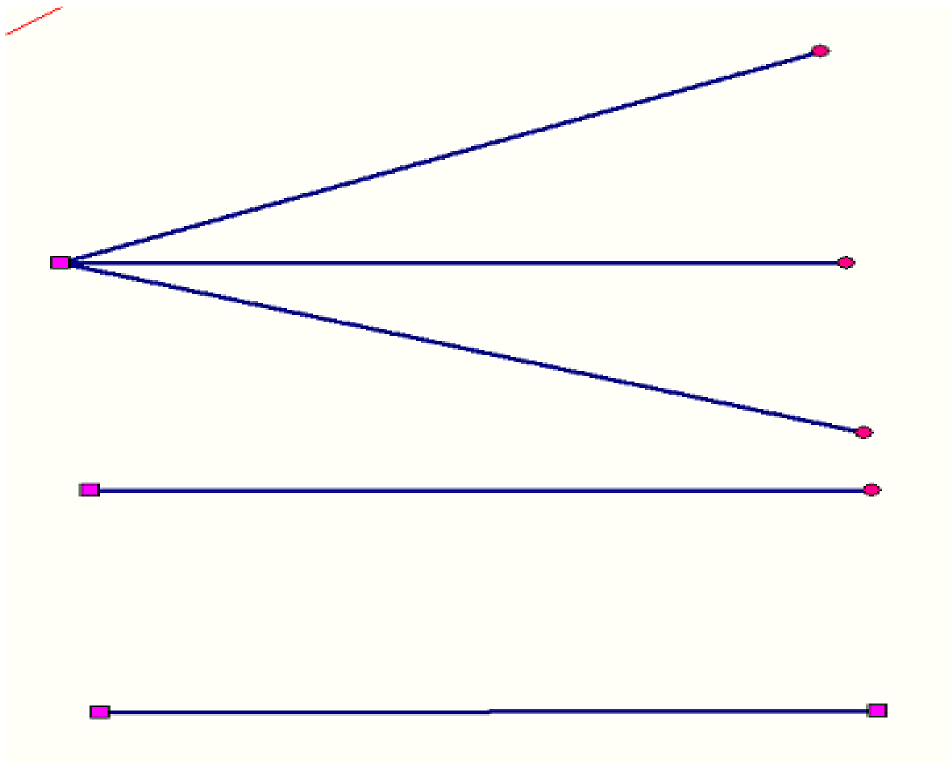
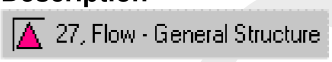


Figure 5.93: Examples of valid configurations with Flow - Boundary nodes

Note: when you attach more than one *Flow - Branch* to one *Flow - Boundary* node, you can only apply a *water level* boundary condition to it. The reason for this is that in case of a discharge condition, it would not be clear how to divide the discharge over the connected branches.

5.3.10 Flow - General Structure

Description



In this chapter, the *Flow - General structure node* is described. The *general structure* combines weir and gate flow in one structure type. It offers you much freedom in defining geometry and dimensions of the structure. Discharges through a general structure are computed on basis of upstream and downstream energy levels. The crest level, crest width and gate lower edge level [m AD] can be varied by a controller. A structure inertia damping factor (see Numerical parameters) can be defined for each General structure.

◇ For a detailed description of this node's input parameters: see the "input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the "topology" section from the Reference Manual
◇ For a detailed description of this node's underlying mathematical equations: see the "General structure" section in the "Technical Reference Manual.

Input screens

When starting the model data editor for an *Flow - General Structure* node type, the following tabs will be available for input:

General structure tab:

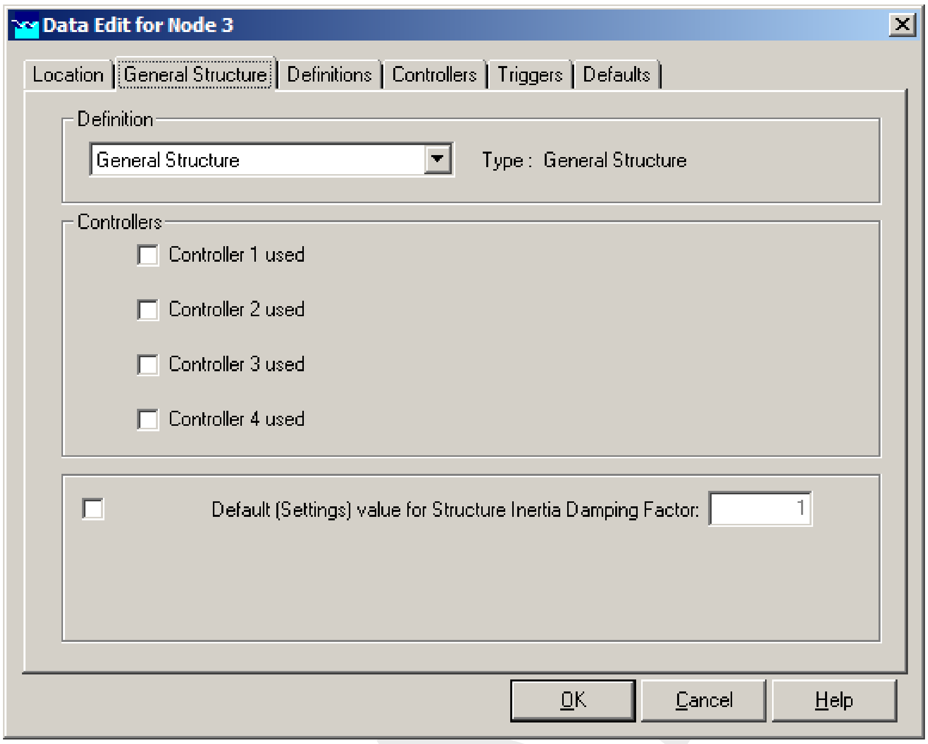


***Figure 5.94: Data Edit** window,* General Structure *tab*

On this tab, you can select a previously defined *structure definition*. If there is no definition available yet, go to the "definitions" tab first, to create one.

Also, you can select the controllers that apply to this structure. If there are no controllers available to be selected yet, go to the "controller" tab first to create them.

You can use the defined value for structure inertia damping factor as defined in Setting, Tab – numerical parameters, or you can define a value for structure inertia damping factor for this particular General structure. For meaning of structure inertia damping factor see Numerical parameters.

Definitions tab:

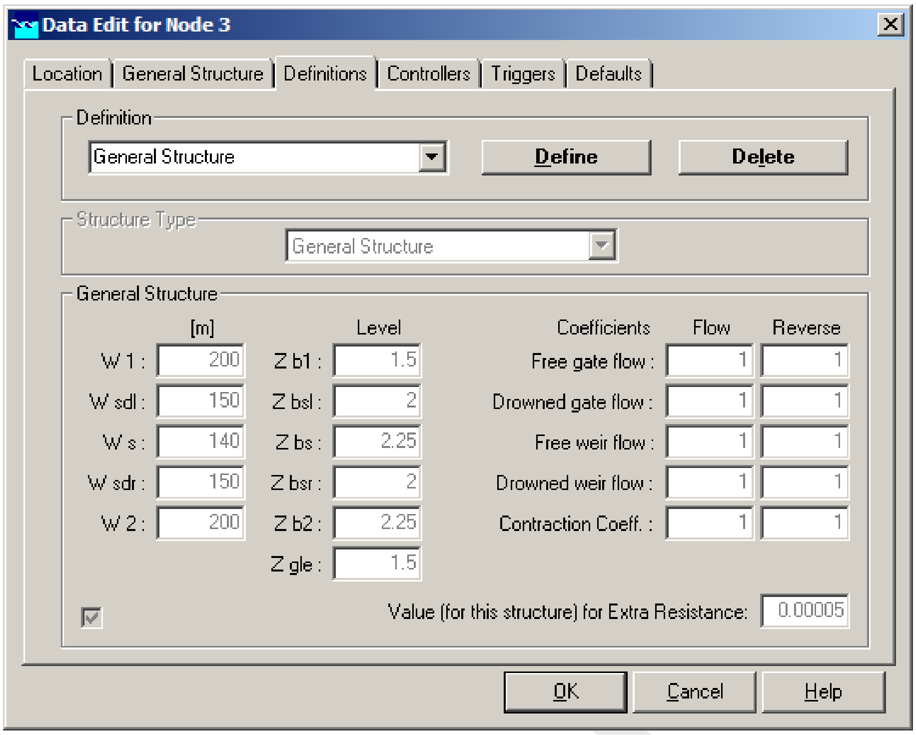


***Figure 5.95: Data Edit** window,* Definitions *tab*

You must enter five dimensions for the flow width along the structure, plus five bed elevations and the *gate lower edge level (Zgle)* See for the use and meaning of each dimension see the Technical Reference Manual, Flow modules, General Structure.

You can use the default value for Extra resistance as defined in Setting, Tab – numerical parameters, or you can define a value for Extra resistance for this particular General structure. For meaning of Extra resistance see Numerical parameters.

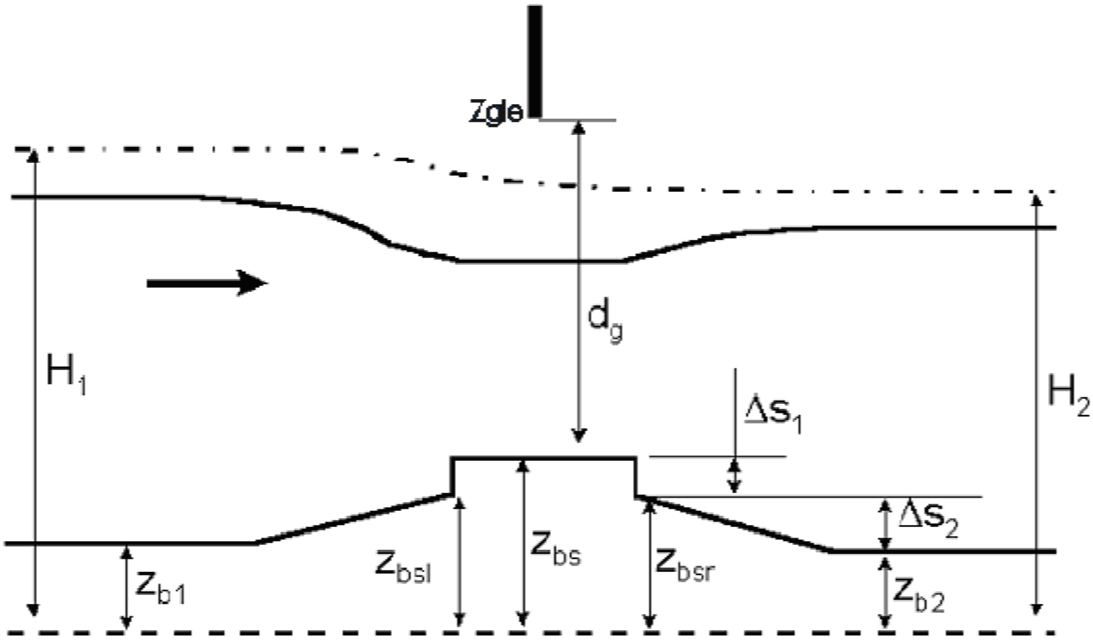


Figure 5.96: *Cross-sectional view on general structure; definitions of vertical distances*

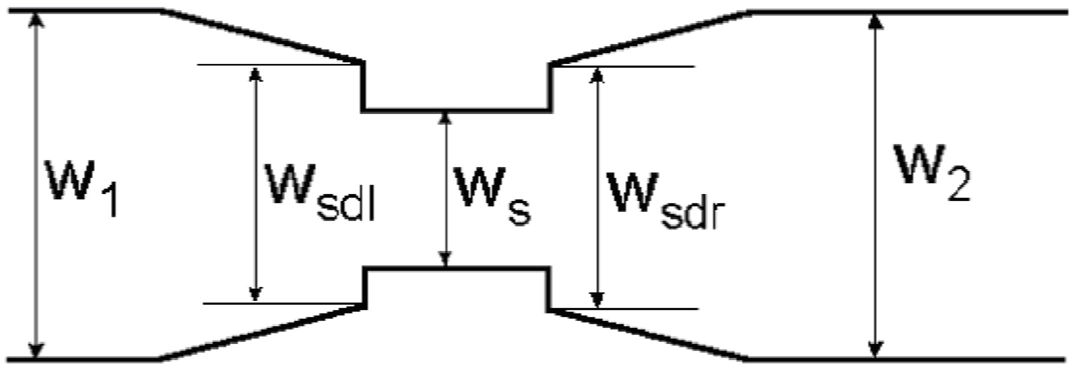


Figure 5.97: *Top view on general structure; definitions of width distances*

Depending on the downstream water level free or drowned (submerged) flow can occur for the weir or for the gate. For each of these conditions you must specify a reduction factor; if you wish you can also enter values for the condition of reverse flow. The reduction factors are used in the formulas given in the last part of the description in the Technical Reference Manual.

◇ Free gate flow cgf
◇ Drowned gate flow cgd
◇ Free weir flow cwf
◇ Drowned weir flow cwd
◇ Zgle Gate lower edge: the lower crest level of the gate.

The default value for each coefficient is 1.00. Finally you must enter a contraction coefficient for gate flow.

Controllers & Triggers tabs: see section 5.8.

Topology

Nodes of the type *Flow - General Structure* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - General Structure* node to your schematisation in the following way:

- Select the appropriate node type (Flow - General Structure)
- Use the "add node"  button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node"  button.

Note:

1 Before adding a general structure to your schematisation, you should have at least one of the above Channel types present in it.
2 SOBEK by default places a computational point 0.5 m upstream and 0.5 m downstream of the general structure location. Hence the general structure is default located on a branch having a length of 1 m.

5.3.11 Flow - Lateral Flow

Description

A "Flow - Lateral Flow Node" can be used to model the lateral inflow (positive values) of water and/or the lateral outflow (negative values) of water in a branch. For how lateral discharges are assigned to a particular calculation point, reference is made to section 6.1.11.2.

Input screens

When starting the model data editor for an *Flow - Lateral Flow* node type, the following tab will be available for input:

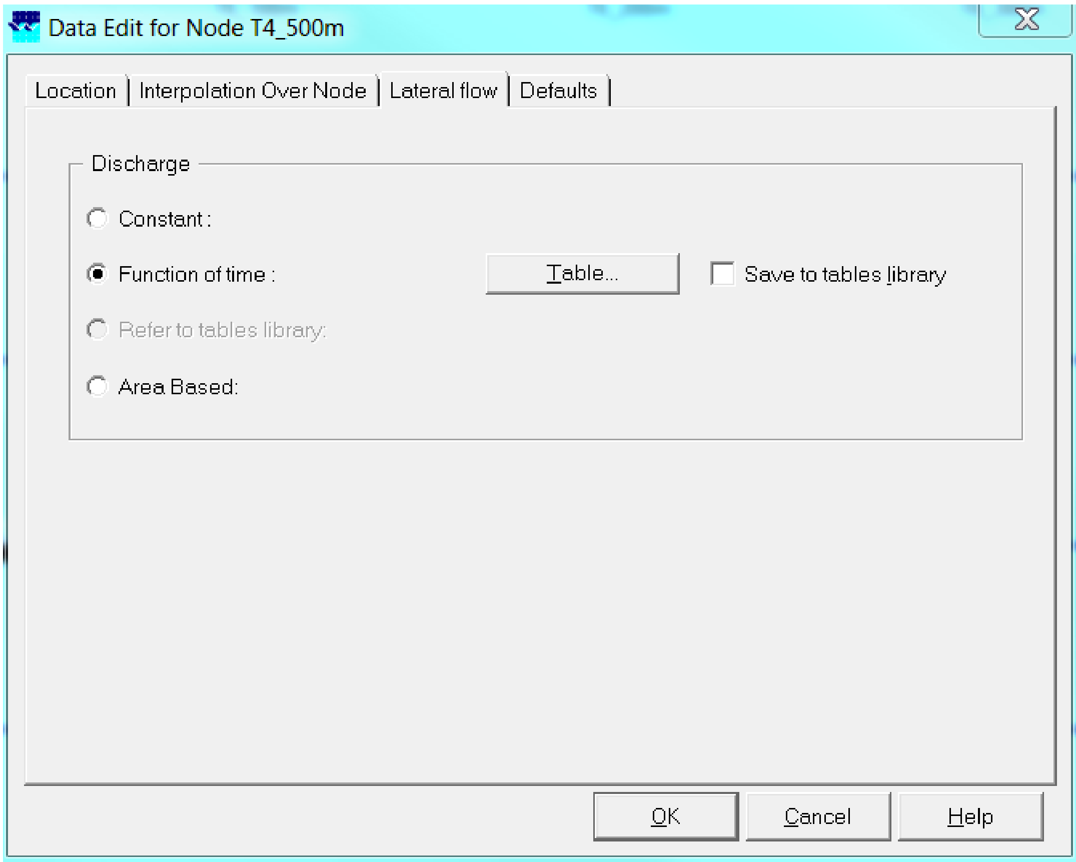


***Figure 5.98: Data Edit** window,* Lateral flow *tab*

Here, the lateral discharge can be specified.

◇ **Constant discharge** When choosing this option, a constant lateral discharge is applied. A positive value means inflow of water, a negative value means outflow of water.
◇ **Function of time** Here a lateral inflow and or outflow as function of time can be specified. Clicking the *Table* button opens a window like the one depicted below:

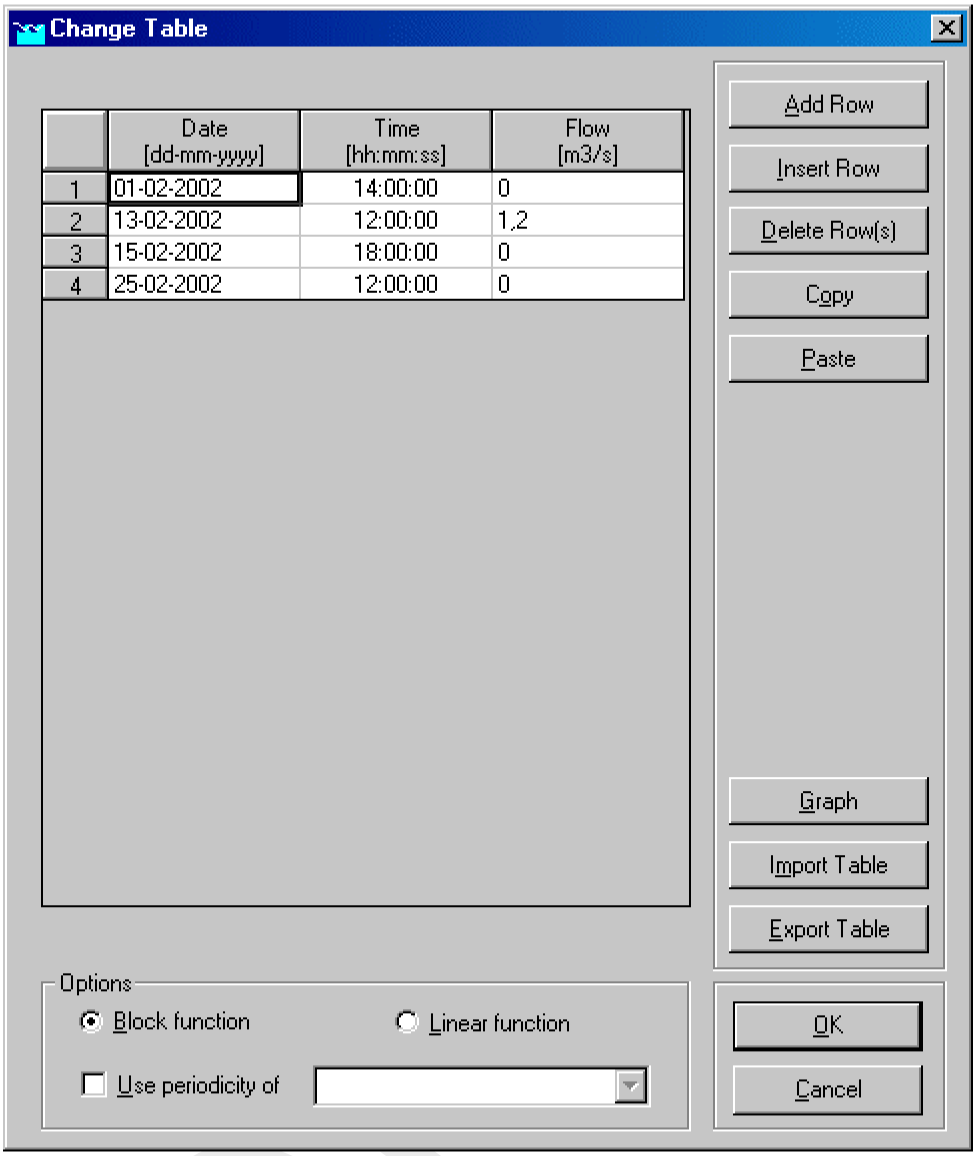


***Figure 5.99: Change Table** window*

In this table, the modeller can submit lateral discharges as function of time. More information about the input options of such tables can be found in the following chapter: section 5.13.4.

◇ **Area based method** The Area based method calculates lateral discharges based on the runoff area, design factor, rainfall, seepage and/or infiltration (see Equation (6.14)). When choosing the Area based method, the input window looks like this:

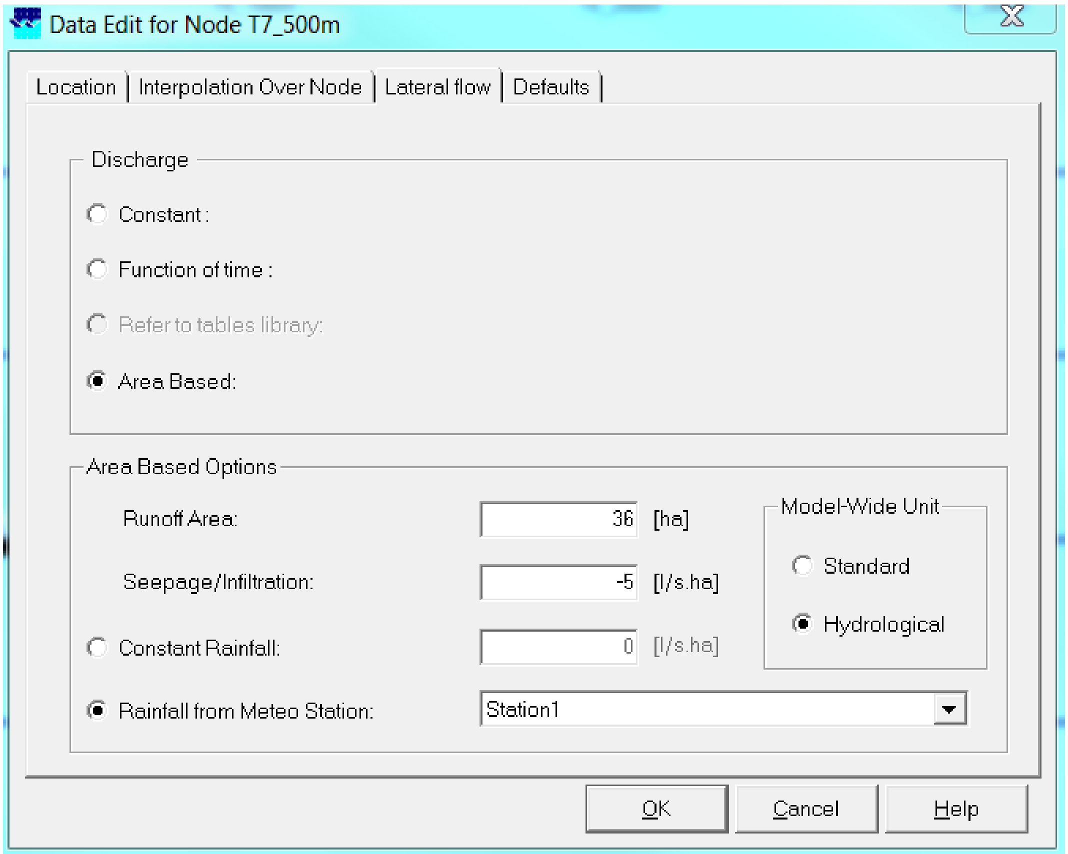


***Figure 5.100: Data Edit** window,* Lateral flow *tab*

- *Model-Wide Unit*: Either Standard units or Hydrological units can be selected. In case of Standard units: the runoff area is to be given in [square metres]; while seepage, infiltration and constant rainfall are to be given in [milimetres/seconds]. In case of Hydrological units: the runoff area is to be given in [hectares]; while seepage, infiltration and constant rainfall are to be given in [liters/second/hectares]. Please note that selecting the option "Rainfall from Meteo Station" means that irrespective of the selected unit (Standard or Hydrological), the unit of the rainfall equals the unit defined for the concerning Meteo station. Please further note that changing the selected unit (f.i. from Standard unit into Hydrological unit) at a specific "Flow - Lateral Flow Node", means that you change the unit for all lateral flows in your entire model schematization that are computed using the Area based method .
- *Runoff Area*: represents the surface area of the catchment, that is assigned to the Lateral Flow node. For the unit of runoff area, see item "Model-Wide unit" above.
- *Seepage/Infiltration*: Only constant Seepage (positive value, inflow) or or constant Infiltration (negative value, outflow) can be defined. For the unit of seepage and infiltration, see item "Model-Wide unit" above.
- *Constant Rainfall:* Here you can define a constant rainfall. For the unit of constant rainfall, see item "Model-Wide unit" above.
- *Rainfall from Meteo Station*: Here you can define that the rainfall (including unit) from a particular Meteo station, defined in the Meteorological Task Block, is to be used.

Topology

Nodes of the type *Flow - Lateral Flow* need to be attached to one of these branch types:

Flow Channel:

Flow Channel with lateral discharge

Add a *Flow - Lateral Flow* node to your schematisation in the following way:

- ◇ Select the appropriate node type (*Flow - Lateral Flow*)
- ◇ Use the *add node*  button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- ◇ If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node"  button.

Example of how a network could contain a Lateral Flow node:

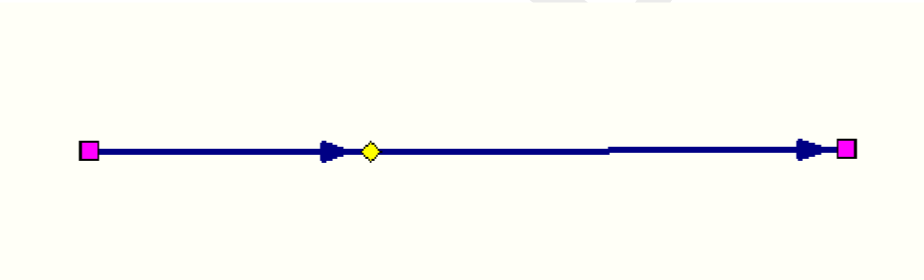


Figure 5.101: *Example of network with a Lateral Flow node*

5.3.12 Flow - Flow manhole

5.3.12.1 Flow - Flow manhole (basic)

Flow Manhole description

This chapter describes the *Flow - Manhole node*. These nodes form a basic modelling element for all urban schematisations. Manholes can be interconnected by Flow-Pipes and/or internal structures. At Flow-Manholes the Water levels are calculated. On Flow-Pipes, discharges are calculated.

- ◇ For a detailed description of this nodes input parameters, see the "Flow - manhole input screens" section from the Reference Manual.

Flow Manhole input screens
When starting the model data editor for an *Flow - Manhole* node type, the following tabs will be available for input:

Storage tab:

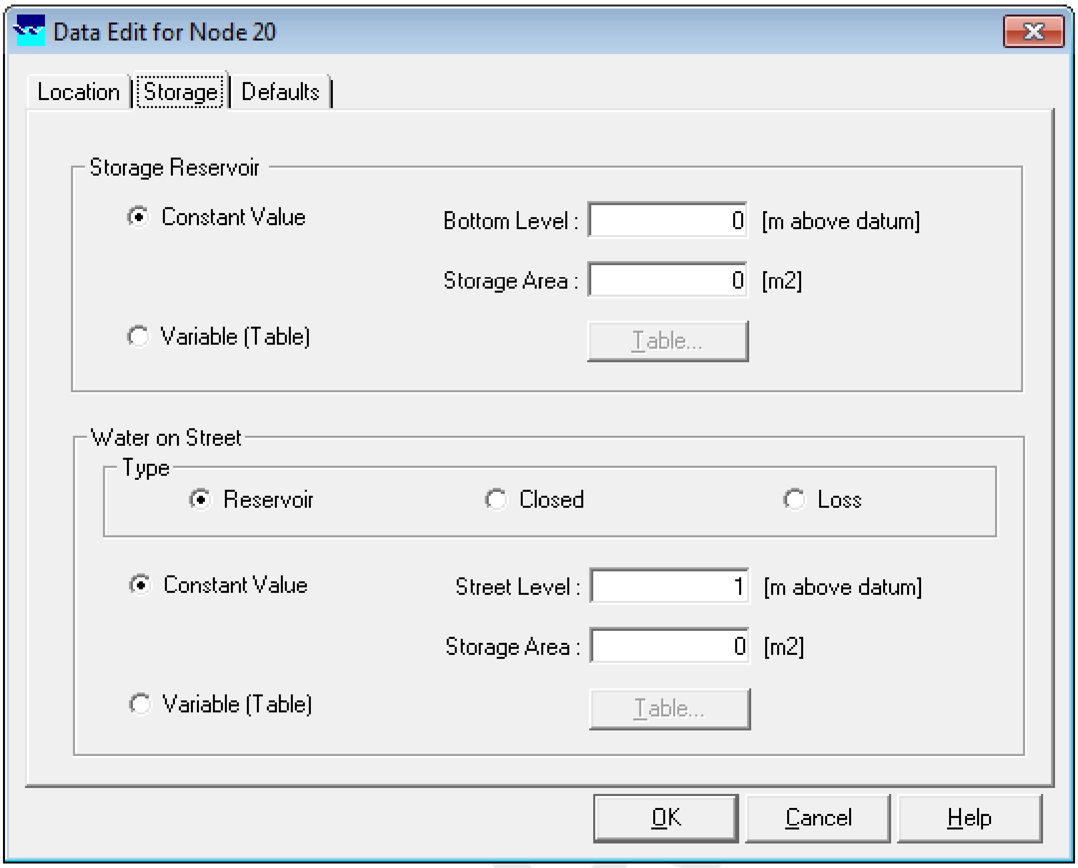


***Figure 5.102: Data Edit** window,* Storage *tab*

On this tab, the following data can be entered:

◇ **The bottom level level and storage area of the manhole.**
A manhole is in fact a small reservoir that interconnects two or more sewer pipes. Therefore, it has a certain storage volume. This volume is defined by the bottom level, the surface level and the area of the manhole. (Surface level - Bottom level)*surface area = total storage capacity of the manhole.

On this tab, the modeller can also choose between three manhole types:

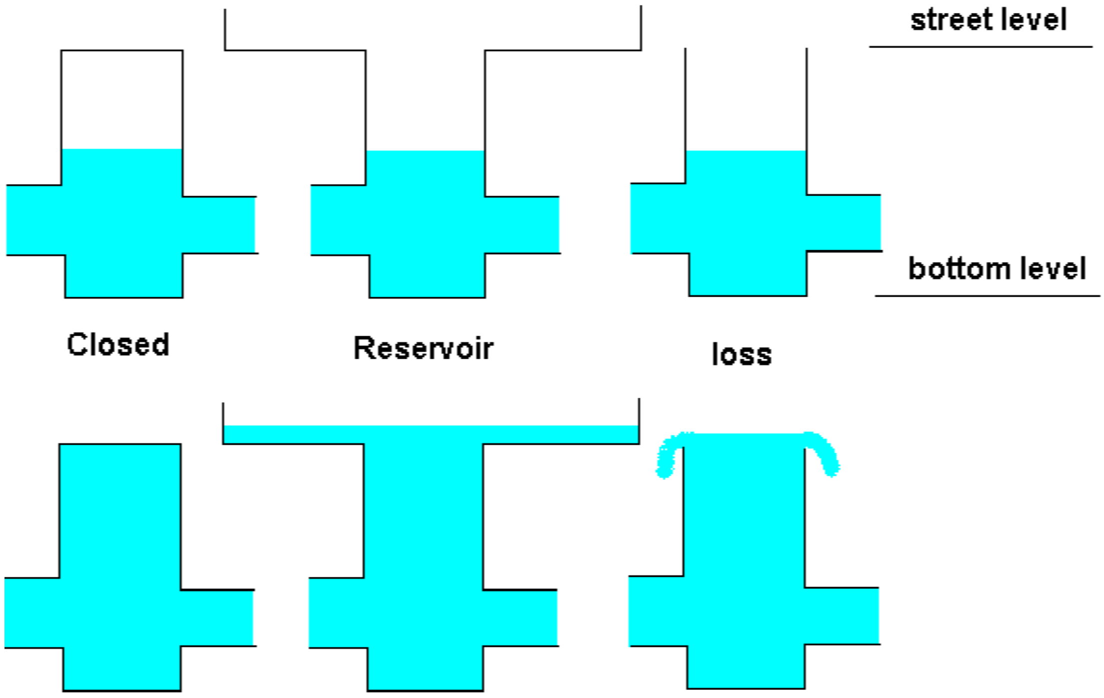


Figure 5.103: The difference between Flow-manholes of the type "closed", "reservoir" and "loss"

The difference between Flow-manholes of the type "closed", "reservoir" and "loss"

Three manhole types that can be chosen. **Note:** In case the 'loss' type is chosen, all water that exceeds the surface level, will be removed from the model. However, if you connect it with a 2D grid (Overland Flow module), it will exchange water with the grid.

- **Reservoir** In case the manhole type "Reservoir" is chosen, there will be an open connection between the manhole and the street surface. This means that, if the water level in the manhole exceeds the street level, it will inundate the 'storage area' that can be entered in the section below.Choosing the type 'reservoir' is useful if you desire to connect your sewer model with a 2D grid (Overland Flow module). That will allow interaction between flow in the pipes (1D) and flow over the land (2D).
- **Closed** Choosing a manholes of the type "Closed" means that you have selected a manhole that has a lid at street level, thus there will be **no** interaction between the water within the manhole and the street surface.
- **Loss** Manholes of the type "Loss" will have all water that exceeds the entered street level flow out.Thus, you might want to use this manhole type if you need to connect it to a 2D grid (Overland Flow module). Water that exceeds the street level will then start flowing over the 2D grid.

5.3.12.2 Flow - Flow manhole with level measurement

Measurement stations

Measurement stations represent locations where structure controllers get the required hydraulic information. Flow-Measurement Pipes are Flow-Pipes with a discharge measurement station. These branches have a length, a cross section and a resistance. Flow-Manholes with Level Measurement are Flow-Manholes with a water level measurement station. Before you can specify a controller you must define a measurement location in the schematisation.

5.3.12.3 Flow - Flow manhole with runoff

Flow-manhole with runoff - description

This chapter describes the *Flow - Manhole with Runoff*.

Basically, these nodes are equal to Flow - Manholes. The only difference is that *Flow - manholes with runoff* have an extra option to transfer rainfall into a discharge into the pipes (rainfall-runoff).

- ◇ For a detailed description of this node's input parameters defining its physical dimensions **(Sewer Flow)**, see the "Flow - manhole input screens" section from the Reference Manual.
- ◇ For a detailed description of this node's **Rainfall-Runoff** input parameters defining the catchment dimensions, see the "Module - Rainfall runoff section of the Reference Manual"
- ◇ For a detailed description of this node's possible network configurations: see the "Flow - Manhole topology" section from the Reference Manual

5.3.13 Flow - Measurement station

Description

In this chapter, the Flow Measurement station node is described.

Measurement stations represent the locations from where structure controllers get their required hydraulic information. Before you can specify a controller you must place at least one measurement station in the schematisation.

- ◇ For a detailed description of this node's possible network configurations: see the "Measurement station node topology" section from the Reference Manual

Note that, after placement of a measurement station, your schematisation needs to be saved before a structure will know that it exists! After saving, you'll notice that when editing model data for structures, various controlling options become available under the "controller" tab.

Topology

Nodes of the type 'Measurement Station' need to be placed on a Flow Branch. Do this by using the  (add node) button.

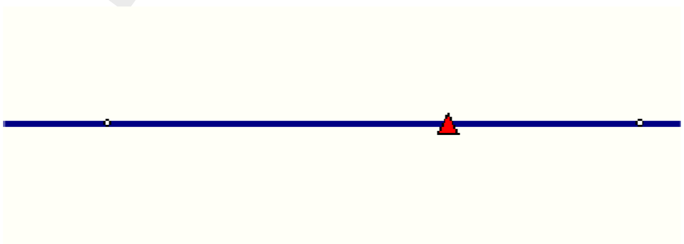


Figure 5.104: *Placement of a Flow Measurement Station node*

Note: the measurement station will derive information about the water level from the nearest ζ-calculation point.

5.3.14 Flow - Orifice node

Description

In this chapter, the *Flow - Orifice node* is described.

With a *Flow - Orifice node* you can simulate rectangle-shaped gates through which water within a channel or river is led. Orifices are used to regulate the water flow through or towards a channel or river. For a detailed description of this node's underlying mathematical equations: see section 6.1.16.8.

Some applications are:

- ◇ In rivers: used to regulate the division of flows over an upstream split-point.
- ◇ In deltas: to protect the delta against storms on the coast (the orifices close during a storm).
- ◇ In smaller channels: to allow water to flow into an area in periods of drought.

Input screens

When starting the model data editor for an *Flow - Orifice* node type, the following tabs will be available for input:

Orifice:

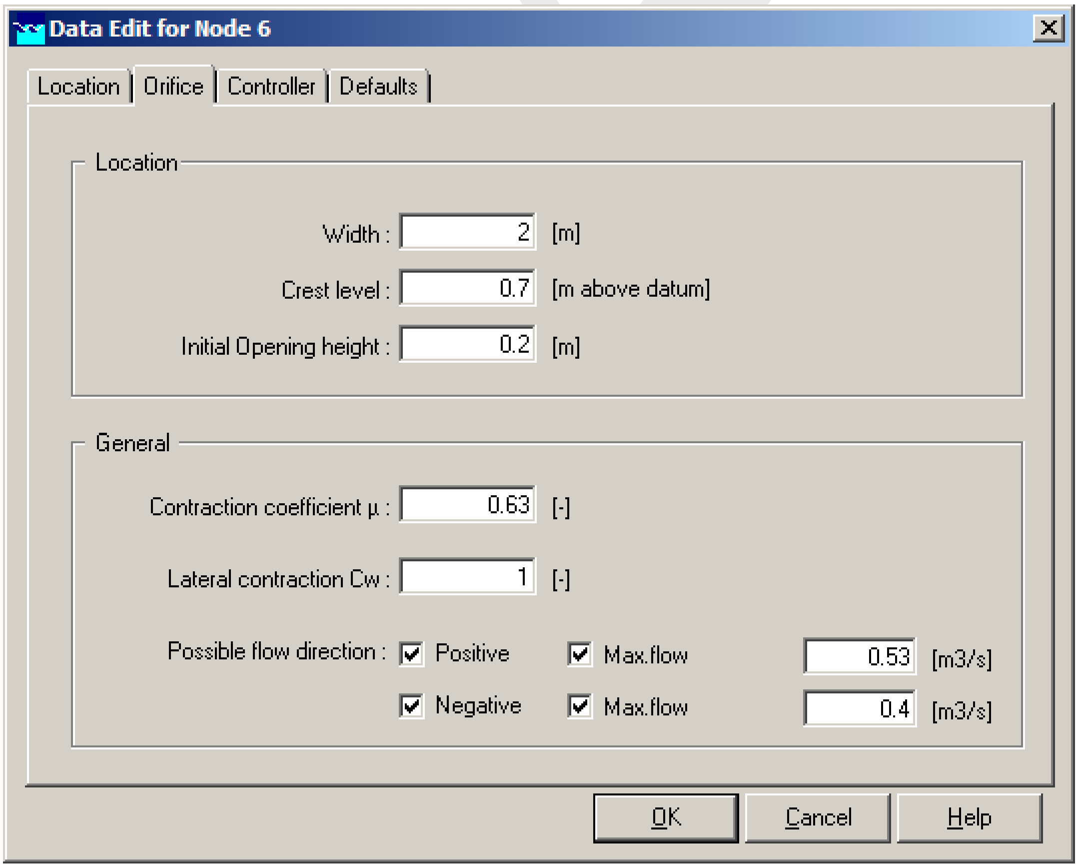


***Figure 5.105: Data Edit** window,* Orifice *tab*

On this tab, you can fill in the general dimensions and parameters for the orifice:

- **Width:** This parameter represents the width of the gate through which the water will flow.
- **Crest level:** This parameter represents the *bottom* crest level of the gate.
- **Initial openings height:** Here you can enter the vertical dimension of the gate for the *start of the simulation*. For orifices it is common to apply a controller, which will define the gate height during the simulation. If you don't define a controller, the initial openings height will remain valid throughout the entire simulation.
- **Contraction coefficient** μ: This parameter represents the energy loss that is caused by contraction of the flow towards the orifice. This phenomenon generally occurs when the weir's crest is less wide than the channel. See the Technical Reference Manual for the exact mathematical implementation of this parameter.
- **Lateral Contraction** C_w: This parameter represents the energy loss that is caused by contraction of the flow towards the orifice. This phenomenon generally occurs when the weir's crest is less wide than the channel. See the Technical Reference Manual for the exact mathematical implementation of this parameter.
- **Possible flow directions:** This parameter defines in which direction(s) water can flow through the orifice. The directions are defined with relation to the defined branch direction. Positive means: in the same direction as the defined branch direction; negative means: in the opposite direction than the defined branch direction. If you don't know the defined direction of the branch, read the Flow - Branch topology section of the Functional Reference Manual. In addition to defining the flow directions, you can also define a maximum flow through the gate for each flow direction. To do so, activate the check boxes next to the word "Max. flow" and enter a value.

Controller:

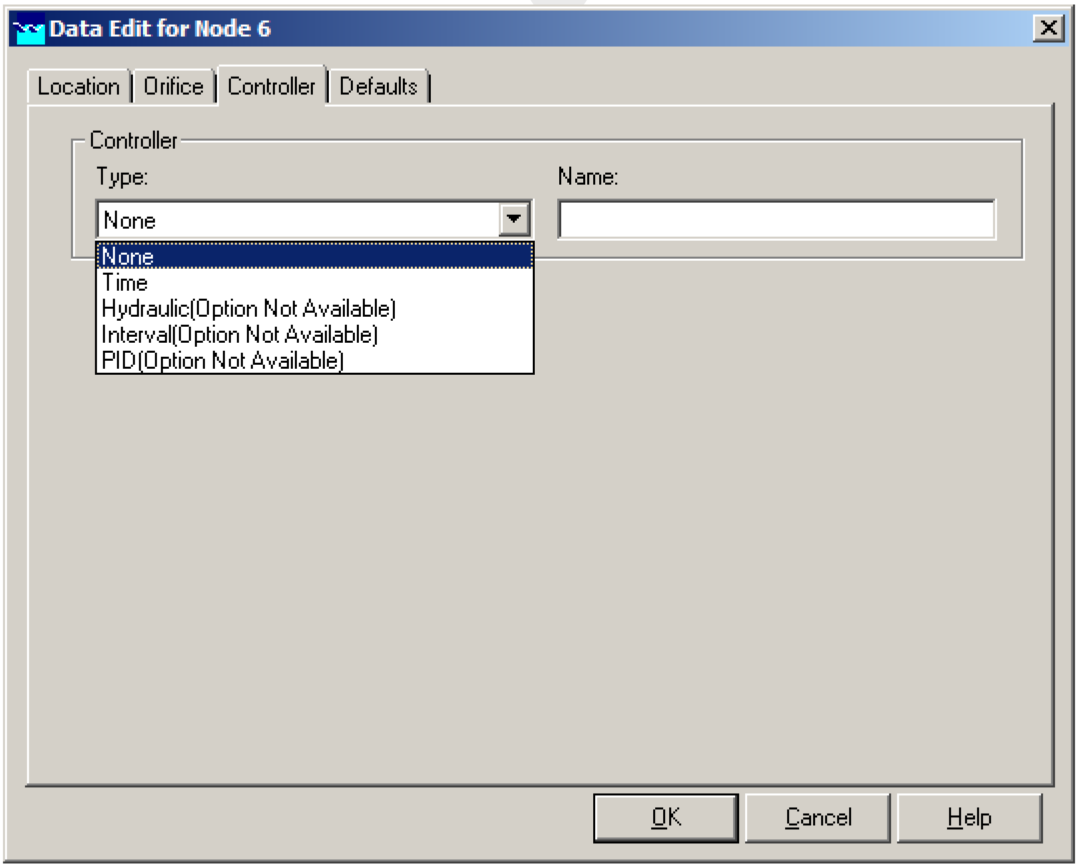


***Figure 5.106: Data Edit** window,* Controller *tab*

On this tab, you can define a controller for your orifice. A controller will overrule your general settings for the orifice's **opening height** and apply the chosen controlling rules to it. Read more about them in section 6.4.1.

Note that three of the four available controller options only become available after you have defined a Flow - Measurement station within the schematisation and saved that schematisation.

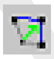
Topology

Nodes of the type *Flow - Orifice* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - Orifice* node to your schematisation in the following way:

- ◇ Select the appropriate node type (Flow Orifice)
- ◇ Use the "add node"  button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- ◇ If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node"  button.

Note: Before adding an orifice to your schematisation, you should have at least one of the above Channel types present.

5.3.15 Flow - Pump station node

A *Flow - Pump station* can be used to remove excess water or to supply water for drought prevention. In the section Topology it is explained how to place a *Flow - Pump station* on a network. The data input screens are described in the section Data input screens. See section 6.1.16.9 for detailed information on:

- ◇ Pump discharge direction with respect to the positive x-direction along the branch,
- ◇ Pump stages,
- ◇ Controller options at a Pump station,
- ◇ Capacity reduction table,
- ◇ Pump station output parameters,
- ◇ Dead-band triggering algorithm,
- ◇ Conventions for switch-on and switch-off levels.

Topology

Nodes of the type *Flow - Pump Station* need to be attached to one of these branch types:

Flow Channel:

Flow Channel with lateral discharge

Add a Flow - Pump Station node to your schematisation in the following way:

- ◇ Select the appropriate node type (Flow Pump Station)
- ◇ Use the "add node" button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- ◇ If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node" button.

Note: Before adding a pump station to your schematisation, you should have at least one of the above Channel types present in it.

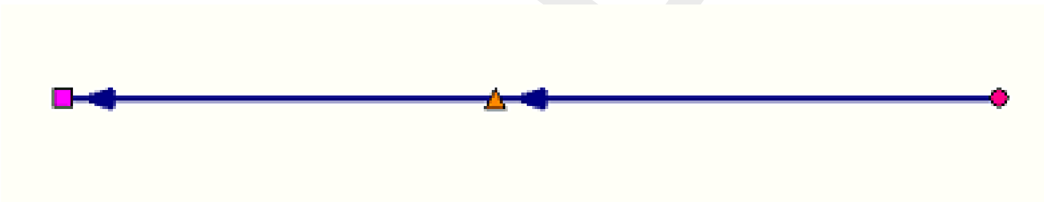


Figure 5.107: *Example of a valid network configuration containing a pump station.*

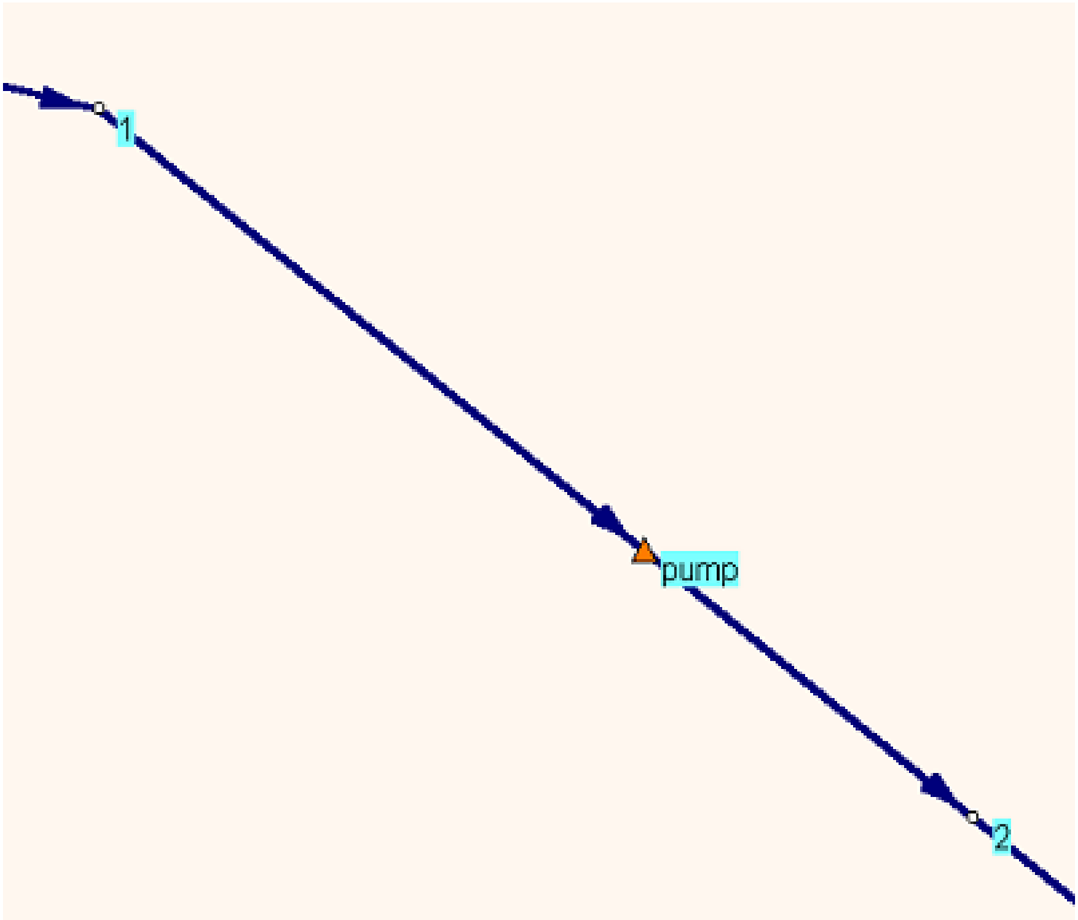


Figure 5.108: Example of the application of a Flow - Pump Station. If you would define a "positive pump direction", here node #1 would be the suction side and node #2 would be the pressure side.

Data input screens

After starting the model data editor for an *Flow - Pump Station* node type, the Pump tab (see Figure 5.109) becomes available. On the Pump tab two separate windows can be opened, respectively the reduction table window (see Figure 5.110) and the pump data window (see Figure 5.111) For detailed information on the various pump station input parameters, reference is made to section 6.1.16.9.

Following pump characteristics can be defined on the Pump tab (see Figure 5.109):

- ◇ Number of different pump stages
- ◇ Reduction factors on pump discharge capacity (see also Figure 5.110)
- ◇ Unit of the specified pump discharge capacity
- ◇ Location of dead-band triggers (Suction-side only, Delivery-side only, both Suction-side and Delivery-side)
- ◇ Pump stage data (see also Figure 5.111)

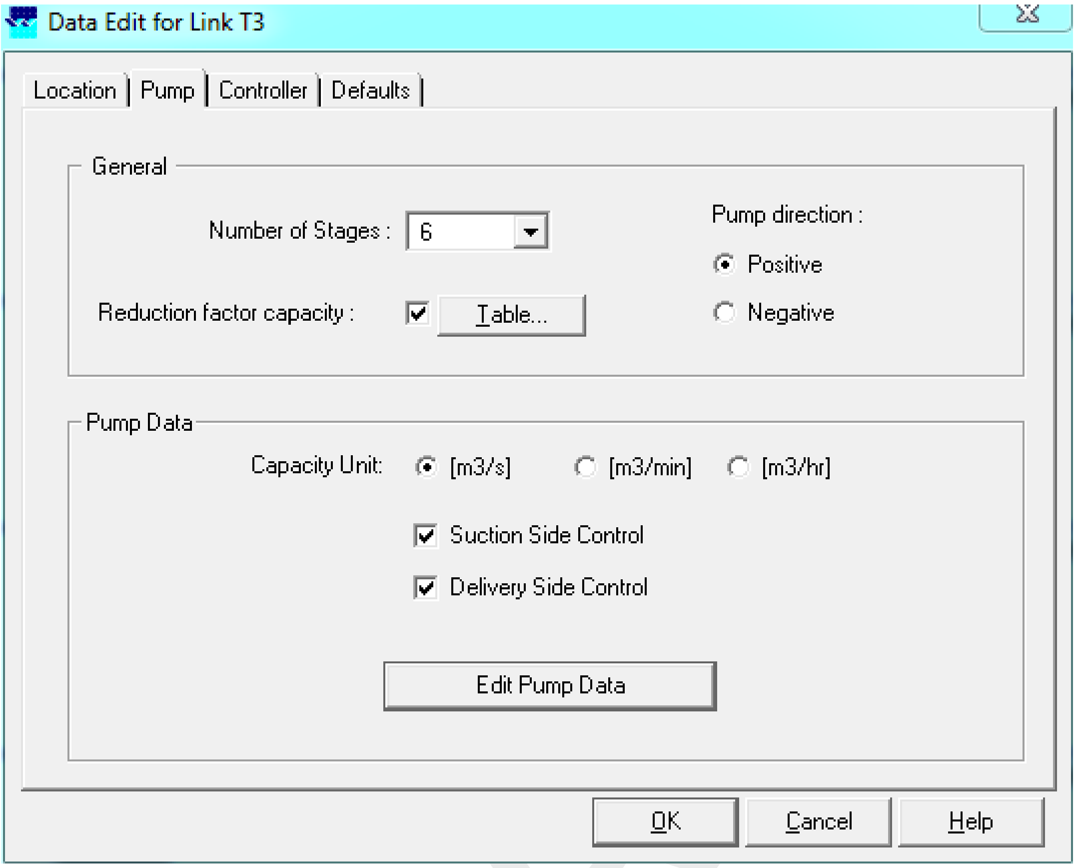


***Figure 5.109: Data Edit** window,* Pump *tab*

In the capacity reduction window (see Figure 5.110), the capacity reduction factor can be defined as function of pump head (e.g. water level at delivery-side minus water level at the suction-side).

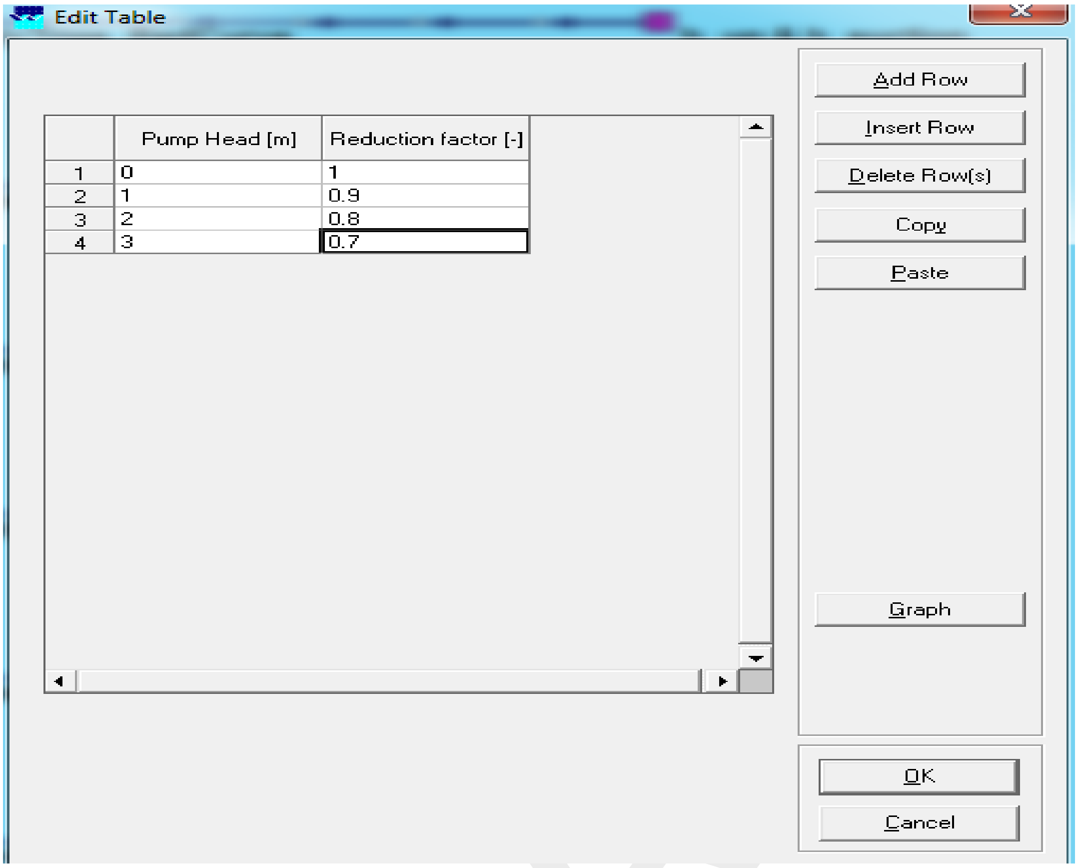
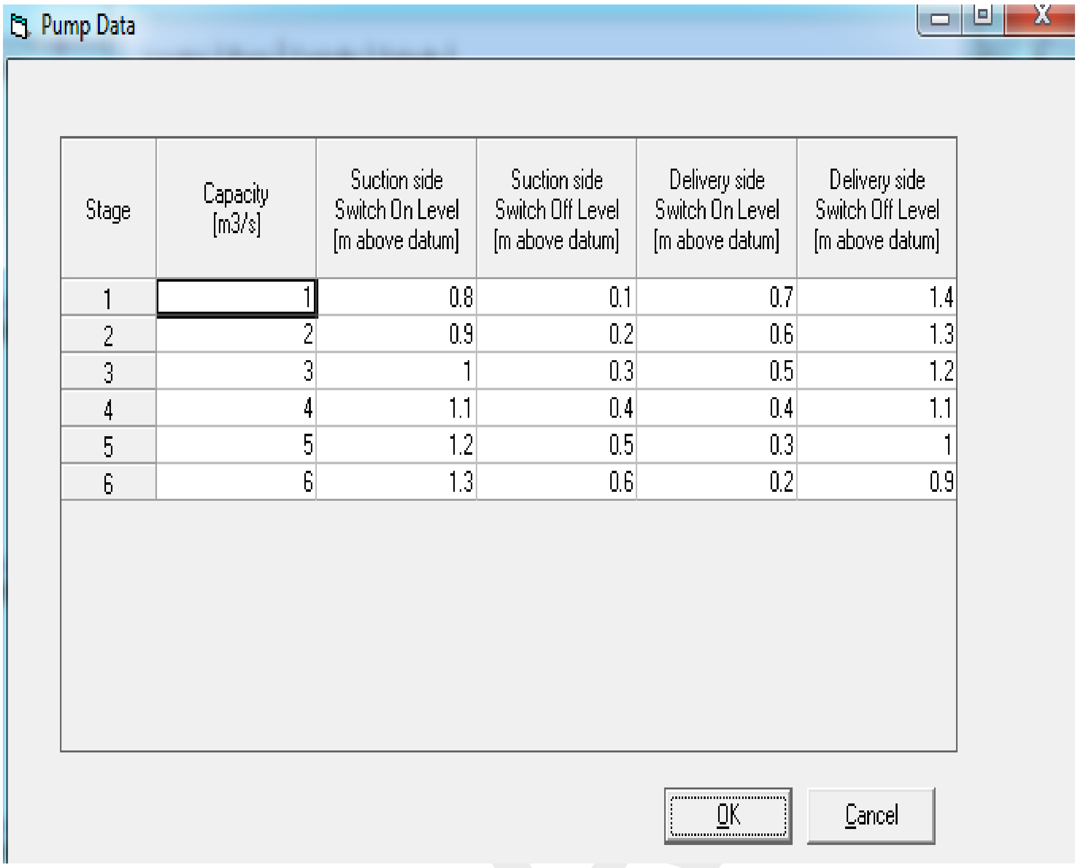


***Figure 5.110: Data Edit** window,* Capacity Reduction Table *tab*

In the Pump data window (see Figure 5.111), for each pump stage following parameters can be defined:

- ◇ Pump stage capacity
- ◇ Switch-on-level at the Suction-side (if applicable)
- ◇ Switch-on-level at the Delivery-side (if applicable)
- ◇ Switch-off-level at the Suction-side (if applicable)
- ◇ Switch-off-level at the Delivery-side (if applicable)



Stage	Capacity [m3/s]	Suction side Switch On Level [m above datum]	Suction side Switch Off Level [m above datum]	Delivery side Switch On Level [m above datum]	Delivery side Switch Off Level [m above datum]
1	1	0.8	0.1	0.7	1.4
2	2	0.9	0.2	0.6	1.3
3	3	1	0.3	0.5	1.2
4	4	1.1	0.4	0.4	1.1
5	5	1.2	0.5	0.3	1
6	6	1.3	0.6	0.2	0.9

***Figure 5.111: Data Edit** window,* Capacity Reduction Table *tab*

5.3.16 Flow - River Advanced Weir

Description

In this chapter, the *Flow - Advanced Weir node* is described.

The advanced weir requires entry of values for crest level, (net) sill width and number of piers. Discharges through an Advanced weir are computed on basis of upstream and downstream energy-levels. The crest level can be varied by a controller. A structure inertia damping factor (see Numerical parameters) can be defined for each Advanced weir.

◇ For a detailed description of this node's input parameters: see the "input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the "topology" section from the Reference Manual
◇ For a detailed description of this node's underlying mathematical equations: see the "Advanced weir" section in the "Technical Reference Manual".

Input screens

When starting the model data editor for an *Flow - Advanced Weir* node type, the following tabs will be available for input:

Advanced weir tab:

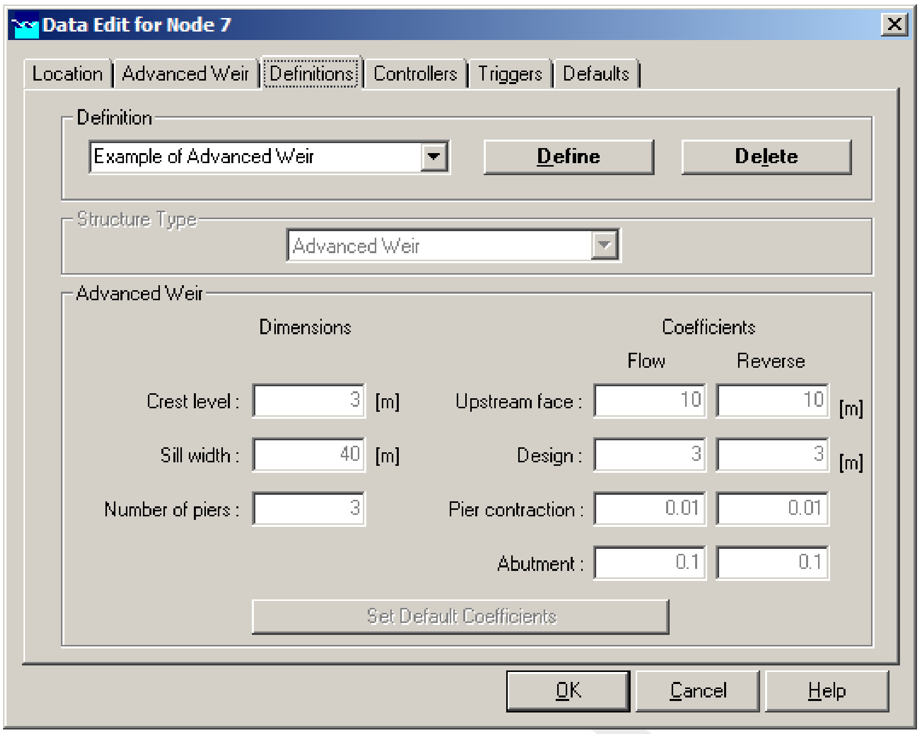


***Figure 5.112: Data Edit** window,* Advanced Weir *tab*

On this tab, you can select a previously defined *structure definition*. If there is no definition available yet, go to the "definitions" tab first, to create one.

Also, you can select the controllers that apply to this structure. If there are no controllers available to be selected yet, go to the "controller" tab first to create them.

You can use the default value for structure inertia damping factor as defined in Setting, Tab – numerical parameters, or you can define a value for structure inertia damping factor for this particular Advanced weir. For the mening of structure inertia damping factor see Numerical parameters.

Definitions tab:

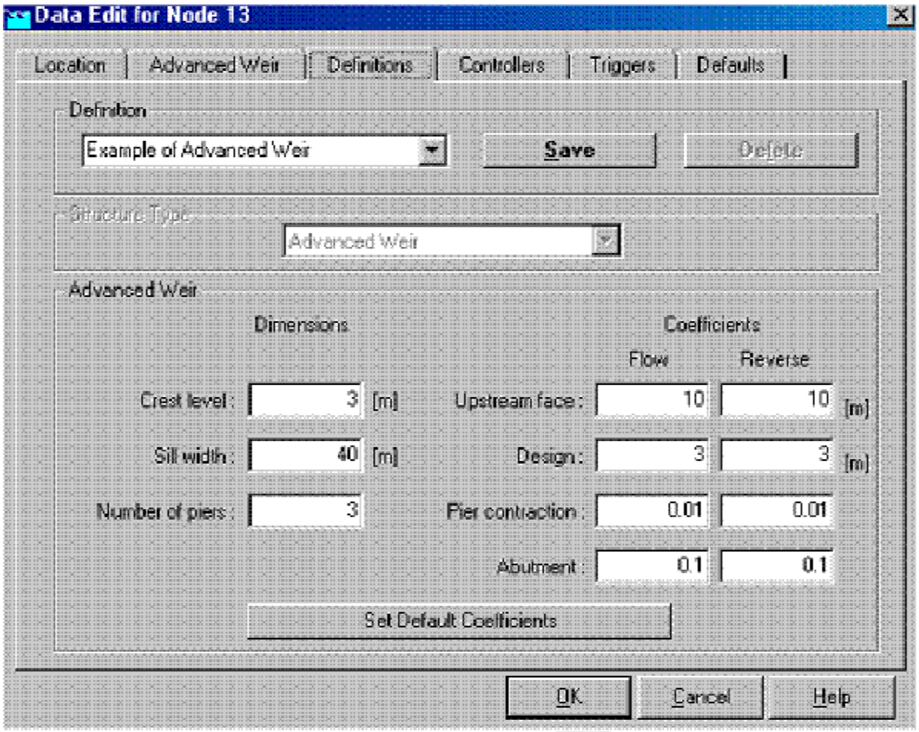


***Figure 5.113: Data Edit** window,* Definitions *tab*

The advanced weir requires entry of values for crest level, (net) sill width and number of piers. The crest level can be varied by a controller. For the following parameters you get default values in the corresponding data fields, which you may adapt to your particular situation.

1 **Upstream face height** The height of the weir relative to the bed level at the upstream sideA default value of 10 m is available, but you should enter the actual value.
2 **Design head** The head for which the structure was designed. A default value of 3 m is available, but you should enter the actual value.
3 **Pier contraction** The contraction coefficient represents the net sill-width reduction due to the presence of piers. It depends on the shape of the piers. Default value is 0.01.
4 **Abutment contraction** The contraction coefficient represents the net total flow width reduction due to the presence of abutments. It depends on the shape of the abutments. Default value is 0.10. If reverse flow may occur you can adapt the values for that condition as well.

Controllers & Triggers tabs: see section 5.8.

Topology

Nodes of the type *Flow - Advanced Weir* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - Advanced Weir* node to your schematisation in the following way:

- Select the appropriate node type (Flow - Advanced Weir)
- Use the "add node" button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node" button.

Note:
1. Before adding an Advanced weir to your schematisation, you should have at least one of the above Channel types present in it.
2. SOBEK by default places a computational point 0.5 m upstream and 0.5 m downstream of the Advanced weir location. Hence the Advanced weir is default located on a branch having a length of 1 m.

5.3.17 Flow - River Pump

A *Flow - River Pump* can be used to remove excess water or to supply water for drought prevention. Controllers can not be assigned to a River pump. In the section Topology it is explained how to place a *Flow - River Pump* on a network. The data input screens are described in the section Data input screens. See section 6.1.16.11 for detailed information on:

- Pump discharge direction with respect to the positive x-direction along the branch,
- Pump stage,
- Constant Reduction Factor,
- Reduction Factor F(Pump Head),
- Pump station output parameters,
- Dead-band triggering algorithm,
- Comparison of a River Pump and a Pump station.

Topology

Nodes of the type *Flow - River Pump* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - River Pump* node to your schematisation in the following way:

- Select the appropriate node type (Flow - River Pump)
- Use the "add node" button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node" button.

Note:

1 Before adding a river pump to your schematisation, you should have at least one of the

above Channel types present in it.

2 SOBEK by default places a computation point 0.5 m upstream and 0.5 m downstream of the River pump location. Hence the River pump is default located on a branch segment having a length of 1 m.

Data input screens

After starting the model data editor for an *Flow - River Pump* node type, the River Pump tab (see Figure 5.114) becomes available. On the River Pump tab you can select a previously defined River Pump definition. Various River Pumps definitions can be specified on the Definition tab (see Figure 5.115).

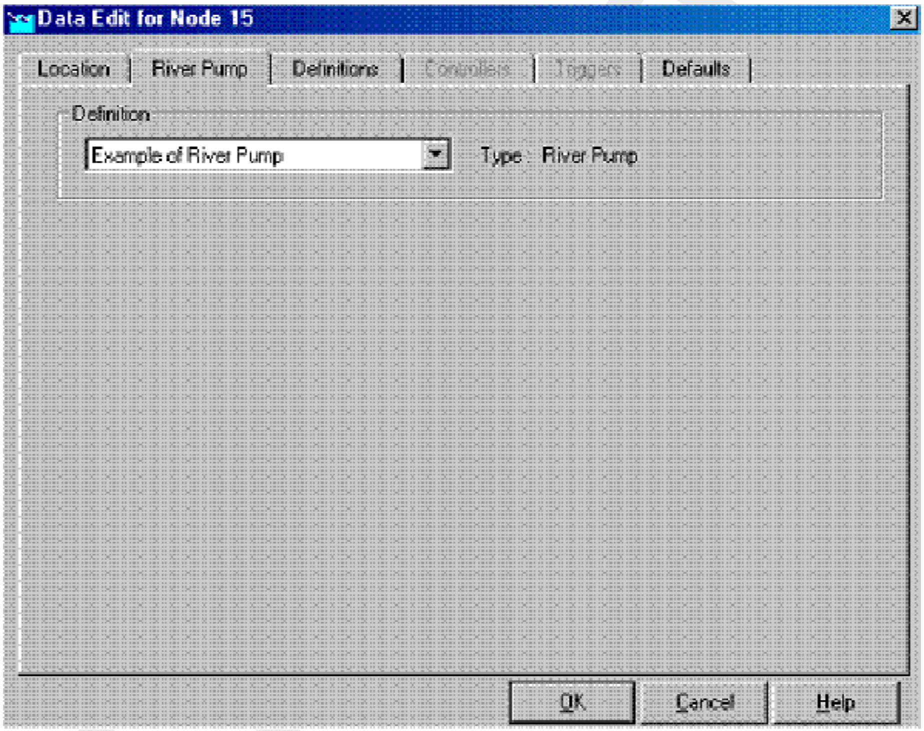


***Figure 5.114: Data Edit** window,* River Pump *tab*

On the Definition tab of a River Pump (see Figure 5.115) following parameters can be defined:

◇ Control direction,
◇ Capacity,
◇ Start level,
◇ Stop level,
◇ A Constant Reduction Factor or a Reduction Factor F(Pump Head),

For detailed information on these various River Pump input parameters, reference is made to section 6.1.16.11.

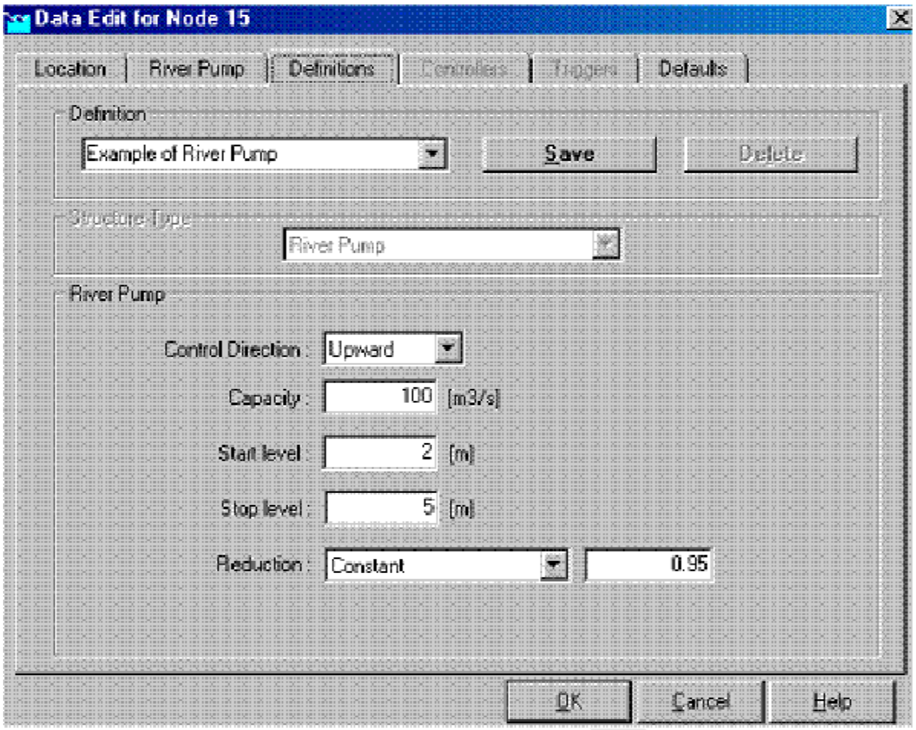


Figure 5.115: Data Edit *window,* Definition *tab*

5.3.18 Flow - River Weir

Description

Different crest shapes (broad crested, triangular, round or sharp) can be defined for a *Flow - River Weir* by selecting a particular "drowned reduction curve" (see Figure 6.38 in section 6.1.16.12). Crest levels of a *Flow - River Weir* cannot vary in height along the weir. Discharges through a *Flow - River Weir* are computed on basis of upstream and downstream energy-level. For the applied mathematical equations, reference is made to section 6.1.16.12.

The crest level and crest width of a river weir can be varied by a controller. A structure inertia damping factor (see Numerical parameters) can be defined to each River weir.

Input screens
When starting the model date editor for an *Flow - River Weir* node type, the following tabs will be available for input:

River weir tab:

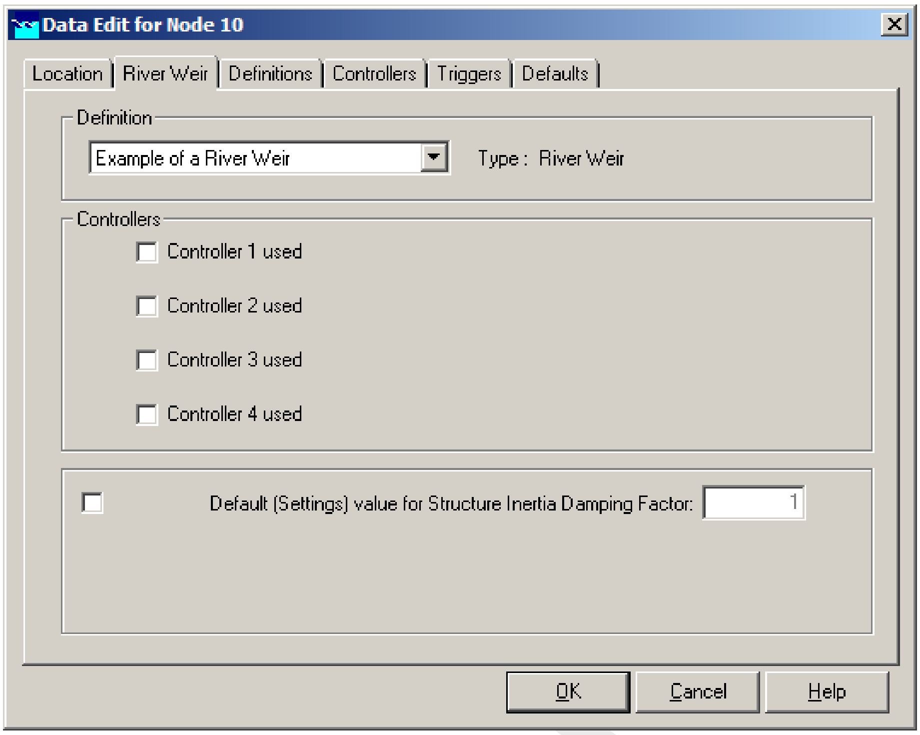


***Figure 5.116: Data Edit** window,* River Weir *tab*

On this tab, you can select a previously defined *structure definition*. If there is no definition available yet, go to the "definitions" tab first, to create one.

Also, you can select the controllers that apply to this structure. If there are no controllers available to be selected yet, go to the "controller" tab first to create them.

Definitions tab:

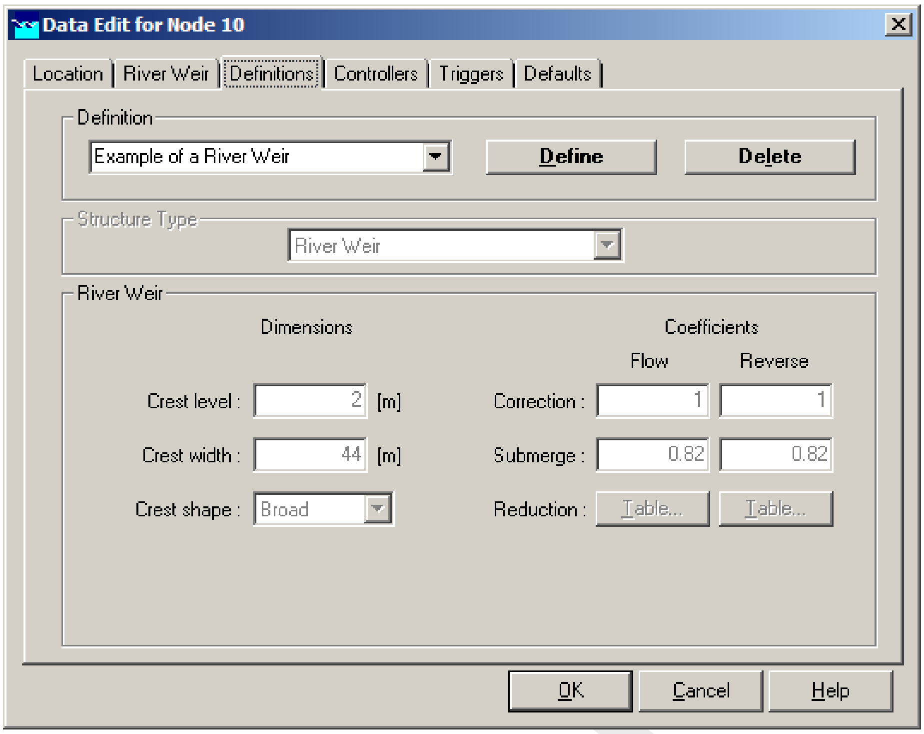


***Figure 5.117: Data Edit** window,* Definitions *tab*

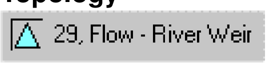
On this tab, you can define the properties of your *River Weir* structure. Select an existing definition from the drop-down box or create one by typing an appropriate name in the "definition" field and then clicking <Define>.

Then you can:

◇ Define the crest level of the weir
◇ Define the crest width of the weir
◇ Define the crest's shape: choose between (broad, triangular, sharp and round)Note that changing the crest's shape will change the default flow coefficients.
◇ Define a correction coefficient: For flow in the positive direction of the branch or in the opposite direction, default correction coefficients cw are proposed by SOBEK for each crest shape (see also Technical Reference Manual, Flow Modules, River Weir). If you wish, you can replace them by other values.
◇ Define the submerge flow coefficient. If the tail water level affects the weir flow, drowned (submerged) flow occurs. Whether this happens is determined by the submergence (or modular) limit. For each crest shape a default value is presented. If the submergence factor Sf ((H2-Zs)/(H1-Zs)) is above the submergence limit, the flow is multiplied by a corresponding drowned reduction factor, which is also a function of the crest shape. The default values are shown in the Technical Reference Manual, Flow Modules, River Weir. If you wish to apply other factors you can enter them in a table, accessible through *Edit*. A similar table is available for reduction factors applying to the reverse flow condition

Controllers & Triggers tabs: see section 5.8.

Topology

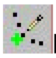
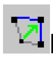


Nodes of the type *Flow - River Weir* need to be attached to one of these branch types:

Flow Channel: 

Flow Channel with lateral discharge 

Add a *Flow - River Weir* node to your schematisation in the following way:

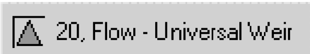
- ◇ Select the appropriate node type (Flow - River Weir)
- ◇ Use the "add node"  button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- ◇ If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node"  button.

Note:

1 Before adding a River weir to your schematisation, you should have at least one of the above Channel types present in it.
2 SOBEK by default places a computational point 0.5 m upstream and 0.5 m downstream of the River weir location. Hence the River weir is default located on a branch having a length of 1m.

5.3.19 Flow - Universal Weir

Description



The *Flow - Universal Weir* is a broad crested weir. Crest levels may vary in height along the weir. For the assumptions made in the parameterization of the *Flow - Universal Weir* as well as for the applied mathematical equations, reference is made to section 6.1.16.14

Input screens
When starting the model data editor for an *Flow - Universal Weir* node type, the following tabs will be available for input:

Universal Weir:

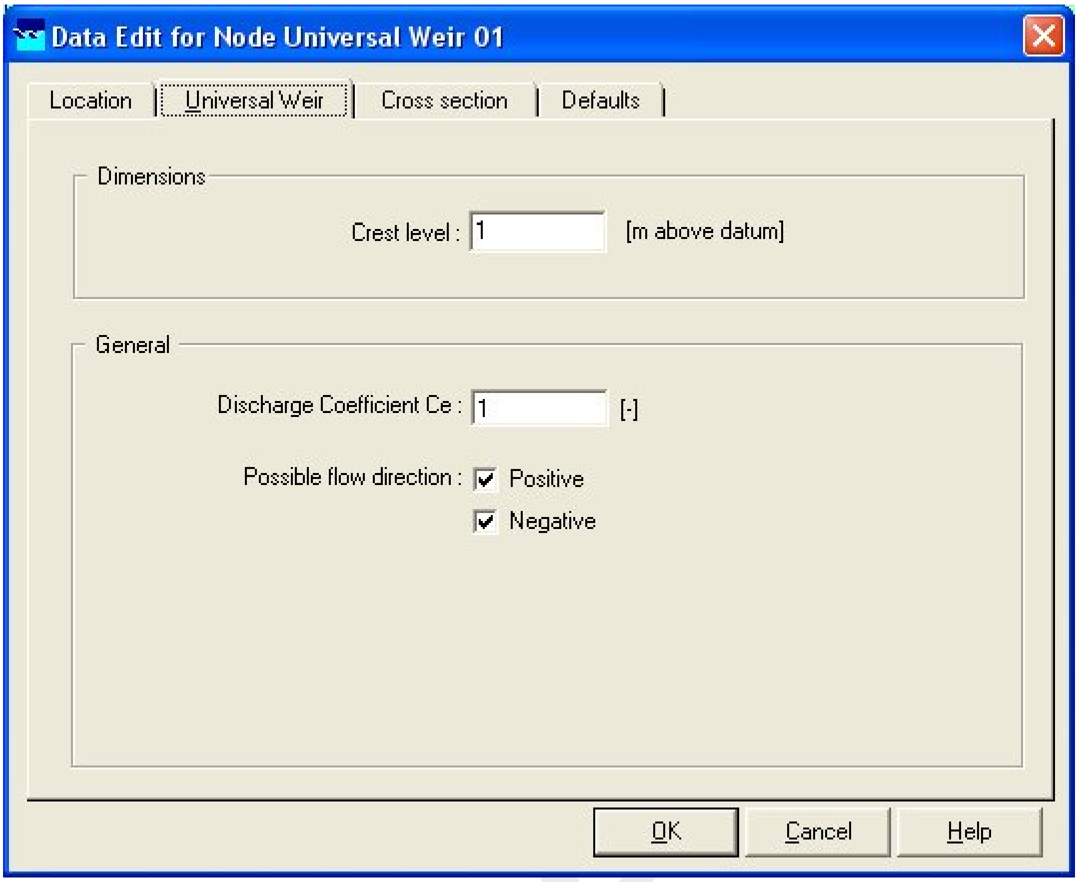


***Figure 5.118: Data Edit** window,* Universal Weir *tab*

On this tab, the general dimensions of the weir can be defined:

- **Crest Level:** This value represents the **lowest** part of the weir's crest. On the cross section tab, later on the exact shape of the crest can be defined.
- **Discharge coefficient Ce:** This parameter represents the discharge coefficient for the weir, as present in the formula that applies to this type of structure. See the "Universal Weir" section in the Technical Reference Manual for the exact implementation of this parameter.
- **Possible flow direction:** This parameter defines in which direction(s) water can flow over the weir. The directions are defined with relation to the defined branch direction. Positive means: in the same direction as the defined branch direction; negative means: opposite of the defined branch direction. If you don't know the defined direction of the branch, read more about it in the Flow - Branch topology section of the Functional Reference Manual.

Cross Section:

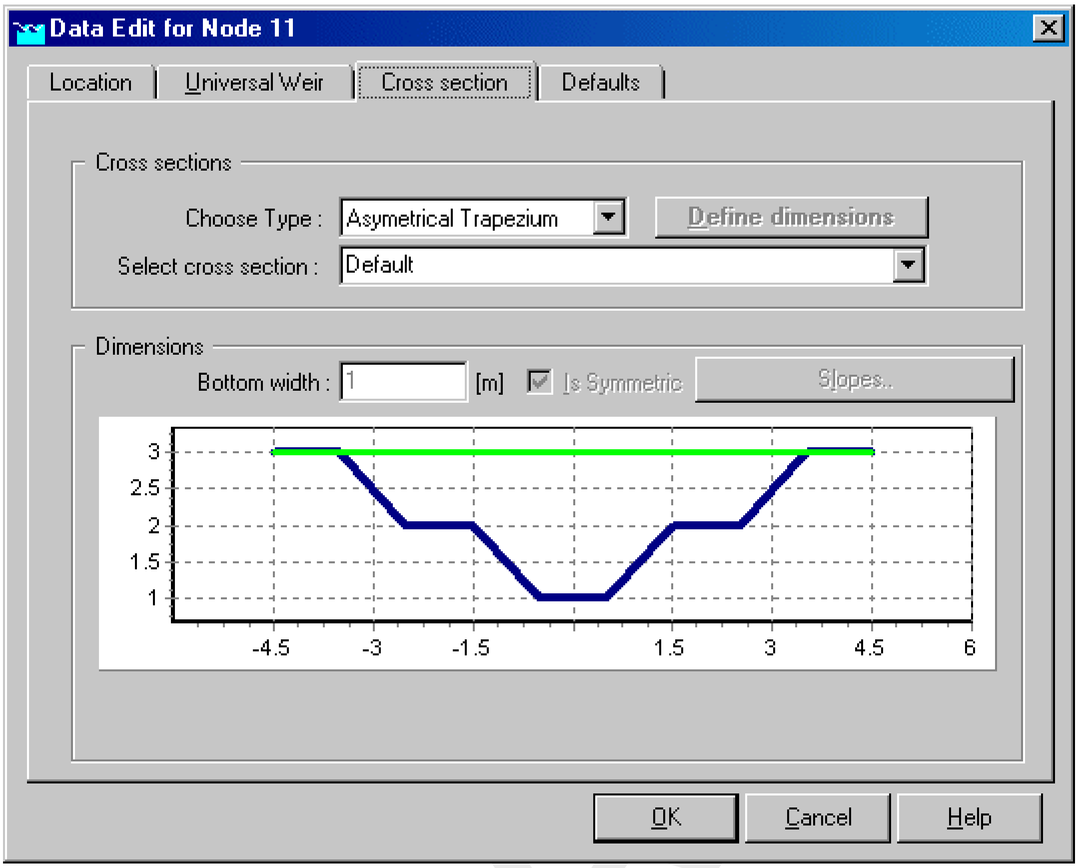


***Figure 5.119: Data Edit** window,* Cross section *tab*

On this tab, the actual shape of the weir's crest should be defined. This is done by means of **cross section definitions**, very similar to the ones you define for the shape of a river bed. The only real difference is that the number of cross section types is limited to two:

- ◇ Y-Z profile
- ◇ Asymmetrical Trapezium

For a detailed description of both cross section types and their input parameter, see the "Functional Reference Manual" - "Channel Flow module" - "Cross section types" section.

5.3.20 Flow - Weir

Description

The *Flow - Weir* is a broad crested weir. Crest levels cannot vary in height along the weir. For the applied mathematical equations, reference is made to section section 6.1.16.16.

Input screens
When starting the model data editor for an *Flow - Weir* node type, the following tabs will be available for input:

Weir:

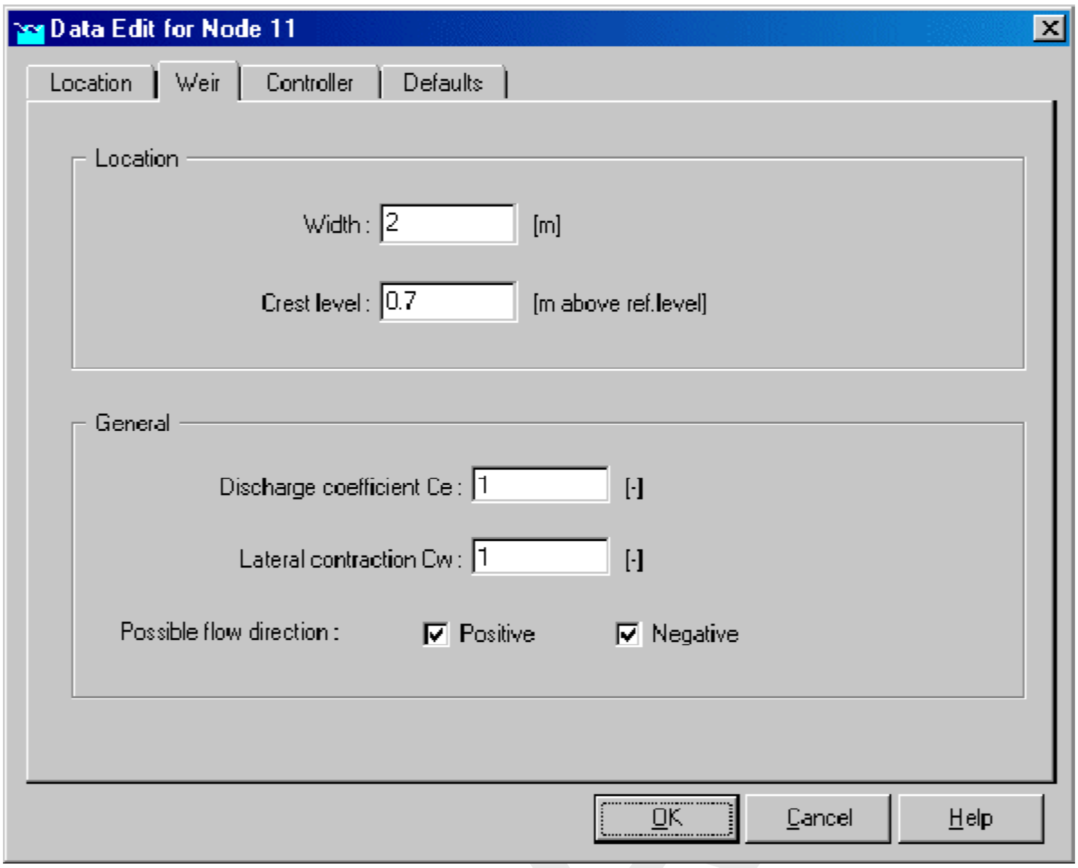


***Figure 5.120: Data Edit** window,* Weir *tab*

On this tab, the weir's general dimensions can be filled in:

- **Width** This parameter represents the width of the crest over which the water will flow.
- **Crest level** This parameter represents the level (with relation to reference level) of the weir's crest. Note that the value you fill in here may be overruled by a controller, if you desire.
- **Discharge coefficient Ce** This parameter represents the discharge coefficient that is used for calculation of the discharge. See the Technical Reference Manual for the exact mathematical implementation of this parameter.
- **Lateral contraction coefficient Cw** This parameter represents the energy loss that is caused by contraction of the flow towards the weir. This phenomenon generally occurs when the weir's crest is less wide than the channel. See the Technical Reference Manual for the exact mathematical implementation of this parameter.
- **Possible flow direction** This parameter defines in which direction(s) water can flow over the weir. The directions are defined with relation to the defined branch direction. Positive means: in the same direction as the defined branch direction; negative means: in the opposite direction than the defined branch direction. If you don't know the defined direction of the branch, read the Flow - Branch topology section of the Functional Reference Manual.

Controller:

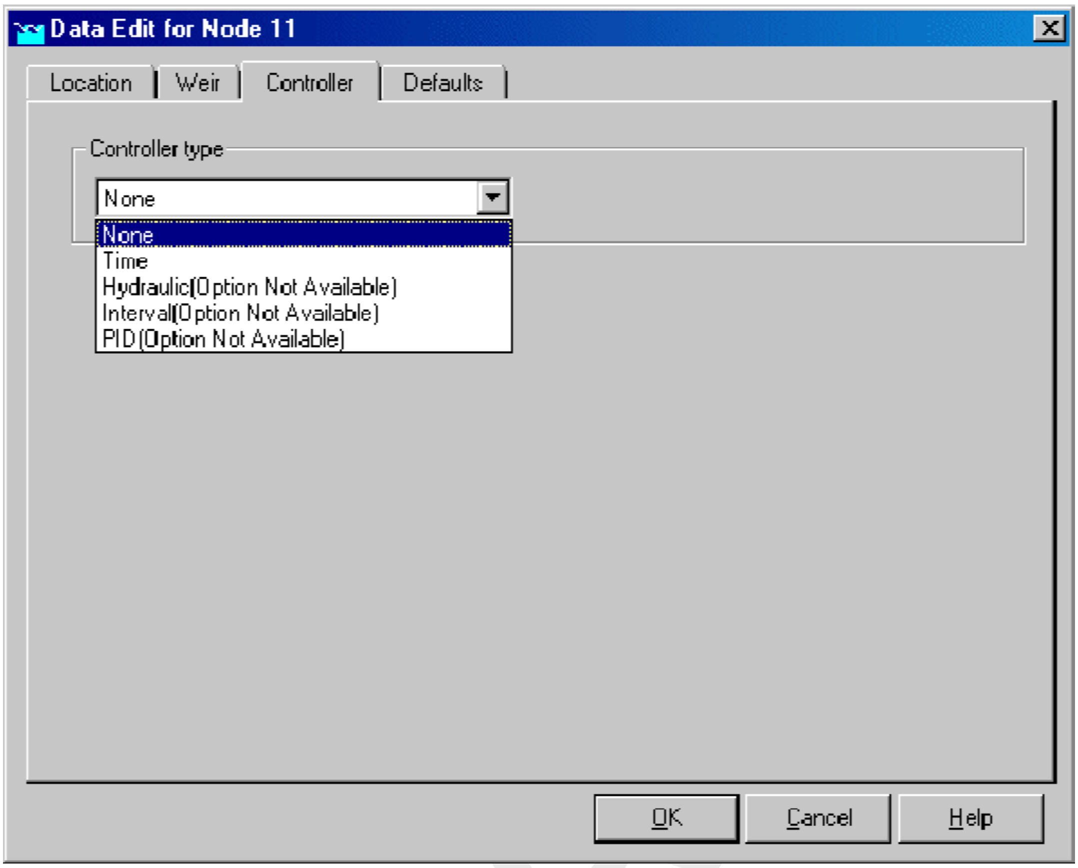


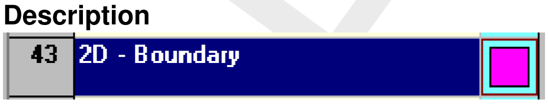
***Figure 5.121: Data Edit** window,* Controller *tab*

On this tab, you can define a controller for your weir. A controller will overrule your general settings for the weir's crest level and apply the chosen controlling rules to it. Read more about them in the Controller section of the Technical Reference Manual.

Note that three of the four available controller options only become available after you have defined a Flow - Measurement station within the schematisation and saved that schematisation.

5.3.21 Flow - 2D-Boundary

Description



General description:

The 2D - boundary node works more or less the same as its counterpart in the 1D-channel flow module. The boundary condition is either a water level or a discharge, both of which can be either constant or varying in time.

Important: the grid cell on which a 2D-Boundary node is placed, should be given the no-data value! (often -9999) If not, this object will **not** work.

How the 2D-boundary node works:

Any 2D grid cell can discharge into any of the four cells directly surrounding it. However, a 2D cell containing a boundary can only discharge into (any) ONE of these cells! By default, it discharges into the cell directly to the right of it. The flow direction can be controlled by defining no-data values in the surrounding cells to which the boundary cell should NOT discharge. This way, if you define 3 no-data values out of 4 surrounding cells, the boundary cell will discharge into the remaining cell (see example Figure 5.122).

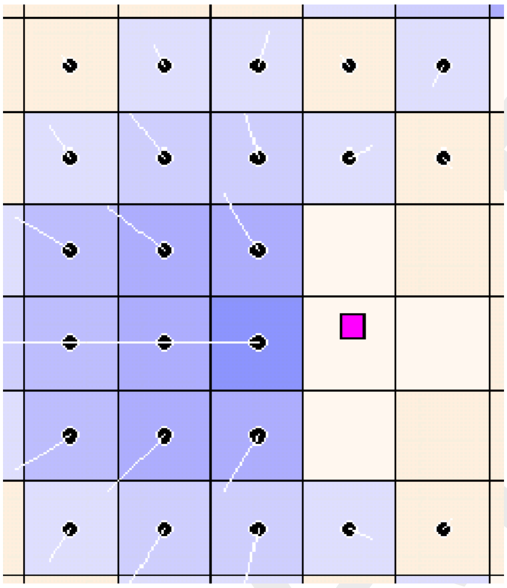


Figure 5.122: *Possible configuration of a 2D boundary node. Note: the white (transparent) cells contain no-data values*

In case of a 2D $h(t)$ boundary condition, the water level is directly imposed at the active grid cell that lies in front of the concerning 2D boundary grid cell. Hence there is no water movement over the concerning 2D boundary grid cell.

In case of a 2D $Q(t)$ or 2D Q-h boundary, an artificial 1D link is created from the concerning 2D boundary grid cell towards the active 2D grid cell that lies in front of this 2D boundary grid cell. The hydraulic properties of this artificial 1D link are based on the properties of the active 2D grid cell lying in front of the 2D boundary grid cell. This implies that in case of a 2D $Q(t)$ or 2D Q-h boundary water movement over the 2D boundary grid cell is taken into account.

Suggestions for the user:

If you decide to use a Discharge boundary condition as part of the 2D schematisation, it is wise to define an initial water level point in the 2D grid cell into which the cell containing the boundary condition should flow. The reason for the initial water level point in this case is purely a numerical one, as sometimes no water will flow at all from the discharge boundary. The water depth can be very small, for example one centimeter.

The initial water level node will (instantly) fill up all surrounding 2D grid cells with water, until cells which have a higher terrain level are branched, or if the edge of the grid is branched. This can be a problem when using a discharge boundary condition on a high point in the grid: an initial water level used for numerical reasons may fill a large part of the grid! This problem can be solved by lowering the terrain level of the 2D grid containing the initial water level point slightly below the level of the surrounding cells.

One more rule to remember when using this node is that the 2D grid cell containing the node should be defined manually as ’no-data value’!. This can be done in ’edit model data’ mode, under the properties of the 2D grid.

Finally, it is not possible to add this type of node to a grid cell that already contains a 1D-boundary node.

2D-Boundary nodes are normally used in situations where the dam break is modelled starting in the 2D schematisation, as opposite to starting in the 1D-schematisation.

Editing 2D-Boundary data

After selecting any 2D-boundary node in the edit model data mode and pressing *Edit*, the data edit window for this node type pops up. The first tab ’location’ displays the (non-editable) id and location of node.

Under tab ’boundary condition’, the user can specify the boundary condition. See Figure 5.123. There is a choice between a Water level boundary condition or a Flow boundary condition. Both of these can be given either a constant value or a value changing in time.

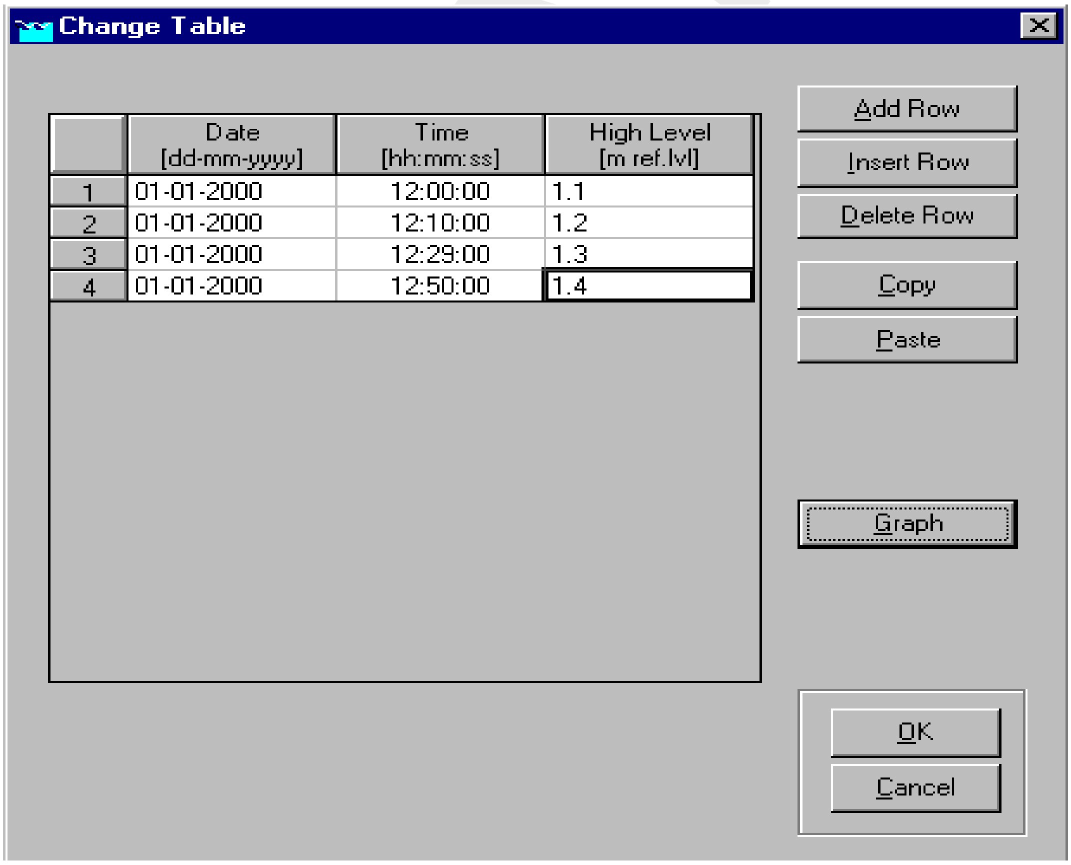


Figure 5.123: *2D-boundary edit window - boundary condition tab*

A variable water level or flow in time can ben entered by pressing *table. . .* and filling in the table, as shown in the example in Figure 5.124:

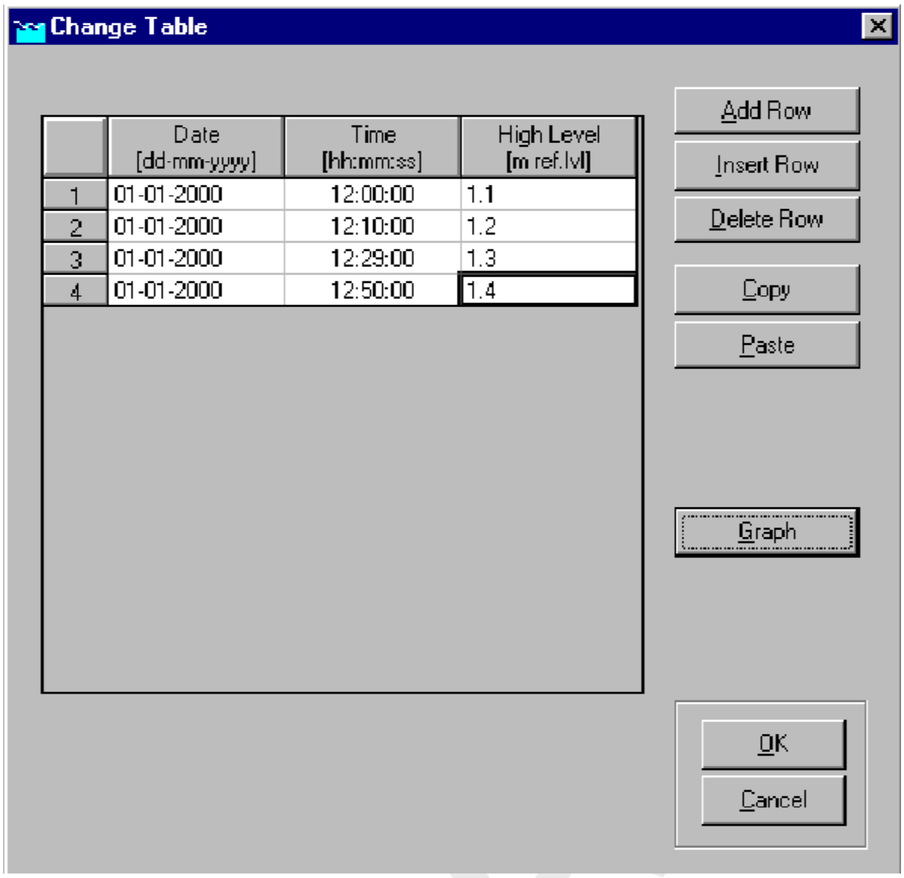


Figure 5.124: *Change table window*

Please note that the water level is defined relative to the reference level (positive above the reference level), and that it is not possible to enter a water depth as a boundary condition.

Also note that a positive value for the flow means input into the system, while a negative value means output.

2D Q-h tabulated boundary condition

Functionality:
The 2D Q-h tabulated boundary condition enables the user to apply a Q-h relationship as the (downstream) boundary condition of a 2D grid. A Q-h relationship is to be defined as discharges as function of water levels $Q = f(h)$. Presently a 2D Q-h tabulated boundary condition may **not** pass over nested grids and should be parallel to either the x-axis or y-axis of the 2D grid. Further on a 2D Q-h tabulated boundary condition is supposed not to pass over a 1D channel flow schematisation.

How to define:

- ◇ Double click the 'Schematisation' task block and thereafter click the *Edit Model* button on the Schematisation window.
- ◇ Click the *Zoom in* button and go to the location where you want to place the 2D Q-h Tabulated boundary condition.
- ◇ Click the *Edit Network* button.
- ◇ Click the *Nodes* button, pull-down the menu and click the Overland Flow Model/ 41, 2D - Corner node.

- ◇ Click the *Branch* button, pull-down the menu and click the Overland Flow Model/ 16, 2D - Line Boundary.
- ◇ Click the *Edit action* button, pull-down the menu and click on Nodes/Add Nodes. Go to the location where you want to place the first 2D Corner node of the 2D Q-h Tabulated Boundary Condition and click with your left mouse button. Place the second 2D Corner node in the same way.
- ◇ Click the *Edit action* button, pull-down the menu and click on Connection/Connect nodes. Click with the left mouse button on your first 2D Corner node. While pressing down the left mouse button drag a line to your second 2D Corner node and release your left mouse button.
- ◇ Click the <Edit Network> button to end the network editing activities.
- ◇ Click on the 2D Q-h Tabulated Boundary Condition and thereafter click with your right mouse button, and click on Model data/ Overland Flow Model. The 'Data Edit' window should appear on your screen.
- ◇ Click the 'Q-h relation' check-box in the 'Boundary condition' Tab of the 'Data Edit' window.
- ◇ Click on the <Table> button in the 'Data Edit' window to enter your Tabulated Q-h relation in the 'Change Table' window.
- ◇ Close both the 'Change Table' and the 'Data Edit' windows by clicking OK.
- ◇ Click the *Save Network* button for saving your defined 2D Q-h Tabulated Boundary Condition.

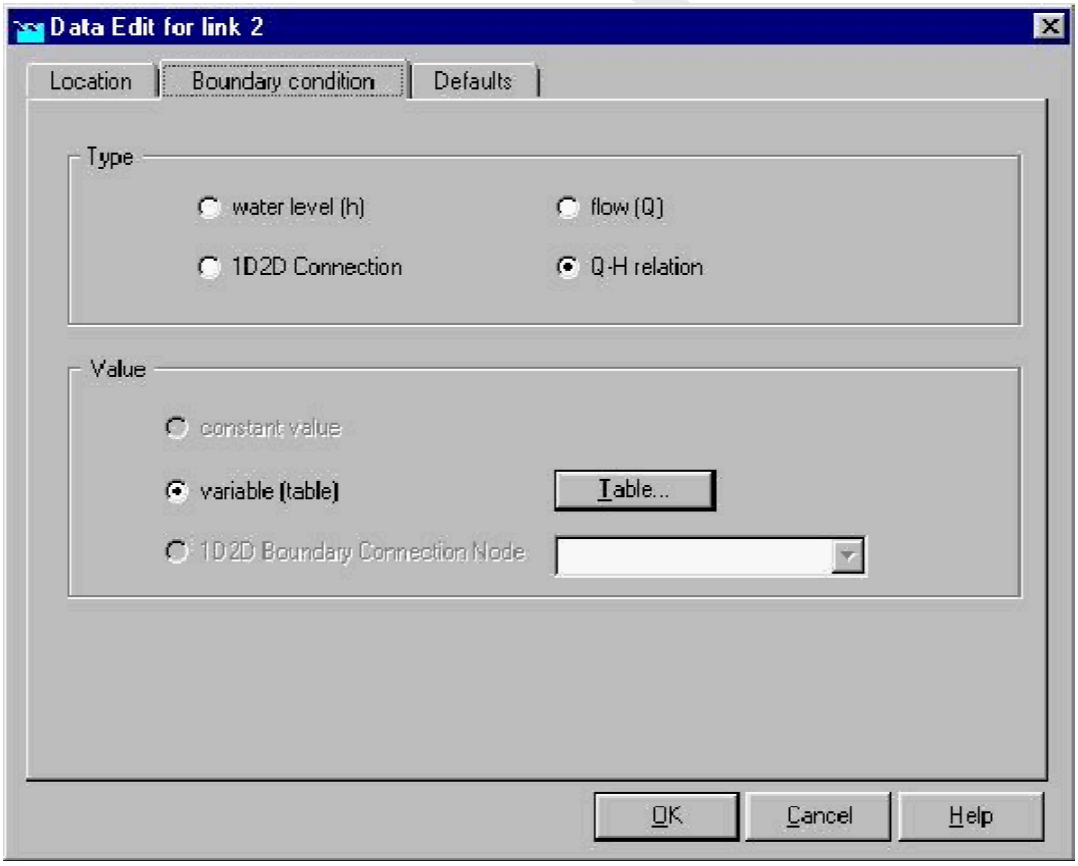


***Figure 5.125: Data Edit** window,* Boundary condition *tab*

How to retrieve information:

Under the 'Results in Maps' Task block

- ◇ Double click the ‘Results in Maps’ task block.
- ◇ Click in the main menu on File/Open Data/Overland Flow Module Result at history stations (i.e. in the ‘Select item’ window)
- ◇ In the ‘View Data’ window select for instance the hydraulic parameters "Water level" and Abs.Disch $[m^3/s]$’.
- ◇ Click the 2D Q-h Tabulated Boundary Condition and then click with right mouse and then on show graph or click on the <graph> button in the ‘View Data’ window.

Under the ‘Results in Charts’ Task block

- ◇ Double click the ‘Results in Charts’ task block.
- ◇ Double click on Overland Flow Module/ Results at history stations (in the ‘Results in Charts’ window).
- ◇ In the ‘ODS-View’ window select under parameters Water level and Abs Disch $[m^3/s]$, under location select location l_x (where l stands for link and x for the ID of the 2D Q-h Tabulated boundary condition), under time-steps select the required time-period. Now click on the *Graph* button or *Export* button to respectively view a graph or export to the data to a particular file.

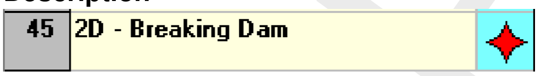
2D Refined Q boundary condition

Functionality:

In effect the 2D refined Q-boundary condition refers to an improvement of the existing 2D Q-line boundary condition. In the past imposed discharges were equally distributed over the number of concerning 2D grid cells. Nowadays imposed discharges are distributed in accordance with the conveyance capacity of the concerning 2D grid cells.

5.3.22 Flow - 2D-Breaking Dam

Description



Dike bbranches can be modeled either in 1D (see Branch – Flow Dam Break) or in 2D using a **2D Breaking - Dam Node** (see Figure below). You have two options for controlling the bed level of the 2D grid cell that lies underneath a 2D Breaking – Dam:

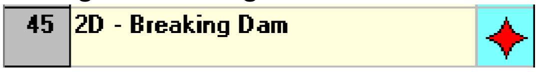
- ◇ Using the Decrease in Height Time Table,
- ◇ Using a controller that can be overruled by RTC or RTC-Matlab. For more information see Module – SOBEK Real Time Control (RTC).

For information on how to use these control options, see Editing Node 2D Breaking Dam.

Figure 5.126: *Example of 2D- Breaking Dams located on 2D grid cells*

Remark:
◇ A 2D breaking dam node may **not** lay on an active 2D grid cell that is surrounded by a boundary condition cell.

Editing 2D-Breaking Dam data



The 2D-Breaking Dam enables you to control the bed level of its underneath lying 2D grid cell. Following control options are available:

◇ Option 1: The Decrease in Height Time Table,
◇ Option 2: A controller that can be overruled by RTC or RTC-Matlab. For more information see Module – SOBEK Real Time Control (RTC).

You can select either option 1 or option 2 by checking its corresponding radio bullet (see Figure 5.127 below).

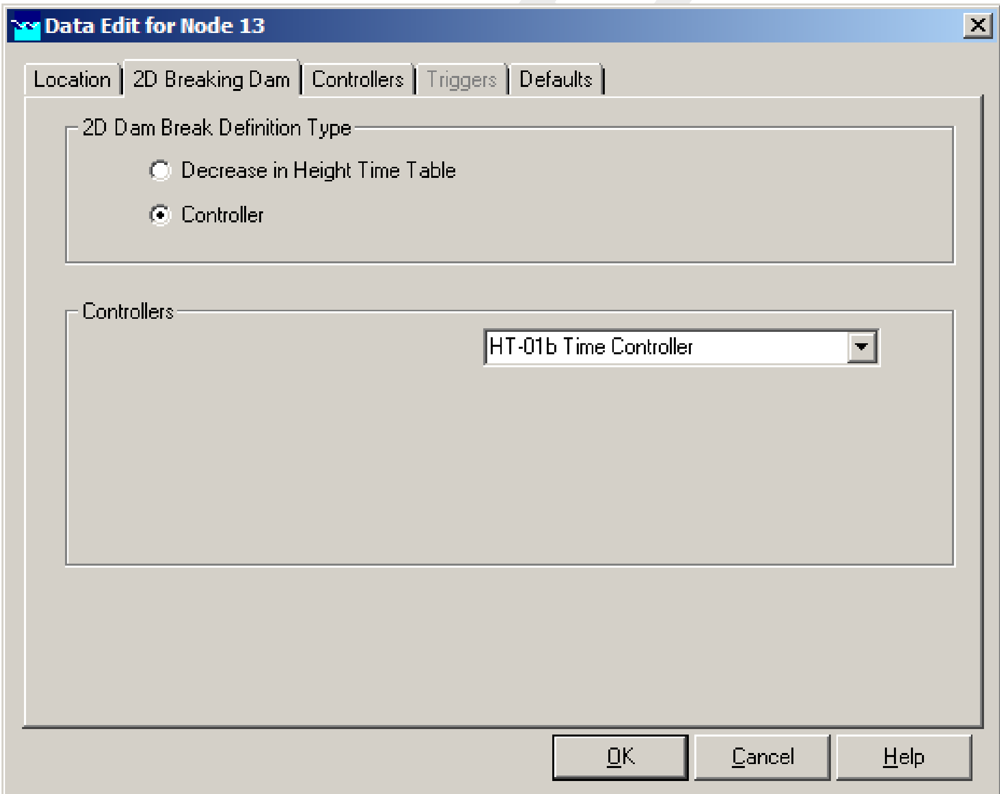


Figure 5.127: *Example of selecting the controller option for a 2D Breaking - Dam*

Option 1: The Decrease in Height Time Table

Check the “Decrease in Height Time Table” radio bullet and click on the *Table* button. You can now specify the lowering of the 2D grid cell lying beneath your 2D Breaking – Dam as function of time (see Figure 5.128).

Note:

1 You specify the lowering with respect to the start level (or initial bed level) defined in your 2D bed level grid (i.e. <*.asc> file). Positive and negative values respectively mean that your grid cell will moved downwards or upwards,
2 SOBEK applies linear interpolation in the Decrease in Height Time Table,
3 In case the actual computational time is before the first time or after the last time specified in your Table, SOBEK will respectively apply the value on the first row or the value on the last row of your Time Table.

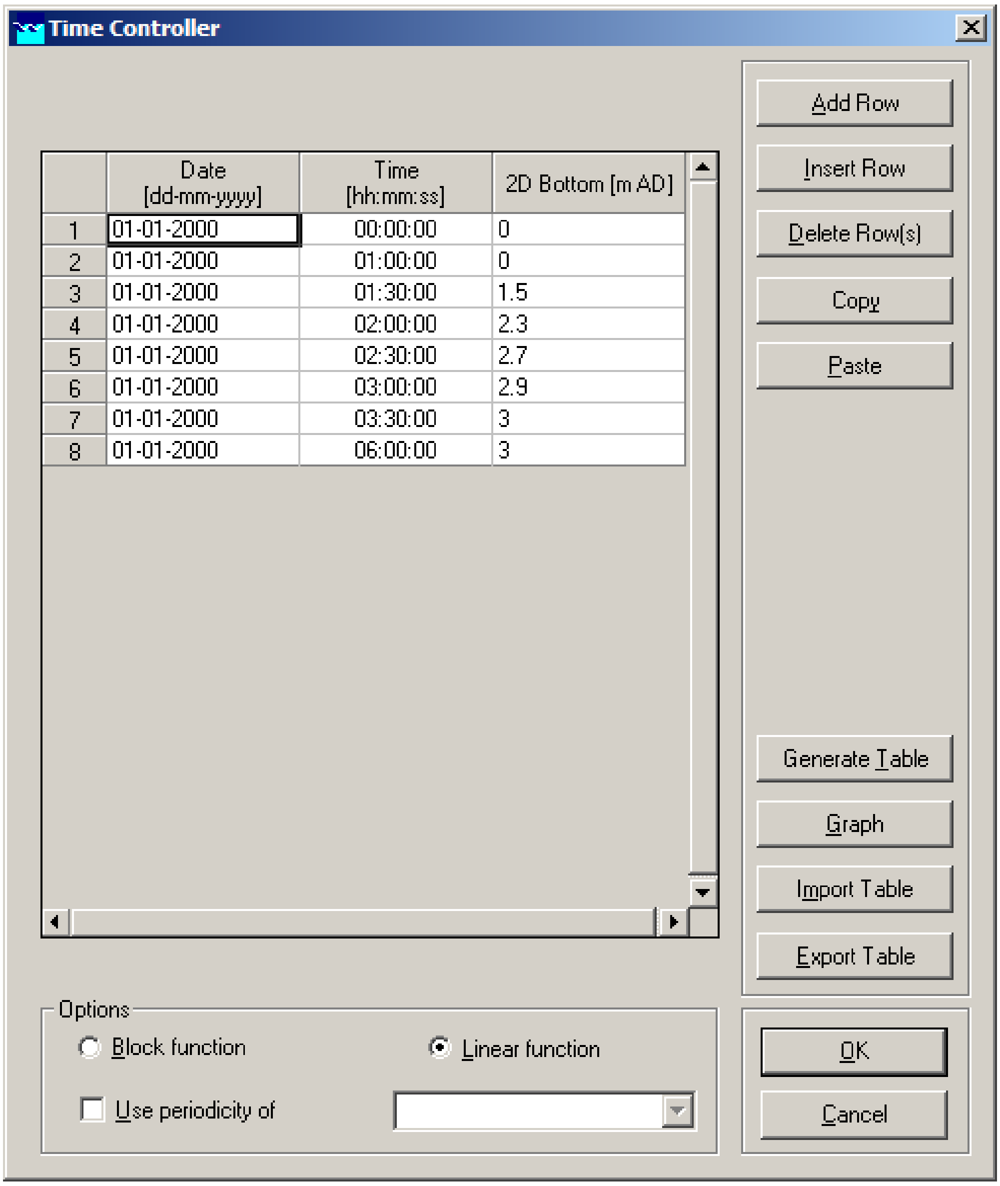


Figure 5.128: *Example of a Decrease in Height Time Table available at a 2D Breaking – Dam*

Option 2: A controller

Firstly you have to define a controller. Thereafter you can select this controller as the one providing set points for the bed level of the 2D grid cell lying beneath your 2D Breaking –

Dam.

Defining a controller for a 2D Breaking – Dam:

Click on the "Controllers" Tab (see Figure 5.129) and type a name in the "Definition" scroll box. Now click the *Define* button and specify your controller. Don't forget to click the *Save* button in order to save your controller definition.

Remarks:

- ◇ Presently only a Time Controller providing bottom(bed) levels as set point for a 2D Breaking – Dam can be defined,
- ◇ Update frequency=1, means that the set point of controller is updated after each hydrodynamic time step, defined in Settings; Update frequency=10, means that the set point of the controller is updated every 10 hydrodynamic time steps,
- ◇ dValue/dt=0.002 m/s, means a maximum speed for the change in bed level (either positive or negative) of 0.002 m/s. Hence in one minute a maximum bed level change of 0.12 m (=0.002 m/s * 60 s) is allowed,
- ◇ Click on the *Time Controller Table* button for specifying your set points.

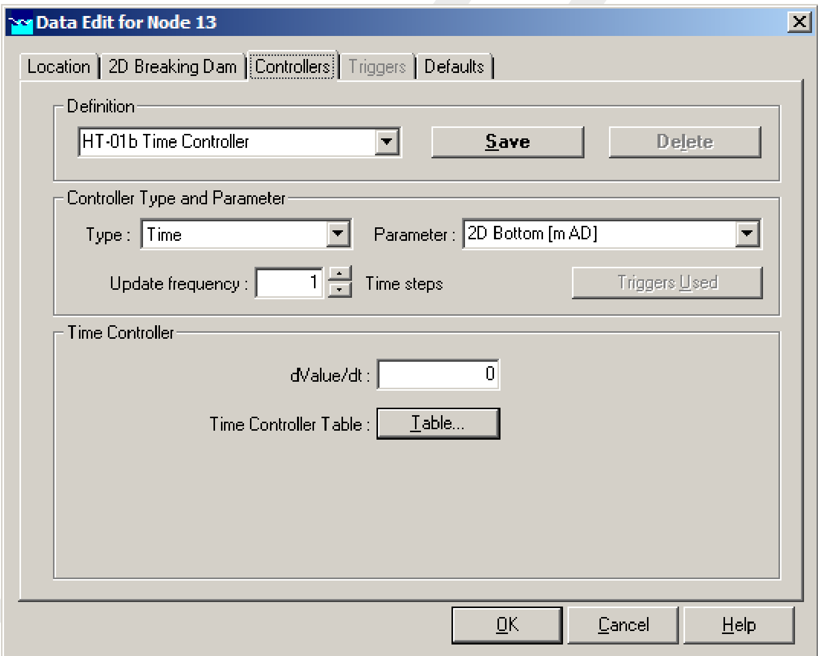


Figure 5.129: *Example defining a controller for a 2D Breaking – Dam*

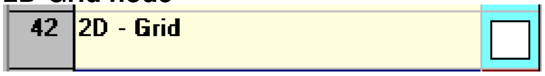
Assigning a controller at a 2D Breaking – Dam:
Click on the 2D Breaking – Dam Tab and check the "Controller" radio button. Now select your previously defined controller in the "Controllers" scroll box (see Figure 5.129 above).

Remark:

- ◇ In the "Controller" scroll-box you can only select so-called 2D Flow controllers. Hence you cannot select a 1D Flow controller or a RR controller.

5.3.23 Flow - 2D-Grid

2D-Grid node



This is one of the most important building blocks for the modelling of 2D systems, and the one you would usually start with. It is referred to in the user interface as a 'node' which at first sounds confusing, as this node actually represents a whole grid.

A grid can be either imported from a GIS system, or defined within SOBEK as a new grid.

Importing a '2D grid' is possible by selecting the 'import 2D grid' option, an option which has been added to the standard SOBEK edit network menu (see Figure 5.130). Note that this option is only available after you have selected the '2D grid' node as a building block.

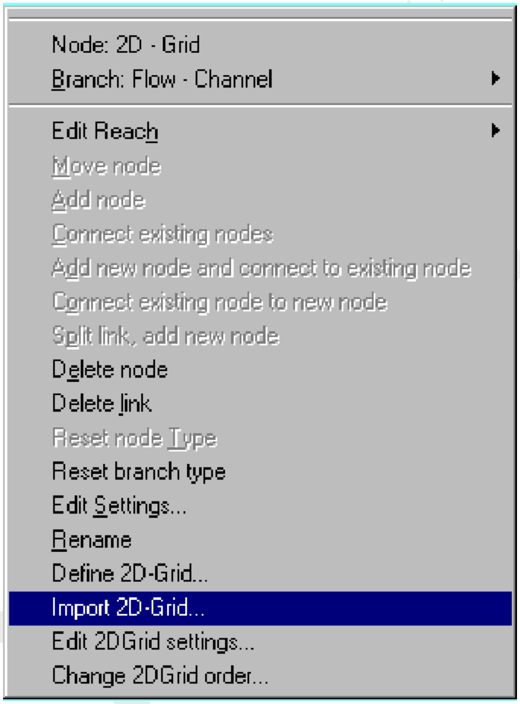


Figure 5.130: Edit network *menu,* Import 2D-Grid... *menu*

Edit network menu

After you have selected this option, you need to select the <asc>-file that contains the grid information you want import. As mentioned before, this <asc>-file is a standard grid definition file, which can be generated as output by for example ARCView (see appendix for an example). The file contains the following information about the grid:

- the number of columns
- the number of rows
- x-coordinate of the bottom left corner of the grid
- y-coordinate of the bottom left corner of the grid
- cell size of the grid elements (same for x and y size)
- the no_data value ('missing value'), usually -999 or -9999
- For every cell the terrain level.

In theory, you can select the file from any given path, for example <c:\gisfiles\grid1.asc>. However, we strongly recommend using the default directory for these filenames, which is the <*projectname*\fixed\> directory. This makes it easier to make a copy of an existing project and give it to somebody else.

After you have successfully imported the grid, it should appear on the map at the coordinates specified in the file. You can then visualise the terrain levels of all grid cells by turning on the *active legend* (under *options*) and activate the *z-data* (model data) from the legend panel. The active legend will be explained later on.

Most of the time, grids will (need to) be imported from GIS based programs. It is however also possible to **define the 2D grids** by hand, by selecting the 'define 2D grid' option from the *edit network menu* (see Figure 5.131). This means that the user will not only have to define the exact location and size of the grid, but also the terrain level for every grid element. In theory, one can place the grid at any given location and make it any given size. However, the way the grids are defined have a major impact on the accuracy of the results, so accurate definition of 2D grids is very important. There are a number of considerations important in this matter:

- ◇ The number of grids and the number of elements determine the simulation time. The less elements there are, the shorter the simulation takes.
- ◇ The smaller the grid elements are, the more accurate the results can be. It is therefore a good idea to choose a coarse grid in (most) places where accuracy is not required, and a more fine grid in places where results are critical, for example close to the inlet point/ dam break point. Another option is to choose the resolution of the grid in the same order as the resolution of the available terrain level input data.
- ◇ The grid elements containing a no-data value don't participate in the calculations; they are used to specify the barrier around the area of interest, and possibly as no-flow locations within the area of interest.

Grid properties

For every grid you need to specify a number of properties, some of which need to be specified immediately, while others can be defined later in the 'model data' mode.

Figure 5.131 shows the properties that need to be defined.

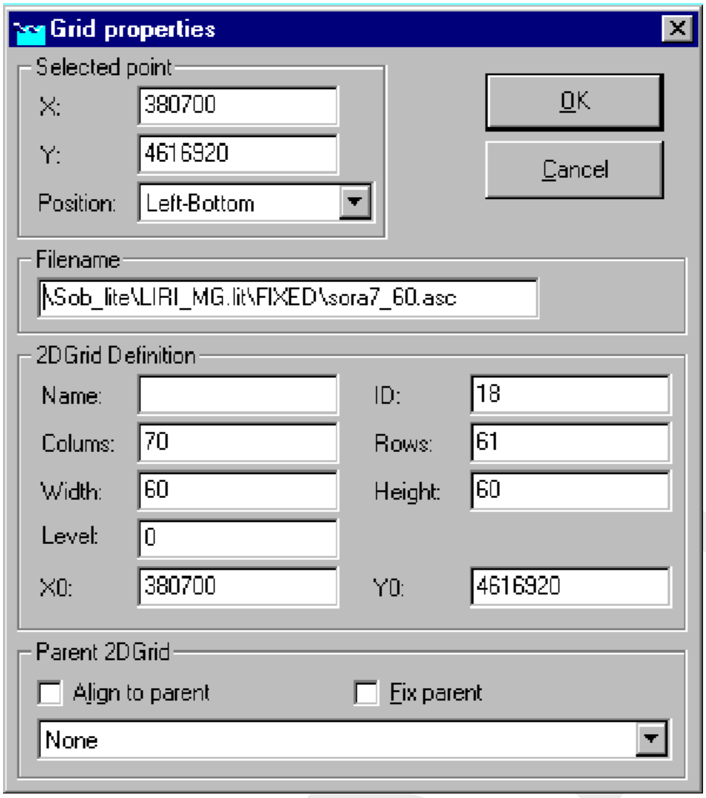


Figure 5.131: *Grid properties*

Selected point information

The 'X' and 'Y' coordinates are the coordinates of the map position you selected by pressing the mouse button just before entering this menu. They represent a (any) corner of the 2D grid you are about to define. If you are not satisfied with the coordinates, you can alter them manually here.

The Position represents which of the four corners should appear at the X-Y coordinate you defined. By default it's the left-bottom corner.

Filename

The default location for putting the .asc files is the <*project directory*\fixed\> directory. Make sure you select a path before saving the grid file.

2D grid definition

◇ Name If you want to, you can specify a unique grid name. This is not absolutely necessary, as a default name will be defined for you if you don't specify a name here.
◇ Columns/Rows Every grid is rectangular in shape, the size of which is determined by the number of rows and columns. The area of interest is inside the rectangular grid, which is filled up with the no-data values.
◇ Width/Height [m] The width and height of every cell are uniform for the whole grid, so it's not possible to specify varying grid size elements throughout one grid. However, different grids within one schematisation can have different grid sizes.
◇ Level [m] (above OR below reference level, dependent on your definition in SETTINGS) When using the define 2d grid option, it is only possible to define one uniform terrain level.

Later on, in the edit model data mode, there is the possibility to change terrain levels for single grid elements.

◇ X0/Y0 This is the same coordinate as specified under selected point.

Parent 2D-grid

This option is only relevant when using multiple grids that are either **overlapping** or **nested.**

If you select the *align to parent* option, the child grid will be replaced according to the parent grid. If you select *Fix parent*, the position of the parent grid will be changed according to the position of the child grid. Note that the position of the top left corner of the grid selected will be changed, and not the number of rows or columns. So, make sure to check both the top left and bottom right corner of the child grid afterwards to check the alignment. It may be necessary to adjust the size (columns/ rows) of the child grid manually.

Note: due to an error in the options 'align to parent' and 'fix parent', these options should not be used at the moment!

Edit 2D-grid. . .

This option is available under the edit network menu (see Figure 5.130). Opening the grid properties window (see Figure 5.131), it gives the user the possibility to change grid properties, after the grid has been defined. If you decide to resize the grid (number of columns and/ or rows, don't forget to edit the terrain levels in the <asc>-file accordingly.

Before you can edit the properties of any grid, you will first have to select the node that represents the grid. And in order to be able to select this type of node, they first have to be made visible. This can be done by selecting *options* → *network options* → *nodes. . .* select the 2D grid node and turn on visibility (see Figure 5.132). The 2D-grid nodes should now be visible in the schematisation.

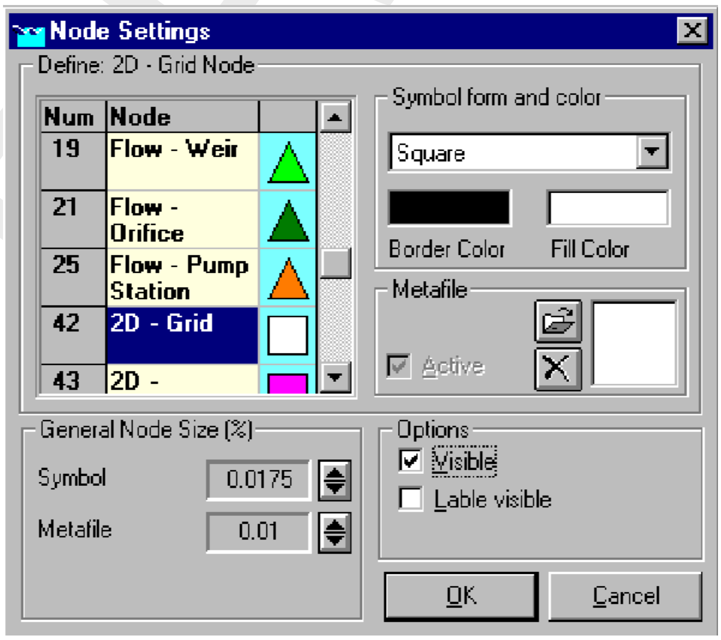


Figure 5.132: *Node settings*

Change 2D-grid order. . .

After selecting this option from the edit network menu, Figure 5.133 appears:

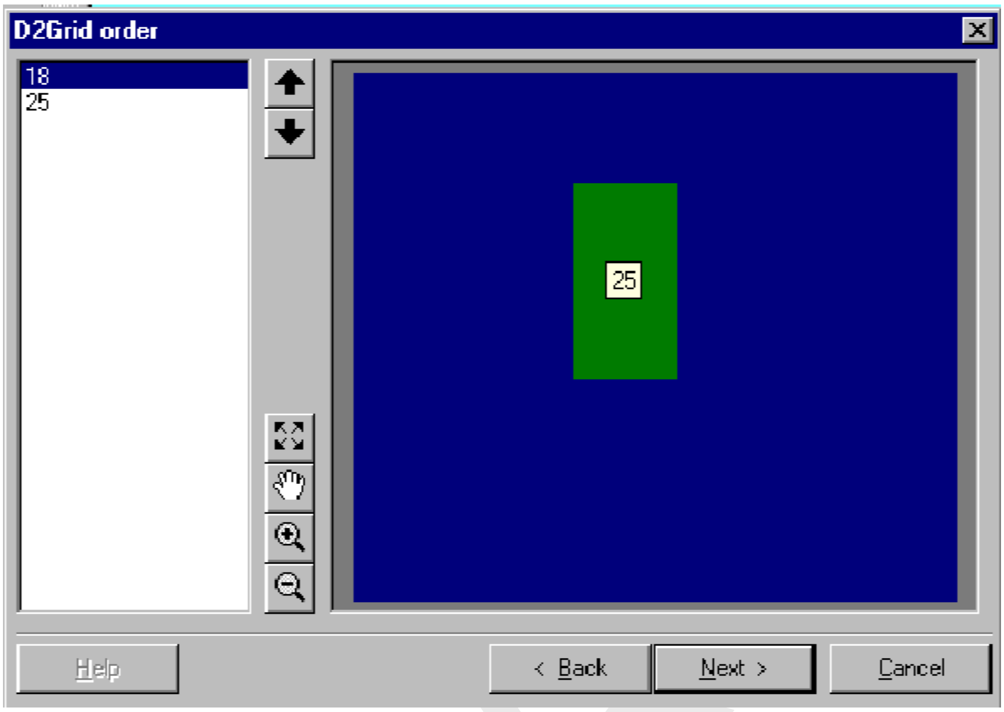


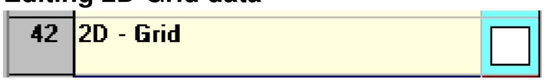
Figure 5.133: *2D Grid Order Window*

Here you can change the order of nested grids according to the rules specified in the FAQ on multiple grids. In short, the child grid (in Figure 5.133 the small green grid, grid 25) should be 'on top of' the parent grid (the blue grid, grid 18).

This means that in the table on the left, the parent grid should be mentioned BEFORE the child grid. If the grid order is wrong (clearly visible in the 2D grid order window because the child grid is hidden behind the parent grid), the simulation will stop. By default, the grid order is defined by the order in which the grids are defined in the first place. So if you first define the parent grid, and then the child, the grid order will be correct. If you first define the child, and the parent, the order needs to be changed using this option.

So, it is very important to always check this option after defining new grids!

Editing 2D-Grid data



After a 2D grid node has been selected for editing, window Figure 5.134 appears:

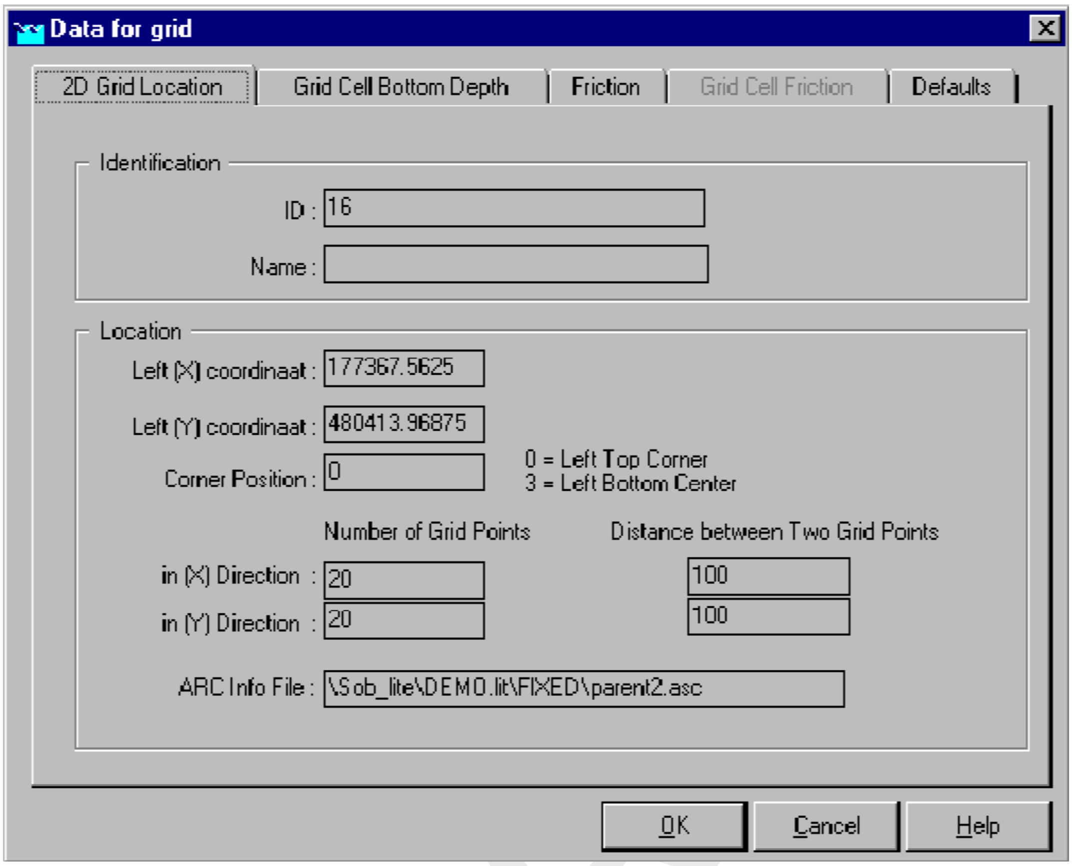


***Figure 5.134: Grid data** window* - 2D Grid location *tab*

This tab displays general information about the selected grid, which was already defined during the edit network mode. If you want to change any of the data specified here, you need to go back to the edit network mode and use the 'Edit 2D grid settings' option.

The next tab contains the terrain levels for all 2D grid elements:

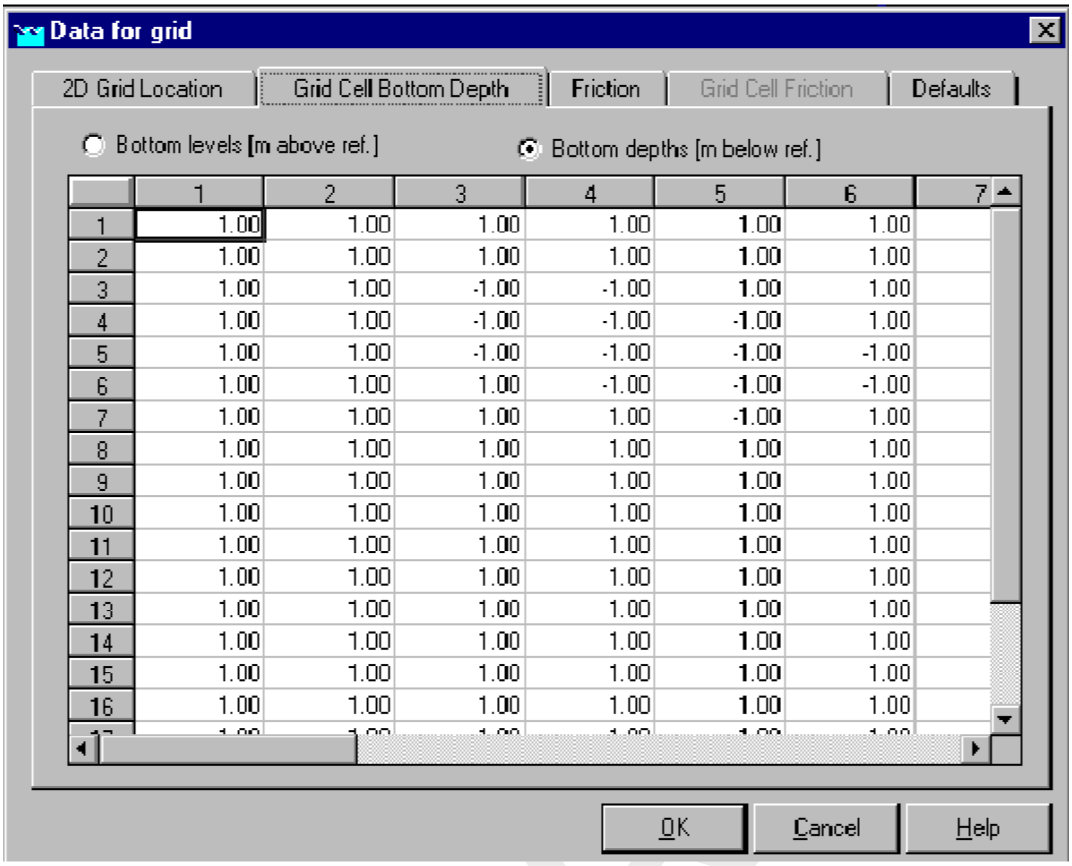


***Figure 5.135: Grid data** window* - Grid Cell Bottom Depth *tab*

In this window, two aspects are of interest:

◇ The terrain levels in this window can be displayed as either bottom levels (m above ref.lev.) or bottom depths (m below ref.lev.), whichever one the user prefers. The option selected here does not influence the simulation in any way, nor the way the results are displayed.
◇ The table contains terrain levels (either depth or height) for all grid-cells. Cell (1,1) represents the top-left corner of the grid. The user is free to alter any of these values. When the user presses *OK* after anything has been changed, the changed .ASC file containing the terrain levels can be saved under the same name or another. Please verify that the file is written to the correct directory (preferably <*casename*\fixed>), and when you enter a new filename, don't forget to add the <asc> extension.

The next tab contains the friction values for all grid cells:

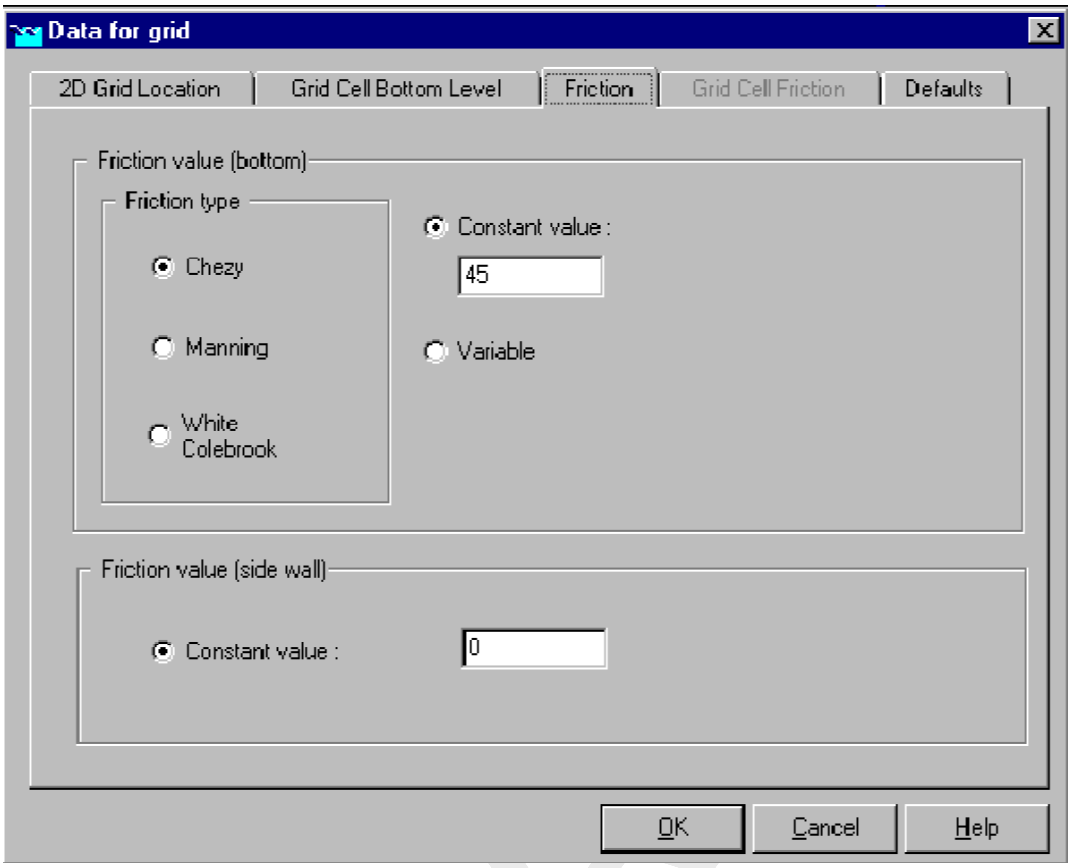


***Figure 5.136: Grid data** window* - Friction *tab*

The user can choose between three types of friction formulations, Chézy, Manning or White-Colebrook. Only one type of friction can be selected per grid. The user can either define one (uniform) friction value for the whole grid, or select a <asc>-file containing friction values for all 2D grid elements as a way to model distributed friction. This <asc>-file should have exactly the same format (including number of columns and rows) as the height definition file. It is possible, however, to have no-data values in 2D grid cells in the height <asc>-file where there ARE values in the friction <asc>-file. The other way around is not possible.

Note: that in the user interface, the number on the horizontal axis have been replaced by letters, ranging from 'A' to 'XX'.

The next tab, *grid cell friction*, displays a table with all friction values. This tab is only available when a distributed friction file has been selected first. It is not possible to modify any of the values seen here. If the user want to change any of these values, he/she will have to edit the friction <asc>-file directly (outside SOBEK).

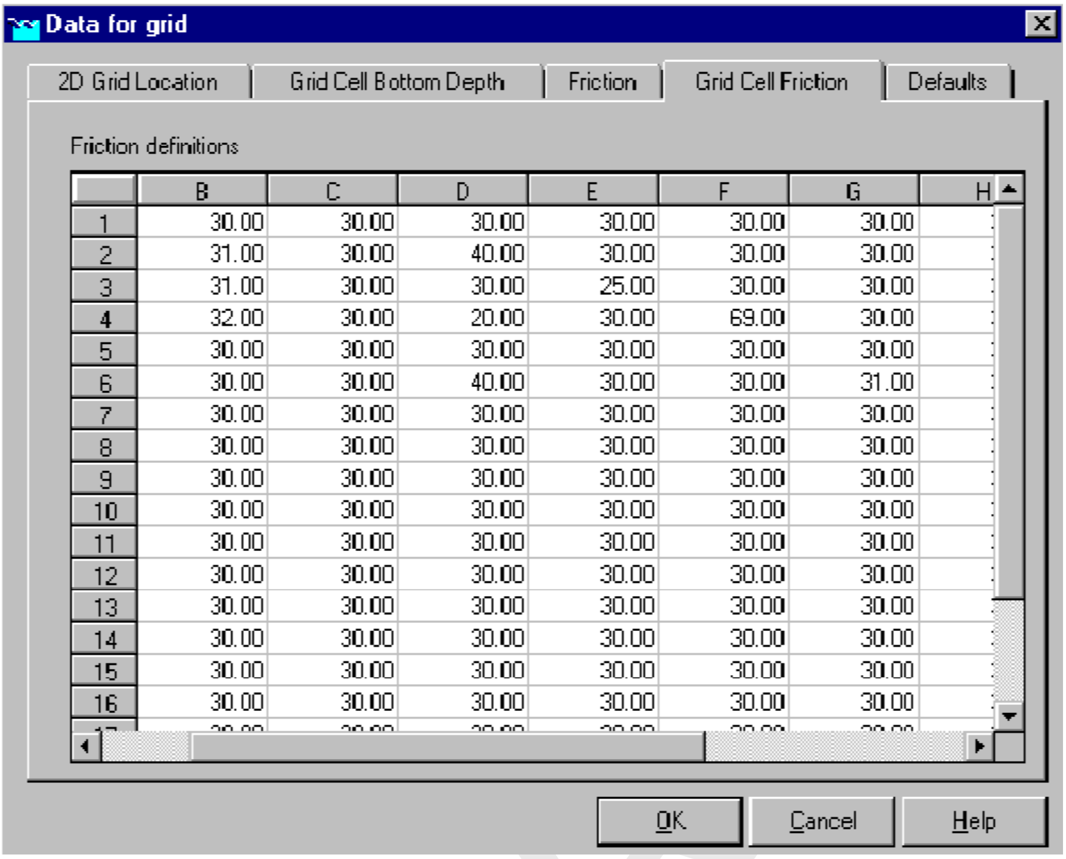
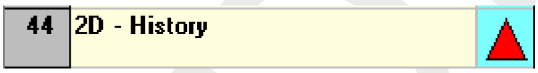


***Figure 5.137: Grid data** window* - Grid Cell Friction *tab*

The last tab, *defaults*, gives the user the possibility to save or load certain grid-related parameters as default. For example, if you save the 'friction' as default (only with uniform friction!), the next grid you define will have the same friction as the current grid.

5.3.24 Flow - 2D-History



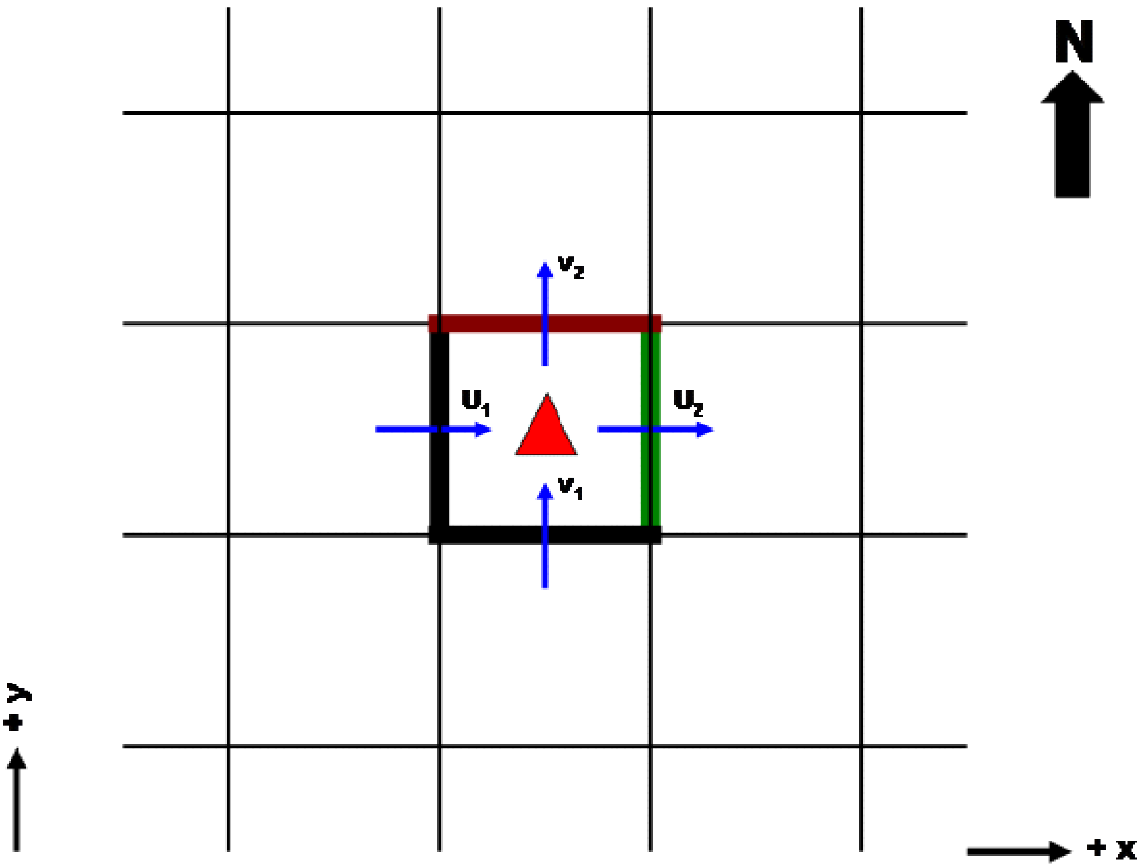
The 2D-history node is a special type of object which can be used to specify (in the Schematisation Taskblock) those 2D grid cells for which output is to be provided. After the simulation has finished, the output will be available as "Results at history stations" in the Result in Maps Taskblock.

In case a 2D-History node is located on a 2D grid cell that has a missing value for the bed elevation, no output for such 2D-History node will available, meaning that all output parameters (see below) will have missing values only. The same yields if a 2D-History node is located on a 2D grid cell of a nested grid that has a missing value, while the underlying parent 2D grid might have a real value for its bed elevation.

Following output is provided for each 2D-History node:

- ◇ Bottom, the bed elevation of its underlying 2D grid cell in m.
- ◇ Waterdepth, the water depth above its underlying 2D grid cell in m.
- ◇ Waterlevel, the water level in m.

- ◇ U-Velocity, the average flow velocity ($\overline{u}$) in x-direction in m/s ($\overline{u} = (u_1 + u_2)/2$; see figure below).
- ◇ V-Velocity, the average flow velocity ($\overline{v}$) in y-direction in m/s ($\overline{v} = (v_1 + v_2)/2$; see figure below)
- ◇ Abs Velocity, the resulting none-directional flow velocity (C) in m/s ($C = \sqrt{\overline{u}^2 + \overline{v}^2}$).
- ◇ Specific U-Discharge, the discharge per meter length (i.e. in m^2/s) flowing in x-direction through the eastern side (green line in figure below) of the underlying 2D grid cell. More precisely, the specific U-Discharge equals the water depth times the u_2-flow velocity at the eastern side of the underlying 2D grid cell.
- ◇ Specific V-Discharge, the discharge per meter length (i.e. in m^2/s) flowing in y-direction through the northern side (brown line in figure below) of the underlying 2D grid cell. More precisely, the specific V-Discharge equals the water depth times the v_2-flow velocity at the northern side of the underlying 2D grid cell.



5.3.25 Flow - 2D initial water level point

Description

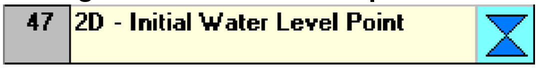
This node can be used to create an initial water level in a part of the grid. This is useful in two cases:

- ◇ Initially there really is water in part of the 2D system, for example a lake.
- ◇ A discharge boundary condition is used as part of the 2D schematisation. In this case, it

is best to define an initial water level point somewhere to make sure that the 2D grid cell adjacent to the cell containing the boundary condition is not initially dry. So, the reason for the initial water level point in this case is purely a numerical one, as sometimes no water will flow at all from the discharge boundary. The water depth can be very small, for example 1 cm.

The initial water level node will (instantly) fill up all surrounding 2D grid cells with water, until cells are branched that have a higher terrain level, or the edge of the grid is branched. This can be a problem when using a discharge boundary condition on a high point in the grid: an initial water level used for numerical reasons may fill a large part of the grid! This problem can be solved by lowering the terrain level of the 2D grid containing the initial water level point slightly below the level of the surrounding cells.

Editing 2D-Initial Water level point data



The only variable that needs to be specified for this node type is the initial water level, relative to the reference level. Note that a positive value means above the reference level, as opposite to terrain levels, which are defined as positive in downward direction.

5.4 Node description (Rainfall-Runoff)

5.4.1 RR - Boundary

RR RR - Boundary node

In this chapter, the *RR boundary* node is described. This node type is used to define boundary conditions for a Rainfall-Runoff schematisation.

◇ For a detailed description of this node's input parameters: see the "boundary node input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the "boundary node topology" section from the Reference Manual

What does an RR boundary node do?
RR boundary nodes hold a certain water level, which will form a boundary condition for the schematisation. This boundary condition is then used to calculate the interaction with the other RR nodes that are connected to it. For example: if an RR unpaved node is directly connected to an RR boundary node, on every time step the outflow of the unpaved area towards that boundary depends on the groundwater level and the boundary value.

Input screens for the RR - Boundary node type
When starting the model data editor for an *RR - boundary node*, the following tab will be available for input:

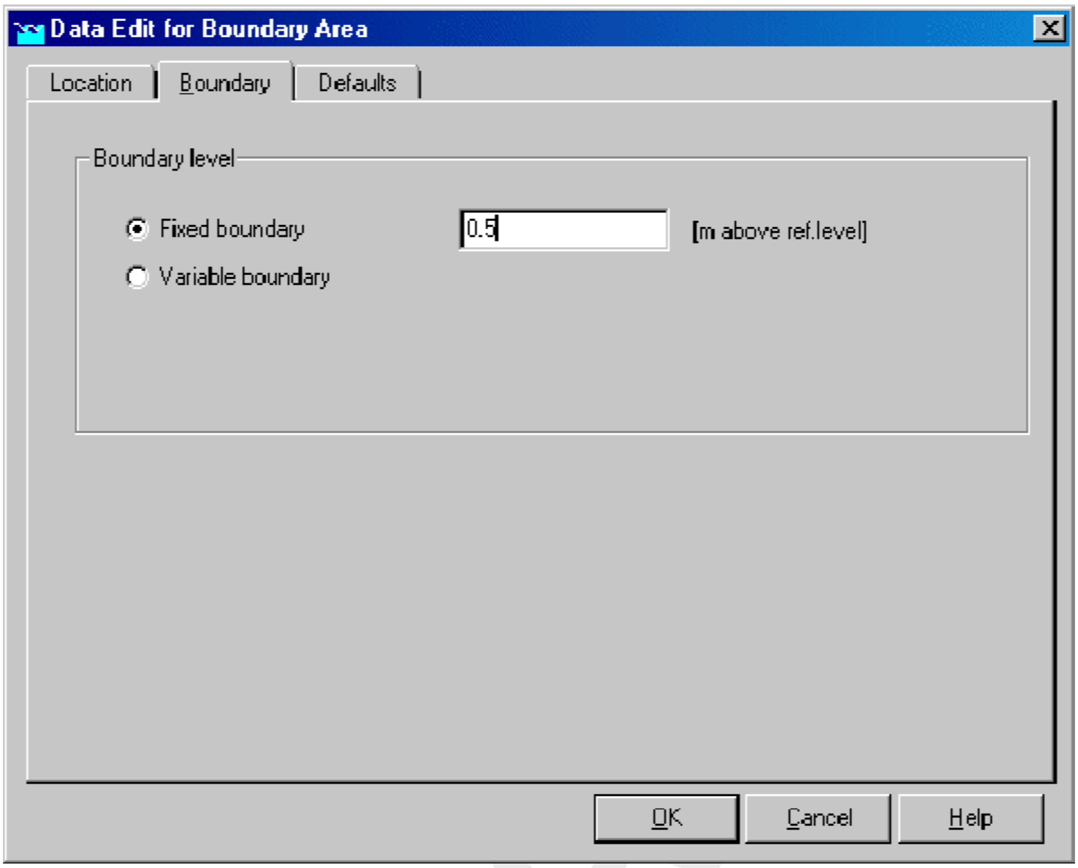


***Figure 5.138: Data Edit for Boundary Area** window* - Boundary *tab*

Here, the water level that applies to the model's boundary should be entered. There are two options:

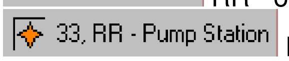
- **Fixed water level:** In this case, the water level on the boundary is constant.
- **Variable water level (from a table):** With this option, the water level on the boundary node can be given as a function of time.Notice that the option "online from flow module" is dimmed. This option is only available for types of nodes that connect the Rainfall Runoff module with one of the Flow modules (RR on Flow connection node and the RR - connection on Channel)

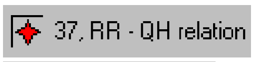
Boundary node topology

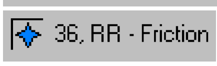
Nodes of the *RR - boundary* type can be connected to nearly all of the other RR nodes:

- 

RR - weir node
- 

RR - orifice node
- 

RR - pump station node (either applied as a pump or as an inlet)
- 

RR - QH relation node
- 

RR - friction node This node is considered deprecated functionality. Use is not recommended.
- 

RR - unpaved node
- 

RR - paved node

◇ 

RR - greenhouse node
◇ 

RR - Sacramento node
◇ 

RR - industry node
◇ 

RR - wastewater treatment plant node

The only node types that can **not** be directly connected to an RR boundary node are:

◇ 

RR Connection on Channel nodes
◇ 

RR Connection on Flow connection node(These nodes actually form boundary nodes themselves)
◇ 

Flow RR Open water node(If such a node would be connected directly to a boundary, it would immediately get a water level equal to the boundary value. Thus, that would be of no use at all)

The picture below gives some examples of how RR boundary nodes can be connected to other node types. Note that these are only some of the possibilities.

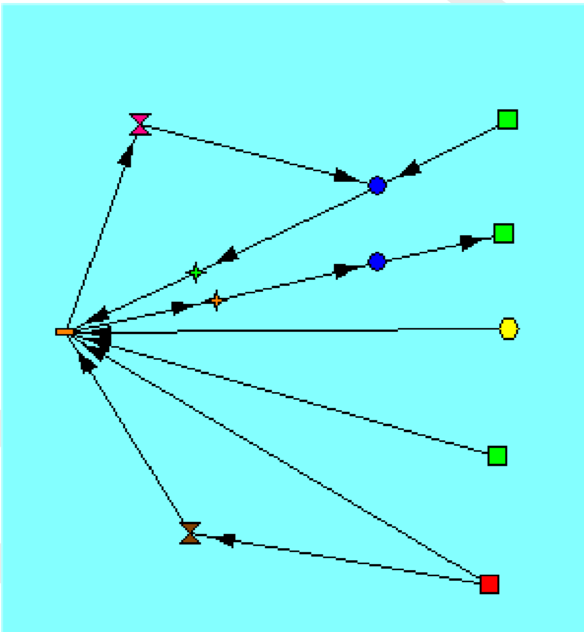


Figure 5.139: *An example of RR boundary nodes connected to other nodes*

In this example you can see the following phenomena:

◇ Upper chain: an industry node that withdraws water from the boundary (demand), and injects water into an RR open water node
◇ Second chain: an unpaved area that drains towards an Open Water node, which on its turn is connected to the boundary via a weir.
◇ Third chain: a pumping station that is used as an inlet (see the link directions) to supply an RR open water node & unpaved area during periods of drought.
◇ Fourth chain: a greenhouse node that drains off towards a boundary node.
◇ Fifth chain: an unpaved area node that drains off directly towards a boundary node.

◇ Sixth & Seventh chain: a paved area where the sewer discharge is pumped towards the boundary via a Waste Water Treatment Plant node, and where the sewer spills directly towards that boundary.

5.4.2 RR - Flow-RR Connection on Channel node

Description

In this chapter, the *Flow - RR Connection on Channel node* is described. This is one of two node types through which an RR schematisation can be linked to a CF schematisation. The other such node type is the *Flow-RR connection on Flow connection node*.

The large difference between both nodes is that this node is a derivative of a normal Flow - Lateral Flow node. This means that this node type should be attached to a branch by simply using the "add node" button. The other node type is a derivative of a Flow - Connection node, thus that one should form the start or end of a branch itself.

◇ For a detailed description of this node's input parameters: see the "Flow - RR Connection on Channel node input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the "Flow - RR Connection on Channel node topology" section from the Reference Manual

Input screens
When starting the model data editor for an *Flow - RR connection on Channel* node type, the following tabs will be available for input:

Boundary:

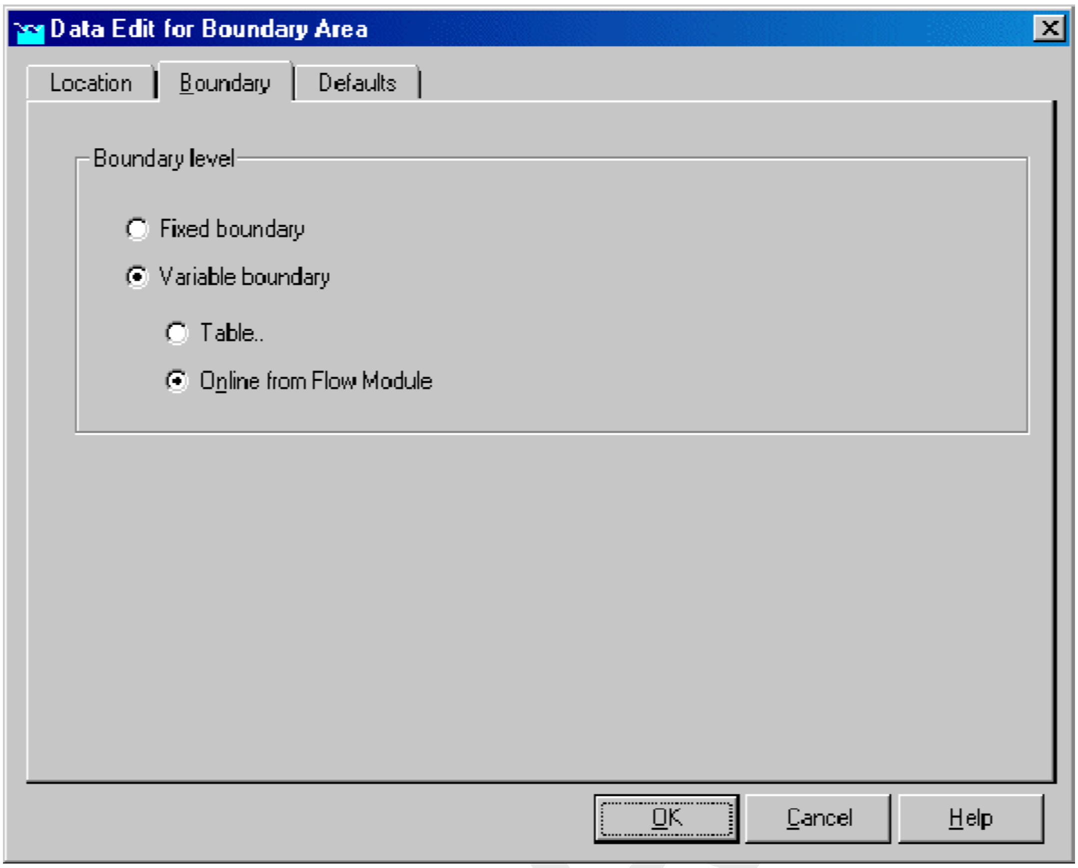


***Figure 5.140: Data Edit for Boundary Area** window* - Boundary *tab*

The parameters that need to be filled in here will form the boundary condition for the Rainfall - Runoff module. The availability of the options is dependent on your simulation settings:

The "online from flow module" option is only available when you have the RR and CF simulation settings set to "simultaneous". This can be done in the SETTINGS task block:

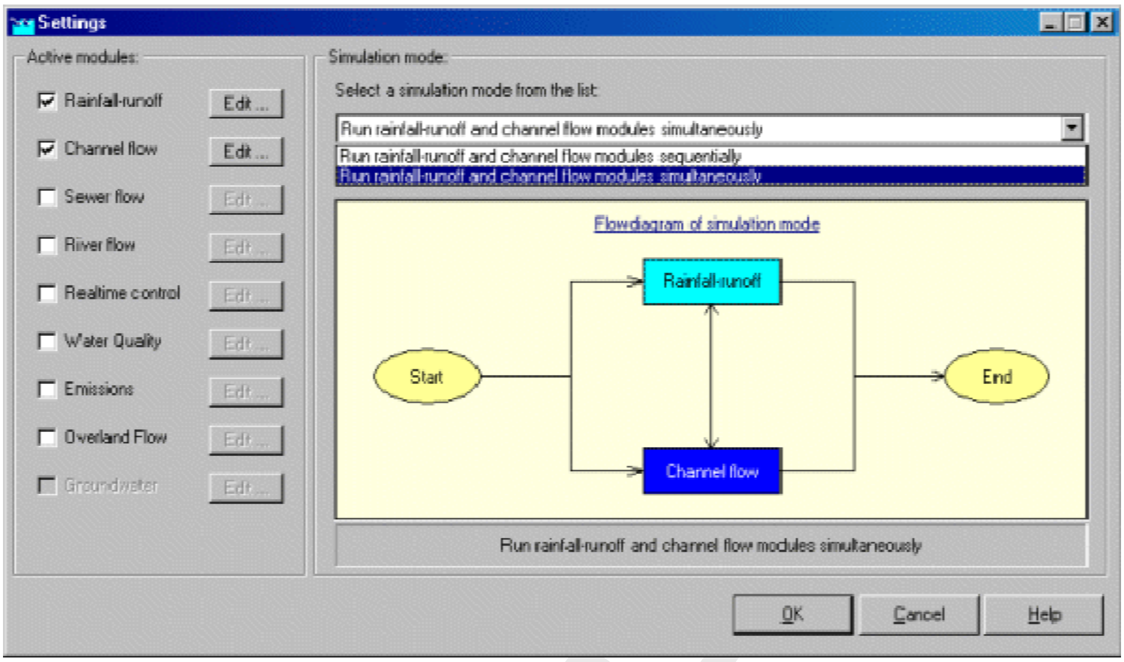


***Figure 5.141: Settings** window*

◇ **Fixed boundary:** If you choose for a fixed boundary value, the Rainfall-Runoff process will take place **completely independent** of the Flow processes. This means that alternating water levels on the channel will **not** affect the Rainfall-Runoff process. The RR module will have a fixed boundary condition to calculate its in- or outflow towards CF.

◇ **Variable boundary - table:** Also if you choose for a **table** with varying boundary values, the Rainfall-Runoff process will take place **completely independent** of the Channel Flow processes. This means that alternating water levels on the channel will **not** affect the Rainfall-Runoff process. The RR module will have a fixed boundary condition to calculate its in- or outflow towards CF.

◇ **Variable boundary - online from Flow module:** This option allows the Rainfall Runoff module to receive its boundary conditions **every time step** from the Flow module. This means that changing water levels in the Flow module will cause the boundary conditions for the Rainfall- Runoff module to change too, thus influencing its in- or outflow.

Topology

26, Flow-RR Connection on Channel

Nodes of the type *Flow - RR Connection on Channel* need to be attached to one of the Channel Flow branch types depicted below:

1, Flow - Channel, 2, Flow - Channel with lateral discharge

On the Rainfall - Runoff side, it should be linked with one of the following RR node types:

36, RR - Friction, 30, RR - Greenhouse, 40, RR - Industry, 35, RR - Orifice, 25, Flow - Pump Station, 28, RR - Paved, 29, RR - UnPaved, 33, RR - Pump Station, 37, RR - QH relation, 34, RR - Weir, 41, RR - Wastewater Treatment Plant

The link with RR should be made through one of the following types:

, 

Please keep in mind that the RR links should point **from** the RR nodes **towards** the *Flow - RR Connection on Channel* node.

Add a *Flow - RR Connection on Channel* node to your schematisation in the following way:

- ◇ Select the appropriate node type (Flow - RR Connection on Channel)
- ◇ Use the "add node"  button to add it to the schematisation. When clicking, SOBEK will automatically find the nearest channel to attach it to.
- ◇ If desired, the node may later-on be moved to other locations on the branch, or even to another branch by using the "move node"

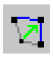
 button.

Figure 5.142: *Move node icon*

Note: Before adding a *Flow - RR Connection on Channel node* to your schematisation, you should have at least one of the above channel types present.

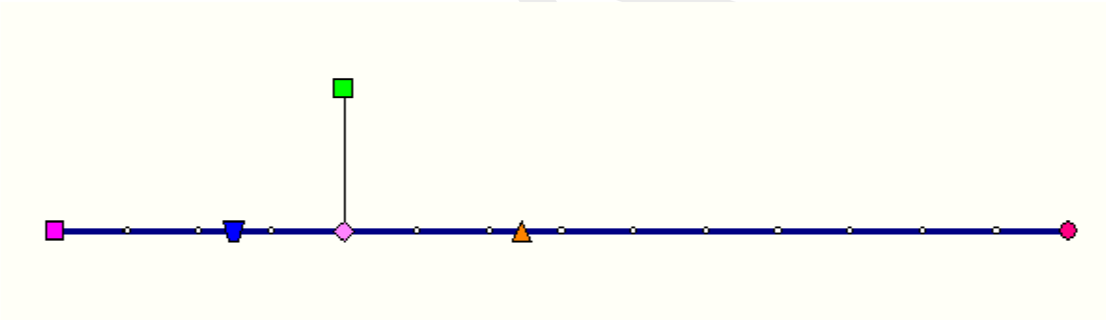
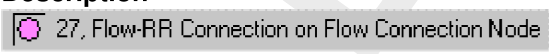


Figure 5.143: *An example of a valid network*

5.4.3 RR - Flow-RR Connection on Flow Connection node

Description



In this chapter, the *Flow - RR Connection on Flow Connection node* is described. This is one of two node types through which an RR schematisation can be linked to a CF schematisation. The other such node type is the *Flow-RR Connection on Channel*.

The large difference between both nodes is that this node is a derivative of a normal Flow - Connection node. This means that this node type should **not** be attached to an existing branch, but that itself it should form the start or end of a branch.

The other node (*Flow-RR Connection on Channel)* is a derivative of a normal Flow - Lateral Flow node, which means that it can be attached to a branch by simply using the "add node" button.

◇ For a detailed description of this node's input parameters: see the "Flow - RR Connection on Flow Connection node input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the "Flow - RR Connection on Flow Connection node topology" section from the Reference Manual

Input screens
When starting the model data editor for an *Flow - RR connection on Flow connection node* node type, the following tabs will be available for input:

Boundary tab:

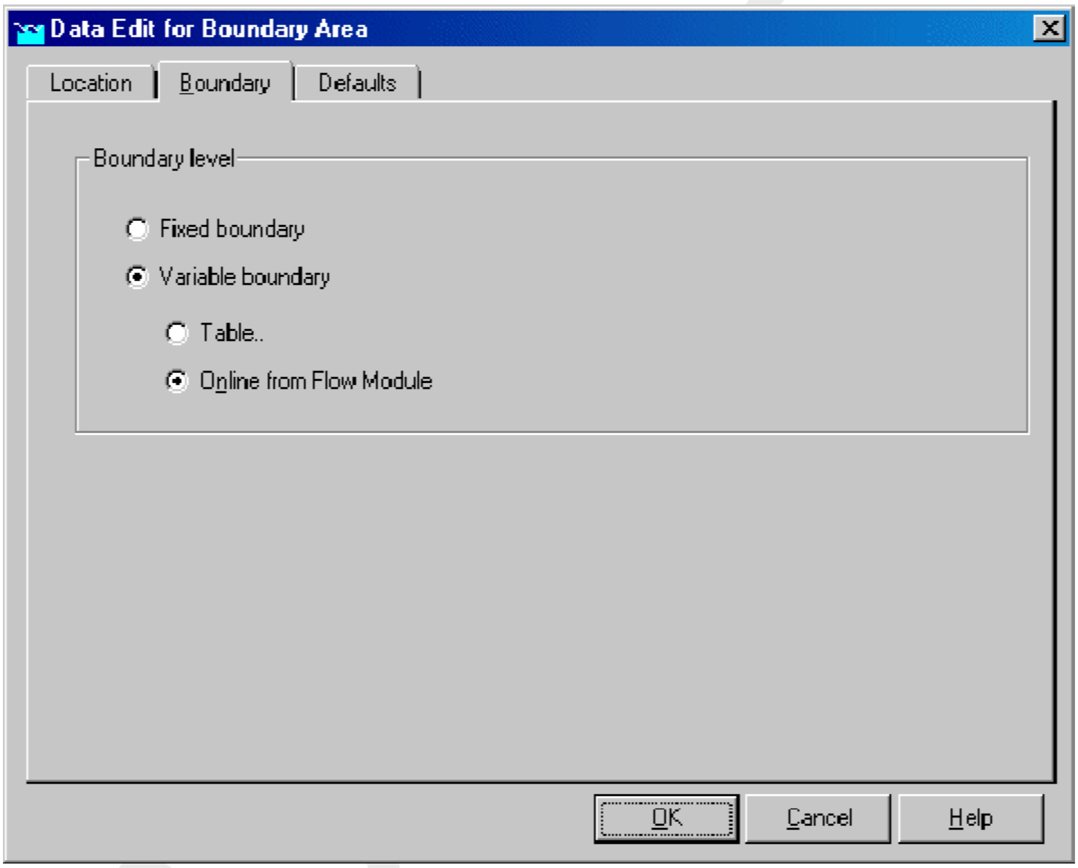


***Figure 5.144: Data Edit for Boundary Area** window* - Boundary *tab*

The parameters that need to be filled in here will form the boundary condition for the Rainfall - Runoff module. The availability of the options is dependent on your simulation settings:

The "online from flow module" option is only available when you have the RR and CF simulation settings set to "simultaneous". This can be done in the SETTINGS task block:

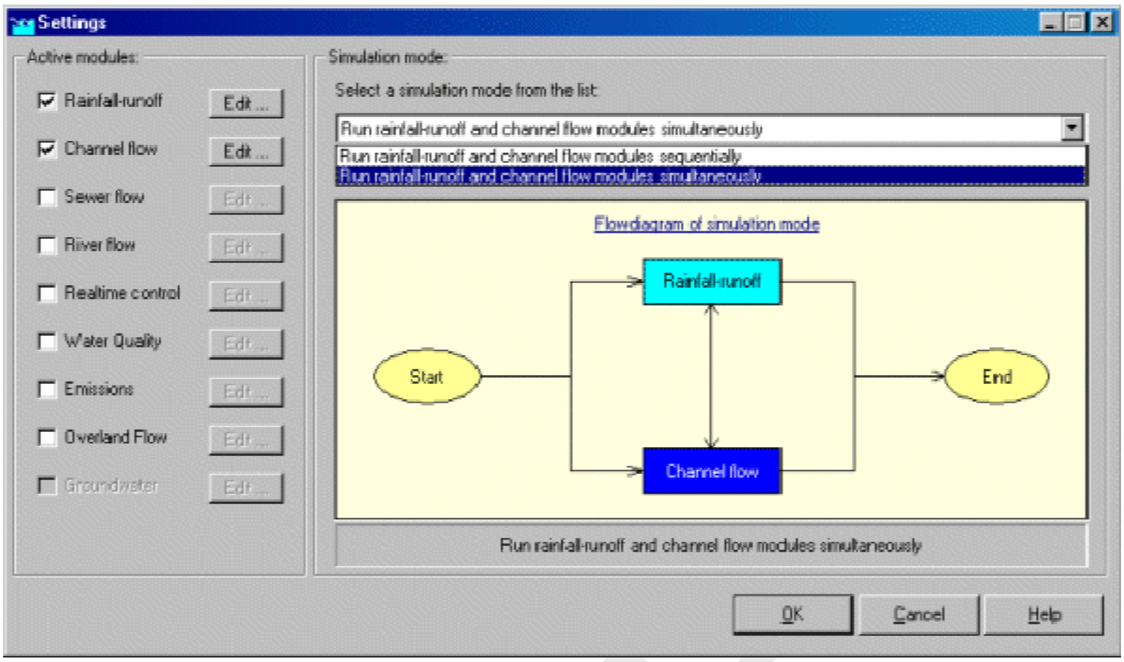


*Figure 5.145: **Settings** window*

- **Fixed boundary:** If you choose for a fixed boundary value, the Rainfall-Runoff process will take place **completely independent** of the Flow processes. This means that alternating water levels on the channel will **not** affect the Rainfall-Runoff process. The RR module will have a fixed boundary condition to calculate its in- or outflow towards CF.
- **Variable boundary - table:** Also if you choose for a **table** with varying boundary values, the Rainfall-Runoff process will take place **completely independent** of the Channel Flow processes. This means that alternating water levels on the channel will **not** affect the Rainfall-Runoff process. The RR module will have a fixed boundary condition to calculate its in- or outflow towards CF.
- **Variable boundary - online from Flow module:** This option allows the Rainfall Runoff module to receive its boundary conditions **every time step** from the Flow module. This means that changing water levels in the Flow module will cause the boundary conditions for the Rainfall- Runoff module to change too, thus influencing its in- or outflow.

The Storage tab depicted below is only activated when the Urban 1DFLOW (Sewer flow) module is activated.

Storage tab:

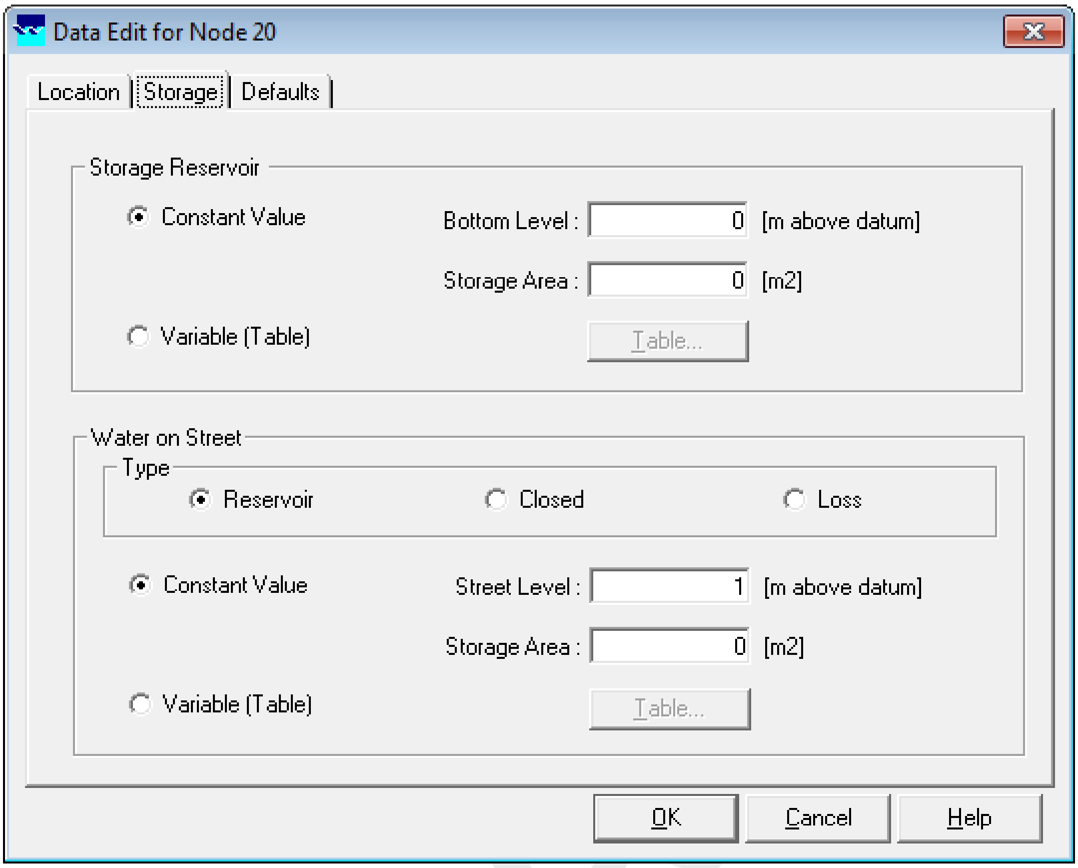
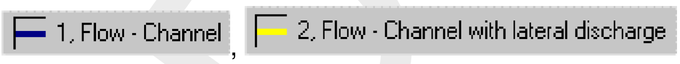


***Figure 5.146: Data Edit for Node** window* - Storage *tab*

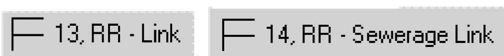
For a more detailed description of the Storage tab, see section 5.3.12.1 paragraph input screens

Topology
Nodes of the type *Flow - RR connection on Flow connection* need to **form the starting or end point** of one of the Channel Flow branch types depicted below:



On the Rainfall - Runoff side, it should be linked with one of the following RR node types:

The link with RR should be made through one of the following types:



Please keep in mind that the RR links should point **from** the RR nodes **towards** the *Flow - RR connection on Flow connection* node. Remember that *Flow - RR connection on Flow* *connection* nodes *should always be part of a branch themselves.* A correct working method is:

- ◇ Switch to the "edit network" mode in NETTER
- ◇ Select the node type *Flow - RR connection on Flow connection*
- ◇ Select the "add node" option 
- ◇ Place the node in the schematisation by clicking
- ◇ Select the appropriate branch type (Flow Branch or Sewer Pipe)
- ◇ Select the "connect nodes" option 
- ◇ Connect the node that you just added to another one by clicking and dragging.

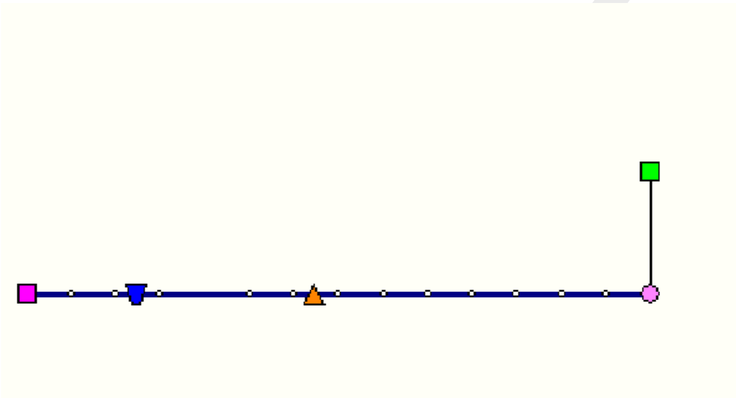
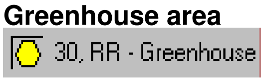


Figure 5.147: *An example of a valid schematisation containing a* Flow - RR connection on Flow connection *node*

5.4.4 RR - Greenhouse area

Greenhouse area



A greenhouse horticulture area can be schematised with a greenhouse node.

- ◇ For a detailed description of this node's input parameters: see the "greenhouse node input screens" section from the Reference Manual;
- ◇ For a detailed description of this node's possible network configurations: see the "greenhouse node topology" section from the Reference Manual
- ◇ For a detailed description of this node's underlying mathematical equations: see the "Technical Reference" manual.

What does a greenhouse node do?
The rainfall-runoff process on greenhouses is described by volume balances in two storage reservoirs:

- ◇ storage on the greenhouses; and
- ◇ storage in rainwater basins.

Storage-on-greenhouses represents the storage of water on greenhouse glass surface area (roofs). Rainfall can be stored on the roofs, before it evaporates or flows into the rainwater storage basins above- or underground.

The above-ground basins take in runoff-water from the glass surface as well as from direct precipitation. The amount of water stored is reduced by evaporation, water use in greenhouses, and possible pumping to the subsoil (under ground storage). When the maximum storage capacity is exceeded, the excess water flows into the adjacent open water. The figure below shows the volume balance in the greenhouse areas.

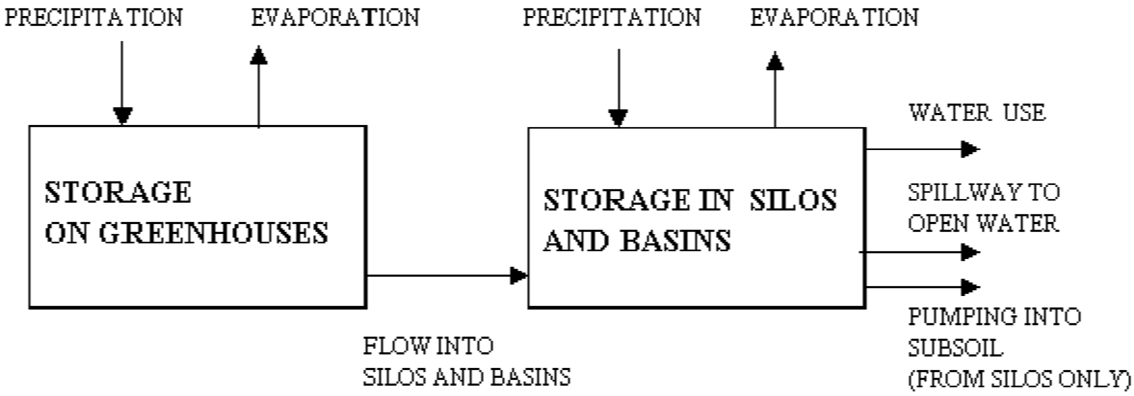


Figure 5.148: *Volume balance in the greenhouse areas*

Representation of the rainfall-runoff process in greenhouse areas

In SOBEK the rainwater basins have been divided into ten categories, depending on their volume per hectare of draining glass surface. SOBEK uses the lower limits of the categories. For example: all basins with a storage between 2500 and 3000 cubic meters per hectares of glass are considered as basins with a capacity of 2500 cubic meters per hectare of glass.

SOBEK-rr distinguishes itself from most other models with respect to the way greenhouse areas are treated. They are considered a special area type with provisions for the way horticulture makes use of rainwater storage basins. For instance, the initial filling percentage can be defined. Since the rainwater storage basins are usually not completely filled at the start of a rainfall period, this possibility often leads to a more realistic description of the flow into open water.

The (remaining) storage present in the basins at the beginning of the computation is an important variable determining whether spilling from the basins will occur or not. Therefore, SOBEK also provides historical data about the development of the storage in basins. To that end a separate computation has been carried out for each of the ten basin categories for the period 1951–1994, using the detailed greenhouse model of the Staring Centre-LDO. This accurate model is based on the water usage by a standard glass culture firm. This model takes into account aspects such as the management of rainwater basins by the market gardeners, return flow of water due to the combustion of natural gas etc. All assumptions have been rounded conservatively, so that the remainder storage in basins are under-estimated.

Greenhouse area input screens

When starting the model data editor for an *RR - Greenhouse area* node type, the following tabs will be available for input:

Area tab

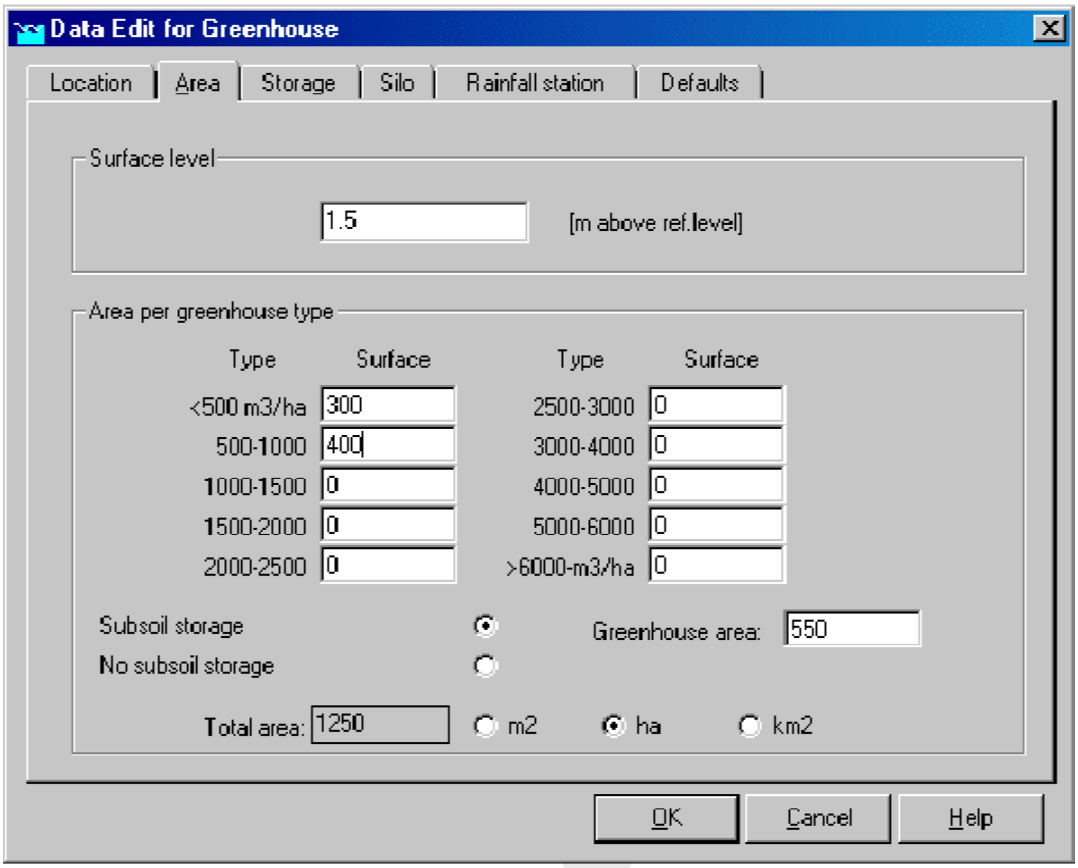


***Figure 5.149: Data Edit for Greenhouse**, Area tab*

On this tab, all data related to the area of different greenhouse types should be entered:

- **Surface level:** Represents the surface level of the soil.
- **Area per greenhouse type:** As explained in the Functional Reference Manual chapter "greenhouse area description", the rainwater basins have been divided into ten categories, depending on their volume per hectare of draining glass surface. On this tab, the surface area for each of these categories can be entered. Of course, the sum of all areas should correspond with the total surface of the greenhouses.
- **Subsoil storage:**For some greenhouses, additional water can be stored in subsoil silos. Here the greenhouse area that uses such storage method should be entered. After choosing this option, the tab "Silo" becomes active.

Storage tab

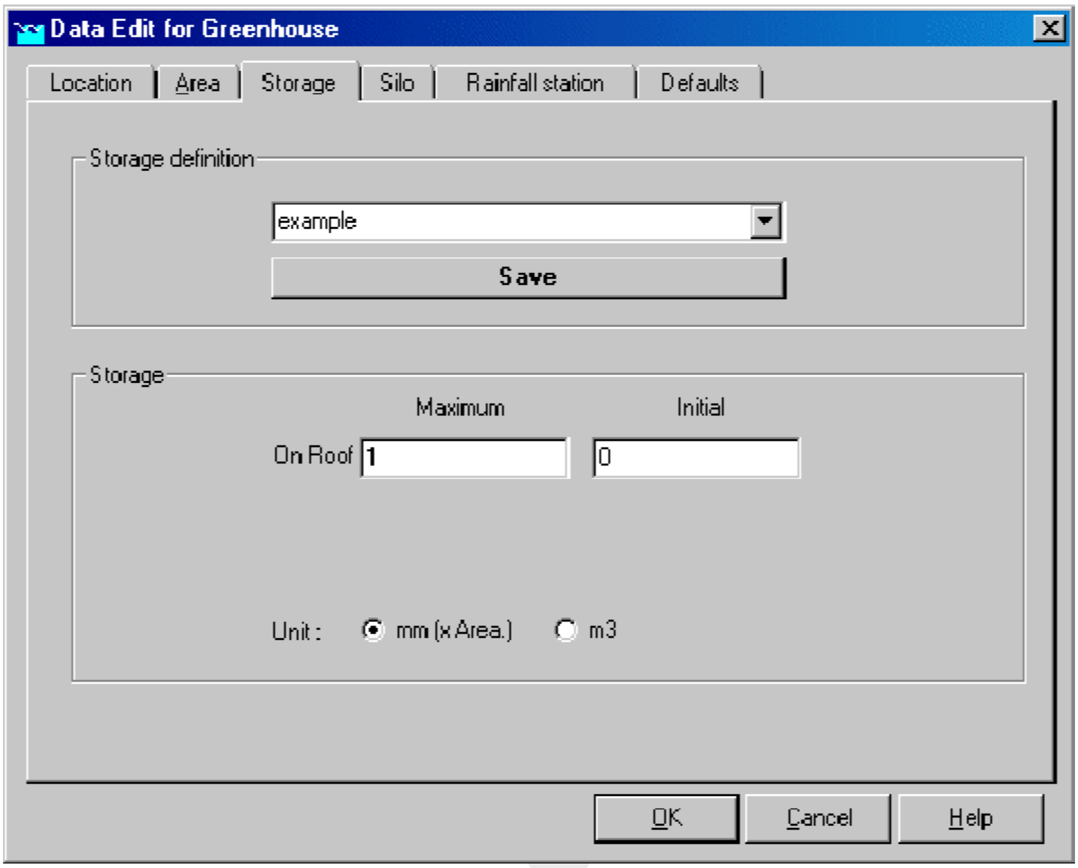


***Figure 5.150: Data Edit for Greenhouse**, Storage tab*

On the storage tab, the maximum and initial amount of water that can be stored on the roof of the greenhouses is defined. The chosen values can be saved in a separate "storage definition" so that the same values can easily be applied to other greenhouse nodes.

Enter the data as follows:

- ◇ first type a name in the "storage definition" field
- ◇ click the *Define* button
- ◇ enter the appropriate values for maximum and initial storage;
- ◇ click the *Save* button;
- ◇ note that you can re-use this storage definition on other greenhouse nodes.

Explanation of the parameters:

- ◇ **Maximum storage:** Equals the amount of water that can be stored on the rooftop without being drained off. Only after the rainfall amount exceeds this amount, water starts draining off the roofs.
- ◇ **Initial storage:** If your simulation starts right after a wet period, it might be the case that the storage capacity of the roof itself is already partially filled with water. That amount of water should be entered here.

Silo Tab

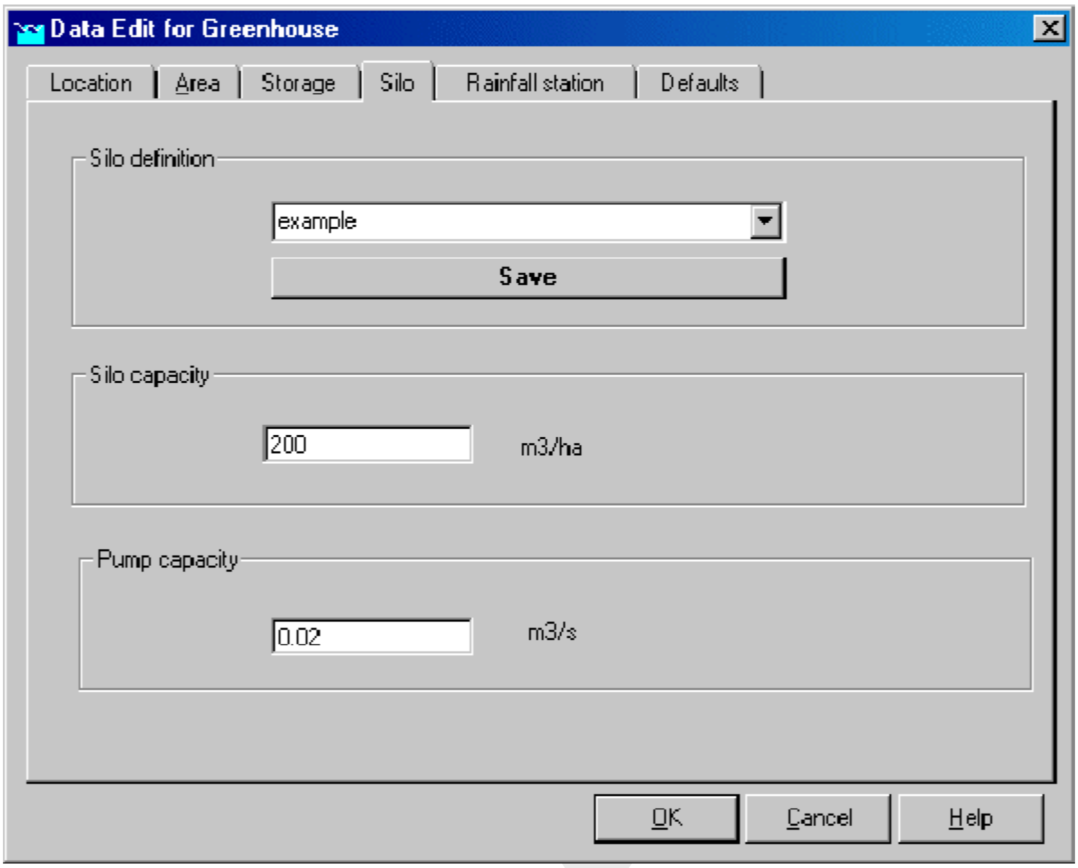


***Figure 5.151: Data Edit for Greenhouse**, Silo tab*

If the "subsoil storage" option on the "Area" tab has been chosen, this tab will become active. Here the storage and pumping capacity of a subsoil silo can be entered.

Due to rainfall on the greenhouse area that is connected to this silo, the silo is filled with water. In the meantime, water is being pumped out in order to supply the plants with water. If, due to heavy rainfall, the silo's storage capacity is exceeded, the excess amount will "overflow" towards a connected RR - Open water node or boundary node.

- **Silo capacity:**This parameter represents the storage capacity of the subsoil silo in volume per unit of greenhouse area. When multiplying this value with the "Greenhouse area" value on the "Area" tab, the total volume of the silo is obtained.
- **Pumping capacity:**This parameter represents the capacity of the pumps that supply water from the silo to the plants within the greenhouse. In the model, this amount of water is simply defined as an outflowing term.

These values too are entered as a "definition". This allows for re-using that definition on other greenhouse nodes.

Enter the data as follows:

- first type a name in the "silo definition" field
- click the *Define* button
- enter the appropriate values for maximum and initial storage
- click the *Save* button;

◇ note that you can re-use this silo definition on other greenhouse nodes.

Rainfall station tab:

Here, the rainfall station that applies for this node can be selected. Rainfall stations can be created in the "Meteorological data" task block.

Greenhouse area node topology

Nodes of the *RR - Greenhouse area* type may drain off towards one of the following nodes:

◇ 

RR - open water node

◇ 

RR - boundary node

Commonly, a greenhouse node is connected with an open water node.

5.4.5 RR - Industry

RR - Industry node general description

In this chapter, the *RR - Industry node* is described. With an *RR - Industry node* you can simulate lateral discharges to and from the water system. This node type is commonly used to simulate industrial water extraction and -return (e.g. water for process cooling).

But this node type can also be used to simulate lateral discharges (extractions and discharges) of known amounts.

◇ For a detailed description of this node's input parameters: see the "RR - Industry node input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the "RR - Industry node topology" section from the Reference Manual

RR - Industry node input screens

When the model data editor for an *RR - Orifice* node type is started, the following tabs will become available for input:

Industry tab:

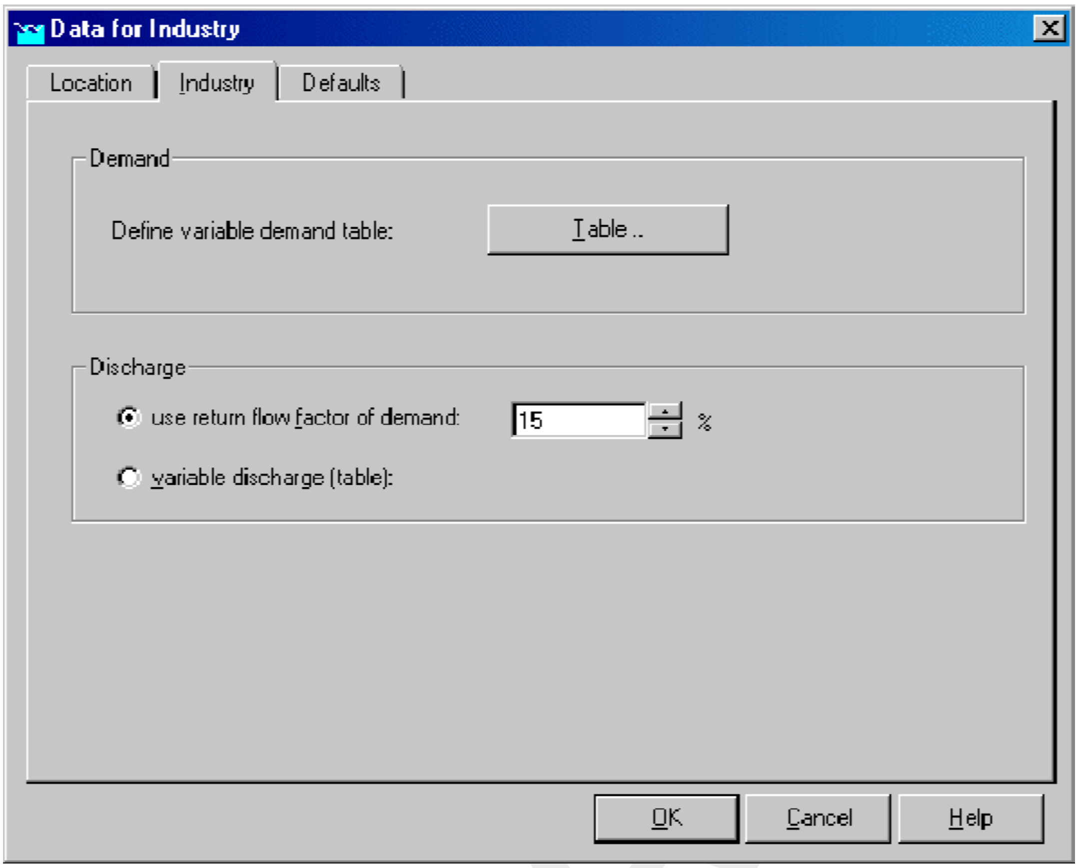


***Figure 5.152: Data Edit for Industry**, Industry tab*

On this tab, lateral discharges (discharges and extractions of water) can be defined. Extractions are defined as "demands" and "discharges" as "injections".

◇ **Demands:** Here, the amount of water that is extracted from the system should be entered (as a function of time). Click the <Table> button to open the table where the values should be entered.Note that demands are always extracted from an "upstream" connected node. See the chapter on Industrial node topology for more information.
◇ **Discharge:** In this section, the amount of water that is injected into the system should be entered (as a function of time). This can be done with two options:

1 "Use return flow factor of the demand". Using this option, the amount of injected water will be a percentage of the amount that is extracted according to the Demands settings. Fill the appropriate percentage in into the box next to this parameter name.
2 "Variable discharge (table)". Using this option, the amount of injected water will take place independently of the demand, but can be defined manually as a function of time. Click the <Table> button to open the table where the values should be entered.

Note: that discharges are always injected on a "downstream" connected node. See the chapter on Industrial node topology for more information.

RR - Industry node topology

The RR - Industry node requires an upstream node to extract its demands and a downstream node to inject its discharges. Note that the arrows on the RR links indicate which nodes are defined as "upstream" and which ones as "downstream".

RR Industry nodes can be connected to the following node types:

- 

RR - Open Water nodes:
- 

RR - boundary nodes:
- 

CF RR on Flow connection nodes:
- 

CF RR on channel connections.

5.4.6 RR - Open water

Open water node descriptions

In this chapter, the open water node is described. It is a commonly used modelling object within the SOBEK Rural Rainfall-Runoff module.

What does the open water node do?
It can represent the area which is occupied by a number of water courses. Such area may vary with the prevailing water level, due to existing bank slopes. Usually, the open water node receives water from connected nodes of the following types:

- Unpaved area node
- Paved area
- Greenhouse area
- Industry
- Waste Water Treatment Plant
- Sacramento

Furthermore it can exchange water with other open water nodes, a Flow model (Channel Flow or River Flow module) or a model boundary, via structures or a friction node.

Note: in dry periods the above described procedure can be reversed to let water in. In that case water is deliberately led from a boundary or Flow model towards the open water node, and finally infiltrated in the unpaved area.

Open water node input screens
When the model data editor for an *RR - Open Water* node type is started, the following tabs will become available for input:

Tab Surface:

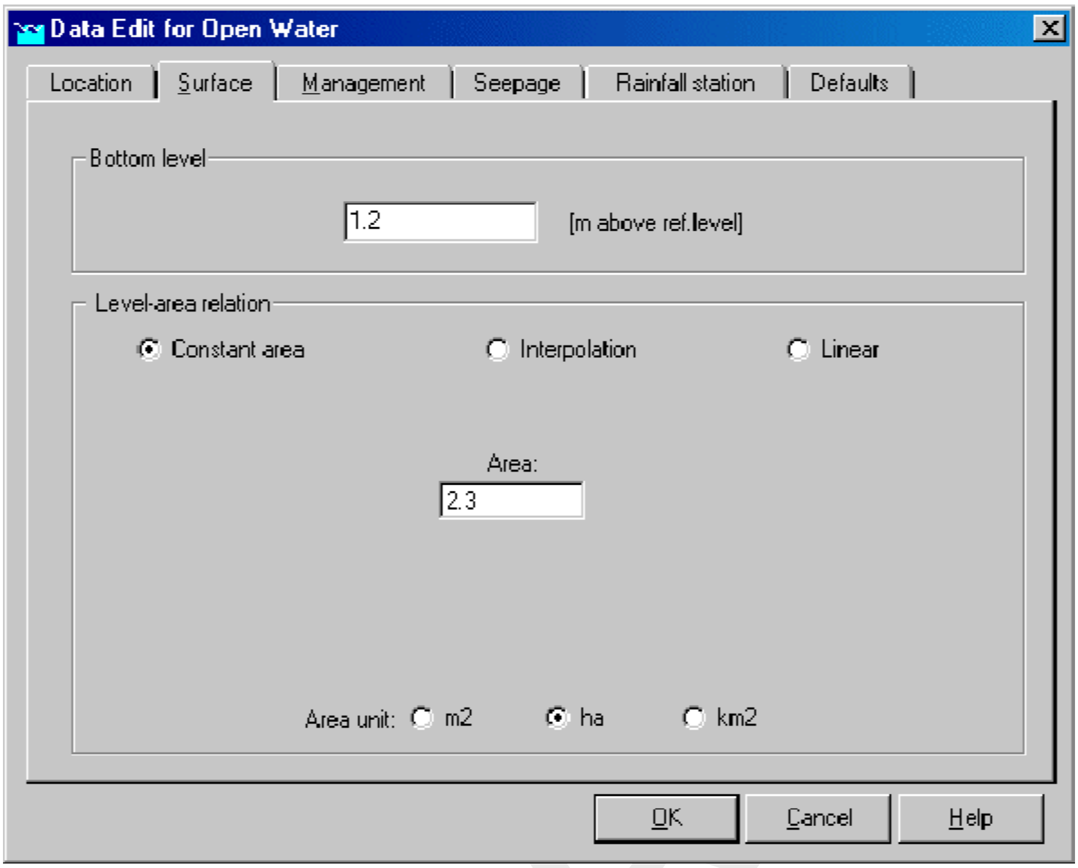


***Figure 5.153: Data Edit for Open Water**, Surface tab*

Note that an RR Open Water node represents a series of channels through a certain **volume** of water, thus:

- ◇ The area that is covered by the open water (may be dependent of the water level)
- ◇ The water depth, which is calculated during every time step.

The following variables have to be specified:

- ◇ **Bed level:** Represents the bed level of the channels/ditches that are represented by the open water node.
- ◇ **Level-Area relation:** In this section, the open water area (as seen from above) is entered. It may be constant (independent of the water level; this is the case if all ditches have vertical walls) or an area that varies with the water level (if the ditches have side slopes).The options:

  1 A constant area.
  Independent of the water level.
  2 Interpolation from values in a table.
  This option lets you fill in six water levels for which the open water area is known. The area for other water levels is then interpolated from this table. **Note:** when using this option, it is wise to include the open water area at the bottom level in the table. Otherwise the calculation of the area at bottom level will be calculated by linear extrapolation of the lowest two values from the table. Sometimes this may lead to negative areas.
  3 Linear.
  Allows to fill in the area of open water at the target level and the percentage by which

it increases with rising water level. You can check the interpolation table which is generated and used in SOBEK by switching towards the "interpolation" option after having specified your values for linear interpolation.

Tab Management:

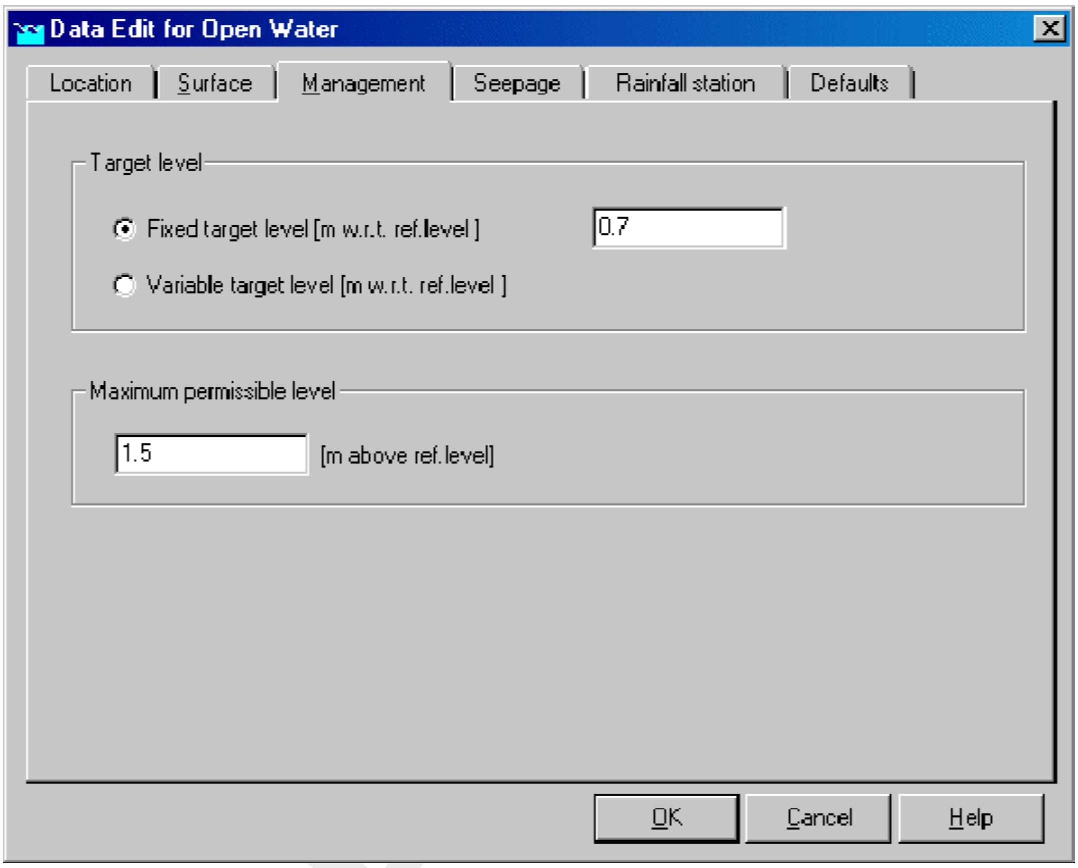


***Figure 5.154: Data Edit for Open Water**, Management tab*

On this tab, the hydrological management characteristics for this open water node need to be entered:

- **Target Level:** Generally, in second order ditches, a certain target level is being maintained by weirs or other structures. This kind of target level needs to be filled in here. It can either be a fixed level or variable in time.
  Note: the target level value on itself will not have any effect on the water management. The actual water levels are determined by the operation of structures (such as weirs) that are connected to this area. You'll need a structure to actually maintain the desired target level. The reason to enter a value for target level here is mainly for the creation of output files that refer to that value.
- **Maximum permissible level:** This parameter does not have any effect on the calculations. It is only used to calculate the amount and duration by which a certain level is exceeded.

Tab Seepage:

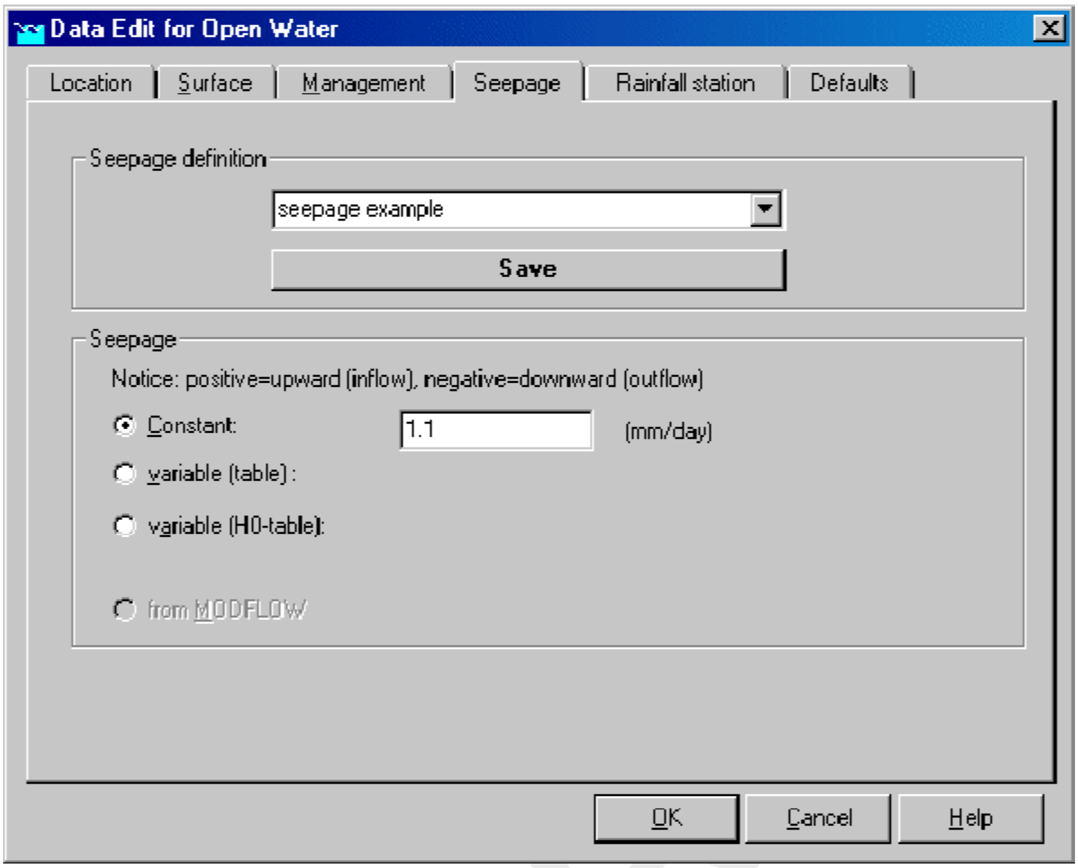


***Figure 5.155: Data Edit for Open Water**, Seepage tab*

The Seepage tab refers to the exchange of water between the top layer (all soil that's higher than the bottom of the ditches) and the groundwater underneath

First, enter a name for the seepage definition, then click on <Define> and enter the appropriate value. A positive value represents inflow, a negative value represents outflow.

The values can either be:

- ◇ Constant
- ◇ Variable (in time)
- ◇ Calculated from a variable H0 value (pressure head in the underlying aquifer) and the resistance value of a confining layer in between.

Tab Rainfall Station:

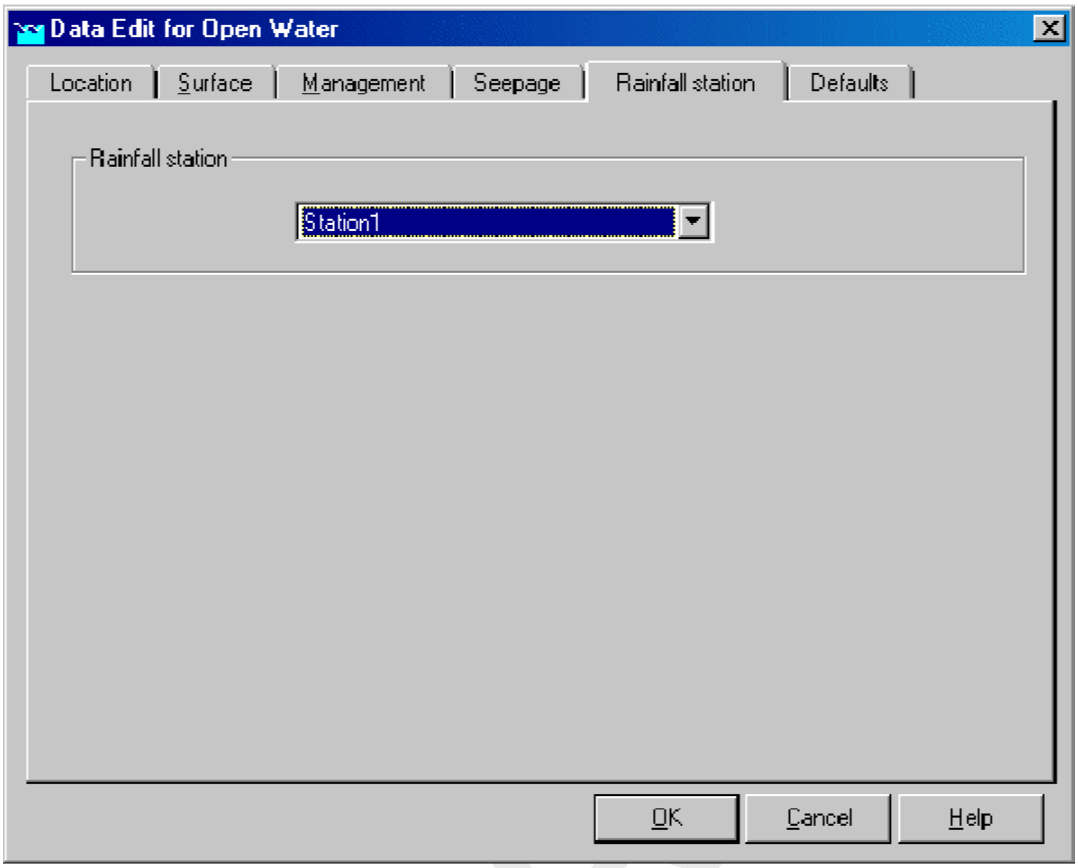
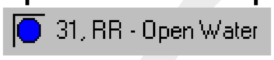


***Figure 5.156: Data Edit for Open Water**,* Rainfall station *tab*

On this tab you choose the meteorological station that will apply to the Open Water node. Rainfall and evaporation will apply to the area open water on every time step. Wind data will have no influence because the Open Water node is a 0-dimentional node type (point objects without physical outstretches in the horizontal plain).

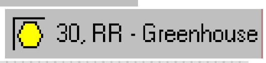
Open water node topology



An RR open water node can only be connected to other RR objects. On the upstream side, it may be connected with:

- RR Unpaved nodes 

- RR Paved nodes 

- RR Greenhouse nodes 

- RR Sacramento node 

- WWTP node 

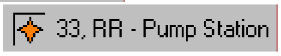
- Industry node 

Notice that the first four node types transfer a rainfall intensity to a certain outflow. When connecting these nodes to an RR Open Water node, this outflow will therefore flow into the RR Open Water node.

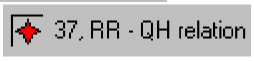
On the downstream and upstream side, the RR Open Water node may be connected to RR structures or a friction node:

- ◇ RR Weir nodes 

- ◇ RR Orifice nodes 

- ◇ RR Pump station nodes 

- ◇ RR Friction nodes 

- ◇ RR QH relation nodes 

Note: on their turn, these structures should be connected to another RR Open Water node or an RR boundary node.

Optionally, one may connect an additional RR Industry node 

to an RR Open Water node in order to withdraw or inject a certain amount of water in time. Typical schematisations covering RR Open water nodes could look like this:

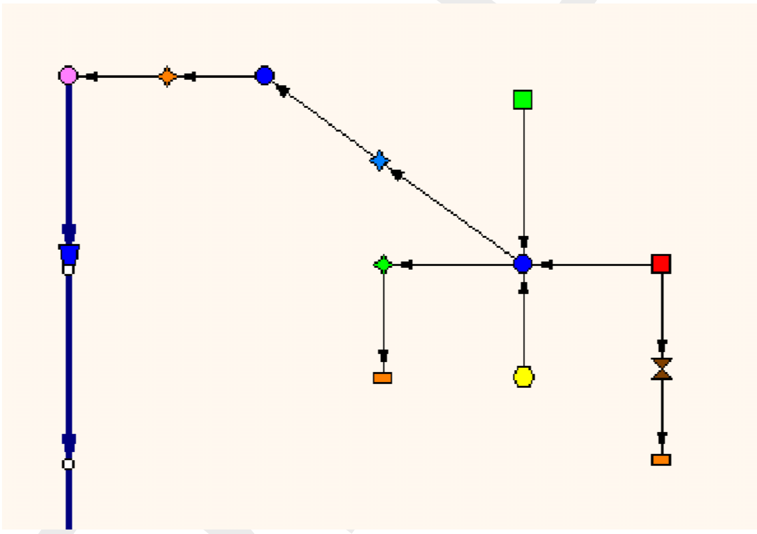


Figure 5.157: *Example of RR Open Water node connections.*

In this example an RR Open Water node receives water from an Unpaved area, a Paved area and a Greenhouse area. The node is connected to a Boundary node via a Weir, thus when the water level exceeds the crest level of the weir, water is discharged to the boundary node. Furthermore there is a connection with another Open Water node via a friction node, causing a slight difference in water level between both Open Water nodes. The downstream Open Water node is connected to the SOBEK Channel Flow module via a RR Pump Station (which could have been any RR structure).

5.4.7 RR - Orifice node

RR - Orifice node general description

In this chapter, the *RR - Orifice node* is described. With an *RR - Orifice node* you can simulate a structure with a controllable gate height, through which water flows.

- For a detailed description of this node's input parameters: see the "RR - Orifice node input screens" section from the Reference Manual;
- For a detailed description of this node's possible network configurations: see the "RR - Orifice node topology" section from the Reference Manual
- For a detailed description of this node's underlying mathematical equations: see the "Orifice section from the Technical Reference Manual".

RR - Orifice node input screens
When the model data editor for an *RR - Orifice* node type is started, the following tabs will become available for input:

Options Tab:

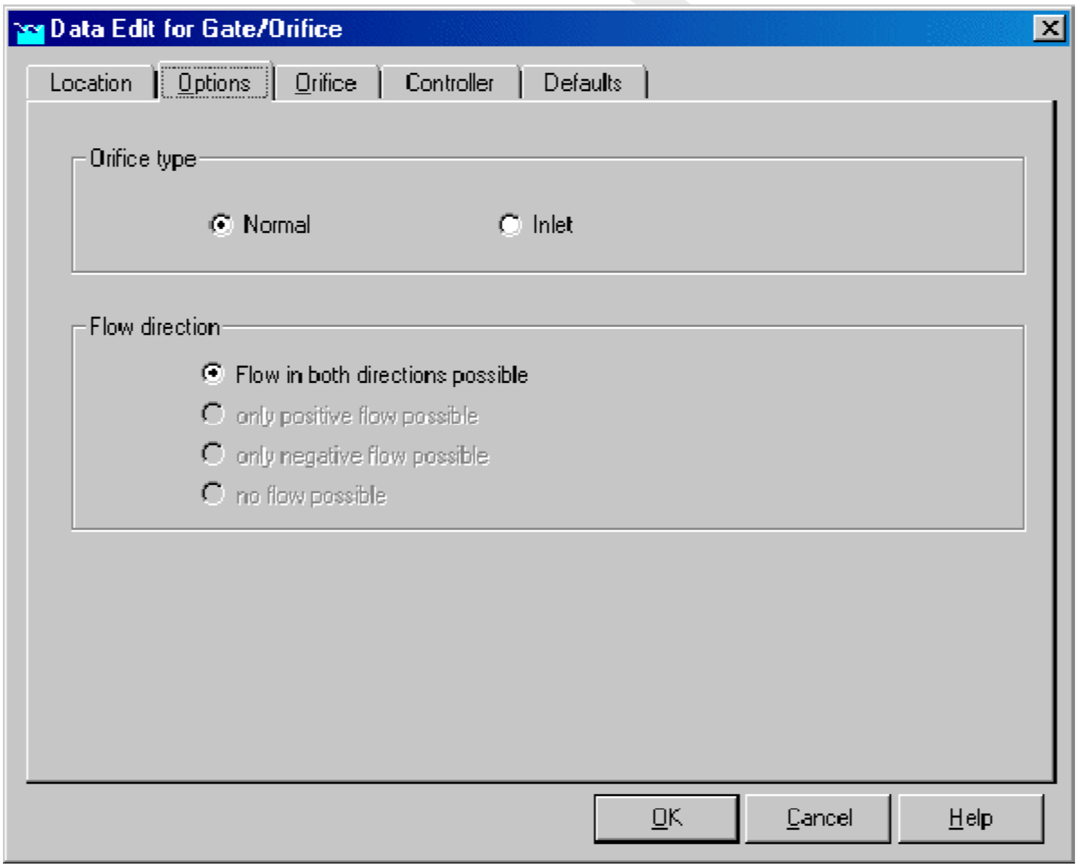


Figure 5.158: Data Edit for Gate/Orifice, Options *tab*

On this tab, you can define how the orifice is to be used:

- **Orifice type: normal**This option means that water can flow through the orifice in both directions.

◇ **Orifice type: inlet**This option means that water may only flow through the orifice in positive direction. The arrows on the links that connect the orifice to other objects indicate the positive flow direction.

Orifice tab:

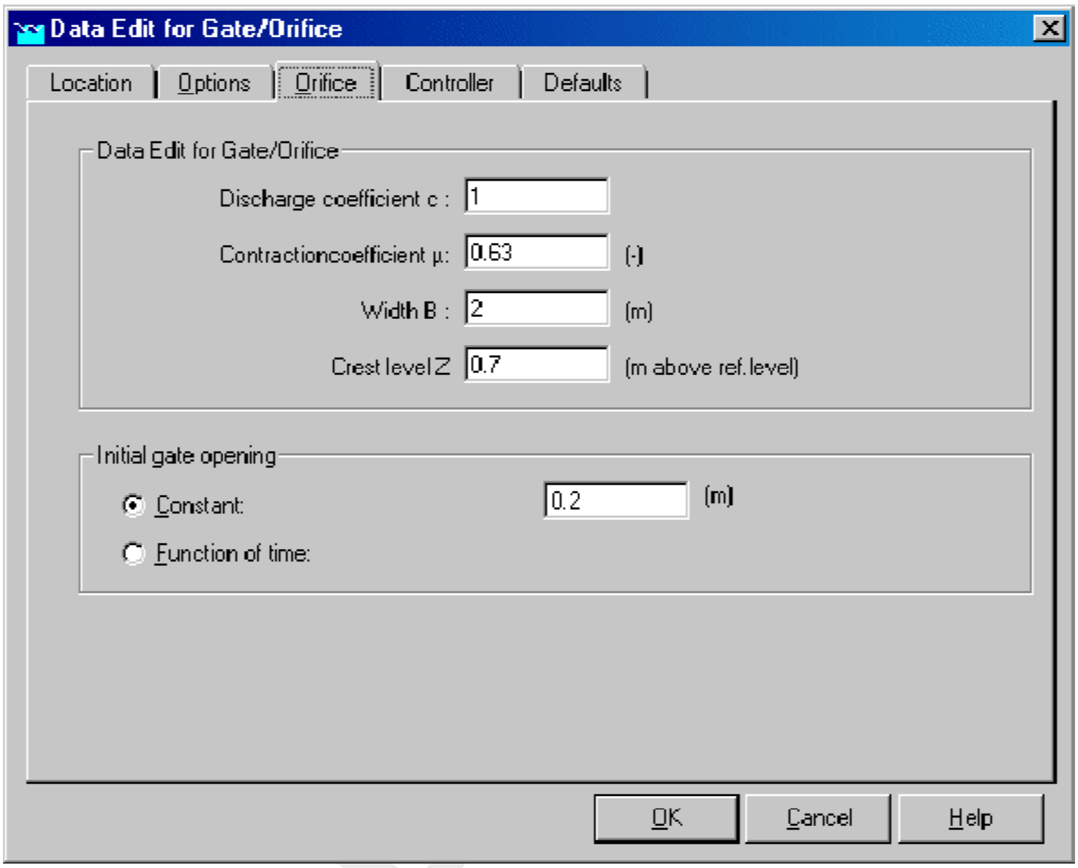


***Figure 5.159: Data Edit for Open Water**, Orifice tab*

On this tab, the dimensions of the orifice should be entered:

◇ **Discharge coefficient c:** See the "Orifice" section of the Technical Reference Manual for the exact implementation of this parameter in the discharge formula of this structure type.
◇ **Contraction coefficient** μ This is a coefficient that represents the energy loss due to contraction of the water mass that flows through the orifice. See the "Orifice" section of the Technical Reference Manual for the exact implementation of this parameter in the discharge formula of this structure type.
◇ **Width B:** Represents the width of the gate through which the water flows.
◇ **Crest level Z:** Represents the crest level of the gate, thus the bottom of the gate opening.
◇ **Initial gate opening:** This parameter represents the orifice's gate opening. Notice that it is called "**initial**" gate opening height. This name has been chosen because the opening may eventually be overruled by a controller. If there's no controller active for the orifice, the gate opening will remain equal to the "initial gate opening" throughout the entire simulation.

Controller Tab:

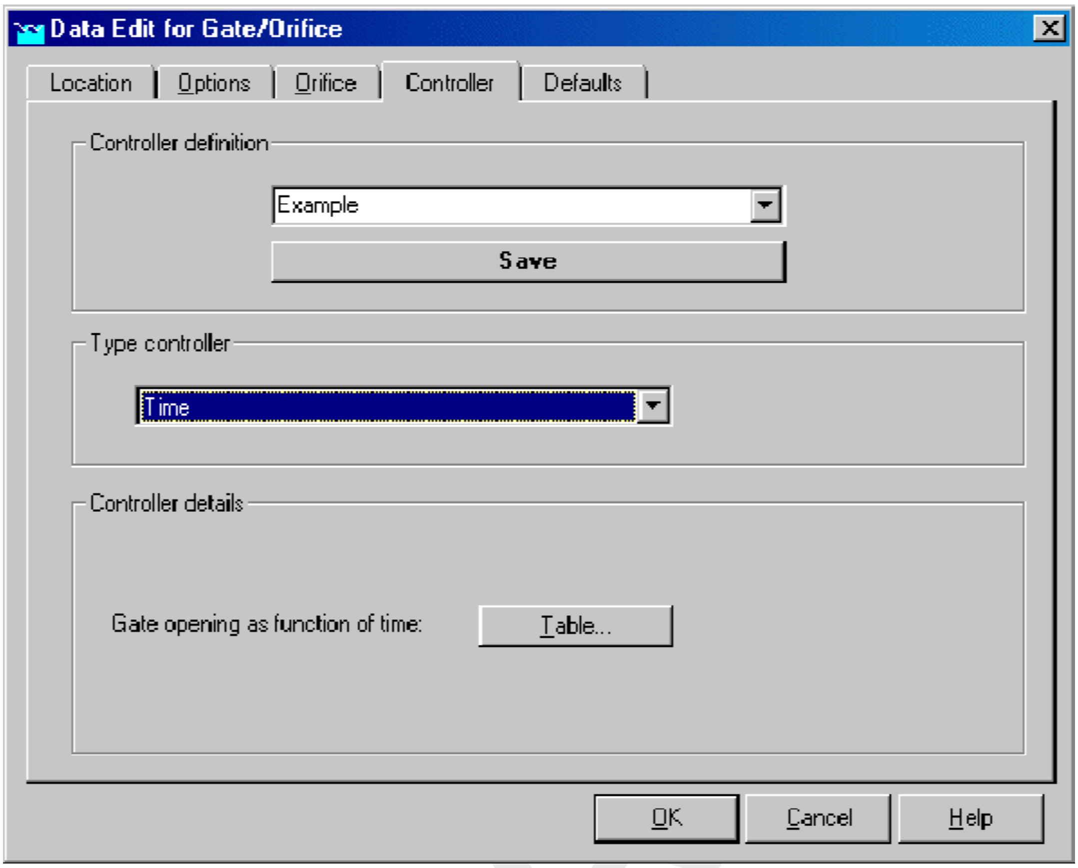


Figure 5.160: Data Edit for Open Water, Controller *tab*

On the controller tab, the behaviour of the orifice can be controlled (optional). With a controller, the initial opening heights, as defined on the "Orifice" tab, are overruled.For *RR - Orifices*, only a so-called time controller is available. This means that the gate height can be defined as a function of time.

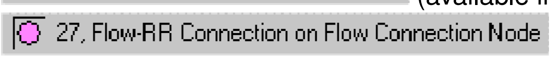
RR - Orifice node topology
An RR - Orifice should always be connected to at least one RR - Open Water node, as it is meant to let water flow in or out of an open water section.

RR - Orifices can be connected to nodes of the following types:

- 

- 

- 

(available in the Flow modules only)
- 

(available in the Flow modules only)

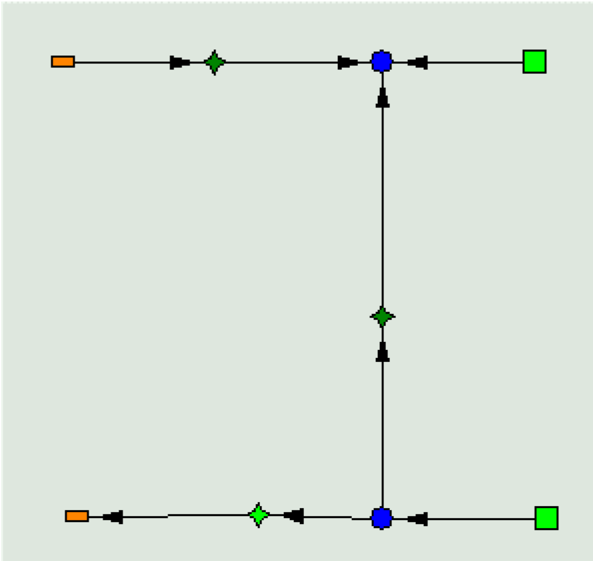


Figure 5.161: An example of a network where RR Orifices are used

5.4.8 RR - Paved node

Paved area node; general description

In paved areas water can be stored on the surface ("on-the-street storage") and in a sewer system. The first one represents the storage on paved areas like roofs and roads. The second one represents the water stored in sewer mains of separated or combined sewer systems. The representation of the rainfall-runoff process in paved areas is shown.

The storage on the street and the sewer storage can be considered to be two reservoirs. The rainfall-runoff module calculates a water balance of these reservoirs. When precipitation occurs on the paved area, first the on-the-street storage reservoir is filled. If this reservoir is full, it starts spilling into the sewer reservoir. The amount of on-the-street storage is reduced by evaporation.

Water can enter the sewer reservoir in two ways: first by spilling from the on-the-street storage, and second by return flow from domestic water use (dry weather flow). Depending on the type of sewer system, the inflow from the surface and the dry weather flow are mixed in one sewer storage reservoir, or put into separate sewer storage reservoirs.

When the sewer storage reservoir contains water, the sewer pumps are switched on, and water is pumped from the sewer to the local open water or to a boundary representing a waste-water treatment plant outside the system. If the sewer is full, it can also spill directly into the open water. Flows from paved to unpaved areas and vice versa are neglected.

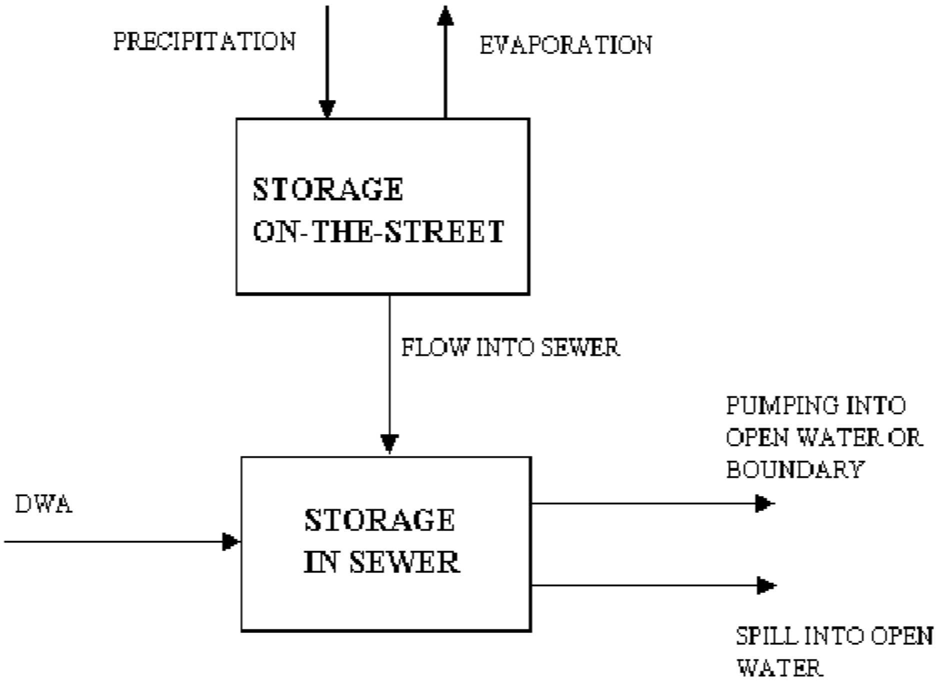


Figure 5.162: *The storage on the street and the sewer storage can be considered as two reservoirs*

Representation of the rainfall-runoff process in paved areas.

You can model multiple types of paved area by dividing the total area into sub-areas with specific characteristics. In this way areas with and without sewers can be modelled. By definition, there is no infiltration to groundwater in paved areas. When modelling urban areas, as a rule of thumb about 50 percent of the urban area can be considered as paved area, and 50 percent can be considered as unpaved area (private gardens and public parks).

Paved area input screens

When starting the model data editor for a *paved area* node type, the following tabs will be available for input:

The Area tab

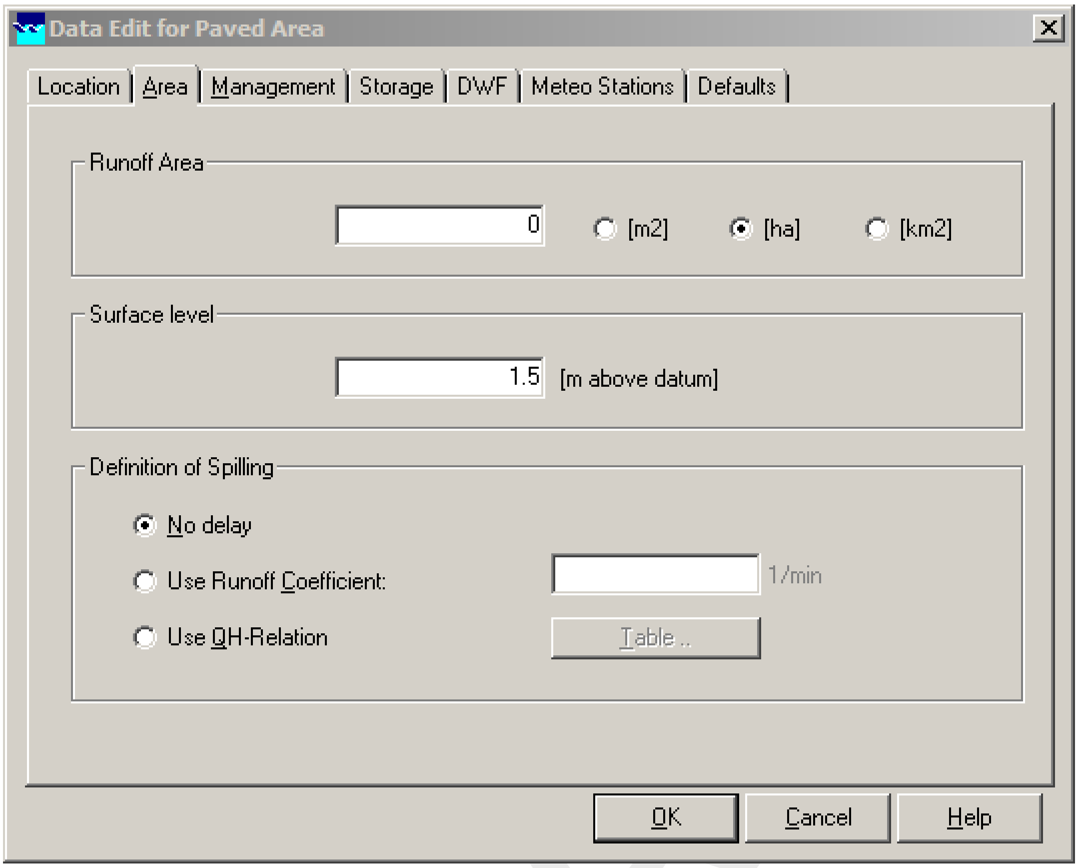


***Figure 5.163: Data Edit for Paved Area** window, the Area tab*

On this tab, the following data can be entered:

- **Runoff Area:** The value filled in here will represent the paved area on which rainfall from the selected rainfall station for this node will fall. When the maximum storage capacity of the surface (storage tab) is exceeded, the water will immediately start flowing into the sewer system (management tab).
- **Surface level:** Surface level equals the street level for the paved area node. If the water level in a connected open water node will exceed this level, the water will inundate the paved area.
- **Definition of Spilling:**
  - **No delay:** All surface runoff branches the outflow point in the same timestep.
  - **Use Runoff Coefficient:** A runoff delay coefficient, similar to the RR-Urban runoff model.
  - **Use QH-Relation (Beta functionality):** Limit the discharge Q (mm/s) based on the water level H (m above datum).

For more information on these options, see section 6.5.1.24.

The Management tab

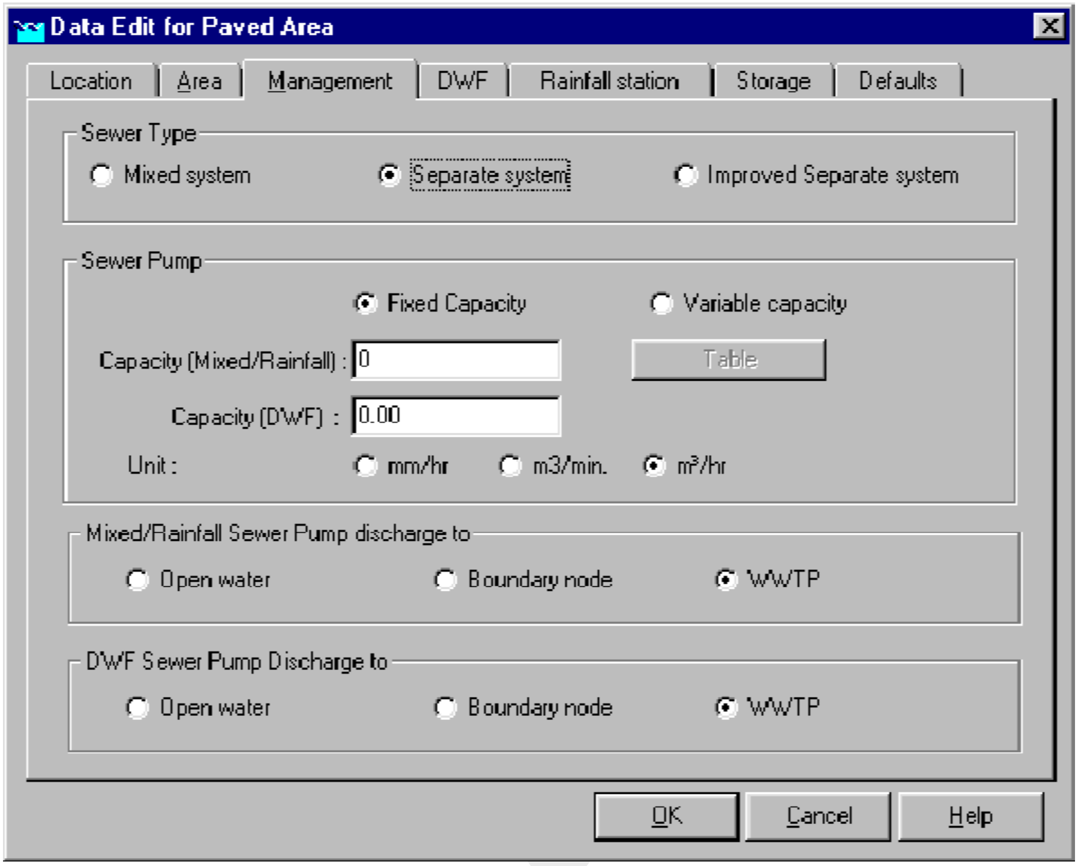


Figure 5.164: Data Edit for Paved Area *window, the Management tab*

On the Management tab, one can define the operation of the sewer system into which the intercepted rainfall flows.

◇ **Sewer Type:** There are three sewer types: Mixed system, Separate system and Improved separate system. The images below show how they work:

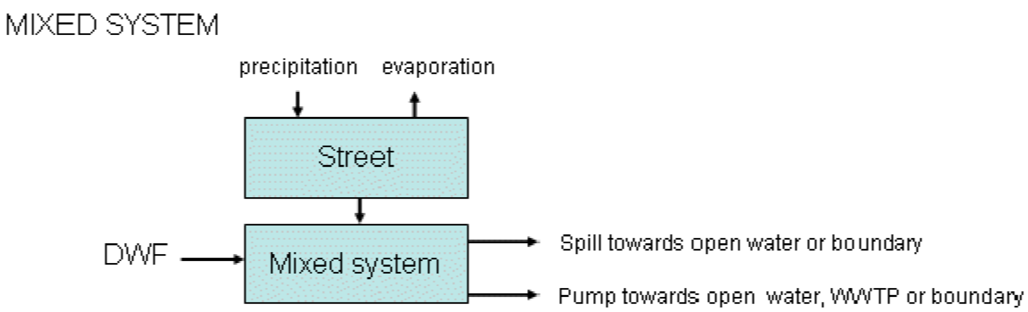


Figure 5.165: *Sewer type: Mixed system*

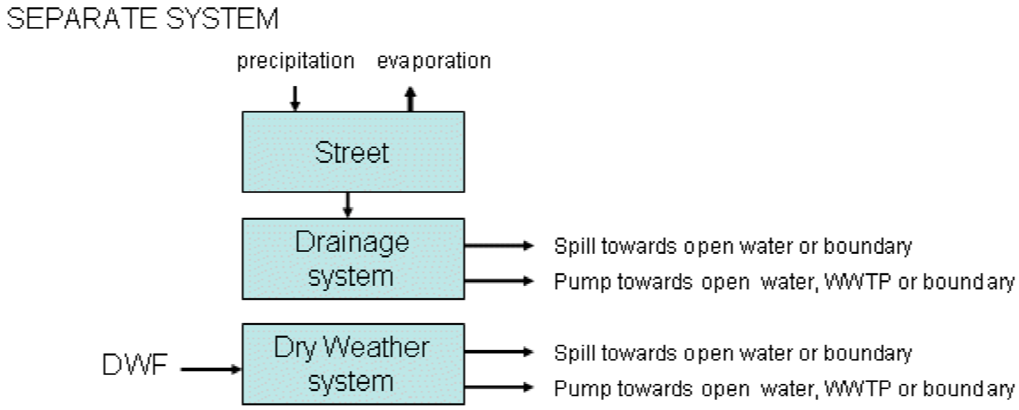


Figure 5.166: *Sewer type: Separate system*

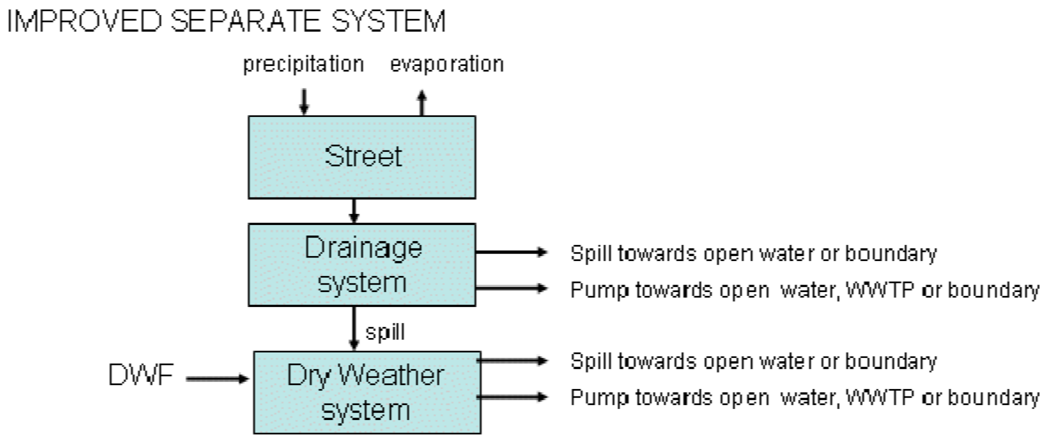


Figure 5.167: *Sewer type: Improved separate system*

- ◇ **Sewer pump:** Here, the pumping capacities for the sewer pump[s] should be specified. These pump will attempt to keep the systems empty. In case of a separate system, each subsystem has its own pump.
- ◇ **Pump discharge to:** You should also define whereto the sewer pump[s] should transport its/their discharge. This can either be an RR open water node, a boundary node or a Waste Water Treatment Plant. If you want to have the discharge pumped towards a Channel Flow connection, select 'boundary'. The Rainfall Runoff module sees RR-Flow connections as boundaries.

The Dry Wether Flow (DWF) tab

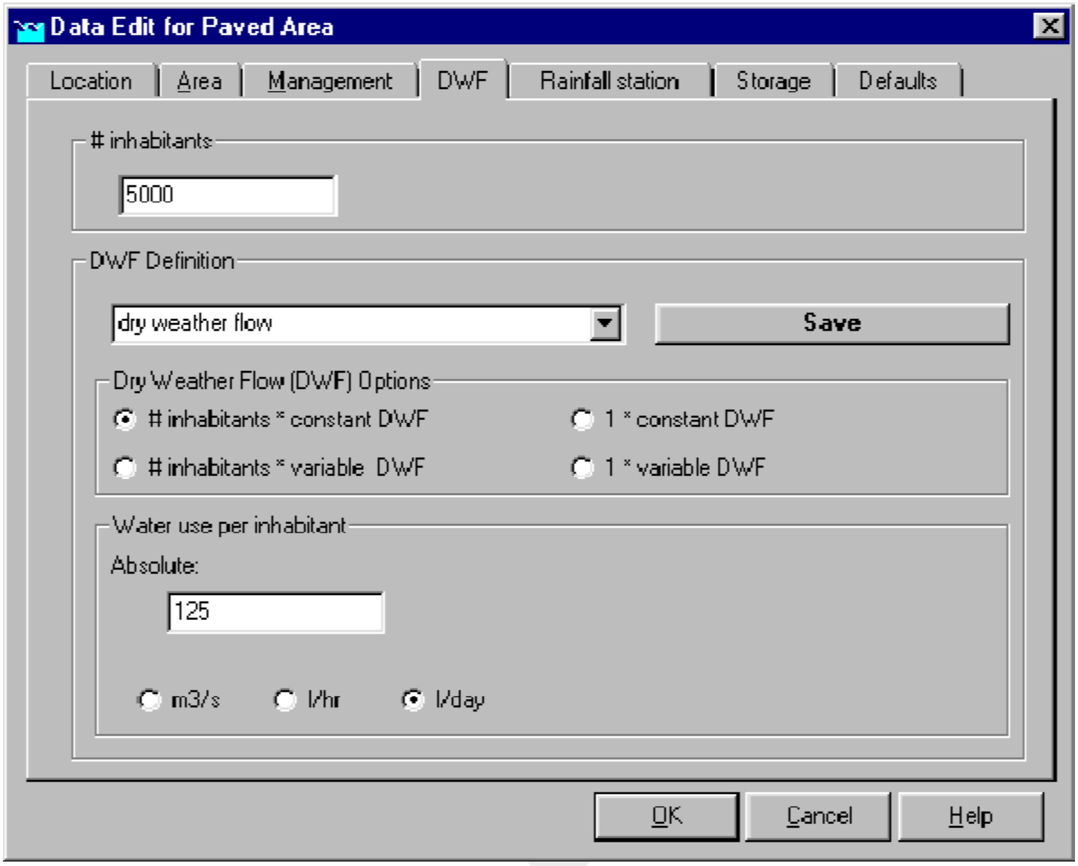


__Figure 5.168: Data Edit for Paved Area__ window, the Dry Weather Flow tab

On this tab, you should define the amount of inhabitants and the amount of water they use daily. Both numbers are then multiplied and simulated as an inflow towards the paved nodes sewer system.

The Rainfall Station tab

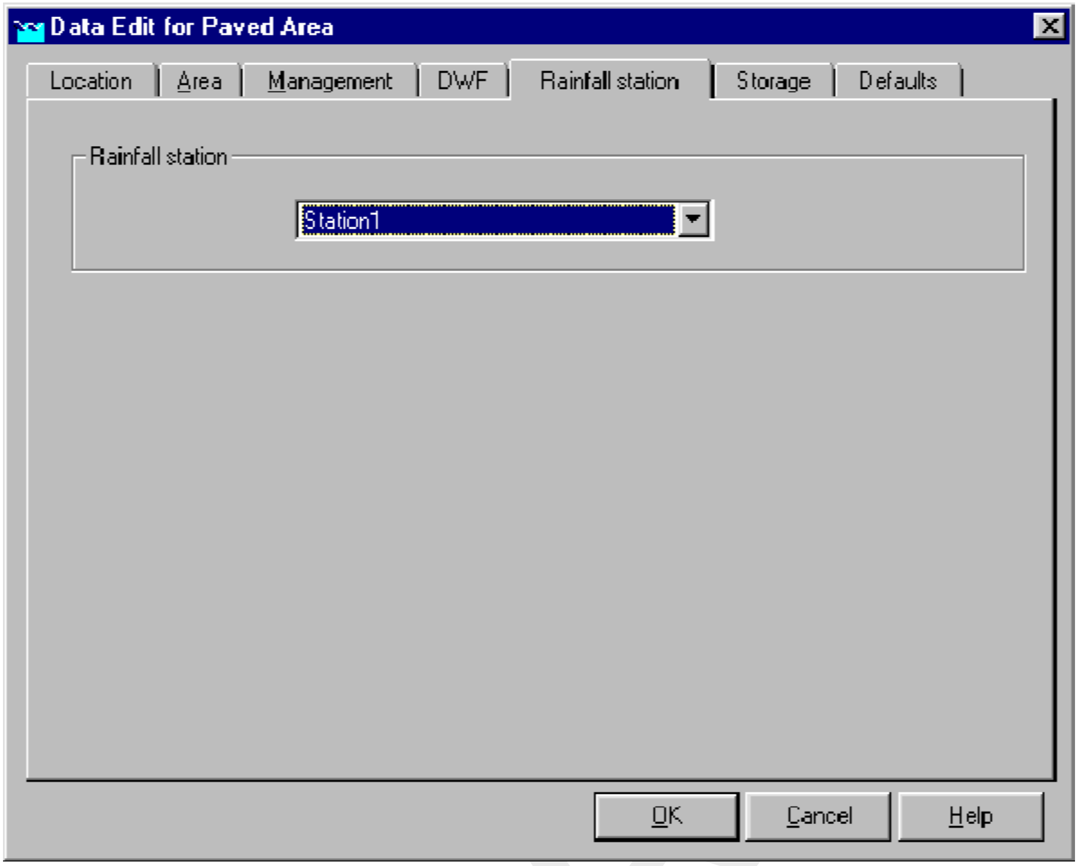


Figure 5.169: Data Edit for Paved Area *window, the Rainfall Station tab*

Select here the rainfall station that applies for this node. Rainfall stations can be created in the "Meteorological data" task block.

The Storage tab

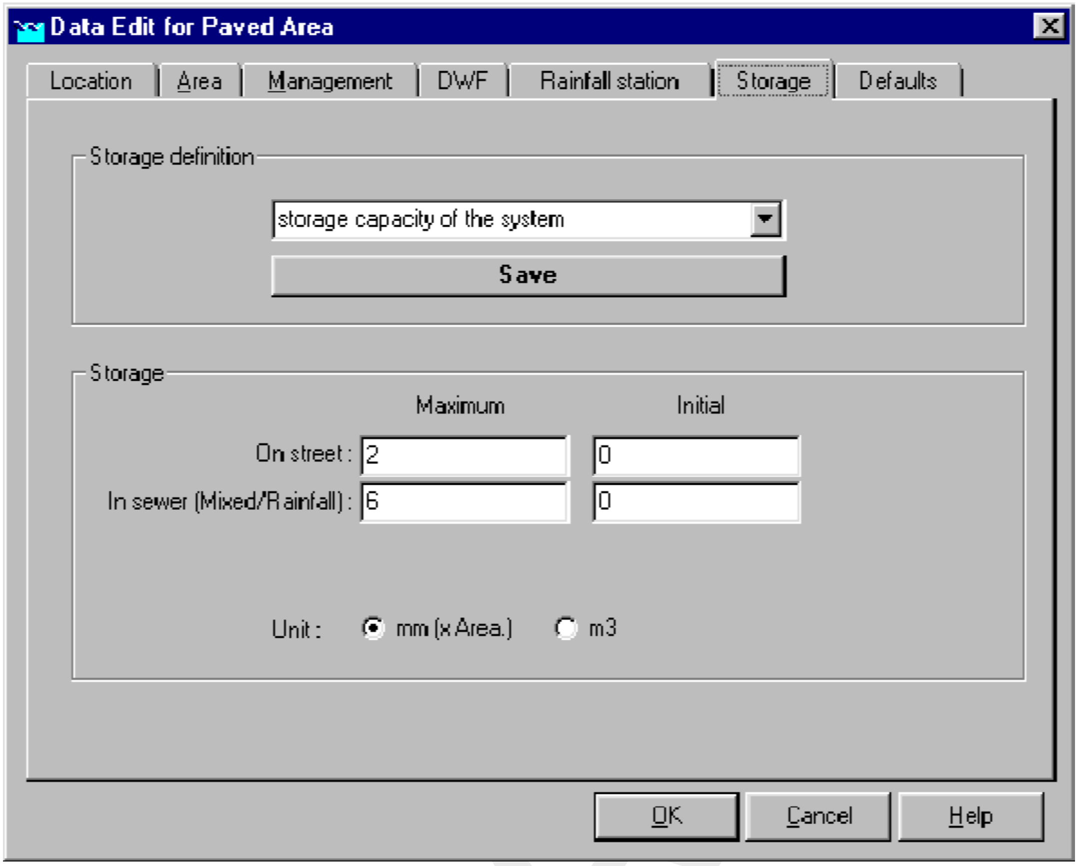


***Figure 5.170: Data Edit for Paved Area** window, the Storage tab*

On the storage tab, the maximum storage capacities of the street level and the sewer system should be specified.

How does it work:
When the rainfall intensity exceeds the maximum storage on street, the remaining amount will flow into the sewer system. On its turn, when that discharge exceeds the maximum storage plus pumping capacity of the sewer, the remaining discharge is spilled (see Management tab).

The initial storage fields can be used when part of the available storage is already filled up at the start of a simulation.

Paved area node topology

RR - Paved nodes give a choice to create one or two connections.

- ◇ In case only one connection is used, both the sewer pump discharges and the sewer overflows (spills) will flow towards the same node.
- ◇ In case two connections are used, the user may, for example, choose to have the sewer pump discharges flow towards a boundary node, and the sewer overflows towards an RR open water node. By default, all sewer overflows (spills) flow towards open water nodes.

RR - Paved nodes can be connected to either one of the following node types:

- ◇ An RR Open Water node 

- ◇ An RR on channel connection 

- ◇ An RR on Flow connection node 

- ◇ An RR Boundary node 

- ◇ An RR Waste Water Treatment Plant node 

When connecting a paved area node to one of these nodes, make sure you have selected the *RR-link* connection type. Notice: the usage of an RR-Sewerage Link is optional. The difference with a regular RR link is only for presentation purposes.

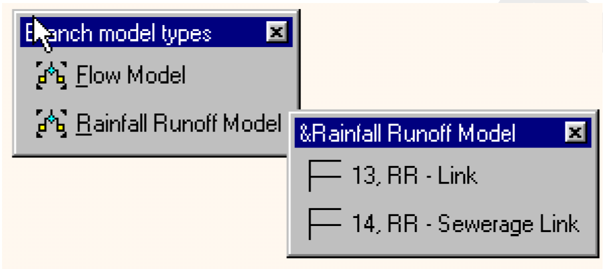


Figure 5.171: *Select RR-Paved node connection*

Below, some examples of typical RR Paved connections are given.

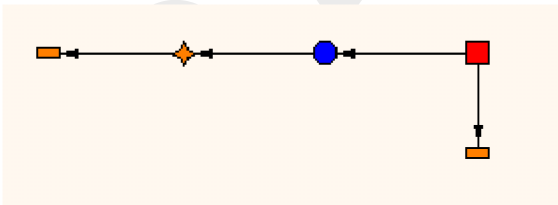


Figure 5.172: *Connecting an RR Paved node to an RR open water for the sewer overflows and surface runoff and an RR boundary for the sewerage discharge.*

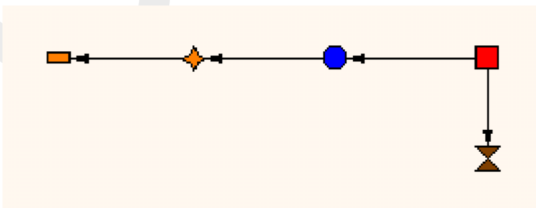


Figure 5.173: *Connecting an RR Paved node to an RR open water for the sewer overflows and surface runoff and an RR Waste Water Treatment Plant for the sewerage discharge. Note that the WWTP node on its turn should be connected to another node.*

5.4.9 RR - Pump Station

RR - Pump station node general description

In this chapter, the *RR - Pump station node* is described. With an *RR - Pump station node* you can simulate a pump that is activated or deactivated through changing water levels. RR - Pump stations can either be applied to withdraw excess water or to supply water in periods of shortage.

- ◇ For a detailed description of this node's input parameters: see the "RR - Pump station node input screens" section from the Reference Manual;
- ◇ For a detailed description of this node's possible network configurations: see the "RR - Pump station node topology" section from the Reference Manual

RR - Pump station node input screens

When the model data editor for an *RR - Pump Station* node type is started, the following tabs will become available for input:

Options tab

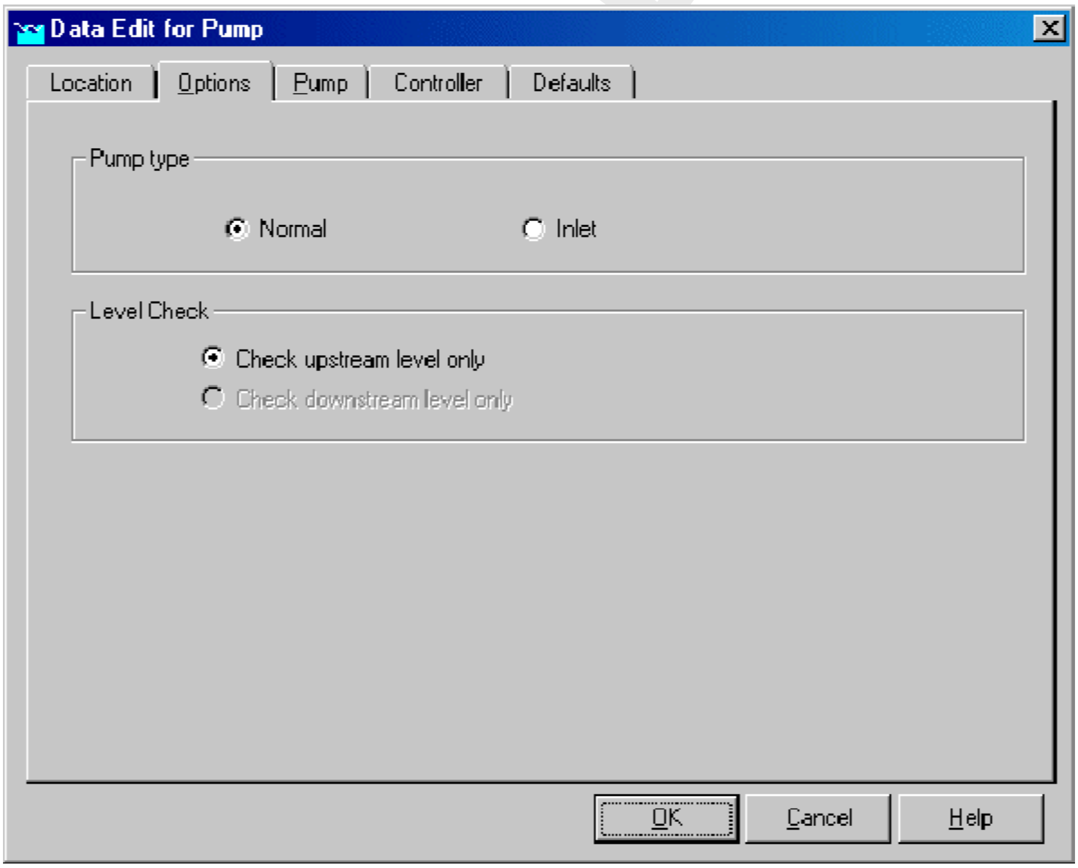


Figure 5.174: Data Edit for Pump*, the Options tab*

On the "Pump" tab, one can define how the pump will be used:

- ◇ **Normal:** The pump will operate as a normal pump that starts **extracting** water when the water level on the upstream side (suction side) exceeds the switch-on level. It will then continue pumping until the water level drops below the switch-off level. In this case, the switch-on level will be higher than the switch-off level.

◇ **Inlet:** The pump will **supply** water to prevent water shortage in the downstream area (pressure side) when the water level there drops below the switch-on level. It will then continue pumping until the water exceeds the switch-off level again. In this case, the switch-on level will be lower than the switch-off level. **Note:** look at the direction of the arrows on the RR-links to find out which side of the pump is defined as the pressure side, and which one as the suction side. You may need to reverse the directions in order to make the pump work the way you want it to.

Pump tab:

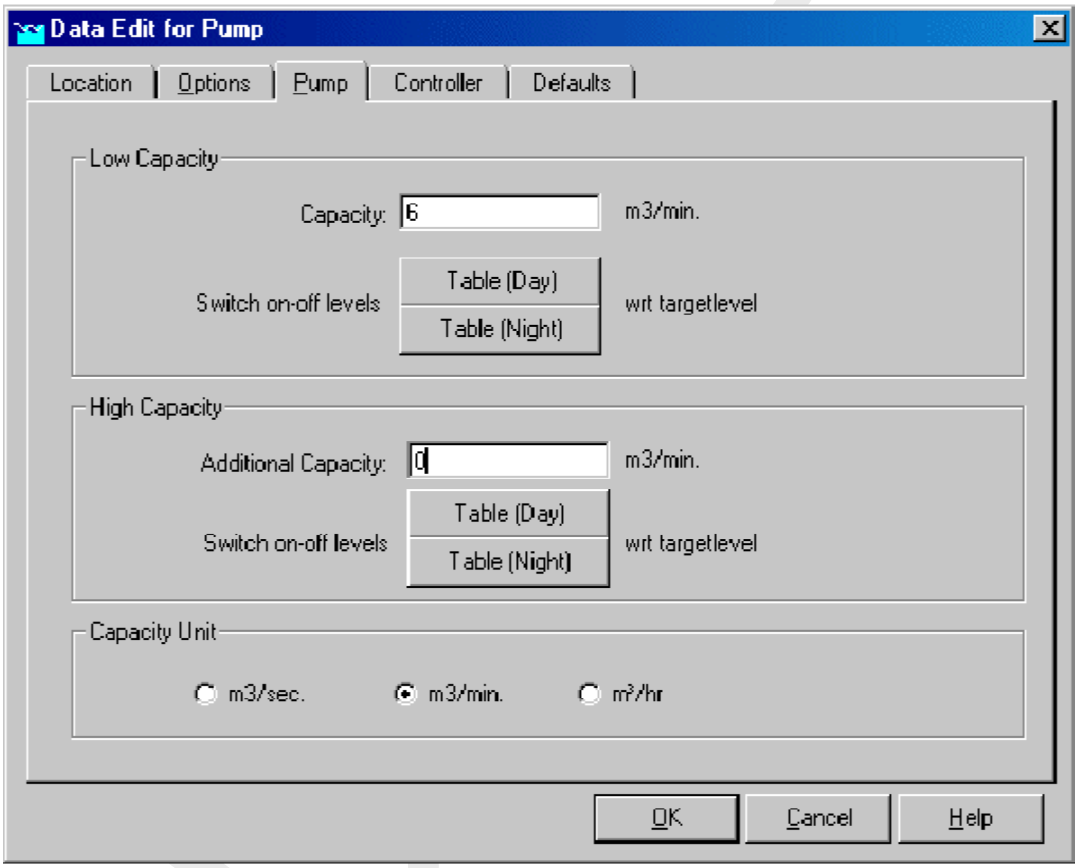


***Figure 5.175: Data Edit for Pump**, the Pump tab*

On this tab, pump capacities and switch-on and -off levels should be entered. Two pump capacities can be entered. For each pump capacity, the corresponding switch-on and switch-off values for day- and nighttime should be entered.

The distinction between day- and night is made because the usage of electricity is cheaper during nighttime. During nighttime, pumps are usually switched on earlier and switched off later. The night- and day hours can be specified in the SETTINGS task block for the Rainfall-Runoff module.

Notice that, when both capacities are active at the same time, both values are summed up in order to get the total capacity of that moment.

Controller tab:

On this tab, a time controller can be applied. With a time controller the pumping capacity can be defined as a function of time, and the capacities according to the values on the "Pump" tab are overruled.

****************** IMPORTANT **************************************

The controller will <u>only</u> adjust the pump **capacity**. **Not** its status of *active* or *inactive*. The status of active or inactive is controlled by the values on the "Pump" tab.

In other words: if the pump would be inactive judging from the settings on the "Pump" tab, it cannot be re-activated by the controller! Therefore, make sure that, if you want the controller to be active all the time, your pumping station should **also** be active according to the settings on the "Pump" tab. Only then, the controller will be able to adjust the capacity to your desired values in time.

****

RR - Pump station node topology

An RR - pump station should always be connected to at least one RR - Open Water node, as it is meant to let water flow in or out of an open water section.

RR - pump station can be connected to nodes of the following types:

◇ 31, RR - Open Water
◇ 32, RR - Boundary
◇ 26, Flow-RR Connection on Channel (available in the Flow modules only)
◇ 27, Flow-RR Connection on Flow Connection Node (available in the Flow modules only)

5.4.10 RR - QH relation node

RR-QH Relation node general description

37, RR - QH relation

In this chapter, the *RR-QH Relation node* is described. If the relationship between the water levels in an area and the discharges from that area is known, such knowledge may be applied to a QH-Relation node.

In fact one can simulate any structure type by using a RR-QH Relation node. In hydraulic river models (1D flow) QH-relations are commonly used to define downstream boundary conditions.

◇ For a detailed description of this node's input parameters: see "RR-QH Relation node input screens";
◇ For a detailed description of this node's possible network configurations: see "RR-QH Relation node topology"
◇ For a detailed description of this node's underlying mathematical equations: see QH-relation.

RR-QH Relation input screens

When the model data editor for an *RR-QH Relation* node type is started, the following tabs will become available for input:

Q-h relation tab:

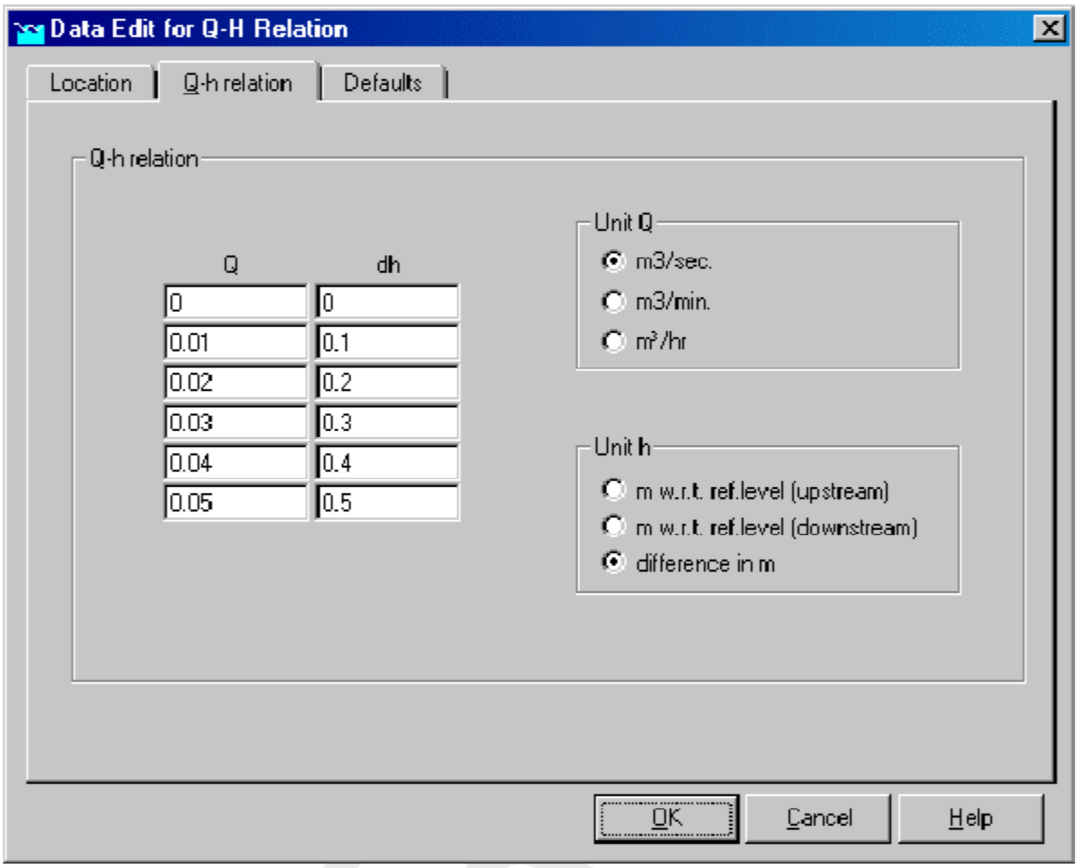


***Figure 5.176: Data Edit for Q-H relation** window,* Q-h relation *tab*

In the table, the relationship between water levels in the area and the discharge through the "structure" should be entered. For six representative water levels (or level differences), a corresponding discharge should be entered.

There are three options for the RR-QH Relation. The options can be chosen in the *Unit-h* section of this tab.

1 **Discharge related to upstream water level only** If this option is chosen, the calculated discharge will only depend on the upstream water level.
2 **Discharge related to downstream water level only** If this option is chosen, the calculated discharge will only depend on the downstream water level.
3 **Discharge related to water level difference** If this option is chosen, the calculated discharge will depend on the water level difference between the upstream and downstream RR nodes.

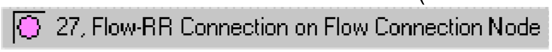
RR-QH Relation topology
Like any other RR structure type, an RR-QH Relation should always be connected to at least one RR - Open Water node, as it is meant to let water flow in or out of an open water section.

RR-QH Relations can be connected to nodes of the following types:

- 

- 

- 

(available in the Flow modules only)
- 

(available in the Flow modules only)

5.4.11 RR - Sacramento node

Sacramento node general description

In this chapter, the Sacramento node is described. Nodes of this type host the Sacramento Rainfall-Runoff modelling concept, which has been implemented in the SOBEK RR module.

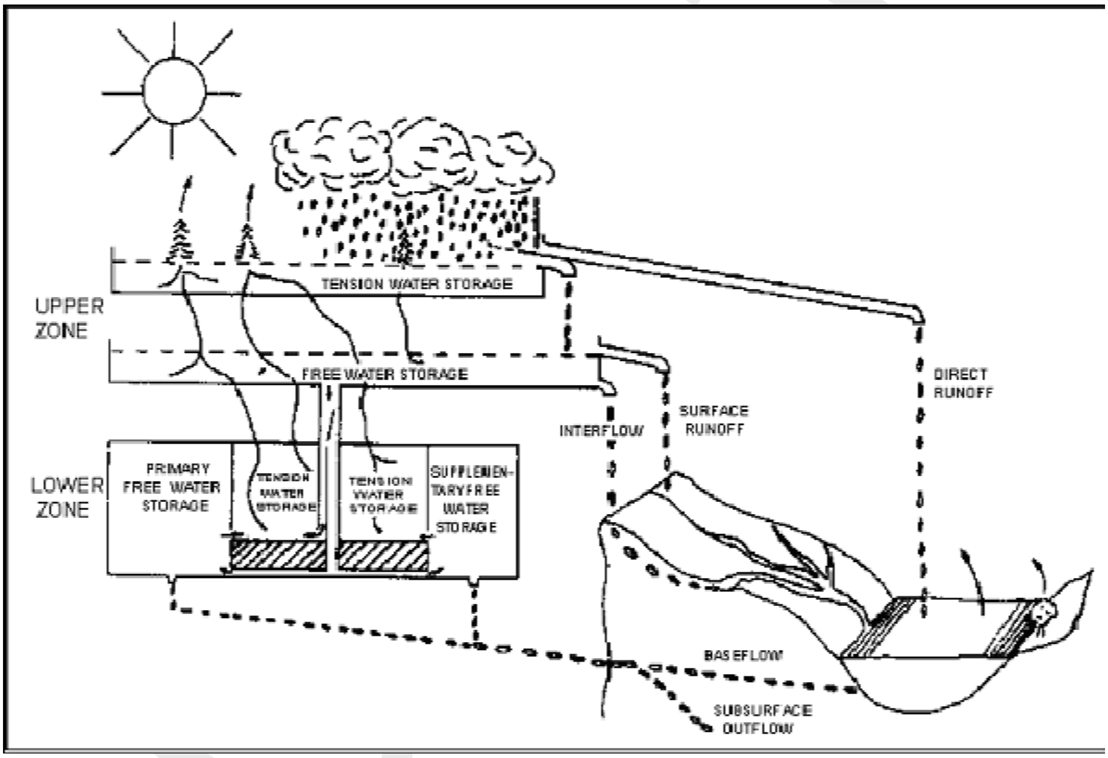


Figure 5.177: *The rainfall-runoff processes that are included in the Sacramento concept.*

What is Sacramento?
Sacramento is a **concept** for rainfall-runoff modelling. It has originally been derived from the Stanford Watershed model, and designed for the Sacramento river system, United States. Today, it is one of the most popular rainfall-runoff modelling concepts. It describes the mathematical equation that count for each process within the transformation of rainfall into an outflow towards a river.

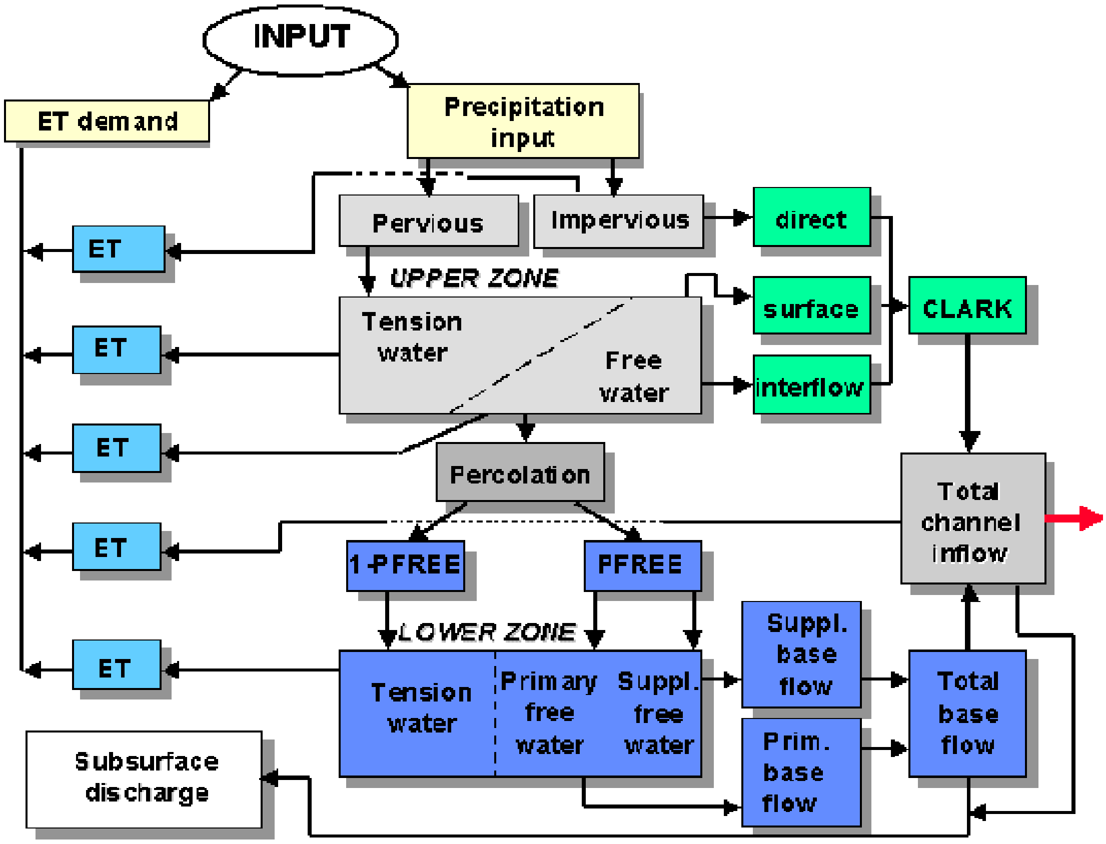


Figure 5.178: *Flowchart of the processes that are covered by the Sacramento concept.*

◇ For a detailed description of this node's input parameters: see the Sacramento node input screens section from the Reference Manual;

◇ For a manual on the Sacramento rainfall-runoff model concept: see the Technical Reference Manual for the Rainfall Runoff module.

◇ For guidelines on Sacramento parameters: see the document on Estimation of Sacramento segment parameters in the Technical Reference Manual for Rainfall Runoff

The application of the Sacramento model in the way it has been integrated in SOBEK is based on a semi-distributed approach. It implies that a catchment is divided into a number of segments, which are interconnected by channel branches as shown in the figure below:

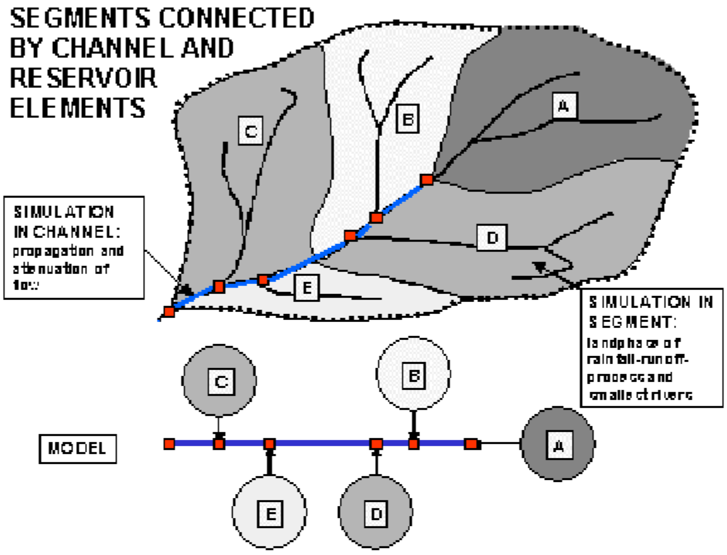


Figure 5.179: Catchment divided into a number of segments

In each segment, rainfall is transformed into runoff towards the main river system. Within each segment, areal homogeneity of rainfall input and basin characteristics is assumed. The contributions of the segments may be linked to a SOBEK Channel Flow or River Flow schematisation.

Note: to apply hydrodynamics to your schematisation, link the Sacramento nodes to your SOBEK Flow schematisations by using RR connection on Flow connection nodes, or RR connection on channel nodes.

Sacramento node input screens
When starting the model data editor for a Sacramento node type, the following tabs will be available for input:

Area tab:

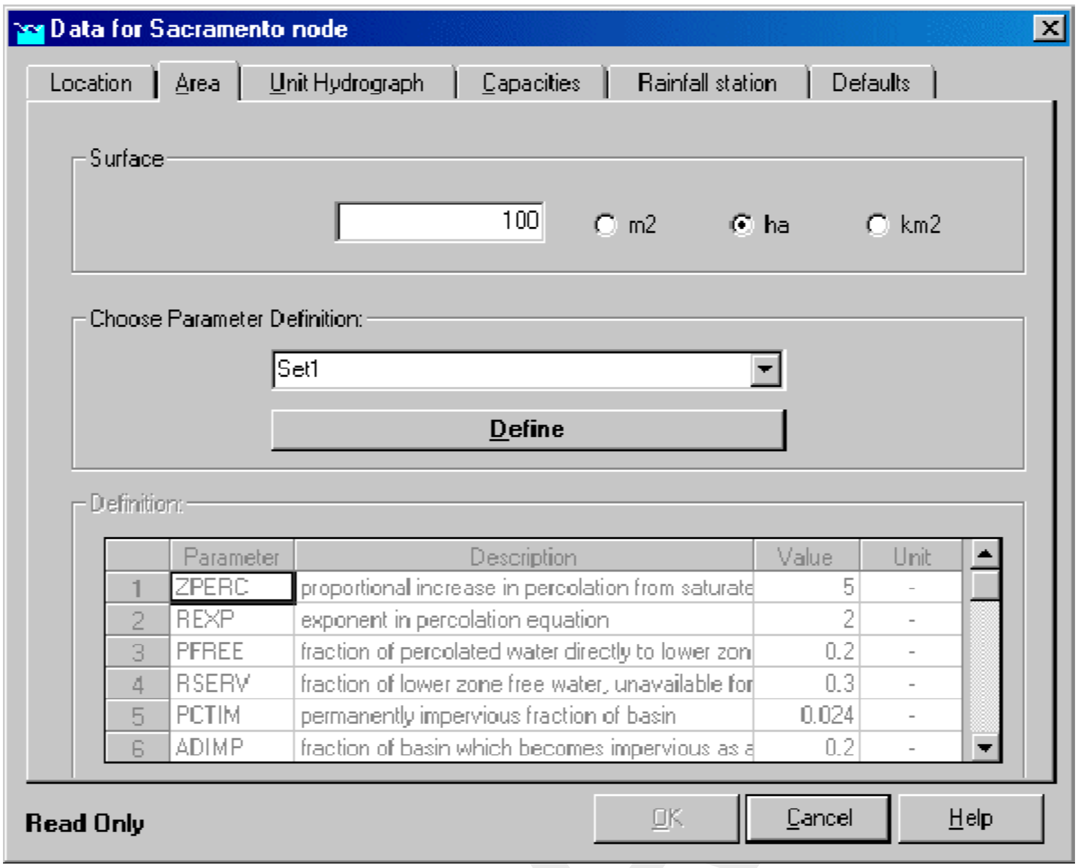


***Figure 5.180: Data Edit for Sacramento node** window,* Area *tab*

Here, the following parameters can be specified:

- ◇ **Surface** Enter the total surface , i.e. pervious and impervious area.
- ◇ **Parameter set definition** You may want to apply a unique combination of parameter values for multiple nodes. For that reason, such a combination can be saved as a parameter set. Enter a name for the set in the drop down box, and click the *Define* button. Then, enter the appropriate parameter values. Finally click *Save*, and the set can be selected for any Sacramento node by choosing it from the drop down box.
- ◇ **ZPERC** The proportional increase in percolation from saturated to dry condition is expressed by the term ZPERC. The value of ZPERC is best determined through computer trials. The initial estimate can be derived by sequentially running one or two months containing significant hydrograph response following a dry period. The value of ZPERC should be initially established so that a reasonable determination of the initial run-off conditions is possible.
- ◇ **REXP** The exponent in the percolation equation which determines the rate at which percolation demand changes from the dry condition, (ZPERC + 1) * PBASE, to the wet condition, PBASE. Figure VIII.3.4 illustrates how different values of the exponent affects the infiltration rate. It is recommended that an initial estimate of this exponent is made from the same record which is used in determining an initial estimate of ZPERC. The interaction between PBASE, ZPERC and REXP may require a shift of all three terms whenever it becomes clear that a single term should be changed. Visualising the percolation curve generated by these three terms helps to ascertain the necessary changes. The observed range of REXP is usually between 1.0 and 3.0. Generally a value of about 1.8 is an effective starting condition.
- ◇ **PFREE** Fraction of the percolated water which is transmitted directly to the lower zone free

water aquifers. Its magnitude cannot generally be determined from hydrograph analysis. An initial value of 0.20 is suggested. Generally, values will range between 0 and 0.40. The analysis of early season baseflow allows an effective determination of PFREE.

◇ **RSERV** Fraction of the lower zone free water which is unavailable for transpiration purposes. Generally this value is between zero and 0.40 with 0.30 being the most common value. This factor has very low sensitivity.

◇ **PCTIM** Permanently impervious fraction of the basin contiguous with stream channels. It can be determined from small storms after a significant period of dry weather. Then the volume of direct runoff (= observed runoff - baseflow) divided by the volume of rain gives the percentage impervious fraction of the basin. PCTIM should not be close to 1!

◇ **ADIMP** Fraction of the basin which becomes impervious as all tension water requirements are met. It can be estimated from small storms after a very wet period. As before, the volume of direct runoff divided by the volume of rain gives the total percentage of impervious area. The estimate for ADIMP follows from:ADIMP = Total Percentage Impervious - PCTIM

◇ **SARVA** Fraction of the basin covered by streams, lakes, and riparian vegetation, under normal circumstances. The SARVA area is considered to be the same as or less than PCTIM. Detailed maps may be referred to in order to estimate the extent of paved areas which drain directly to the streams so that differences between PCTIM and SARVA can be approximated. Generally, SARVA appears to range between 40 % and 100 % of the PCTIM value.

◇ **SSOUT** The sub-surface outflow along the stream channel which must be provided by the stream before water is available for surface discharge. This volume expressed in mm/time interval is generally near zero. It is recommended that the value of zero be utilised, and SSOUT is applied only if the log Q vs time plot requires a constant addition in order to achieve a valid recession characteristic. If constant volumes of flow are added to observed stream flow, the slope of the discharge plot will be altered. That value, which is required to linearize the primary recession, is the appropriate value of SSOUT. It should be realised that where SSOUT is required, an effective determination of lower zone free water storages and discharge rates will require inclusion of the SSOUT value.

◇ **PM, PT1, PT2** Time interval increment parameter (PM) and rainfall thresholds (lower rainfall threshold PT1 and upper rainfall threshold PT2).The model simulates the rainfall-runoff process with a time step which is smaller than the time interval of the basic data (usually one day). The number of increments in the time interval is derived from:N∆t = 1 + PM * (UZFWC * F + Peff)where:F = 1 : for Peff $<$ PT1F = (Peff/PT2)0.5 : for PT1 $<$= Peff $<$= PT2F = 1-PT2/Peff : for Peff $>$ PT2 If the input time interval equals an hour, this option is internally skipped.

Unit Hydrograph tab:

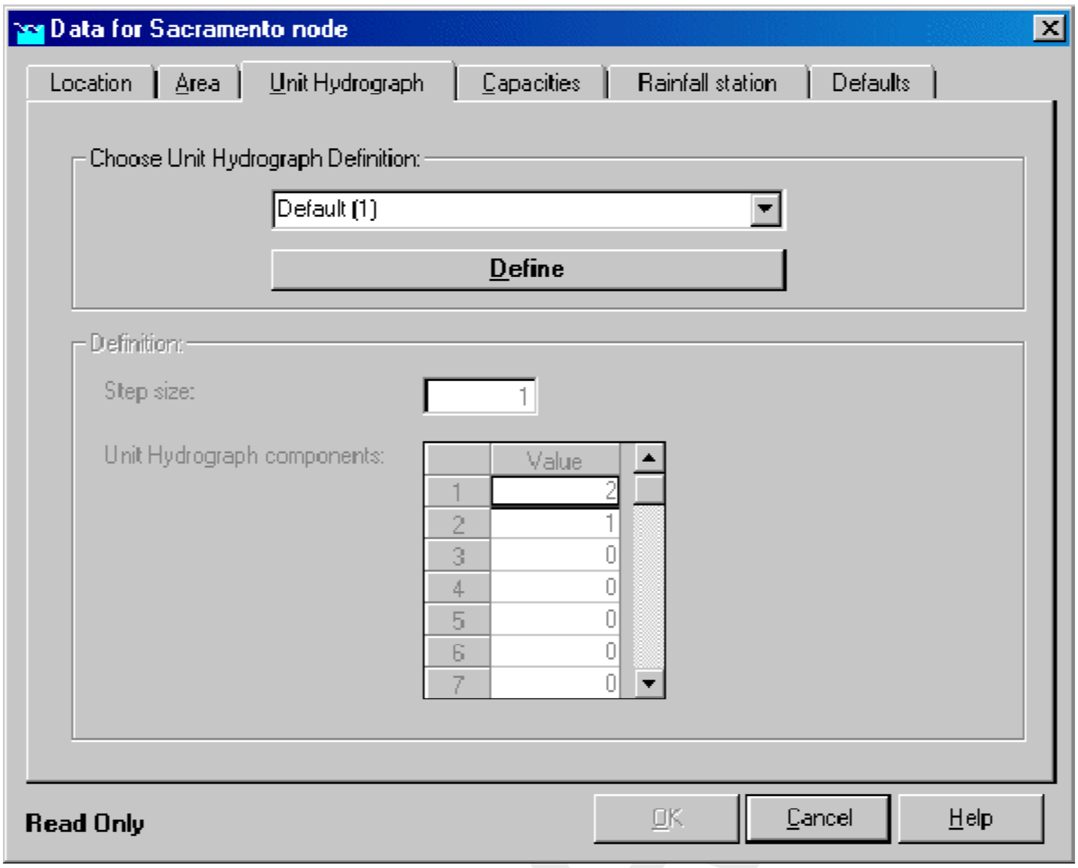


Figure 5.181:* *Data Edit for Sacramento node *window,* Unit Hydrograph *tab*

About the Unit Hydrograph:
On this tab, a Unit Hydrograph can be defined. It is used to transform the runoff from impervious areas (direct runoff), the surface runoff and the interflow into an adapted time distribution of these flow rates. Before the runoff from the impervious areas (direct runoff), the surface runoff and the interflow branches the channel, they may be transformed according to a Unit Hydrograph.

◇ **The unit hydrograph (UH)** runs for hourly and daily intervals only. Hence, only one hour UH or one day UH's can be entered. At maximum 36 (non-zero) UH-ordinates can be entered, together with a step size. The units with which the UH are to be entered are not of importance; they should only be mutually consistent. First the given ordinates (with given step-size) are interpolated down to ordinates at one hour intervals. Then the sum of the ordinates is computed and all ordinates are subsequently divided by the sum, so the sum of the adjusted ordinates will always be 1.
To test the behaviour of the UH for surface runoff, a run can be made with all storages initially filled and with UZK and LZSK e.g. = 10^{-5}. The only reservoir that is depleted then will be the Lower Zone Primary reservoir = LZPWC * LZPK. Note that LZPK, LZSK and UZK are to be given as fraction of runoff per day. To get the hourly value the daily value is internally adjusted as follows:UZK(hr)=(1-(1-UZK(day))1/24), etc. for the other linear reservoirs.
Example:

1 Given unit hydrograph components : 2. 4. 3. 2. 1. . . . .
2 Step size = 2
3 Computed unit hydrograph components:

0.042 0.083 0.125 0.167 0.146 0.125 0.104 0.083 0.063 0.042 0.021

4 Given the 5 (non-zero) ordinates at 2 hour intervals a time base of 12 hours is considered.
The ordinates at two hourly intervals first are converted to ordinates at 1 hour intervals: 1, 2, 3, 4, 3.5, 3, 2.5, 2, 1.5, 1, 0.5. Its sum is 24. Dividing the 1 hour ordinates by 24 results in the above shown computed components.

Note:

- ◇ In the SOBEK Sacramento node the UH is available to transform the direct runoff + surface runoff + interflow as described above.
- ◇ The original Sacramento model concept offers some methods to transform the total inflow hydrograph (i.e. direct runoff + surface runoff + interflow + 2 baseflow components). These methods are the Standard Muskingum method or the Clark method. **Note:** The latter is not available in the SOBEK Sacramento node.
- ◇ Instead of using routing methods like Muskingum, one can use SOBEK-Channel flow for routing the inflow hydrograph from a SOBEK Sacramento node.

The Capacities tab:

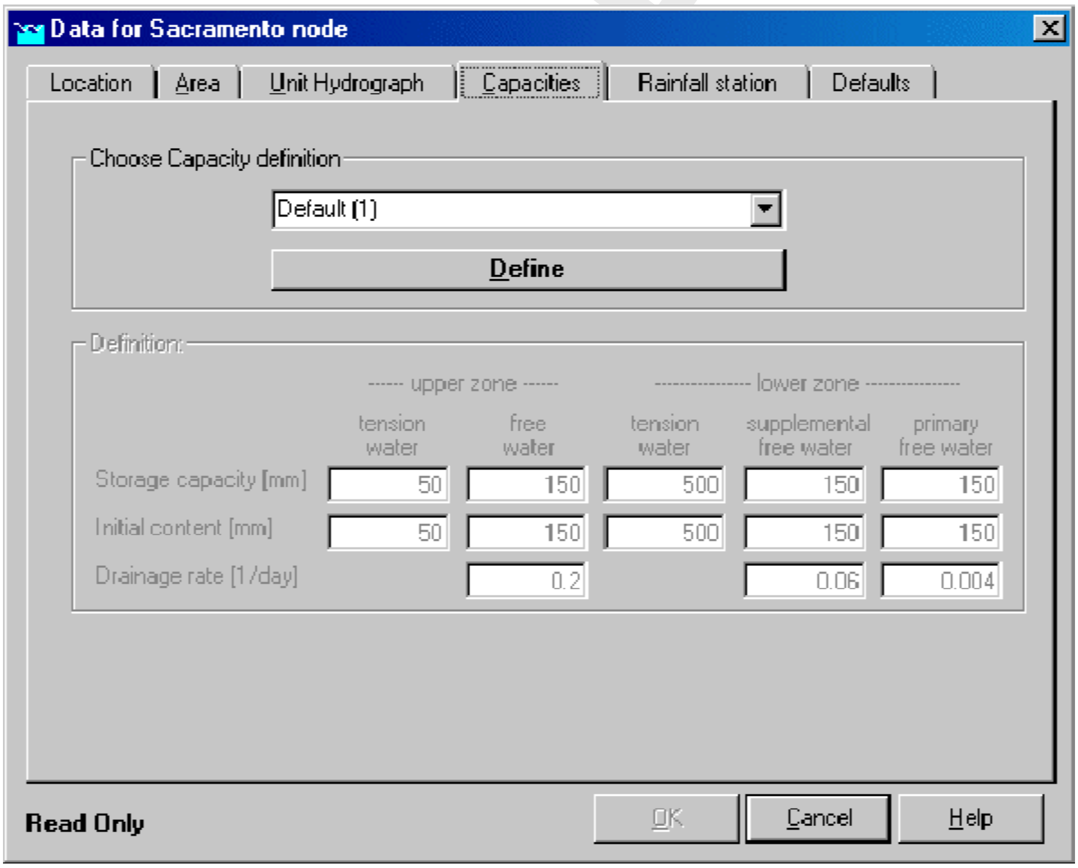


***Figure 5.182: Data Edit for Sacramento node** window,* Capacities *tab*

Here, one can define the storage capacity of the five reservoir types (see the figure in the chapter Sacramento node general description) These types are: Upper zone, tension water Upper zone, free water Lower zone, tension water Lower zone, supplemental free watero Lower zone, primary free water

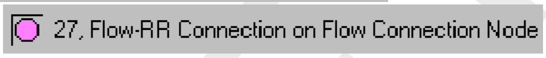
- ◇ **Initial Content** Initial content of the reservoirs, thus the amount of storage capacity that is already occupied at the start of a simulation.
- ◇ **Drainage rate** Upper zone free watero Lower zone, supplemental free watero Lower zone, primary free water

Rainfall Station tab:
Select here the rainfall station that applies to this node. Note: rainfall stations and rainfall data can be specified in the Meteorological Data task block.

Sacramento node topology
Nodes of the Sacramento type can be placed in a network in exactly the same way as RR-unpaved nodes. They can be connected to either of the following three node types:

- ◇ An RR Open Water node 

- ◇ An RR on channel connection 

- ◇ An RR on Flow connection node 

When connecting a Sacramento area node to one of these nodes, make sure you have selected the *RR-link* connection type:

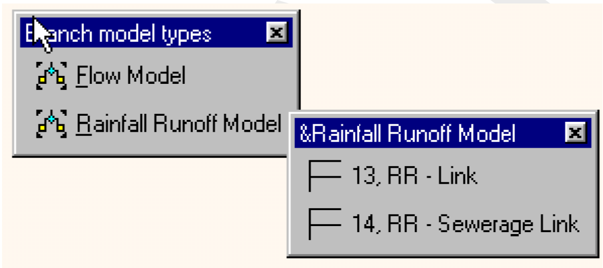


Figure 5.183: *Showing the RR-Link connection type*

The images below show how the Sacramento area node can be connected to the three nodes mentioned.

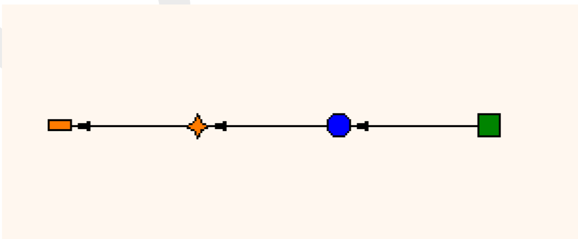


Figure 5.184: *Connecting an RR Sacramento node to an RR Open Water node. Notice the defined link directions.*

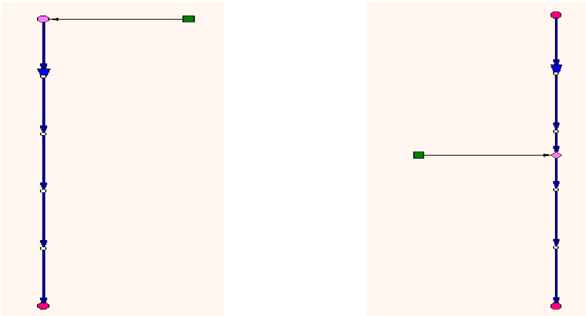


Figure 5.185: Two ways to connect an RR Sacramento node to a channel (only available if you have a license for the flow module).

5.4.12 RR - Unpaved node

General description of the unpaved area node

In this chapter, the unpaved area node is described. It is by far the most important modelling object within the SOBEK-Rural Rainfall-Runoff module.

◇ For a detailed description of this node's input parameters: see the "unpaved area input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the "unpaved area node topology" section from the Reference Manual
◇ For a detailed description of this node's underlying mathematical equations: see the "Technical Reference" manual.

What does the unpaved area node do?
This node type transforms rainfall that falls on a parcel of land into an outflow towards open water. In order to do so, it catches rainfall, which infiltrates in the soil and/or is stored on the land. The infiltrated water fills up the groundwater, so that the groundwater level will rise. Through different soil layers, each covering its own drainage capacity it will then flow out of the soil towards the connected open water. Also evaporation, infiltration and percolation can be modelled. If desired, unsaturated zone processes can be included in the calculations.

Because the SOBEK Rainfall-Runoff module is a so-called zero-dimensional model, an unpaved node can be seen as a container where water flows in (rainfall and seepage), water is stored (change in groundwater table & unsaturated zone) and water flows out, (evaporation, runoff, surface runoff and infiltration). The picture below gives an impression of the various terms which apply to the unpaved area node.

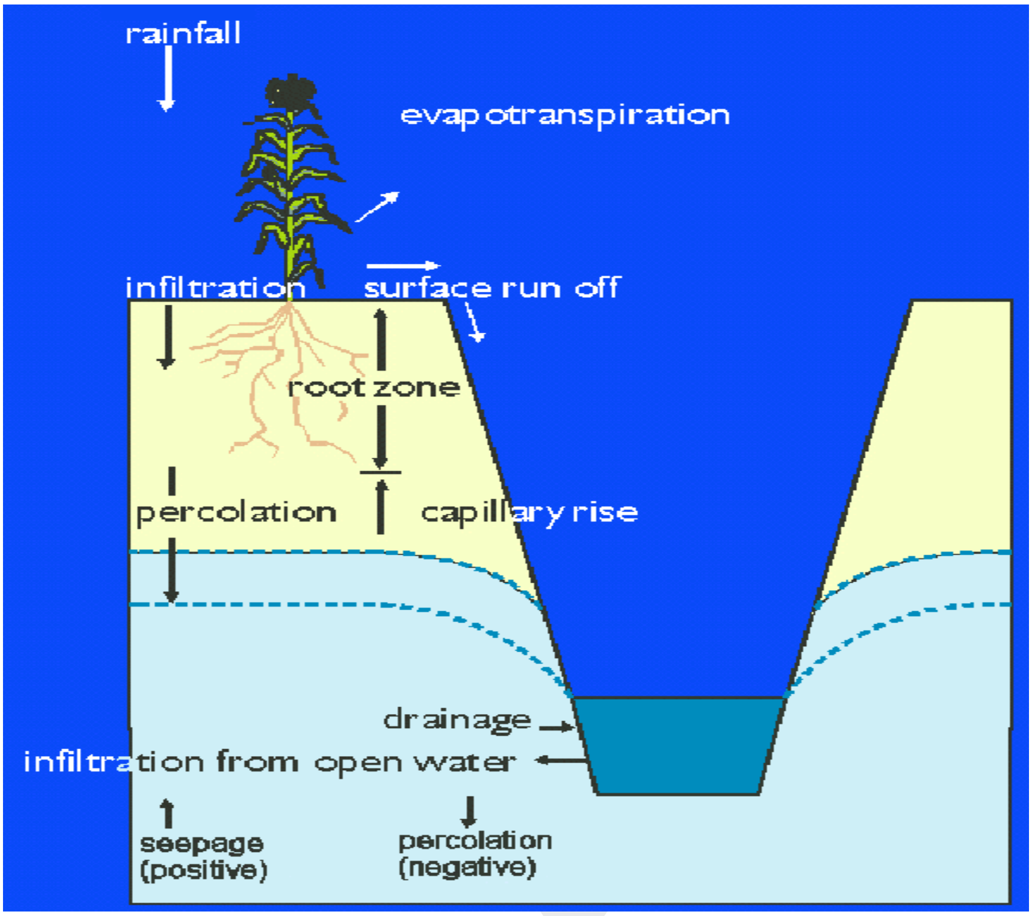


Figure 5.186: *Representation of the rainfall-runoff process in unpaved areas*

In addition, the user can choose whether to use the above described definition of modelling the transport processes in the unpaved area, or to use a more detailed description. In that case the unsaturated zone is modelled by means of root zone reservoir.

The unsaturated zone is modelled as vertically oriented 1-D model using the steady-state approach. The 1-D column model consists of a reservoir representing the root zone and the subsoil. The equilibrium moisture storage in the root zone is defined as the amount of moisture corresponding with a steady-state situation with no-flow conditions to or from the root zone. If the equilibrium moisture storage for the root zone is exceeded, excess water will percolate to the saturated zone. If the moisture storage is less than the equilibrium moisture storage, upward flow from the saturated zone is simulated through capillary rise. The height of the phreatic surface is calculated from the water balance of the subsoil, using a storage coefficient which is dependent on the depth of the groundwater table.

The root zone reservoir is used for calculations of evapotranspiration. The subsoil reservoir is used to calculate the saturated storage coefficient.

Evapotranspiration is determined by the crop and the moisture content in the root zone. For these calculations, recorded values of precipitation and potential evapotranspiration of a reference crop and woodland must be available. The potential evapotranspiration for other crops or vegetation types are derived from the values for the reference crop by conversion.

Some important characteristics of the unsaturated zone, i.e. upward flux, storage coefficient and equilibrium moisture storage of the root zone, are calculated from the soil moisture retention-, and hydraulic conductivity curves using the CAPSIM model which is based on the assumption of steady state soil moisture flow (Wesseling, 1991). For more than twenty different soil types, sets of pre-defined tables are provided with SOBEK.

Input screens for the unpaved area node type
When starting the model data editor for an unpaved area node type, the following tabs will be available for input:

The Area tab

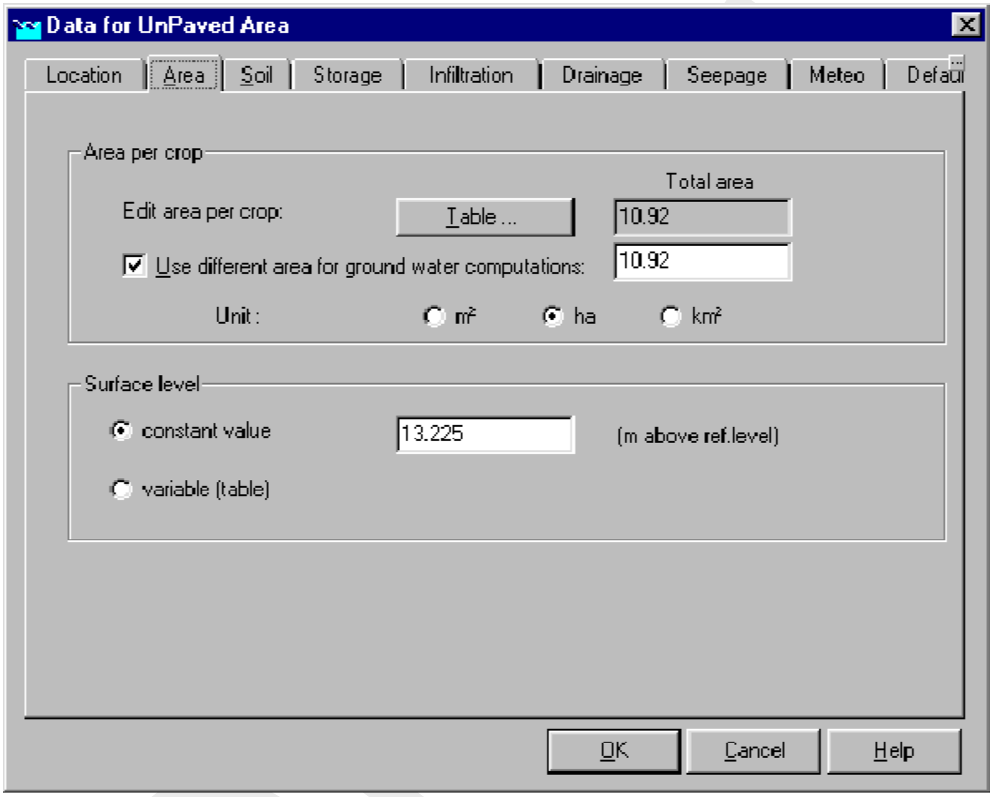


***Figure 5.187: Data for UnPaved Area** window,* Area *tab*

Here, the following data can be entered:

- **Area per crop:** Every crop type has it's own evaporation characteristics. Therefore it is important to define which crops are grown on the area you want to model. The sum of all crop-areas should be equal to the total area you want to model with this unpaved node.
- **Use different area for groundwater computations:** Sometimes, a part of the area you want to model might be paved, having it's own runoff system. Such areas do not contribute to the rainfall-runoff process for your unpaved area. However, the soil underneath the paved area will. For such cases this variable can be used. Example: an area of 10 000 m^2 has 2 000 m^2 of paved area, which has it's own runoff system. The rest is unpaved area where grass is grown. Only 8 000 m^2 of the area will therefore catch rainfall, actually contributing to the rainfall-runoff process. However, the rainfall on these 8 000 m^2 will apply to the groundwater of a 10 000 m^2 large area, because there is a soil underneath the paved area. In this case, we fill in: 8 000 m^2 for the Area per crop, type grass, 10 000 m^2 for the total area (for groundwater computations).
- **Surface level:** Here, the surface level for the area should be filled in. The surface level is important for several processes within the calculations:

1 When the unsaturated zone module "CAPSIM" is not active, the initial groundwater depth (= surface level - initial groundwater level) will be used to calculate the soil storage coefficient. **Note:** CAPSIM can be switched on in the SETTINGS task block for the RR module.
2 When the groundwater level branches the surface level, excess water will be stored on land or run off over land
3 When the water level in a connected node exceeds the unpaved area surface level, the area will be inundated.

The surface level can be entered as 'constant' or 'variable'. When using the 'variable' type, for various levels the percentage of land laying below it can be filled in, thus creating an S-curve. Important: when using the variable surface level type, the value you previously filled in for the constant value, will still be used for calculation of the soil storage coefficient!

The Soil tab

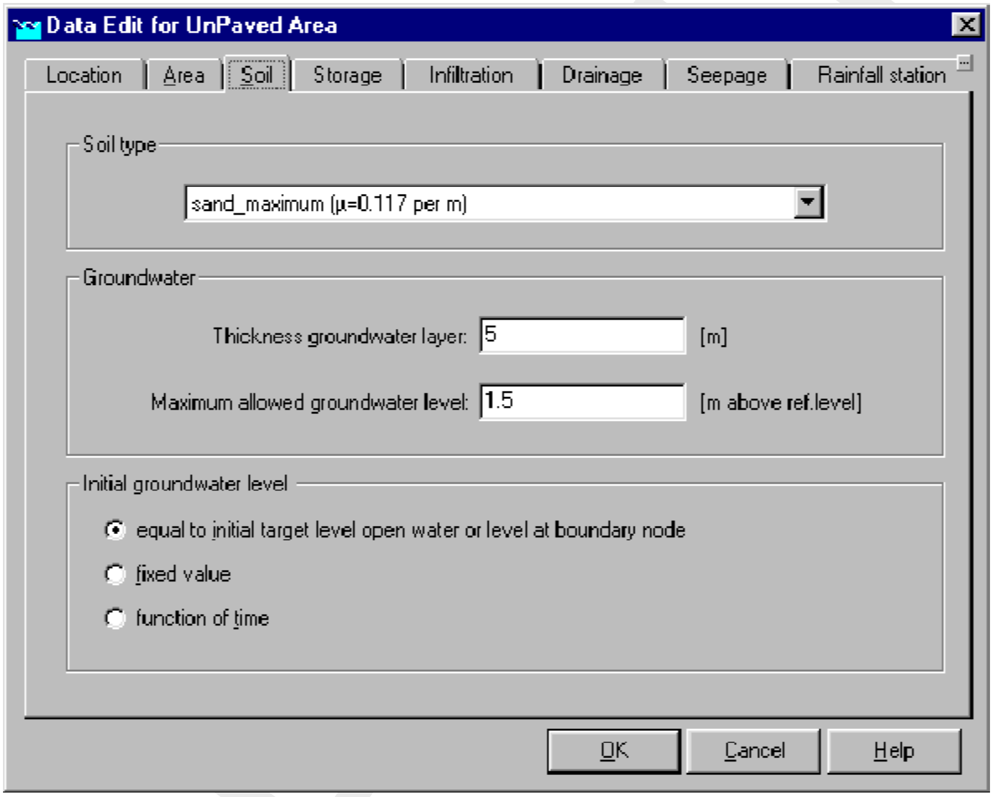


*Figure 5.188: **Data for UnPaved Area** window,* Soil *tab*

On the Soil tab, the following variables can be filled in:

◇ **Soil type:** Choosing a certain soil type, actually implies choosing the soil storage coefficients that will be used during the calculations. These coefficients determine how quickly the groundwater table will rise due to recharge.
Note: is important to know whether the unsaturated zone module CAPSIM is active or inactive. You can activate CAPSIM in the SETTINGS task block for the RR module.

- If CAPSIM is inactive:
At the start of each simulation, SOBEK will determine for all unpaved nodes:— the soil type — the initial groundwater depth (in cm below surface)SOBEK will then search for an appropriate storage coefficient in the file <\SOBEK\FIXED\3B\BERGCOEF>.

The coefficient it finds there, will then be applied to the corresponding unpaved node during the entire simulation. The deeper the initial groundwater level lies, the higher the storage coefficient will be.

- If CAPSIM is active:
  soil storage coefficient will be calculated on every time step. It will depend on:- soil type- actual groundwater level

- **Thickness groundwater layer:** This parameter is only of importance when salinity calculations are done. For calculation of salinity, the water volume of each object should be known. Therefore a groundwater depth should be given (soil volume = surface area * depth)
- **Maximum allowed groundwater level:** This parameter is not used in the calculation itself, but it may be very usable for output. In the post processing phase, one can determine whether, and for how long the maximum allowed groundwater level has been exceeded. This is suitable for the calculation of the damage of floodings to crops.
- **Initial groundwater level:** Unless you're using a restart file for the initialisation of your simulation, the initial groundwater level is the groundwater level which is applied during the first time step of the simulation. Furthermore, it is used to determine the soil storage coefficient (see the points above). Notice that the initial groundwater level is defined in [meters below surface level]. When choosing the initial groundwater level as a function of time, you can have a rainfall events in summertime starting with different initial groundwater levels than rainfall events in wintertime.

The Storage tab

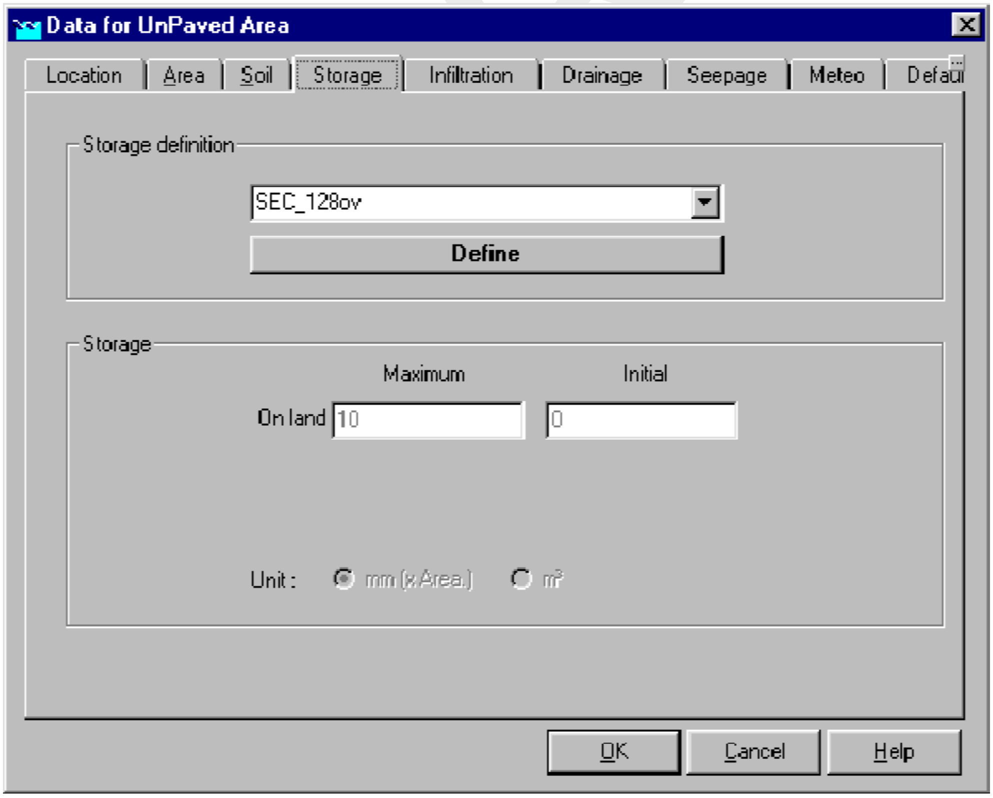


Figure 5.189: ***Data for UnPaved Area*** *window,* Storage *tab*

On the Storage tab, values for the initial and maximum storage on land can be filled in. These values can then be saved in a "Storage definition" so that you can re-use them on other nodes.

The storage on land determines how long it takes before surface runoff will occur. Surface runoff will start when the rainfall intensity is so high that the sum of the infiltration capacity and the maximum storage on land is exceeded. To define the storage parameters, type a name of the storage definition and click *Define*. Then fill in the values for maximum and initial storage on land. Finally click *Save*. On other unpaved area nodes you can select your previously defined values or create a new set.

The Infiltration Tab

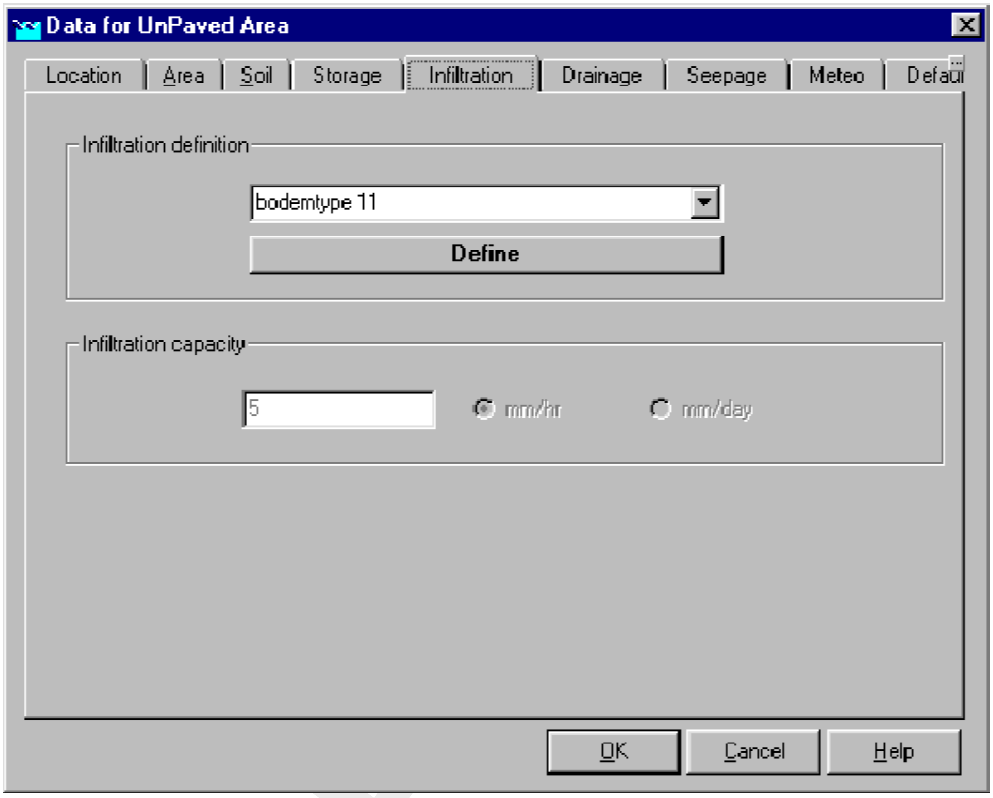


***Figure 5.190: Data for UnPaved Area** window,* Infiltration *tab*

On the Infiltration tab, one may enter values for the infiltration capacity of the soil. If the rainfall exceeds the infiltration capacity, water will be stored on land. To define the infiltration parameters, type a name for the infiltration definition and click <Define>. Then fill in the value infiltration capacity. Finally click <Save>. On other unpaved area nodes you can then select your previously defined values or create a new set.

The Drainage tab

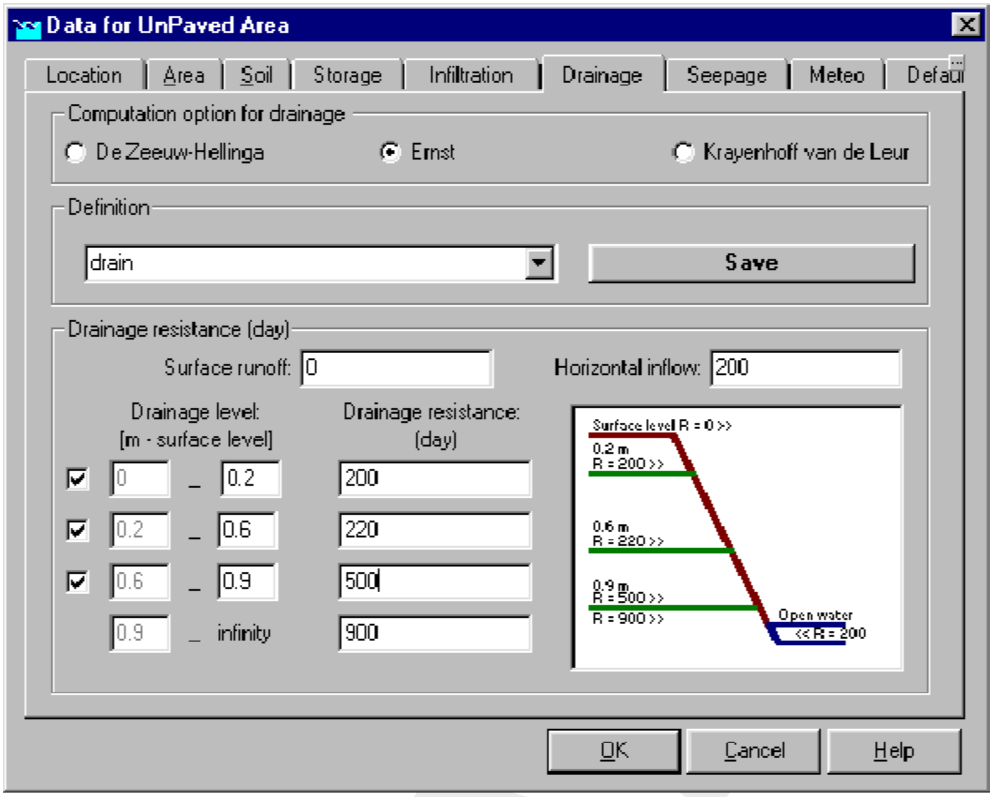


***Figure 5.191: Data for UnPaved Area** window,* Drainage *tab*

This tab is one of the most important ones. Here the drainage resistance values for different soil layers are filled in. The groundwater outflow is determined by:

1 groundwater level
2 drainage resistance values
3 soil storage coefficient
4 downstream water level

◇ **Computation option for drainage:** Three drainage formulas are available:

  1 De Zeeuw - Hellinga,
  2 Ernst and
  3 Krayenhoff van de Leur

  Note:
  If you are using CAPSIM for calculation of the unsaturated zone, you are advised to use the ERNST drainage formula.

◇ **Definition:** The values that you fill in for a certain node, may be re-used on other nodes. Therefore, they can be stored in a "definition". Define a name for the set of values you enter and save it. On the other nodes you can then refer to this previously defined definition.

◇ **The values to be entered:** The name of the section where the values should be entered varies with the chosen computation method.

  - surface runoff: the drainage resistance (Ernst) or reaction factor (Zeeuw-Hellinga) for the surface-runoff process (usually a very quick process, so low drainage resistance or high reaction factor)
  - horizontal inflow: the values for water flowing backwards from surface water into the soil.
  - drainage levels: for different soil layers, different values may apply.

All levels are defined as from x meters below surface level to y meters below surface level.

The Seepage tab

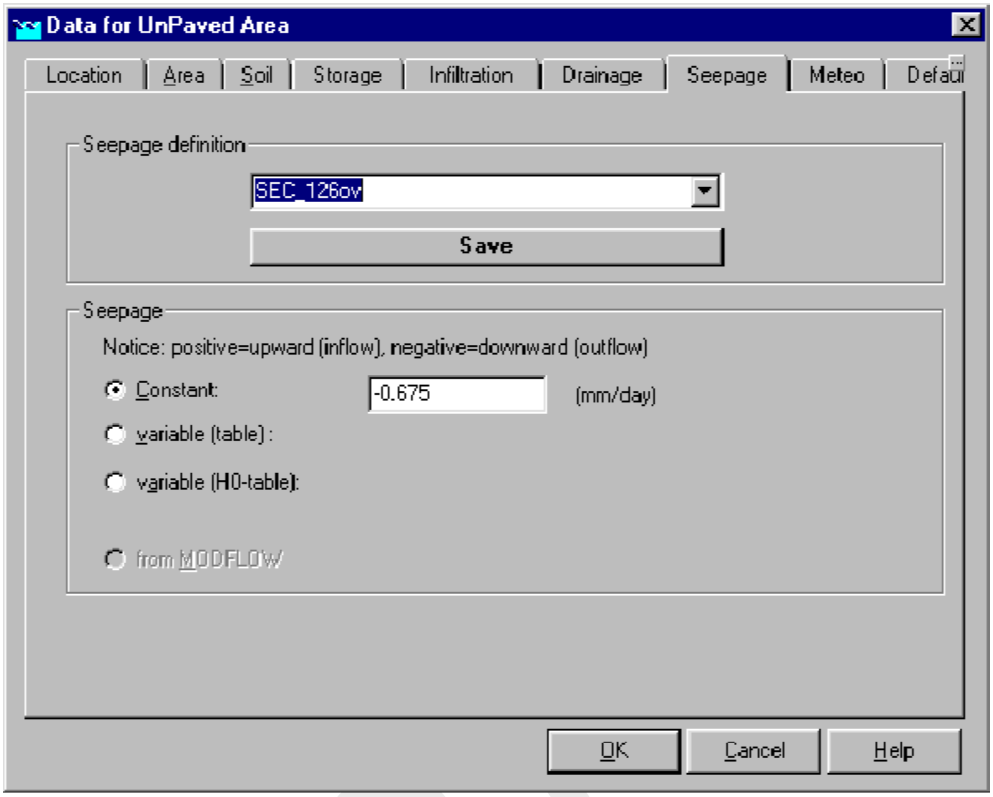


*Figure 5.192: **Data for UnPaved Area** window,* Seepage *tab*

Besides rainfall and evaporation, also seepage and infiltration are input/output terms for unpaved area nodes. On the Seepage tab, one can define the flux from or towards deep aquifers. Various options are available:

- **Constant:** A negative value means that the amount of water will be withdrawn from the unpaved area node; a positive values means that the amount of water is supplied to the node.
- **Variable (table):** The seepage and infiltration can be entered as a function of time
- **Variable (H0-table):** The seepage and infiltration is calculated as a function of: - groundwater table in the unconfined aquifer (as calculated by SOBEK) - groundwater head in the aquifer below (entered as a constant or function of time) - hydraulic resistance value of the aquitard between the unconfined and confined aquifer.
- **Future option: from Modflow:** In this future option, SOBEK RR and the groundwater software Modflow will be able to run simultaneously. Modflow will then supply SOBEK every time step with values for the groundwater head, so that the seepage/infiltration can be calculated according to the difference between phreatic and piezometric head.

The Meteo tab

On the Meteo tab, one selects the rainfall station that applies for the unpaved area node. Rainfall stations and - values can be created in the Meteorological Data Task block.

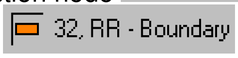
Unpaved area node topology

Nodes of the *unpaved area* type can be connected to four types of nodes:

- ◇ An RR Open Water node 

- ◇ An RR on channel connection 

- ◇ An RR on Flow connection node 

- ◇ An RR boundary node 

When connecting an unpaved area node to one of these nodes, make sure you have selected the *RR-link* connection type:

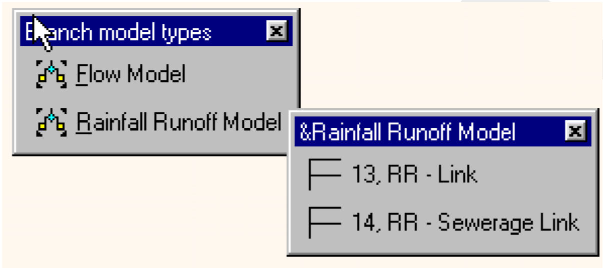


Figure 5.193: *RR-link connection type*

The images below show how the Unpaved area node can be connected to the three nodes mentioned.

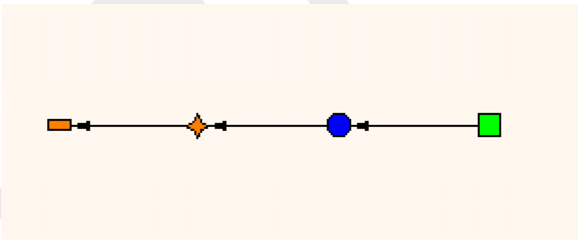


Figure 5.194: *Connecting an RR Unpaved node to an RR Open Water node. Notice the defined link directions.*

Figure 5.195: *Two ways to connect an RR Unpaved node to a channel (only available if you have a license for the flow module).*

5.4.13 D-NAM Input Screens

Hereunder the D-NAM rainfall-runoff model input screens are described. For more information on the various input parameters, reference is made to section 6.5.3.

- **The D-NAM Tab** (see Figure 5.196): On this tab, the catchment area can be defined. To be selected on this tab are a Soil definition, an Initial value definition, a Surface runoff definition, a Base flow definition and a Groundwater pump definition. These definitions can respectively be specified and deleted on the Soil&Soil2 tab, the Initial tab, the Runoff tab, the Base flow tab and the GW Pump tab.

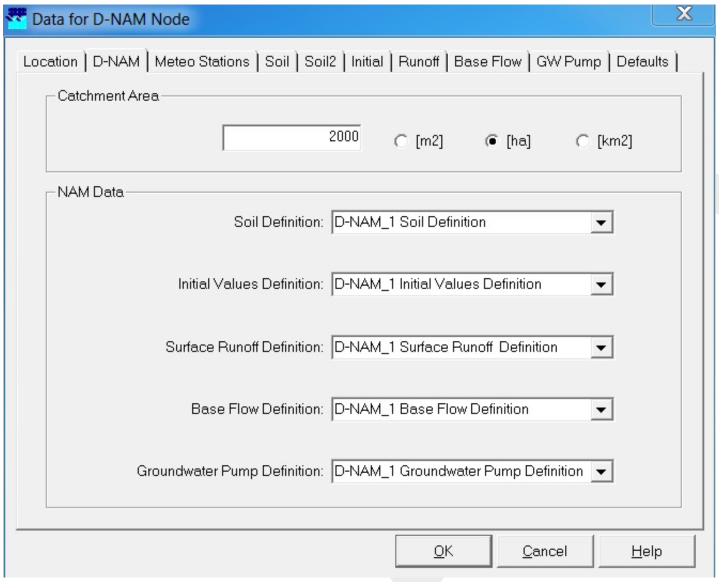


Figure 5.196: *The D-NAM Tab of the "Data for D-NAM Node" Form*

- **The Meteo Stations Tab** (see Figure 5.197): The meteo (rainfall) station of the D-NAM rainfall-runoff model is specified on this tab. In addition, an area adjustment factor is defined, being a multiplication factor applied on rainfall only. Please note that rainfall and evapotranspiration data of meteo stations are defined in the Meteo Task Block.

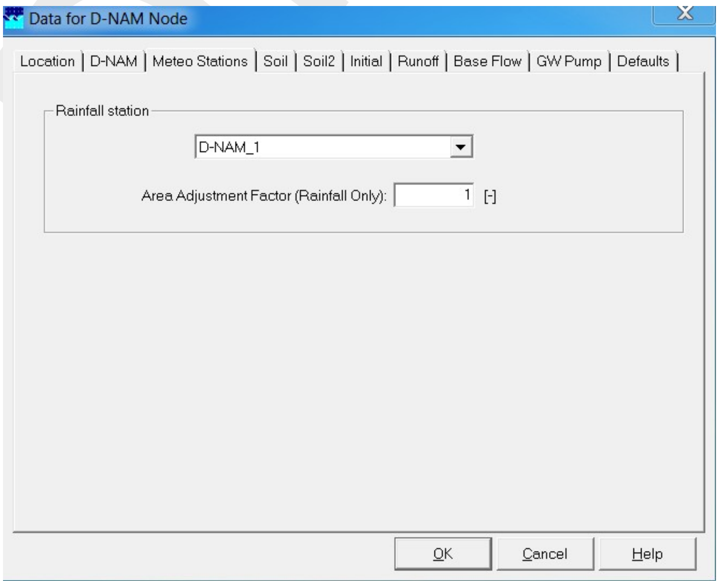


Figure 5.197: *The Meteo Stations Tab of the "Data for D-NAM Node" Form*

◇ **The Soil Tab** (see Figure 5.198): On this tab are specified the maximum soil infiltration rate of water contained in the surface storage [mm/hr], the field capacity of the root zone layer [-], the specific yield of the root zone layer [-] and the specific yield of the subsoil layer [-].
For information only are shown the maximum water depth in the lower zone storage [mm], the maximum water depth in the root zone part of the groundwater storage [mm], the maximum water depth in the subsoil part of the groundwater storage [mm] and the maximum water depth in the groundwater storage [mm]. These maximum water depths are computed using the field capacity and specific yields defined on this tab as well as the values defined on the intial tab for the surface level, the bed level of the root zone layer and the bed level of the subsoil layer.

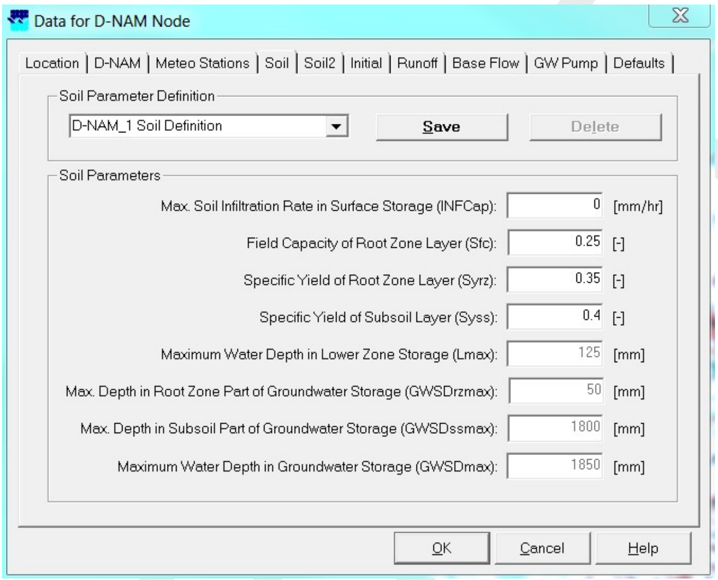


__Figure 5.198:__ The Soil Tab of the "Data for D-NAM Node" Form

◇ **The Soil2 Tab** (see Figure 5.199): On this tab are defined the potential maximum capillary rise [mm/day] and the potential maximum percolation capacity [mm/day] either as a constant or as function of the groundwater table depth.

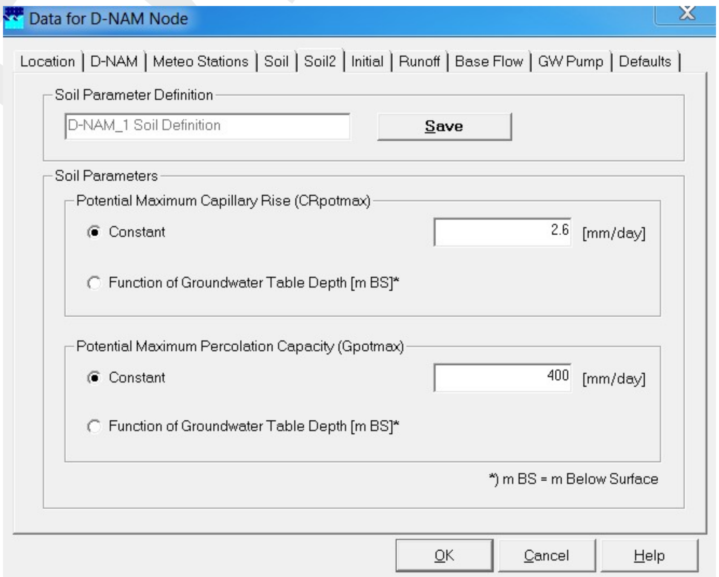


__Figure 5.199:__ The Soil2 Tab of the "Data for D-NAM Node" Form

◇ **The Initial Tab** (see Figure 5.200): On this tab are defined the surface level of the catchment area [m above datum], the bed level of the root zone layer [m above datum], the bed level of the subsoil layer [m above datum], the initial water depth in the surface storage [mm], the initial water depth in the lower zone storage [mm] and the initial water depth in the groundwater storage [mm].
Please note that the resulting initial water depth in the root zone part of the groundwater storage ($GWSD_{i,rz}$) is computed and shown for information only. If $GWSD_{i,rz} > 0$ while the initial water depth in the lower zone storage (L_i) is less than its maximum water depth (L_{max}), means that there is water contained in the root zone part of the groundwater storage that can be retained against gravity. In such case a form pops up (see Figure 5.200) informing on the amount of water depth in the root zone part of the groundwater storage that should be transferred to the lower zone storage. By clicking on the <OK> button, the initial water depth in the lower zone storage (L_i) and the initial water depth in the groundwater storage ($GWSD_i$) are adapted in such way that the root zone part of the groundwater storage does not contain any water that can be retained against gravity.

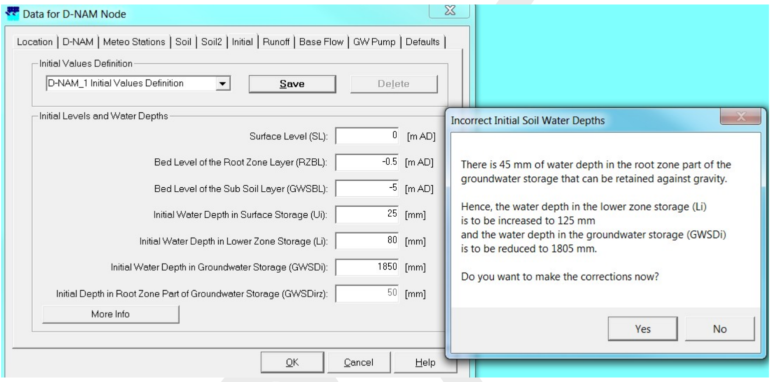


Figure 5.200: *The Initial Tab of the "Data for D-NAM Node" Form*

◇ **The Runoff Tab** (see Figure 5.201): On this tab are defined the catchment length in drainage direction [m], the surface slope [-], the Manning roughness of the catchment land-use [$s/m^{1/3}$], the surface-storage-threshold for overland flow [mm], the interflow reservoir coefficient [days], the surface-storage-threshold for interflow [mm] and the lower-zone-storage threshold for interflow [mm].

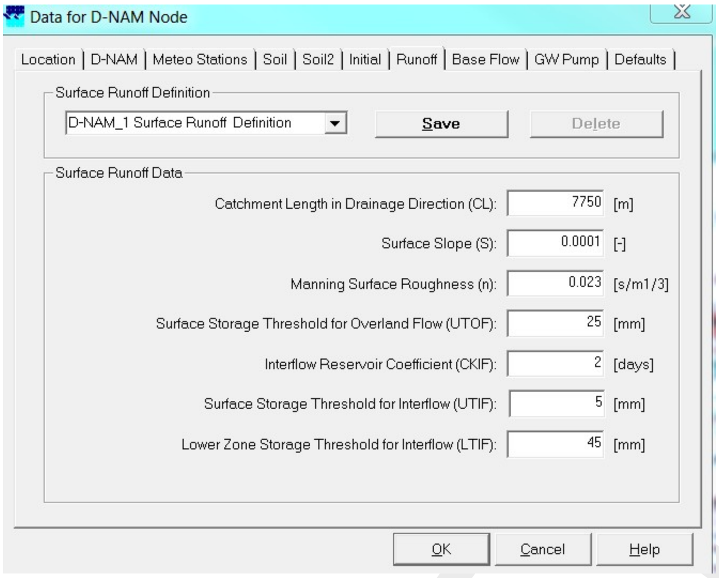


Figure 5.201: *The Runoff Tab of the "Data for D-NAM Node" Form*

◇ **The Base Flow Tab** (see Figure 5.202): On this are defined the fast base flow reservoir coefficient [days], the slow base flow reservoir coeffient [days], the external (ground)water inflow reservoir coeffient [days], the lower-zone-storage threshold for percolation [mm], the groundwater-storage threshold for fast base flow [m above datum] and the groundwater-storage threshold for slow base flow [m above datum].

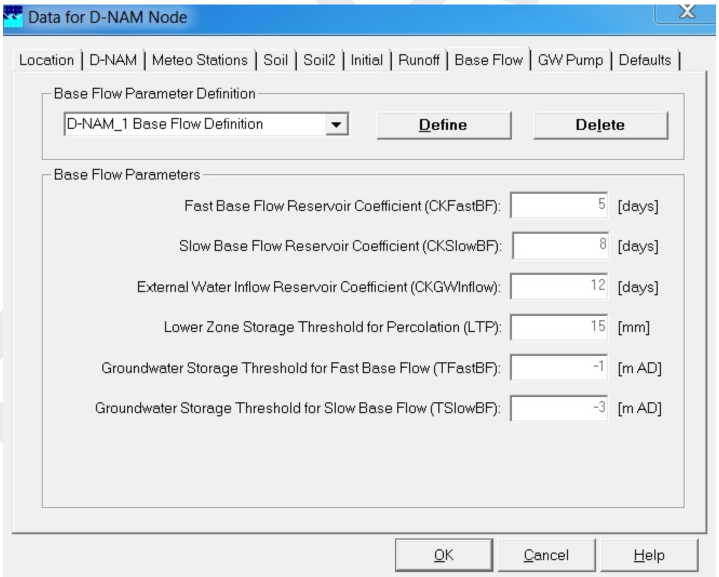


Figure 5.202: *The Base Flow Tab of the "Data for D-NAM Node" Form*

◇ **The GW Pump Tab** (see Figure 5.203): On this tab the discharge of the groundwater pump $[m^3/s]$ is defined either as a constant or as function of time. Please note that a positive groundwater pump discharge means that water is to be abstracted, while a negative groundwater pump discharge means that water is to be supplied.

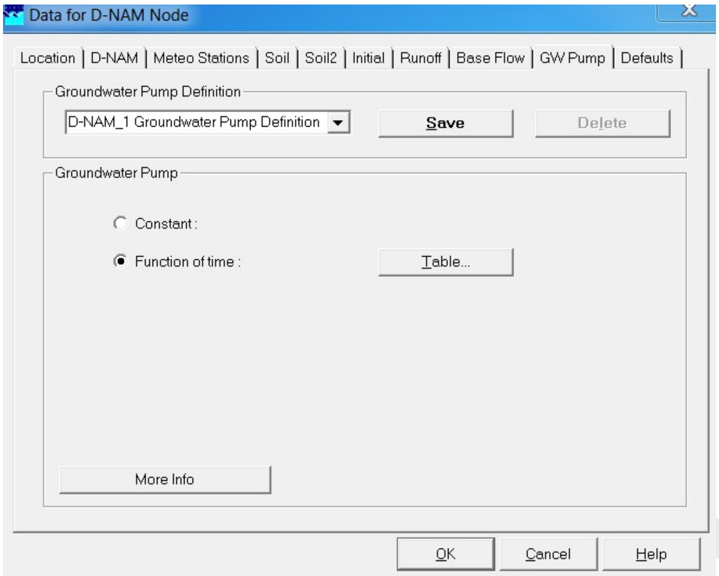


Figure 5.203: *The GW Pump Tab of the "Data for D-NAM Node" Form*

5.4.14 RR - Wastewater Treatment Plant

RR - WWTP node description

A WWTP node can be used when there is measured data available for the discharges from a waste water treatment plant towards a boundary or an open water node that is connected on the downstream side.

◇ For a detailed description of this node's input parameters: see the "RR - WWTP node input screens" section from the Reference Manual;
◇ For a detailed description of this node's possible network configurations: see the "RR - WWTP node topology" section from the Reference Manual

RR - WWTP node input screens

When the model data editor for an *RR - Orifice* node type is started, the following tabs will become available for input:

WWTP Tab:

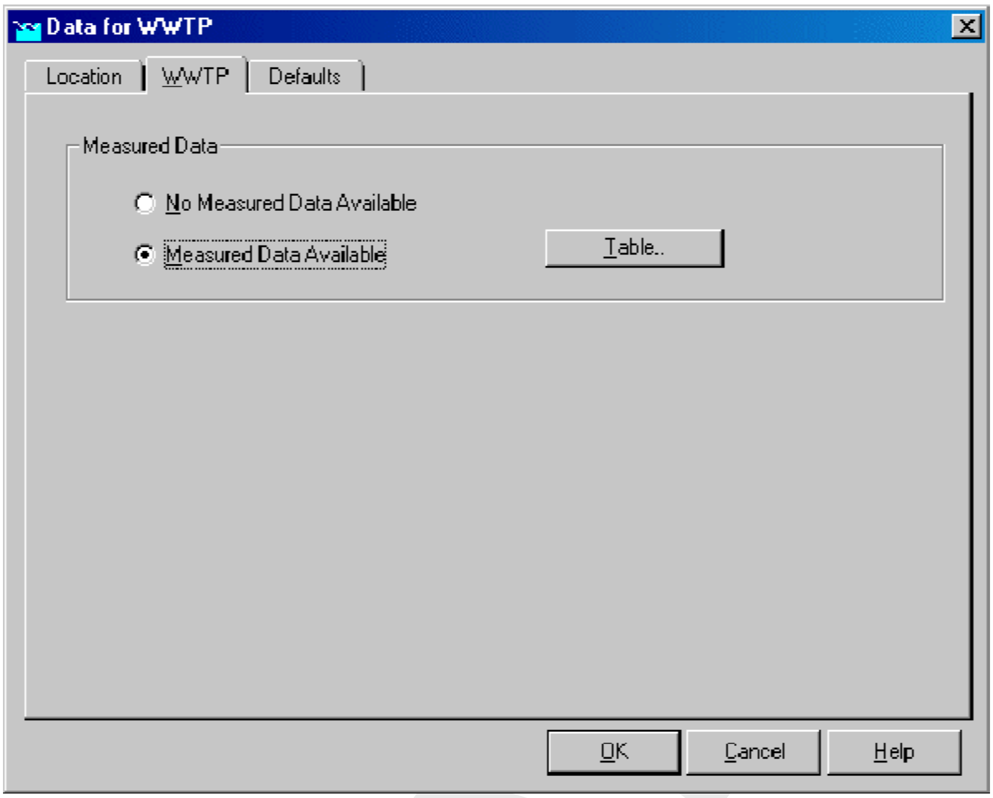


***Figure 5.204: Data for WWTP** window,* WWTP *tab*

Here, the discharges from the Waste Water Treatment Plant to a boundary or Open Water node can be entered. There are two options:

- ◇ **No measured data available:** With this option, the discharges from the WWTP node towards a downstream side connected *Boundary* or *Open Water* node will be derived directly from the combined actual flows of all RR - Paved area nodes that are connected on the upstream side.
- ◇ **Measured data available:** If this option is active, discharges from the WWTP node towards a Boundary node or Open Water node can be entered as a function of time.
  Note: : the specified WWTP discharges **only** apply for the flow *from* the WWTP node towards a downstream Boundary or Open Water node. Not for the flow from the RR paved nodes towards the WWTP. For that flow, the summed up sewer pump discharges apply.
  The difference between the flow towards and from the WWTP will become visible as a storage change in the results for the WWTP.

RR - WWTP node topology

A WWTP node always forms an interconnection between two nodes:

On the upstream side:

- ◇ 28, RR - Paved (multiple nodes of this type are allowed)

On the downstream side:

- ◇ 31, RR - Open Water
- ◇ 32, RR - Boundary
- ◇ 26, Flow-RR Connection on Channel (available in the Flow modules only)
- ◇ 27, Flow-RR Connection on Flow Connection Node (available in the Flow modules only)

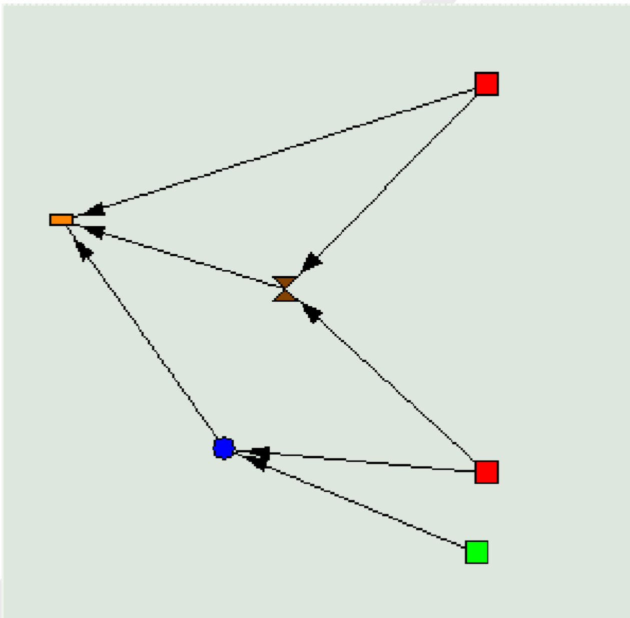


Figure 5.205: *Typical example of a schematisation that contains a WWTP node*

In this example, the upper paved node will pump its sewer discharge towards the WWTP node. The sewer overflows from that paved node will flow directly towards a boundary node.

The lower paved node also pumps its sewer discharge towards the WWTP, but sewer overflows will take place on an RR Open Water node.

On the WWTP node, the modeller now has the option of applying measured data for the flow towards the boundary, or to apply the sewer discharges from the connected paved nodes directly to the WWTP.

5.4.15 RR - Weir

Description of the RR - Weir node type

In this chapter, the RR - Weir node is described. With an RR - Weir node you can simulate a structure over which water flows. Nodes of this type therefore can represent three types of weirs: broad crested weirs (rectangular), v-notch weirs or 2-stage weirs.

- ◇ For a detailed description of this node's input parameters: see the "RR - Weir node input screens" section from the Reference Manual;
- ◇ For a detailed description of this node's possible network configurations: see the "RR - Weir node topology" section from the Reference Manual
- ◇ For a detailed description of this node's underlying mathematical equations: see the "Weir section from the Technical Reference Manual".

Definition of a Weir (barrage):

A weir is a fixed or movable structure that aims to control a certain water level on its upstream side (Dutch Hydrological Society, 2002).

Figure 5.206: *Example of a broad crested weir. Photo: Dutch waterboard "Peel and Maasvallei"*

RR - Weir node input screens

When the model data editor for an *RR - Weir* node type is started, the following tabs will become available for input:

Options Tab:

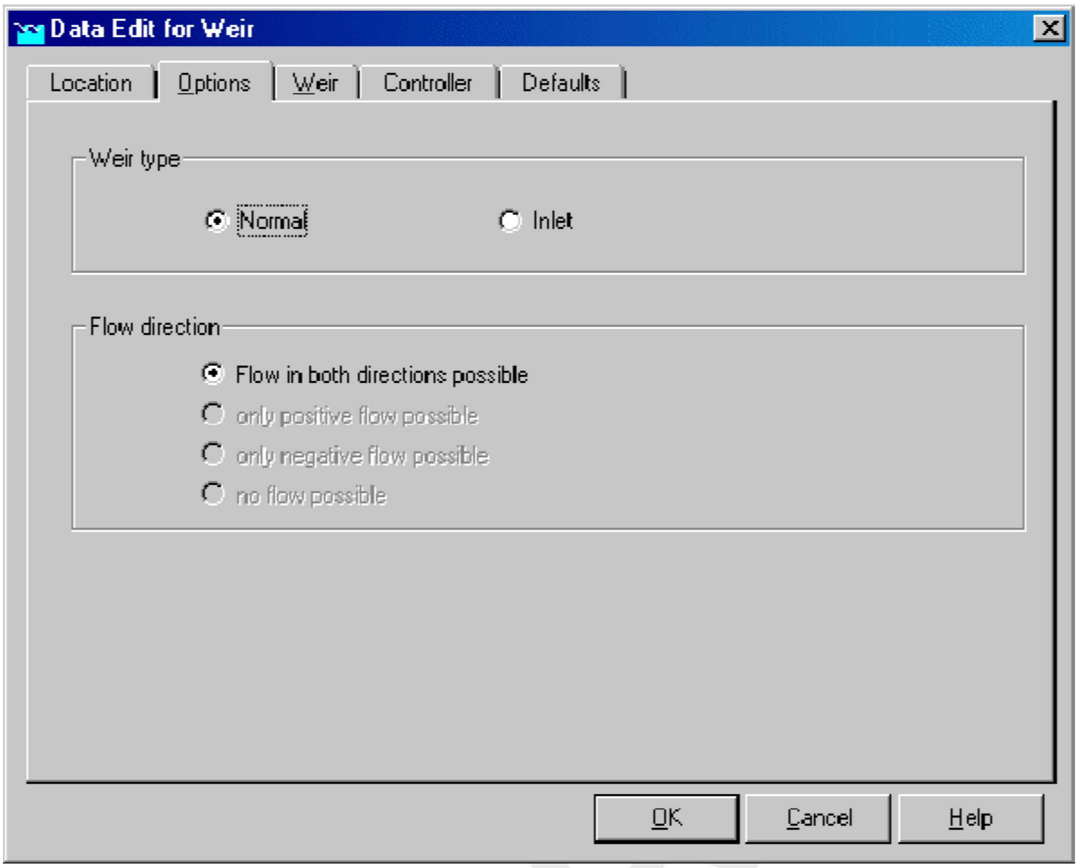


Figure 5.207: ***Data for Weir*** *window,* Options *tab*

On this tab, you can define how the weir is to be used:

- **Weir type: normal** This option means that water is allowed to flow over the weir in both directions.
- **Weir type: inlet** This option means that water may only flow over the weir in positive direction. The arrows on the links that connect the weir to other objects indicate the positive flow direction.

Weir tab (normal type):

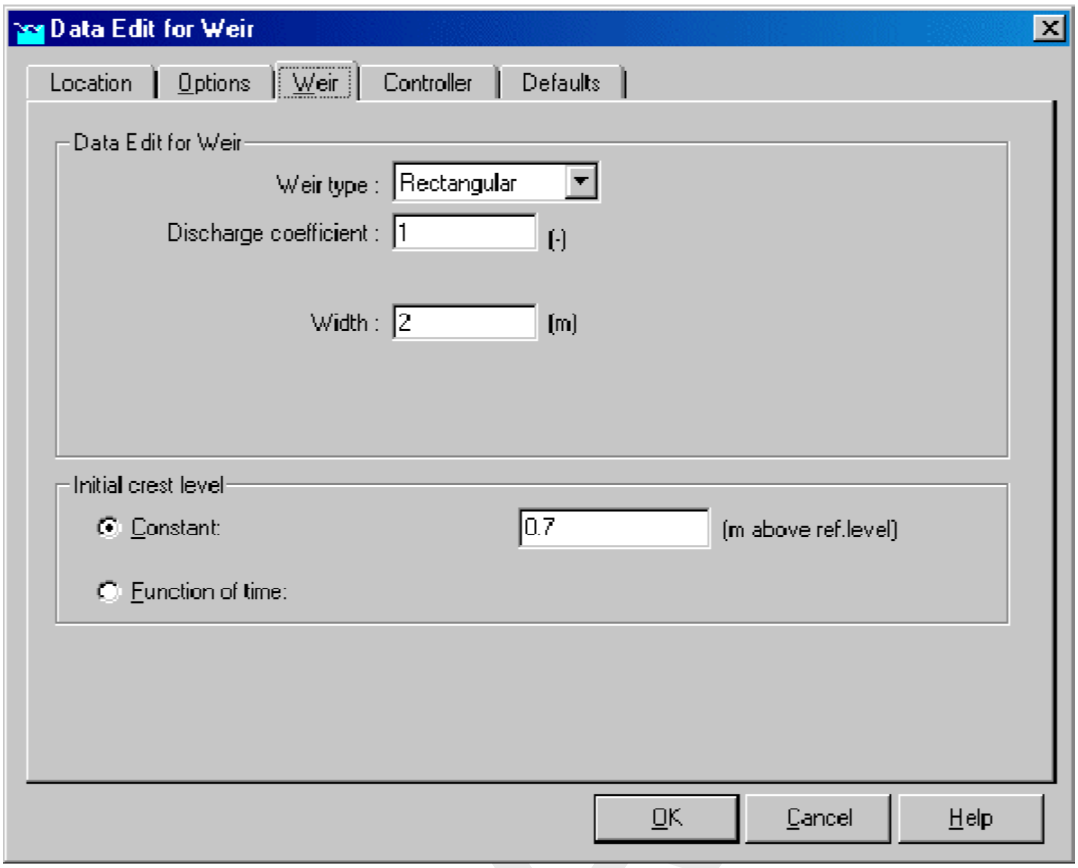


Figure 5.208: ***Data for a normal RR-Weir*** *window,* Weir *tab*

On the "Weir" tab, the shape and crest level of the weir is defined. In the "Weir type" combo box can choose between three types of weirs:

Rectangular weir (broad crest):

Figure 5.209: *Example of a broad crested weir. Photo: South African Department of Water Affairs*

This weir type requires the following input parameters:

◇ **discharge coefficient:** See the "Weir" section of the Technical Reference Manual for the exact implementation of this parameter in the discharge formula of this structure type.
◇ **Width:** Represents the width of the weir's crest. See the "Weir" section of the Technical Reference Manual for the exact implementation of this parameter in the discharge formula.
◇ **Initial Crest level:** This parameter represents the weir's crest level. Notice that it is called "**initial**" crest level. This name has been chosen because the crest level may be overruled in certain cases:

1 by a controller. If there's no controller active for the weir, the crest level will remain equal to the "initial crest level" throughout the entire simulation.
2 if the option "adjust weir setting automatically to $<=$ upstream target level" in the "Settings" task block has been activated.

V-notch weir:

Figure 5.210: *Example of a V-notch weir. Photo: South African Department of Water Affairs*

If the V-notch type is selected for the weir, there will be some other parameters required than in case of a rectangular one:

◇ **Discharge coefficient:** See the "Weir" section of the Technical Reference Manual for the exact implementation of this parameter in the discharge formula of this structure type.
◇ **Slope:** Represents the tangent of the weir's side slopes. Thus a value of 2 results in a steep V-shape (1 horizontally -$>$ 2 vertically)
◇ **Initial Crest level:** This parameter represents the weir's crest level. Notice that it is called "**initial**" crest level. This name has been chosen because the crest level may be overruled in case the option "adjust weir setting automatically to $<=$ upstream target level" has been activated in the "Settings" task block .

2-stage weir:

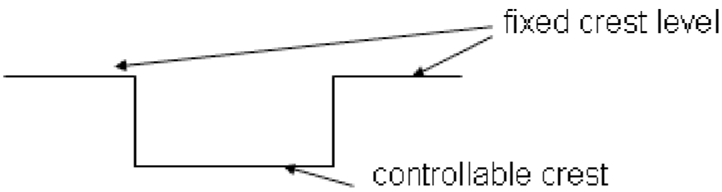
Figure 5.211: *Example of a multiple-stage weir. Photo: South African Department of Water Affairs*

Weirs of this type are a variety of the broad crested type. In stead of one crest level they have two: at the lowest point there is a small crest which is meant to maintain sufficient flow velocities over the weir in periods of low discharge. A little higher, the crest becomes broad. This is meant to allow enough water to flow over the weir in cases of high discharges.

The following parameters are required for this weir type:

- ◇ **Discharge coefficient:** See the "Weir" section of the Technical Reference Manual for the exact implementation of this parameter in the discharge formula of this structure type.
- ◇ **Width:** Represents the width of the crest at its narrowest point.
- ◇ **Additional width 2nd stage:** Represents the amount by which the weir broadens at the crest level of the 2nd stage.
- ◇ **Crest level 2nd stage:** Represents the crest level for the second stage; thus the crest level for the broad part of the weir. Naturally this value should be higher than the crest level for the 1st (narrow) stage.
- ◇ **Initial Crest level:** This parameter represents the weir's crest level. Notice that it is called "**initial**" crest level. This name has been chosen because the crest level may be overruled in two cases:

  1 by a controller. If there's no controller active for the weir, the crest level will remain equal to the "initial crest level" throughout the entire simulation. Note that only the crest for the **first** stage can be controlled:



2 if the option "adjust weir setting automatically to <= upstream target level" in the "Settings" task block has been activated.

Weir tab (Inlet type):

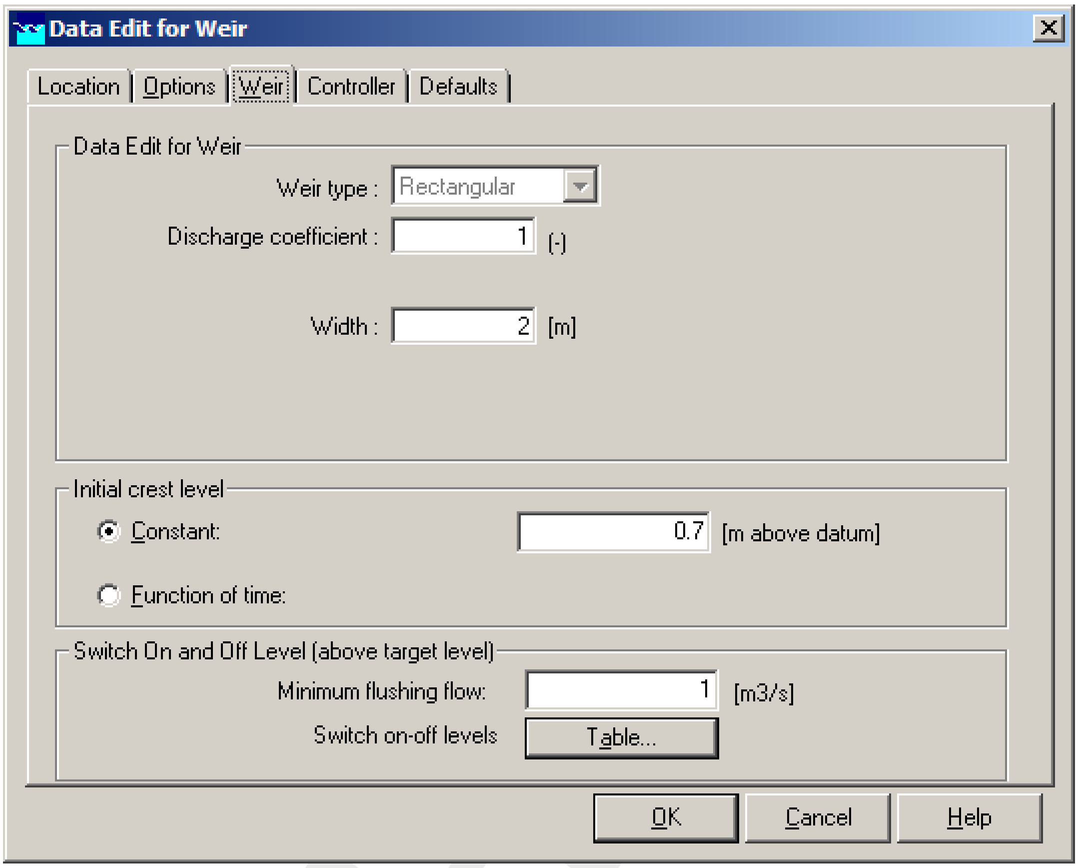


***Figure 5.212: Data Edit for Weir** window,* Weir *tab*

If the option Inlet is selected in the "Options" tab, the "Weir" tab will allow the user to set two additional parameters:

◇ **Minimum flushing flow:** This is the minimum amount of flow when the inlet is active.
◇ **Switch on-off levels:** The inlet will open according to the switch on-off levels (relative to the downstream target level).

Controller tab:

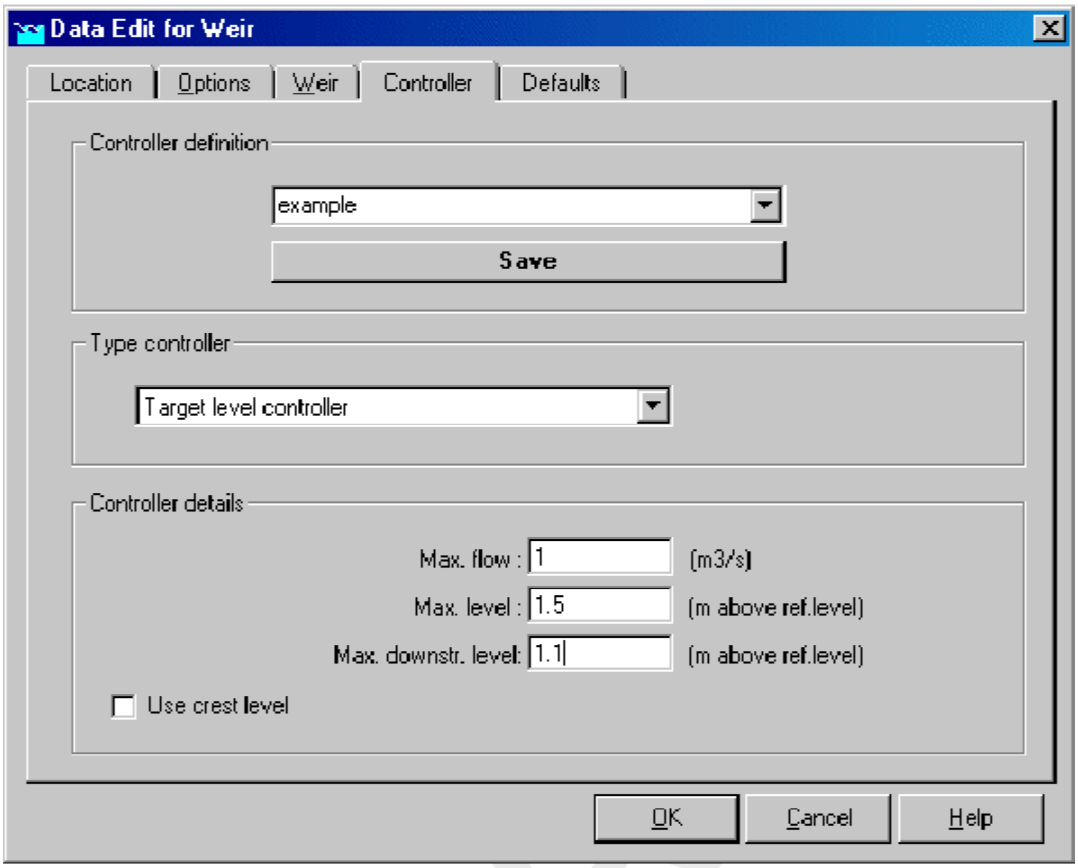


Figure 5.213: Data Edit for RR - Weir *window,* controller *tab*

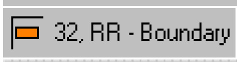
On the controller tab, the behaviour of the weir can be controlled (optional). With a controller, the initial crest levels, as defined on the "Weir" tab, are overruled. For more details on the available controllers and their function, see the Controller section of the RR Technical Reference Manual.

RR - Weir node topology

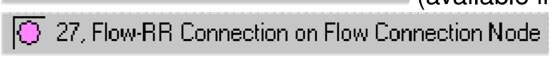
An RR - Weir should always be connected to at least one RR - Open Water node, as it is meant to control the water level on an open water node.

RR - Weirs can be connected to nodes of the following types:

- 

- 

- 

(available in the Flow modules only)
- 

(available in the Flow modules only)

An example of a network where RR Weirs are used is given below:

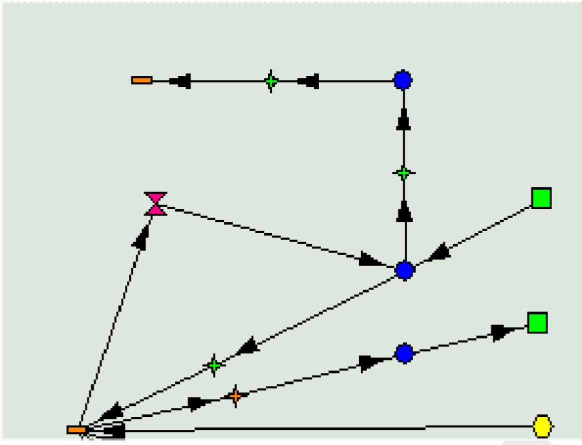


Figure 5.214: *Example of a network with RR Weirs*

5.5 Branch description

5.5.1 Branch - Channel

Description

In this chapter, the *Flow - Channel* branch is described. This type of object forms the basis for any River or Channel. With a *branch* you can define the course of the river or channel that you want to schematise. A branch is always interconnecting two nodes; these may be boundary nodes, or connection nodes. On top of the branch, you may later add other objects, such as weirs, pumping stations etc.

◇ This branch has no input parameters for itself.
◇ For a detailed description of this branch's possible network configurations: see the "Flow - Channel branch topology" section from the Reference Manual
◇ This branch type has no underlying mathematical equations for itself. Its length, the boundary nodes and other objects attached to it determine the flow through branches of this type.

In SOBEK-Rural Water Flow channels are represented by branches of the type 'Flow - Channel'. In contrast to the branches in SOBEK-Rural Rainfall-Runoff, the so-called RR-Links, the lengths of these branches have a physical meaning. Furthermore, they can have objects attached to the, such as cross sections, ζ-calculation points and structures. The cross section is defined per branch by a cross section node. Various branches can be connected by connection nodes which can comprise extra storage capacity and lateral inflow. The connection of the modelled area with the outside world (which can also be a sobek-Rural Rainfall-Runoff schematisation) is represented by the boundary condition nodes.

Topology
The direction of a branch is defined when you create it:

◇ If you create a branch by starting at node A and drawing the mouse to node B, the branch direction is from A to B.
◇ If you create a branch by starting at node B and drawing the mouse to node A, the branch direction is from B to A.

If you don't know the branch direction, do the following:

- ◇ Click the Menu "Options" - "Network Data" - "Links"- Select the option "Defined" under the "show direction" section.
- ◇ Click *OK*

You should now see an arrow that indicates the branch direction.

If you still don't see it, zoom in on the branch.

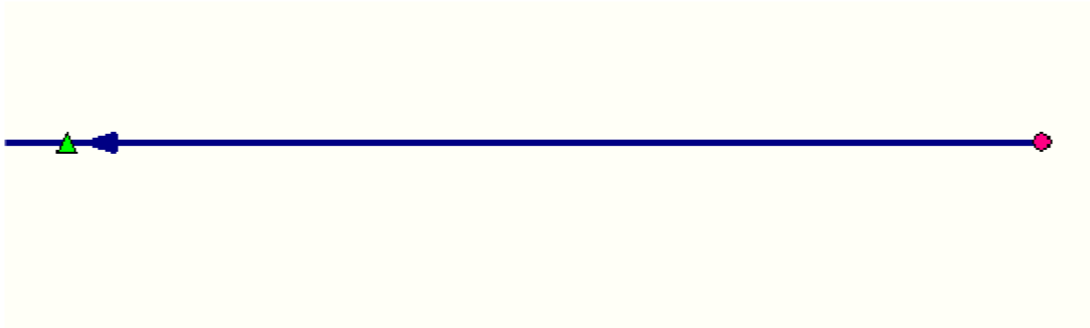


Figure 5.215: *Visualising the defined direction of a branch.*

The branch direction can be reversed with the "reverse branch direction" button 

5.5.2 Branch - Flow 1D Dam Break branch and the Once Hydraulic Trigger

This section describes the "Flow 1D Dam Break branch", including the "Once Hydraulic Trigger":

- ◇ Application examples of the "Flow 1D Dam Break branch" are given in section 5.5.2.1.
- ◇ The method of modelling bbranching and the available bbranch growth options/formulae are discussed in section 5.5.2.2.
- ◇ Options for specifying the point-in-time that bbranching should start (e.g. at an absolute point-in-time or at the point-in-time that a specific hydraulic event occurs) are discussed in section 5.5.2.3.
- ◇ The "Flow 1D Dam Break branch" input screens are discussed in section 5.5.2.4.
- ◇ The output parameters available at a "Flow 1D Dam Break branch" are explained in section 5.5.2.5.

For detailed information on the Verheij-vdKnaap(2002) bbranch growth formula and the vdKnaap(2000) bbranch growth formula both available at "Flow 1D Dam Break branch", reference is respectively made to section 6.3.3.1 and section 6.3.3.2.

5.5.2.1 Application examples of the Flow 1D Dam Break Branch

The "Flow 1D Dam Break Branch" can be used to model the hydraulic consequences of a dam break or dike failure. Possible applications are:

- ◇ **The bbranching of a dike in a 2D model** (see Figure 5.216). The dike is represented by elevated 2D grid cells. The "Flow 1D Dam Break Branch" is located between two

Connection Nodes, that are respectively located at the west-side and east-side of the dike. Due to bbranching, water flows though the "Flow 1D Dam Break Branch" from the west-side towards the east-side of the dike. Please note that if the water at the east-side is higher than the elevation of the 2D dike cells, in addition to 1D flow through the "Flow 1D Dam Break Branch" also 2D water will flow over the 2D dike grid cells from west to east.

- **The bbranching of a river levee** (see Figure 5.217). The river is modelled as 1D flow. The left and right river levees are represented by elevated 2D grid cells. The "Flow 1D Dam Break Branch" connects the 1D river with a connection node located on the 2D grid part, representing the left flood plain of the river. Due to bbranching, river water will flow through "Flow 1D Dam Break Branch" into the left flood plain.
- **Bbranching of a 1D river dam** (see Figure 5.218). A "Flow 1D Dam Break Branch" can be incorporated into a 1D river model to represent a river dam. Due to bbranching, reservoir water stored upstream of the river dam will flow through the "Flow 1D Dam Break Branch" towards the downstream located river sections.

Note: In the application examples give above, the point-in-time that bbranching should start can be specified either as an absolute point-in-time or as the point-in-time that a specific hydraulic event occurs (for instance: the exceedance of water level upstream of the "Flow 1D Dam Break Branch"). For more information, see section 5.5.2.3).

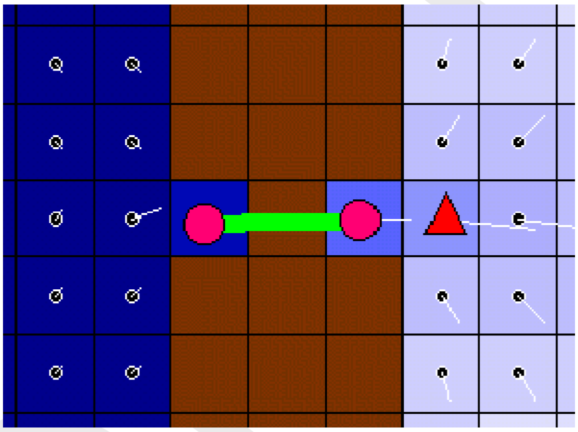


Figure 5.216: *Bbranching of a dike in a 2D model using a "Flow 1D Dam Break Branch"*

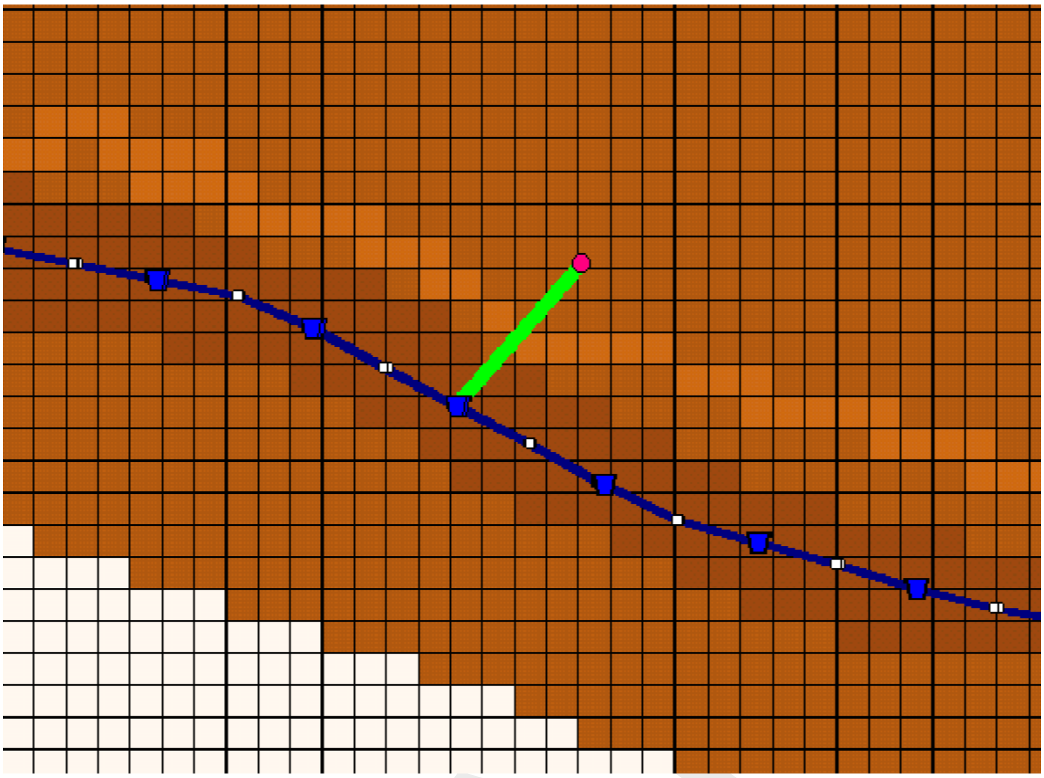


Figure 5.217: Bbranching of a river levee using a "Flow 1D Dam Break Branch"

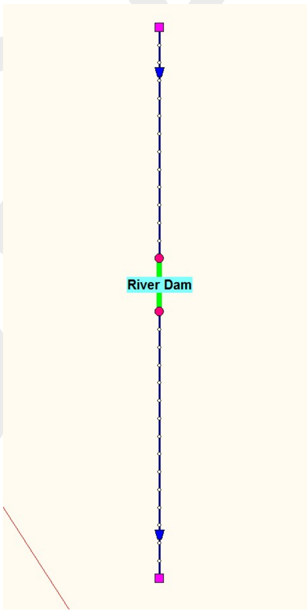


Figure 5.218: Bbranching of a 1D river dam using a "Flow 1D Dam Break Branch"

5.5.2.2 Method of modelling bbranching and Bbranch growth options/formulae

In this section three topics are discussed:

- Method of modelling bbranching.
- Bbranch growth options/formulae.
- Controller table, defining the development of the bbranch area in time.

Method of modelling bbranching

Bbranching (e.g. development of a bbranch, resulting in a dam break or dike failure) is modelled by lowering the crest level and increasing the crest width of a weir, accommodated in the "Flow 1D Dam Break Branch". The discharge over this weir is computed in accordance with the formulae given in section 6.1.16.16, where for the "Flow 1D Dam Break Branch" yields that the dimensionless lateral contraction coefficient (c_w) is set equal to 1.

Two phases are discerned in the development of a bbranch (see Figure 5.219):

- **Phase 1**: At the on-start of the bbranching process, the initial crest level of the weir is set equal to the user-defined initial crest level (Z_0, supposed to coincide with the crown height of the dike) and the initial crest width of the weir is set equal to the user-defined initial bbranch width (B_0). In phase 1 for a constant crest width equal to B_0, the crest level of the weir is lowered from Z_0 to the user-defined lowest crest level (Z_{min}). Please note that for vdKnaap(2000) yields that Z_{min} is equal to Z_0 minus the user-defined maximum bbranching depth.
- **Phase 2**: After phase 1 is completed, the bbranch only grows in width (e.g. crest width of the weir increases). Hence, in phase 2 the crest level of the weir remains equal to Z_{min}.

Note: If the "Flow 1D Dam Break Branch" is connected by means of a Connection node to a 2D grid cell, the elevation of this 2D grid cell will be set equal to the user-defined lowest crest level (Z_{min}).

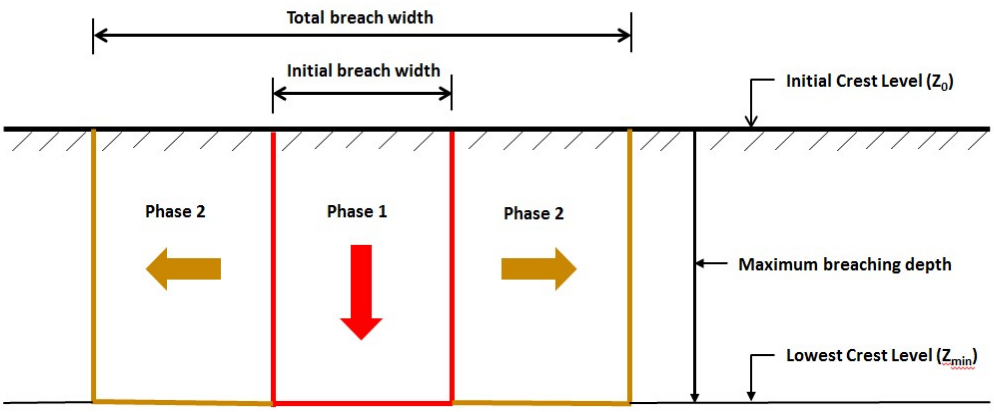


Figure 5.219: *Development of a bbranch in a "Flow 1D Dam Break Branch"*

Bbranch growth options/formulae

Three bbranch growth options/formula are available at a "Flow 1D Dam Break Branch", viz:

- The **Verheij-vdKnaap(2002)** formula: The bbranch width growth in phase 2 (see Figure 5.219) depends on the actual flow velocity in the bbranch (e.g. the average flow velocity over the weir, accommodated in the "Flow 1D Dam Break Branch"). For more information on the Verheij-vdKnaap(2002) formula, reference is made to section 6.3.3.1.
- The **vdKnaap(2000)** formula. The bbranch width growth in phase 2 (see Figure 5.219) is independent of the actual flow velocity in the bbranch. For more information on the vdKnaap(2000) formula, reference is made to section 6.3.3.2).
- **User defined bbranch development**: You can define your own bbranch development independent of the actual flow velocity in the bbranch by manually editting the controller table, that is generated by applying the vdKnaap(2000) formula (see section 5.5.2.4). Hereunder, in paragraph *"Controller table, defining the development of the bbranch area*

in time" it is explained how the controller table defines the development of the bbranch.

Note: For all bbranch growth options/formula yields that the final bbranch width may become larger than the 2D grid cell size.

Controller table, defining the development of the bbranch area in time

During the bbranching process, SOBEK obtains the crest level and crest width of the weir, accomodated in the "Flow 1D Dam Break Branch" from a controller table (see Controller Tab description in section 5.5.2.4), that specifies the bbranch area [m^2] as function of (relative) time.

◇ **Verheij-vdKnaap(2002)**: The controller table describes only phase 1 of the bbranch development and comprise of two rows only. First row [start-time of phase 1 (T_0); 0]) and second row [end-time of phase 1 (T_1); Bbranch area at the end of phase 1 (A_1)], where A_1 = Initial bbranch width(B_0) * (Initial Crest level(Z_0) - Lowest Crest Level(Z_{min})). During phase 1 the actual crest level is determined by linear interpolation in the controller table (e.g. crest level equals Z_0 minus bbranch area divided by B_0). *Please note* that the bbranch width growth (e.g. crest widths) in phase 2 follows from the Verheij-vdKnaap(2002) formula (see section 6.3.3.1) taking into account the actual hydraulic conditions in the bbranch.
◇ **vdKnaap(2000)** and **User defined bbranch development**: The controller table describes both phase 1 and phase 2 of the bbranch development and may comprise of several rows. The first row [start-time of phase 1 (T_0); 0] and the last row [end-time of phase 2 (T_2); Bbranch area at the end of phase 2 (A_2)]. For intermediate rows yields [0<Time<T_2; 0<Bbranch area<A_2]. For the bbranch area at the end of phase 1 (A_1) yields A_1 = Initial bbranch width(B_0) * (Initial Crest level(Z_0) - Maximum bbranching depth). The end-time of phase 1 (T_1) follows from the controller table (e.g. T_1 is point-in-time that the bbranch area equals A_1). In the same way as explained for Verheij-vdKnaap(2002), the crest levels in phase 1 are obtained by linear interpolation in the controller table. Crest levels remain constant in phase 2 (see Figure 5.219). For the crest width (e.g. bbranch width) in phase 2 yields: crest width = bbranch area / (Z_0 - maximum bbranching depth). Crest widths in phase 2 are also obtained by linear interpolation in the controller table.

5.5.2.3 Specifying the point-in-time that bbranching should start in a Flow 1D Dam Break Branch

There are two options for specifying the point-in-time that bbranching should start in a "Flow 1D Dam Break Branch":

◇ Option 1: Start bbranching at an absolute point-in-time, and
◇ Option 2: Start bbranching if a specific hydraulic event occurs.

Option 1: Start bbranching at an absolute point-in-time

In this case the user specifies the absolute point-in-time (dd-mm-yyyy hh:mm:ss) that the bbranching process in the "Flow 1D Dam Break Branch" should start. For more information on how to specify the absolute point-in-time, see section 5.5.2.4

Option 2: Start bbranching if a specific hydraulic event occurs

In this case the bbranching in a "Flow 1D Dam Break Branch" starts if a specific hydraulic event occurs somewhere in the hydraulic model schematization. Such event can for instance be the exceedance of a water level upstream of the "Flow 1D Dam Break Branch".
The bbranching is initiated by a so-called **Once Hydraulic Trigger** that is assigned to a Relative-time controller, having a relative-time controller table (first column: time in seconds; second column: bbranch area). At the on-start of a computation the status of the Once Hydraulic Trigger is "Off". If the trigger condition of the Once Hydraulic Trigger becomes "True", the status of the Once Hydraulic Trigger becomes "On" and remains "On" during the entire computation, this irrespective of the hydraulic conditions occuring after the status change. Hence, the status of a Once Hydraulic Trigger can only change **once** (e.g. from "Off" to "On") during a computation.
The relative times in the relative-time controller table are made absolute in the time-step (Δt), executed after the absolute point-in-time (T_{On}) that the status of the Once Hydraulic Trigger becomes "On". Meaning that the relative times in seconds are replaced by absolute times (dd-mm-yyyy hh:mm:ss); where yields: absolute time = T_{On} + Δt + relative-time in seconds. From this moment onwards, the absolute time controller table is used for determining the bbranching of the "Flow 1D Dam Break Branch". For more information on how to specify an Once Hydraulic Trigger, reference is made to section 5.5.2.4

The trigger condition of an Once Hydraulic Trigger can be specified using one of the following hydraulic parameters:

- ◇ A water level at a 1D measuring station,
- ◇ A discharge at a 1D measuring station,
- ◇ A head difference over a 1D structure,
- ◇ A force difference over a 1D structure,
- ◇ The crest level (value and direction[*lowering or rising*]) of 1D structure,
- ◇ The crest width (value and direction[*decreasing or increasing*]) of 1D structure, and
- ◇ The gate lower edge level (value and direction[*lowering or rising*] of 1D structure.

5.5.2.4 Input screens of the Flow 1D Dam Break Branch

The "Flow 1D Dam Break Branch" comprise of

- ◇ **The Location Tab** (see Figure 5.220): On this tab, the ID and Name of the "Flow 1D Dam Break Branch" are shown.

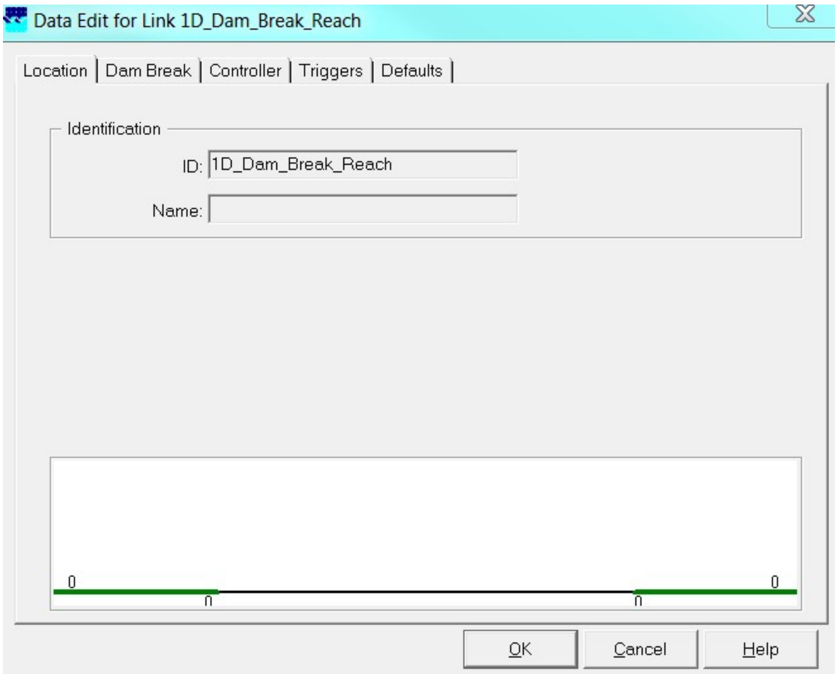


Figure 5.220: *"Flow 1D Dam Break Branch"; Location Tab*

◇ **The Dam Break Tab**: By means of a radio bullet, either the input data (see Figure 5.221) of the Verheij-vdKnaap (2002) bbranch growth formula or the input data (see Figure 5.222) of the vdKnaap(2000) bbranch growth formula can be defined on the Dam Break tab of the "Flow 1D Dam Break Branch". For a description of the input parameters, reference is made to section 6.3.3.1 and section 6.3.3.2.
For defining the vdKnaap(2000) controller table it is necessary to first check the "Use Generation of Table" check-box, enter the input data and thereafter click on the <Generate Table> button. After changing vdKnaap(2000) input data, click on the <Generate Table> button for updating the controller table.
Notice that by checking the "Once Hydraulic Trigger (see section 5.5.2.3)" check-box the "Start bbranch growth" input fields disappear in both Figure 5.221 and Figure 5.222.

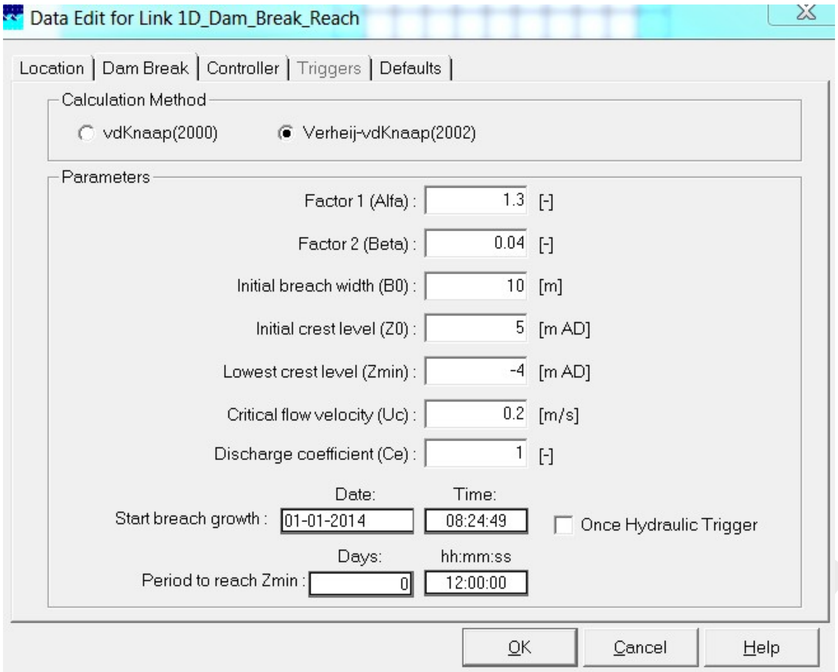


Figure 5.221: *"Flow 1D Dam Break Branch"; the Verheij-vdKnaap(2002) Dam Break Tab*

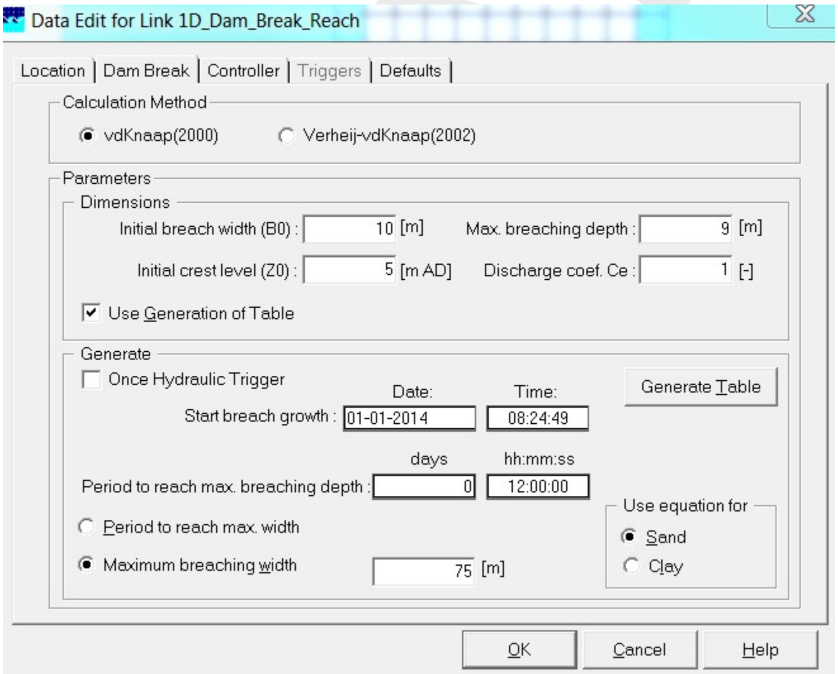


Figure 5.222: *"Flow 1D Dam Break Branch"; the vdKnaap(2000) Dam Break Tab*

- **The Controller Tab**: The defined bbranch development of the "Flow 1D Dam Break Branch" can be inspected on the Controller Tab. There are four options: 1) Verheij-vdKnaap(2002) formula with "Once Hydraulic Trigger (see section 5.5.2.3)"; 2) Verheij-vdKnaap(2002) formula without "Once Hydraulic Trigger"; 3) vdKnaap(2000) formula with "Once Hydraulic Trigger"; and 4) Verheij-vdKnaap(2000) formula without "Once Hydraulic Trigger".
  Option 1 is shown in Figure 5.223. Since the Verhey-vdKnaap(2002) formula is selected, the controller table contains only two rows, defining the bbranch development of phase 1

only (see section 5.5.2.2 and paragraph *"Controller table, defining the development of the bbranch area in time"*). Using an "Once Hydraulic Trigger" implies a "relative-time controller table (see section 5.5.2.3)" with times in seconds.
Option 4 is shown in Figure 5.224. Since the vdKnaap(2000) formula is selected, the controller table comprises of several rows, defining the bbranch development of phase 1 and phase 2 (see section 5.5.2.2 and paragraph *"Controller table, defining the development of the bbranch area in time"*). Not using an "Once Hydraulic Trigger" implies a controller table with absolute times (dd-mm-yyyy hh:mm:ss).

Note: The controller table cannot be edited in case the Verheij-vdKnaap(2002) formula is selected. However, the controller table can be edited if the vdKnaap(2000) formula is selected, which allows for defining an "User defined bbranch development (see section 5.5.2.2)". Please note that clicking on the <Generate Table> button on the Dam Break Tab (see Figure 5.222) will overwrite your edited controller table.

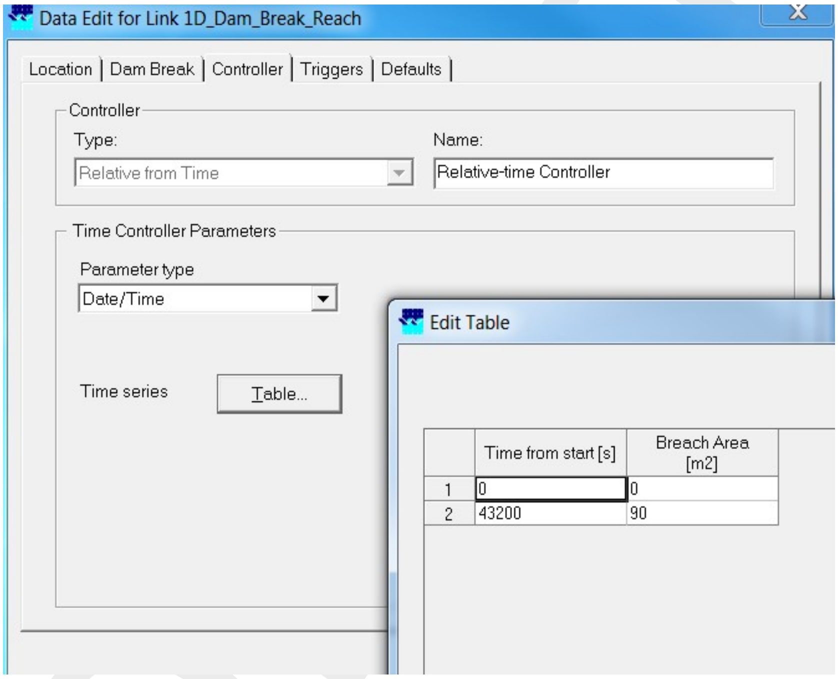


Figure 5.223: *"Flow 1D Dam Break Branch";* ***Option 1"****, Controller Tab for the Verheij-vdKnaap(2002) formula with an "Once Hydraulic Trigger"*

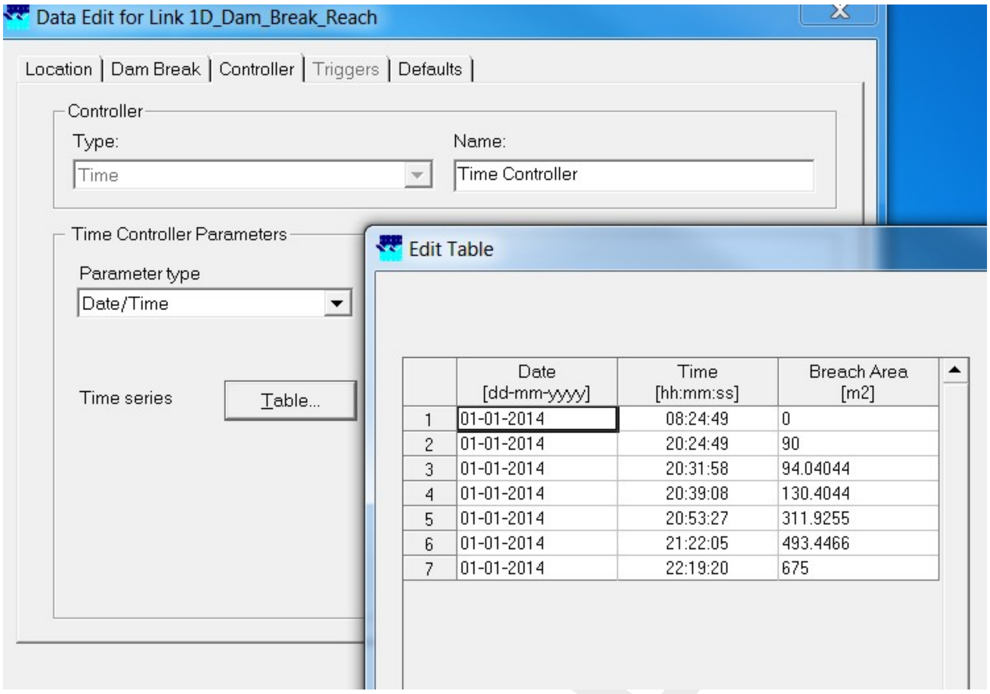


Figure 5.224: *"Flow 1D Dam Break Branch"; **Option 4**, Controller Tab for the vd-Knaap(2000) formula and without an "Once Hydraulic Trigger"*

◇ **The Triggers Tab**: The Triggers Tab is only available if the "Once Hydraulic Trigger" is selected on the Dam Break Tab. At the Triggers Tab (see Figure 5.225) only a trigger of the type "Once Hydraulic Trigger" can be selected in the Trigger Type scroll-box. By typing a Name in the Definition scroll-box a "Once Hydraulic Trigger" can be added and specified. An existing "Once Hydraulic Trigger" can be selected in the Definition scroll-box, and thereafter been edited or deleted. Optional hydraulic parameters are the water level or discharge at a 1D measuring station; the head difference, force difference, crest level, crest width or gate lower edge level of a 1D structure (see Parameter scroll-box). In the Measurement Location scroll-box the ID of the 1D measuring station or 1D structure is to be selected.
The Trigger table (see Figure 5.226) can be obtained by clicking on the <Trigger Table> button. A trigger table is interpreted as a blocked function. In Figure 5.226 it can be seen that the trigger condition of the "Once Hydraulic Trigger" becomes "True": a) if from 01-01-2014 00:00:00 onwards up to 01-01-2014 03:00:00 minus the computational time-step (Δt), the water level at the measuring station becomes above 1.00 m; b) if from 01-01-2014 03:00:00 onwards till the end of the computation, the water level at the measuring station becomes above 1.5 m. Trigger condition is "True" means that the status of the "Once Hydraulic Trigger" is set to "On", that the relative controller table is made absolute and the bbranching starts (for more details, see section 5.5.2.3).

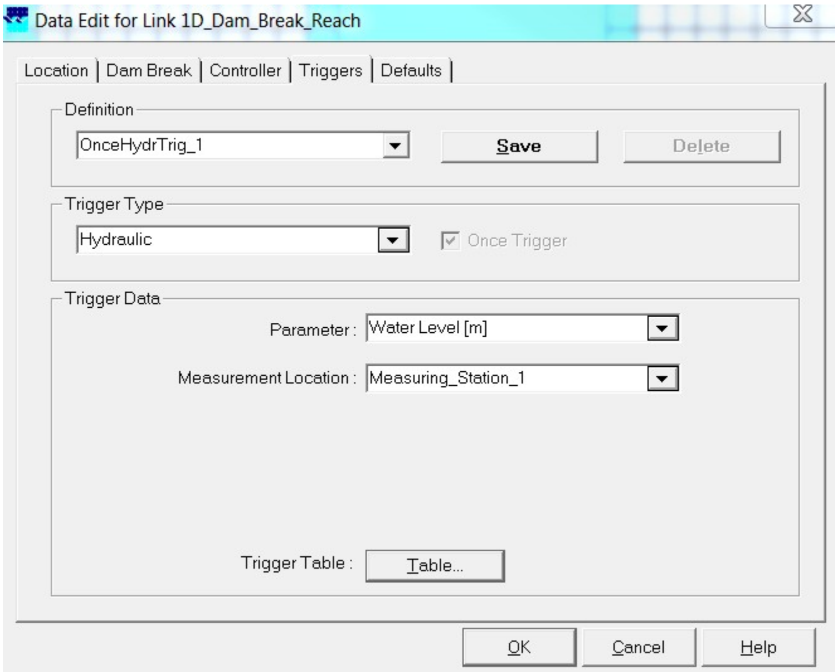


Figure 5.225: *The Triggers Tab of the "Flow 1D Dam Break Branch"*

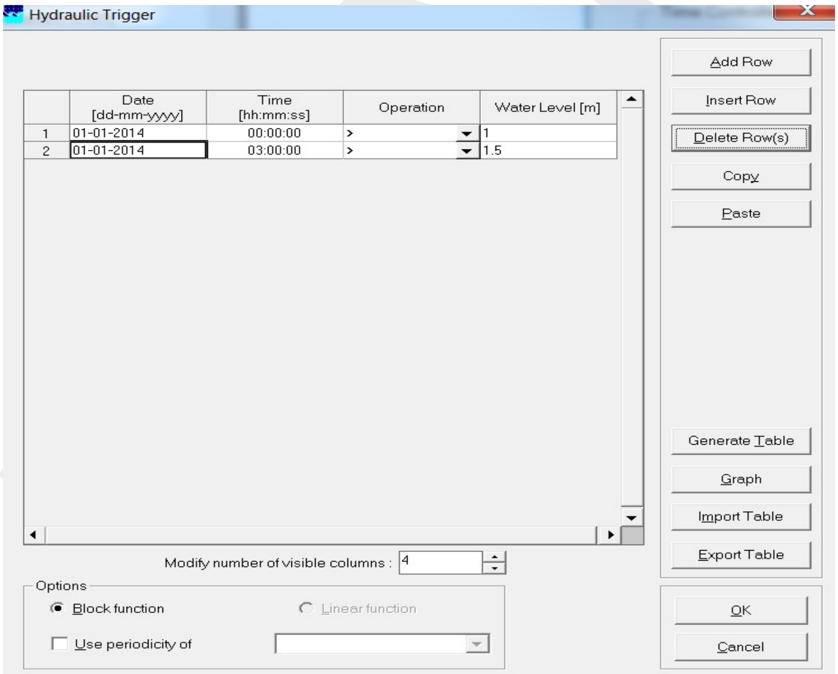


Figure 5.226: *"Flow 1D Dam Break Branch"; Trigger Table of the "Once Hydraulic Trigger"*

5.5.2.5 Output available at a Flow 1D Dam Break Branch

Output parameters available at a Flow 1D Dam Break Branch are in effect structure output parameters, hence:

◇ Firstly, in the "Settings" Task block, click on the <Edit> button of the "1DFLOW" module. Thereafter select the "Output options" Tab and click on the "Structure Tab" available in the "Output parameters" window. Now select your Flow 1D Dam Break Branch output parameters.

◇ After a computation, these output parameters can be viewed in the "Results in Maps" Tasblock (e.g. at a Flow 1D Dam Break branch) as well as in the "Results in Charts" Task block (e.g. under "Results at Structures and Extra Resistance")

The Flow 1D Dam Break Branch output parameters comprise off:

◇ **Discharge**, the discharge through the "Flow 1D Dam break branch" (e.g. discharge through the bbranch) $[m^3/s]$.
◇ **Water Level Up**, water level at the ζ-calculation point having the lowest x-coordinate along the "Flow 1D Dam break branch" $[m]$.
◇ **Water Level Down**, water level at the ζ-calculation point having the highest x-coordinate along the "Flow 1D Dam break branch" $[m]$.
◇ **Head**, the head (=upstream water level minus downstream water level) of the "Flow 1D Dam break branch" (e.g. head over the bbranch) $[m]$.
◇ **Velocity**, the velocity over the weir, accommodated in the "Flow 1D Dam break branch" (e.g. velocity through the bbranch) $[m/s]$.
◇ **Crest Level**, crest level of the weir, accommodated in the "Flow 1D Dam break branch". The crest level represents the bed level of the bbranch (see section 5.5.2.2 and Figure 5.219) $[m]$.
◇ **Structure Flow Area**, flow area above the crest level of the weir, accommodated in the "Flow 1D Dam break branch" $[m^2]$.
◇ **Crest Width**, crest width of the weir, accommodated in the "Flow 1D Dam break branch". The crest width represents the width of the bbranch (see section 5.5.2.2 and Figure 5.219) $[m]$.

Note: The bbranch area at a particular point-in-time is equal to the actual crest width times the difference between the initial crest level and the actual crest level.

5.5.3 Branch - Flow pipe

Pipes
In SOBEK-Urban Water Flow pipes are represented by branches of the type ‘Flow-Pipe’. The Flow-Pipe with Runoff is considered equal to the Flow-Pipe by the Sewer flow module. These branches have a length, a cross section and a resistance. The cross section is defined per branch. Various branches can be connected by various types of Flow-Manholes which can comprise extra storage capacity and lateral inflow.

5.5.4 Flap Gates available for specific type of Pipes

An example of a flap gate is at a tidal outlet; where water can flow into the sea as long as water levels in the pipe are above the sea-level; and where by a hinged flap gate, sea-water is prevented from flowing into the pipe if sea-levels are higher than the water level in the pipe.

A Flap Gate can be accommodated in the following type of pipes only:

◇ Flow - Dry Weather Pipe
◇ Flow - Flow Measurement Pipe
◇ Flow - Pipe
◇ Flow - Pipe with Runoff
◇ Flow - Rain Pipe

On the "Flap Gate" Tab (see Figure 5.227), the user can specify the allowable flow direction(s) in a pipe. Following options are available:

- **None, Flow in Both Directions**, meaning no restriction regarding the flow direction in this pipe (e.g. no flap gate present).
- **Only Positive Flow**, meaning that only positive flow is allowed in this pipe. Flow is positive if flowing in pipe-direction (e.g. flowing from the beginning towards the end of the pipe).
- **Only Negative Flow**, meaning that only negative flow is allowed in this pipe. Flow is negative if flowing in opposite pipe-direction (e.g. flowing from the end towards the beginning of the pipe).

Note: If a "Pipe with Flap Gate" is located adjacent to a 1D Boundary Node, then only a "water level (h)" boundary condition is allowed. Hence, in such case no "flow (Q)" or "Q-h relation" is allowed as boundary condition.

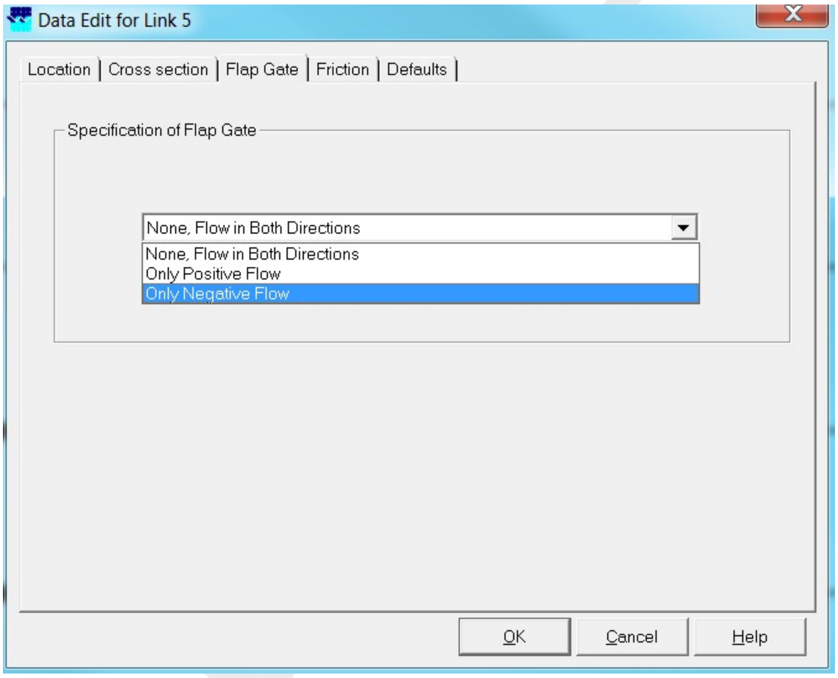


Figure 5.227: *The Flap Gate input screen, that is available for specific type of pipes*

5.5.5 Branch - 1D-2D Internal Boundary Condition

Description

Functionality:

The 1D-2D internal boundary condition enables the user to connect several 2D grid cells to the end of a 1D channel (see picture below). The 1D channel outflow is distributed over the concerning 2D grid cells in accordance with their conveyance capacity. The flow in the 2D grid cells is transferred to the 1D channel. At a 1D-2D Internal boundary condition only 1(one) 1D channel can/may be connected to a particular 2D grid.

Note: At present, a 1D-2D internal boundary condition may <u>not</u> pass over nested grids and should be parallel to either the x-axis or y-axis.

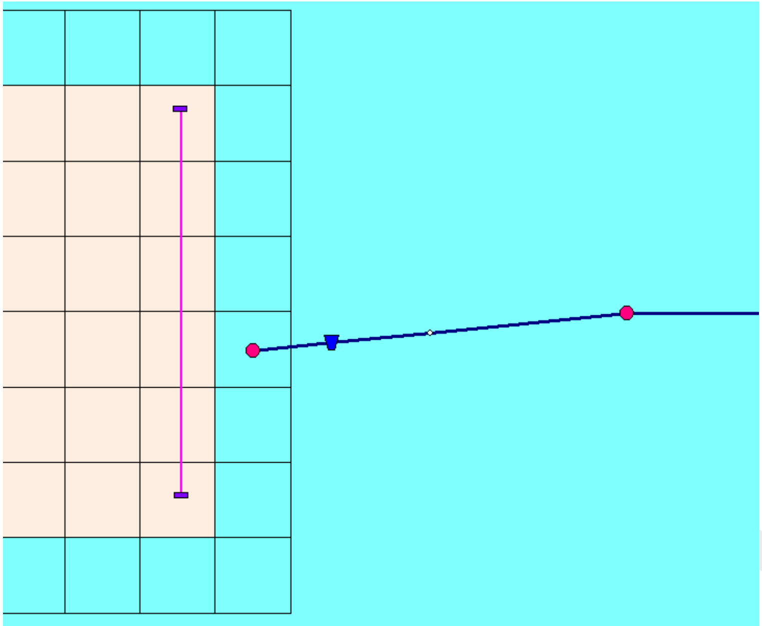


Figure 5.228: *Placing a 1D-2D internal boundary in a model*

How to define:

- ◇ Doubleclick the ‘Schematisation’ task block and thereafter click the *Edit Model* button on the Schematisation window.
- ◇ Click the *Zoom in* button and go to the location where you want to place the 1D-2D Internal boundary condition.
- ◇ Click the *Edit Network* button.
- ◇ Take care that the end node of the 1D Channel, that you like to connect to the 2D grid is a type 36: Flow - 1D2D Internal Boundary Node.
- ◇ Click the *Nodes* button, pull-down the menu and click the Overland Flow Model/ 41, 2D - Corner node.
- ◇ Click the *Branch* button, pull-down the menu and click the Overland Flow Model/ 15, 2D - Line 1D2D Internal Boundary.
- ◇ Click the *Edit action* button, pull-down the menu and click on Nodes/Add Nodes. Go to the location where you want to place the first 2D Corner node of the 1D-2D Internal boundary condition and click with your left mouse button. Place the second 2D- Corner node in the same way. Take care that the two 2D - Corner nodes are located: 1) on either a horizontal or vertical line; 2) on 2D grid cells not having missing values; and 3) that the 2D grid cells lying directly in front of the line through the two 2D Corner nodes are active 2D grid cells (i.e. having no missing value)
- ◇ Click the *Edit action* button, pull-down the menu and click on Connection/Connect nodes. Click with the left mouse button on a 2D Corner node. While pressing down the left mouse button drag a line to the other 2D Corner node and release your left mouse button.
- ◇ It is advised to locate the end node of the 1D Channel on a 2D grid cell, that is part of the 2D grid cells lying under the line connecting the 2D Corner nodes. For moving the end node of the 1D Channel, click the *Edit action* button, pull-down the menu and click on Node/Move Node Click, thereafter click with the left mouse button on the end node and drag it to its new position.
- ◇ Click the *Edit Network* button to end the network editing activities.
- ◇ Click on the 1D-2D Internal boundary condition and thereafter click with your right mouse button, and click on Model data/ Overland Flow Model. The ‘Data Edit’ window should

appear on your screen.
- ◇ Click under Type the 1D2D connection ' check-box' in the 'Boundary condition' Tab of the 'Data Edit' window.
- ◇ Pull down the list-box left of 1D2D Boundary Connection Node and select the ID of the end node of the 1D Channel that you wish to connect to the 2D grid (see Figure below). The ID of a particular node can be obtained by clicking on it, right mouse click, show info.
- ◇ Click the *Save Network* button for saving your defined 2D Q-h Tabulated Boundary Condition.

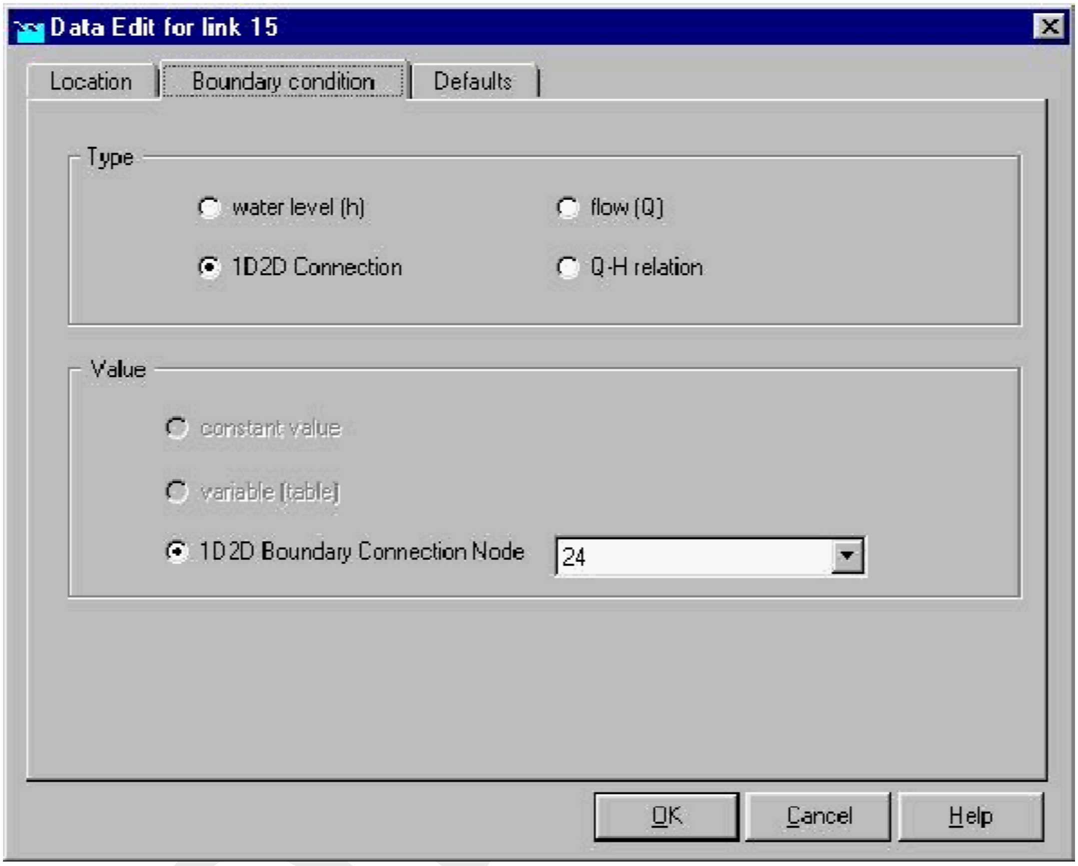


***Figure 5.229: Data Edit**, Boundary condition tab*

How to retrieve information:

Under the 'Results in Maps' Task block

- ◇ Double click the 'Results in Maps' task block.
- ◇ Click in the main menu on File/Open Data/Overland Flow Module Result at history stations (i.e. in the 'Select item' window)
- ◇ In the **View Data** window select the hydraulic parameters "Water level" and "Abs. Disch $[m^3/s]$".
- ◇ Click the 1D-2D Internal boundary condition and then click with right mouse and then on show graph or click on the *graph* button in the 'View Data' window.

  Remarks:
  - ◇ the shown water level is the same as the one at the 1D2D boundary connection node, that is connected to this particular 1D-2D Internal boundary condition;
  - ◇ the discharge is given as an absolute discharge, hence only the magnitude but not the direction of the flow is given,

◇ the direction of the discharge can either be seen from the concerning 1D Channel branch or by providing a discharge-measuring-section on the 2D grid near the 1D-2D Internal boundary condition

Under the 'Results in Charts' Task block

- ◇ Double click the 'Results in Charts' task block.
- ◇ Double click on Overland Flow Module/ Results at history stations (in the 'Results in Charts' window).
- ◇ In the 'ODS-View' window select under parameters Water level and Abs Disch [m^3/s], under locations select the location l_x (where l stands for link and x for the ID of the 1D-2D Internal boundary condition), under time-steps select the required time-period. Now click on the *Graph* button or *Export* button to respectively view a graph or export to the data to a particular file.

5.5.6 Branch - 2D Line discharge measurement

Description
Functionality:

The 2D Line discharge measurement provides the user with information on both the direction and magnitude of the 2D flow passing through the 2D Line discharge measurement as function of time. Using 2D Path, the 2D grid cells that are lying under the 2D Line discharge measurements can be shown. These 2D grid cells are used for determining the actual 2D flow that is passing the 2D Line discharge measurement. From a 2D Line discharge measurement one can obtain the 2D discharge in U direction (i.e. U-Discharge [m^3/s]) and the 2D discharge in V direction (i.e. V-Discharge [m^3/s]) passing through the 2D Line discharge measurement. Discharges in U and V-direction are positive when the flow is respectively flowing in the positive x- and y-direction of the underlying 2D grid. In addition the 2D Line discharge measurement provides the user with information on the direction and magnitude of the effective 2D flow (i.e. Eff.Disch_2D [m^3/s]), passing through the 2D Line discharge measurement as function of time. The sign of this effective 2D discharge depends on the direction in which the user has drawn the 2D Line discharge measurement (see also how to define a 2D Line discharge measurement). In dragging the 2D Line discharge measurement, it is assumed that the user defines an arrow pointing from the starting 2D Corner node towards the ending 2D Corner node. The positive direction of the effective 2D discharge is the clock-wise direction seen from the starting 2D Corner node. In case the effective 2D discharge passes the 2D Line discharge measurement in clock-wise direction or in anti-clock-wise direction, the effective 2D discharge is respectively given a positive or a negative sign (see figure below).

It is to be mentioned that at present discharge-measurement sections must lay on 1 (one) 2D grid and may not pass over nested grids. It is anticipated that in the very-near future the functionality of the discharge-measurement-section will be extended, allowing for a 2D Line discharge measurement to pass several 2D grids (parent and nested 2D grids) as well as areas having no 2D grid at all. In addition information on the effective 1D discharges will be available for the user.

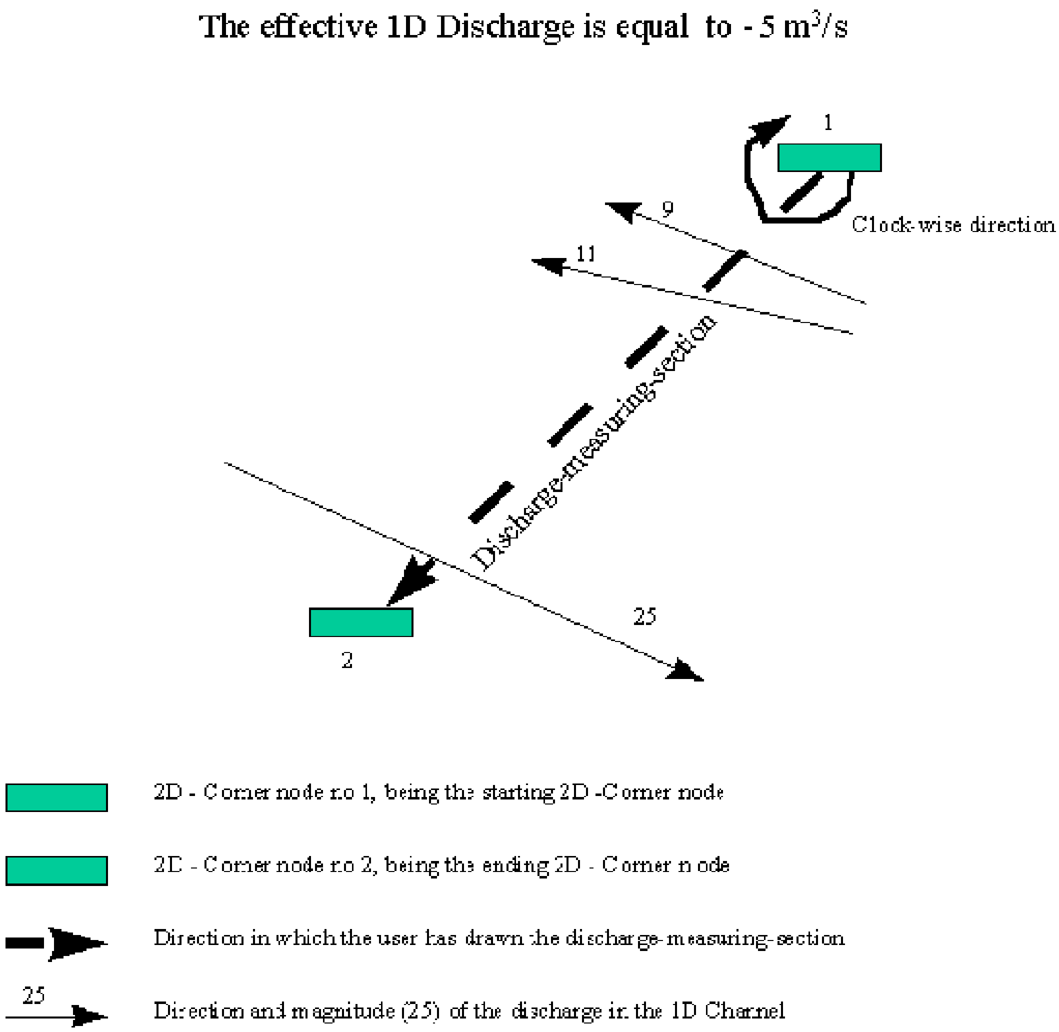


Figure 5.230: *Definition of the sign of the effective 2D Discharge passing a 2D Line discharge measurement*

How to define a 2D Line discharge measurement:

- Double click the ‘Schematisation’ task block and thereafter click the *Edit Model* button on the Schematisation window.
- Click the *Zoom in* button and go to the location where you want to place the 2D Line discharge measurement.
- Click the *Edit Network* button.
- Click the *Nodes* button, pull-down the menu and click the Overland Flow Model/ 41, 2D - Corner node.
- Click the *Branch* button, pull-down the menu and click the Overland Flow Model/ 16, 2D – Line Measurement.
- Click the *Edit action* button, pull-down the menu and click on Nodes/Add Nodes. Go to the locations where you want to place the 2D Corner nodes of the 2D Line discharge measurement and click with your left mouse button.
- Click the *Edit action* button, pull-down the menu and click on Connection/Connect nodes. Click with the left mouse button the starting 2D Corner node. While pressing down the left mouse button drag a line to your ending 2D Corner node and release your left mouse button.
- Click the *Edit Network* button to end the network editing activities.

◇ Click the *Save Network* button for saving your defined 2D Line discharge measurement

How to retrieve information on a 2D Line discharge measurement:

Under the **Results in Maps** Task block

◇ Double click the **Results in Maps** task block.
◇ Click in the main menu on File/Open Data/Overland Flow Module Result at history stations (i.e. in the **Select item** window)
◇ In the **View Data** window select for instance the hydraulic property "Eff. Discharge 2D $[m^3/s]$"
◇ Click the 2D Line discharge measurement and then click with right mouse and then on show graph or click on the *graph* button in the **View Data** window.

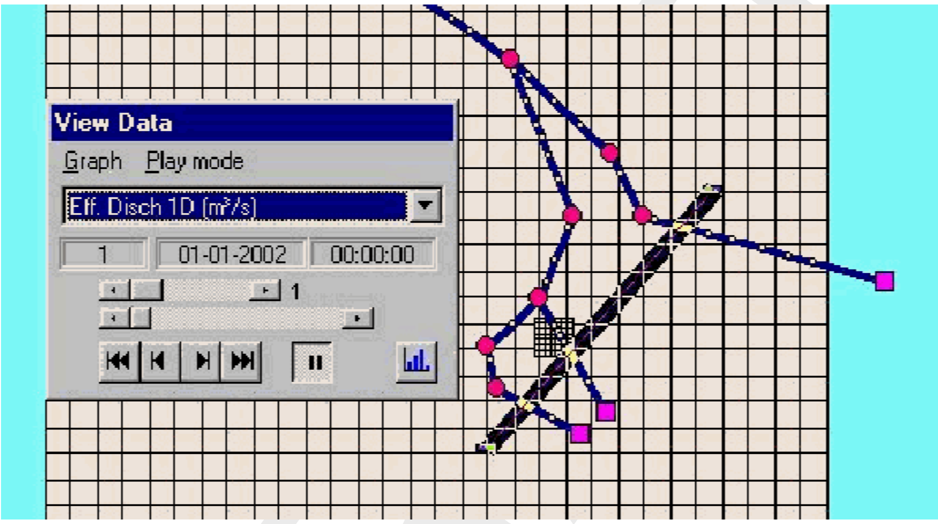


Figure 5.231:* *View Data *window*

Under the 'Results in Charts' Task block

◇ Double click the 'Results in Charts' task block.
◇ Double click on Overland Flow Module/ Results at history stations (in the 'Results in Charts' window).
◇ Select the hydraulic parameter (for instance Eff. Disch 2D $[m^3/s]$), under locations select location l_x (where l stands for link and x for the ID of the 2D Line discharge measurement), under time-steps select the required time-period. Now click on the *Graph* button or *Export* button to respectively view a graph or export to the data to a particular file.

5.5.7 Branch - 2D-Line boundary

Description

The 2D-line boundary can be used instead of the 2D-boundary node when there are multiple grid cells next to each other that have the same boundary condition (i.e. tidal boundary). The boundary condition specified for the line boundary (in edit model data mode) is used for all underlying 2D grid cells.

The 2D-line boundary is made up of a construction of two 2D-corner nodes and one 2D-line boundary. This construction is made by first placing the 2 nodes on the grid, and then connecting them using the '2D-line boundary' branch type and the 'connect existing nodes' option from the edit network menu. The line boundary can be either horizontal or vertical, but not anything else. All underlying 2D grid cells should contain no-data values, and from every boundary cell the water should be allowed to flow to (or from) only **one** other cell. See the Figure 5.232 for an example.

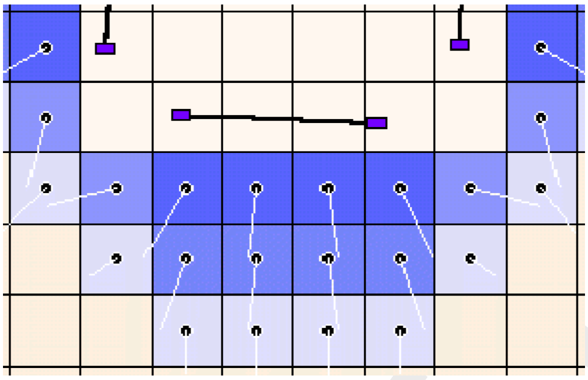
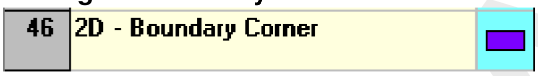


Figure 5.232: *Possible configuration of a 2D Line boundary*

Note: the 2D line boundary should not pass over a nested grid.

Editing 2D Boundary Corner and Line Boundary data



At a 2D-line boundary the user can define a water level or discharge as function of time (i.e. $h(t)$ or $Q(t)$) as an external forcing boundary condition. In case of a water level boundary all 2D grid cells lying under the 2D-line boundary obtain the same water level. In case of a discharge boundary condition, the inflowing discharge is distributed over the 2D grid cells lying under the 2D-line boundary in accordance with the conveyance capacity of each individual 2D grid cell.

5.6 Cross Section types

5.6.1 Overview of available cross-sectional profiles

An overview of the cross-section types that may lay on the same branch is given in Table 5.3. An overview of the properties of the available cross-section types is given in Table 5.4.

Table 5.3: *Overview of Cross-section types that may lay on the same branch*

Overview of Cross-section types that may lay on the same SOBEK branch															
	Arch	Asym. Trapezium	Closed Lumped Y-Z	Closed Tabulated[1]	Cunnette	Egg-shape	Elliptical	Open Lumped Y-Z	Open Tabulated[1]	Open Vert.Segm. Y-Z	Rectangle	River Profile	Round	Steel Cunette	Trapezium
Arch	✓	-	-	✓	✓	-	✓	-	-	-	✓	-	-	✓	-
AsymTrap	-	✓	-	-	-	-	-	-	-	✓	-	-	-	-	-
Closed Lumped Y-Z	-	-	✓	-	-	-	-	-	-	-	-	-	-	-	-
Closed Tabulated[1]	✓	-	-	✓	✓	-	✓	-	-	-	✓	-	-	✓	-
Cunnette	✓	-	-	✓	✓	-	✓	-	-	-	✓	-	-	✓	-

Table 5.3: *Overview of Cross-section types that may lay on the same branch*

Overview of Cross-section types that may lay on the same SOBEK branch															
	Arch	Asym. Trapezium	Closed Lumped Y-Z	Closed Tabulated[1]	Cunnette	Egg-shape	Elliptical	Open Lumped Y-Z	Open Tabulated[1]	Open Vert.Segm. Y-Z	Rectangle	River Profile	Round	Steel Cunette	Trapezium
Egg-shape	-	-	-	-	-	✓	-	-	-	-	-	-	-	-	-
Elliptical	✓	-	-	✓	✓	-	✓	-	-	-	✓	-	-	✓	-
Open Lumped Y-Z	-	-	-	-	-	-	-	✓	-	-	-	-	-	-	-
Open Tabulated[1]	-	-	-	-	-	-	-	-	✓	-	-	-	-	-	✓
Open Vert.Segm. Y-Z	-	✓	-	-	-	-	-	-	-	✓	-	-	-	-	-
Rectangle	✓	-	-	✓	✓	-	✓	-	-	-	✓	-	-	✓	-
River Profile	-	-	-	-	-	-	-	-	-	-	-	✓	-	-	-
Round	-	-	-	-	-	-	-	-	-	-	-	-	✓	-	-
Steel Cunette	✓	-	-	✓	✓	-	✓	-	-	-	✓	-	-	✓	-
Trapezium	-	-	-	-	-	-	-	-	✓	-	-	-	-	-	✓

[1] A Tabulated profile is closed in case the total width at its highest level is less than 20 mm

Table 5.4: *Properties of available cross-section types*

Type	**Open/Closed**	**Summer dike**	**Storage Width**	**Conveyance method**	**No. of Friction Sections**	**Ground layer Thickness**	**Bed Friction definition**	**Ground layer Friction def.**
Arch	Closed	no	no	lumped	1	yes	G&L	G&L
AsymTrap	Open	no	no	V.Segm	n	no	Crs	-
Closed Lumped Y-Z	Closed	no	no	lumped	1	no	Crs	-
Closed Tabulated	Closed	no	no	lumped	1	yes	G&L	G&L
Cunnette	Closed	no	no	lumped	1	yes	G&L	G&L
Egg-shape	Closed	no	no	lumped analytic	1	yes	G&L	G&L
Elliptical	Closed	no	no	lumped	1	yes	G&L	G&L
Open Lumped Y-Z	Open	no	no	lumped	1	no	Crs	-
Open Tabulated	Open	no	yes	lumped	1	yes	G&L	G&L
Open Vert.Segm. Y-Z	Open	no	no	V.Segm	n	no	Crs	-
Rectangle	Closed	no	no	lumped	1	yes	G&L	G&L
River Profile	Open	yes	yes	lumped	3	no	G&L $f(x)$, $f(x,h)$, $f(x,Q)$	-
Round	Closed	no	no	lumped analytic	1	yes	G&L	G&L
Steel Cunette	Closed	no	no	lumped	1	yes	G&L	G&L

***Table 5.4:** Properties of available cross-section types*

Type	**Open/Closed**	**Summer dike**	**Storage Width**	**Conveyance method**	**No. of Friction Sections**	**Ground layer Thickness**	**Bed Friction definition**	**Ground layer Friction def.**
Trapezium	Open	no	no	lumped	1	yes	G&L	G&L

Note:

1 Open/Closed:

- ◇ Closed means that pressurized flow occurs for water depths higher than the inner cross-sectional height.
- ◇ A tabulated profile is closed in case the total width at the highest level is less than 20 millimetres. At all levels of a closed Tabulated profile should yield that the total width equals flow width (hence no storage allowed)

2 Method for computing conveyance (see section 6.1.20):

- ◇ Lumped means that conveyance is computed using the "lumped conveyance method" (see section 6.1.20.1)
- ◇ Lumped analytical means that in computing "the lumped conveyance" use is made of analytical formulae, giving the wetted area and the wetted perimeter as function of the actual water depth.
- ◇ V.Segm. means that conveyance is computed using "the vertically segmented conveyance method" (see section 6.1.20.2).

3 Number of Friction sections

- ◇ For a River profile three different friction sections can be defined (main section, floodplain1 and floodplain2)
- ◇ For a Y-Z and Asym. Trapezium profile yields that a particular friction section may not be smaller than the width of its underlying vertical segment ($W = Y_{i+1} - Y_i$)

4 Ground layer Thickness

- ◇ The surface level of the ground layer equals the lowest level in the cross-sectional profile plus the ground layer thickness.
- ◇ The cross-sectional part, located below the surface level of the ground layer is omitted in the hydraulic computations.
- ◇ The "ground layer width" is the width of the cross-sectional profile at the surface level of the ground layer.
- ◇ For the "ground layer width" a roughness value can be defined that differs from the roughness value applied to the remaining part of the wetted cross-sectional perimeter. The roughness of the ground layer width is referred to as Ground layer friction. The roughness value applied to the remaining part of the wetted cross-sectional perimeter is referred to as Bed friction.
- ◇ The same friction formula (Chézy, Manning etc) is applied for bed friction as well as for ground layer friction.

5 Bed and Ground layer Friction definitions

- ◇ G means constant global (or model-wide) friction value.
- ◇ L means for the entire branch a constant (local) friction value.
- ◇ $f(x)$, $f(x,h)$ and $f(x,Q)$ means that bed-friction along a branch can be defined as function of the x-coordinate, as function of the x-coordinate and the actual water level, or as function of the x-coordinate and the actual discharge.
- ◇ Crs means that bed friction is to be defined for each and every cross-section.

5.6.2 Flow - Cross Section node (Arch type)

Cross sections of the Arch type are closed profiles using the tabulated lumped conveyance approach (see section 6.1.20.1).

The input screen looks as follows:

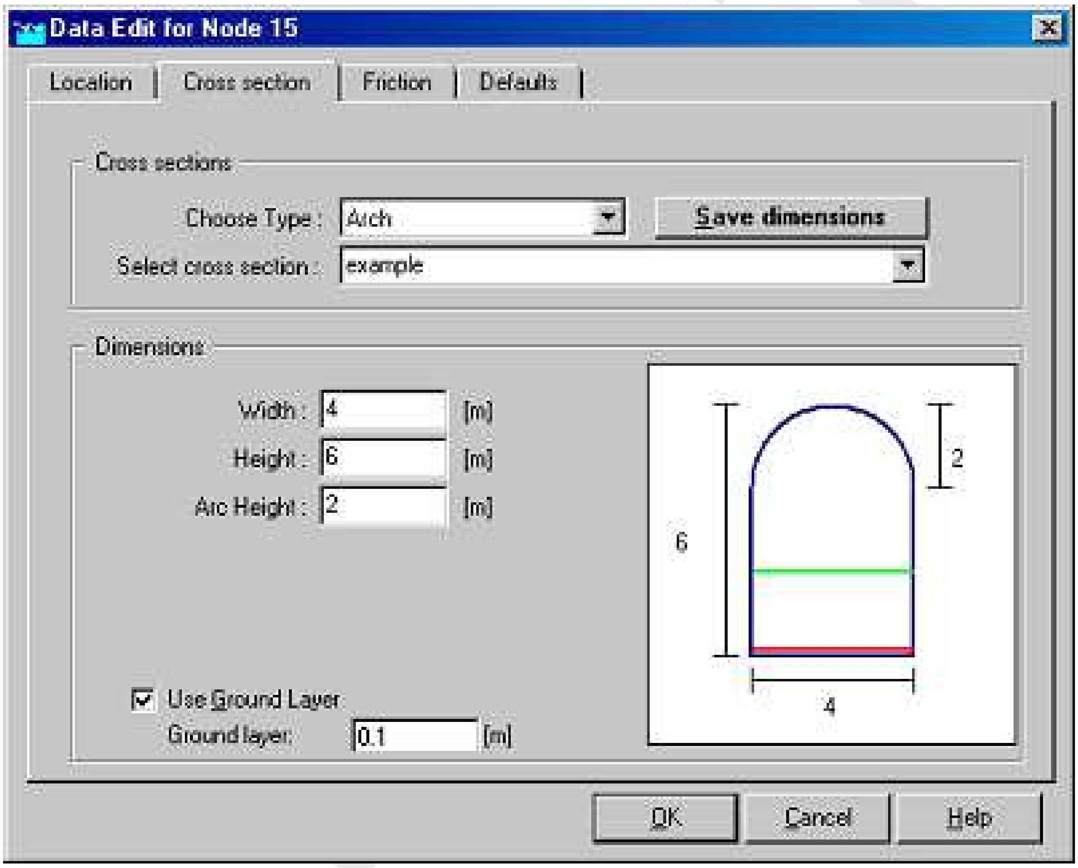


***Figure 5.233: Data Edit**, Cross section tab*

In the right-lower corner of the screen, an impression of the cross section shape that you enter will be plotted. The dimensions that you filled in will be plotted next to it.

Create a cross section definition:
To start creating a cross section definition, do the following:

- ◇ choose the cross section of the "Arch" type in the 'Choose Type' box.
- ◇ Create a new definition by typing a name for it in the "select cross section" field
- ◇ Click the button *Define Dimensions*;
- ◇ Enter the desired values in the parameter fields "width", "height" and "arc height";

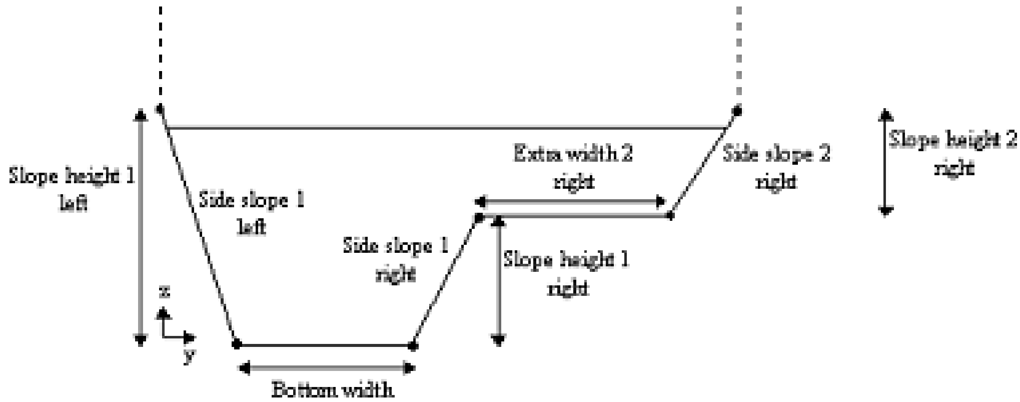
◇ Optionally, turn on the "use ground layer option" and enter a value;
◇ Click the button *Save Dimensions*.

Parameters:
This cross section type requires the following parameters:

◇ Width: represents the width of the arch.
◇ Height: represents the height of the rectangle part of the arch(from the bottom up to the point where the curved section starts)
◇ Arc Height: represents the height of the curved part.
◇ Use Ground Layer: switch on if a layer of sediment lies in the cross section. An individual roughness value may be given for the ground layer. See the 'Friction' tab for this.
◇ Ground layer: enter the thickness of the sediment layer that lies within the cross section. This thickness will be made visible in the image on the right side through a red line.

5.6.3 Flow - Cross Section node (Asymmetrical trapezium type)

Cross-sections of the Asymmetrical trapezium type (see Figure 5.234) are open profiles using the vertically segmented conveyance approach (see section 6.1.20.2).



Center

Bottom width (m)
5

Left: 1 side slope

Side slope 1:n hor:ver	Slope height (m)	Extra width (m)
4	7	0

Right: 2 side slopes

Side slope 1:n hor:ver	Slope height (m)	Extra width (m)
4	4	4
3	3	0

Figure 5.234: *Asymmetrical trapezium cross section*

The bed level that you define on the Location tab becomes the lowest point of this cross section.

There is no restriction to the number of side slopes that are chosen left and right. Of course a trapezium cross section that is symmetrical can be modelled by using the same side slope, heights and extra width at the left and right side of the cross section.

The maximum flow width is limited by the outer left and outer right point of this profile. These points are the left and right bank of the profile. The surface level is equal to the highest point of the cross-sectional profile.

For the Asymmetrical trapezium profile the vertically segmented conveyance approach is applied.

5.6.4 Flow - Cross Section node (Closed Lumped Y-Z type)

Cross-sections of the closed Lumped Y-Z type use the tabulated lumped conveyance approach (see section 6.1.20.1). A ground-layer as well as total widths (i.e. storage) can not be defined for a closed lumped Y-Z profile. Furthermore, only one roughness value and corresponding roughness formula can be defined, which are applied to the entire wetted perimeter. *Please note* that a Closed Lumped Y-Z profile is obtained by selecting the item 'Closed Lumped' in the 'Type of Y-Z profile' list-box (see Figure 5.235), accomodated on the 'Cross-section' Tab (Edit/Model data/ Flow - Cross Section).

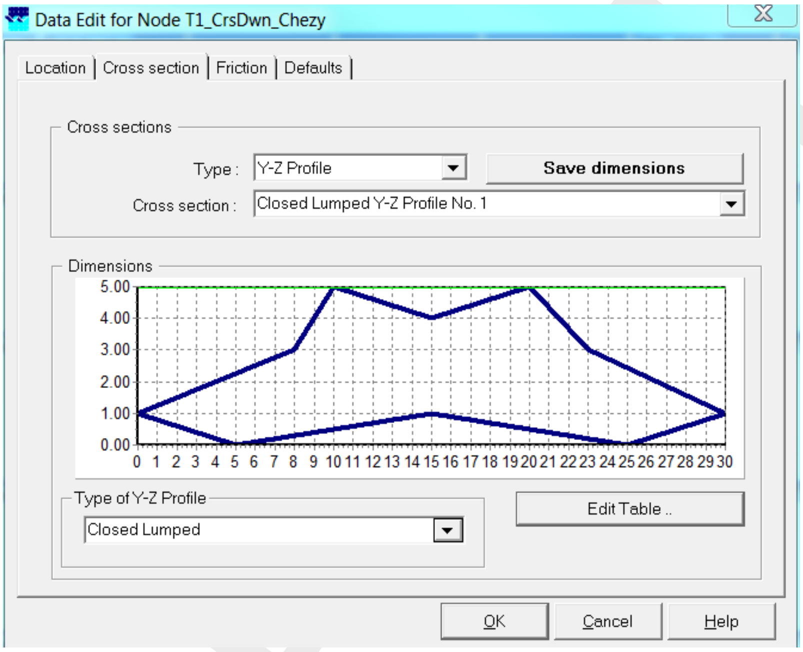


Figure 5.235: *Example of a closed Lumped Y-Z profile defined in ModelEdt*

If the user defines a closed lumped Y-Z profile that is actually not closed (see Figure 5.236), such lumped Y-Z profile will be closed by adding a (Y,Z) point at the end of the user-defined Y-Z table. This added (Y,Z) point equals the first user-defined (Y,Z) point (see Figure 5.237).

The computational procedure is as follows. Firstly, it is verified if a valid closed lumped Y-Z profile is defined and an error message is given if this is not the case. Thereafter, a tabulated closed lumped Y-Z table is constructed (see Figure 5.238). Next, based on the symmetrical tabulated closed lumped Y-Z profile depicted in Figure 5.238, for a particular water level the corresponding flow area and wetted perimeter is determined. Finally, using this information and the defined roughness, the corresponding conveyance is computed (see section 6.1.20.1).

User-defined Closed Lumped Y-Z profile		
No	Y	Z
-	*m*	*m*
1	25.0	3.00
2	5.0	4.00
3	5.0	0.50
4	30.0	1.00
5	55.0	0.50
6	55.0	4.50
7	35.0	3.50

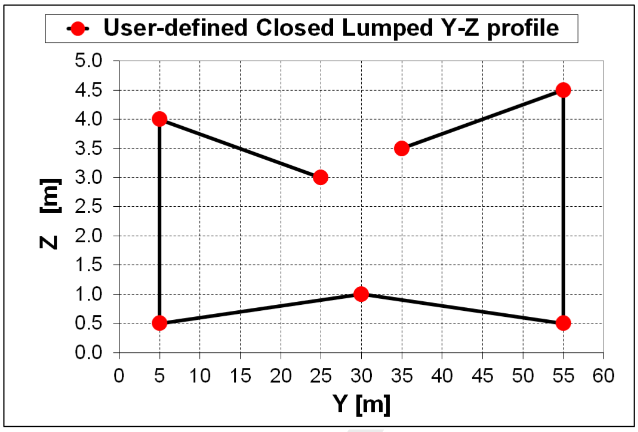


Figure 5.236: *User defined closed Lumped Y-Z profile*

SOBEK Adapted Closed Lumped Y-Z profile		
No	Y	Z
-	*m*	*m*
1	25.0	3.00
2	5.0	4.00
3	5.0	0.50
4	30.0	1.00
5	55.0	0.50
6	55.0	4.50
7	35.0	3.50
8	25.0	3.00

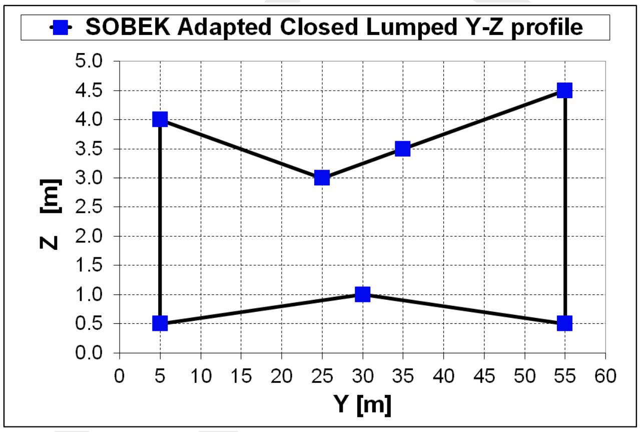


Figure 5.237: *SOBEK adapted closed Lumped Y-Z profile*

Tabulated Closed Lumped Y-Z table	
Elevation	Width
m	*m*
0.50	0.00
1.00	50.00
3.00	50.00
3.50	30.00
4.00	10.00
4.50	0.00

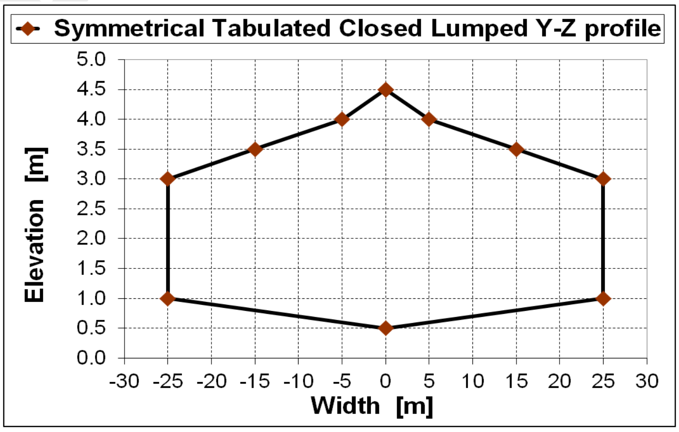


Figure 5.238: *Symmetrical tabulated closed lumped Y-Z profile*

5.6.5 Flow - Cross Section node (Closed Tabulated type)

Cross-sections of the tabulated type use the tabulated lumped conveyance approach (see section 6.1.20.1). A tabulated cross-section is closed if the flow width at the heighest defined elevation is smaller than 20 millimetres, else it is an Open tabulated cross-section (see section 5.6.10). Total widths (i.e. storage) cannot be defined for a Closed tabulated cross-section.

5.6.6 Flow - Cross Section node (Cunette type)

Cross sections of the Cunette type are closed profiles using the tabulated lumped conveyance approach (see section 6.1.20.1).

The input screen looks as follows:

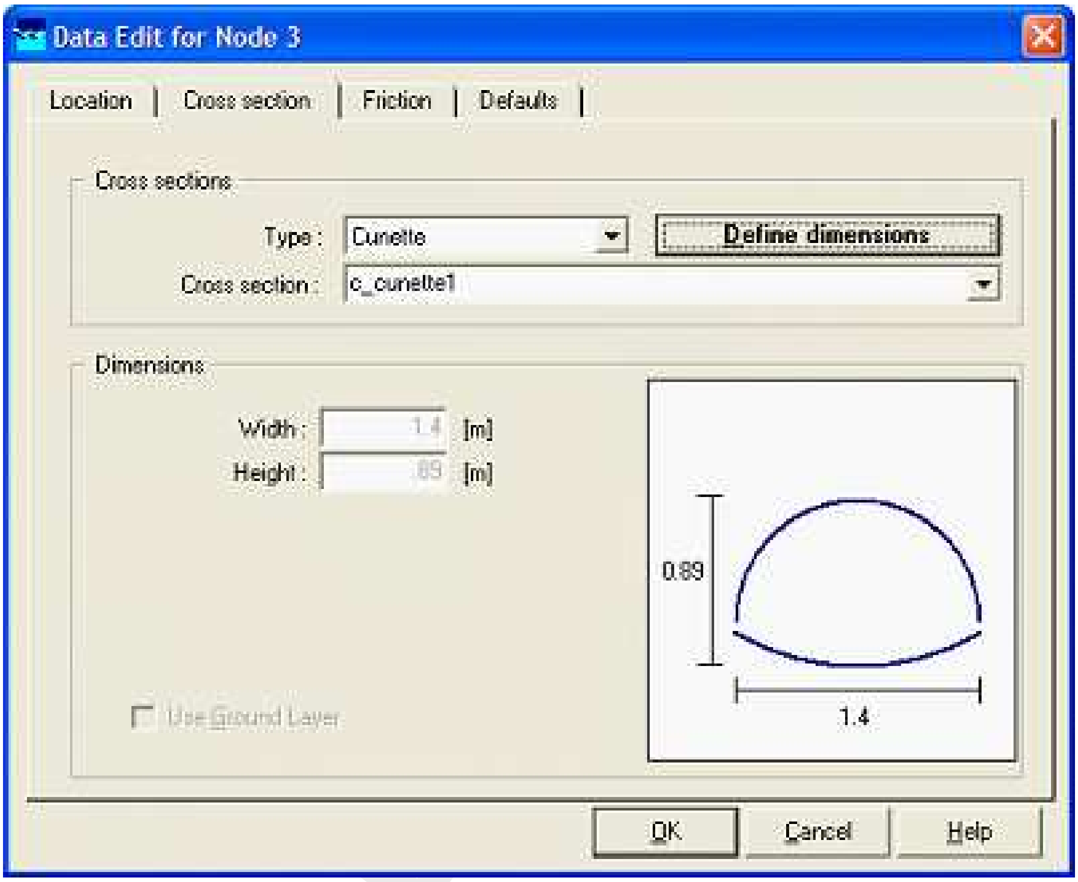


Figure 5.239: *The Cross section tab of a cunette profile*

The parameter Height is determined based on the user-defined Width. Internally, a cunette profile is handled as a tabulated profile. When reopening a cunette profile, SOBEK will show the converted tabulated profile based on the cunette profile as defined by the user. See Figure 5.240. A tabulated type originating from a cunette profile is prefixed by 'c_'.

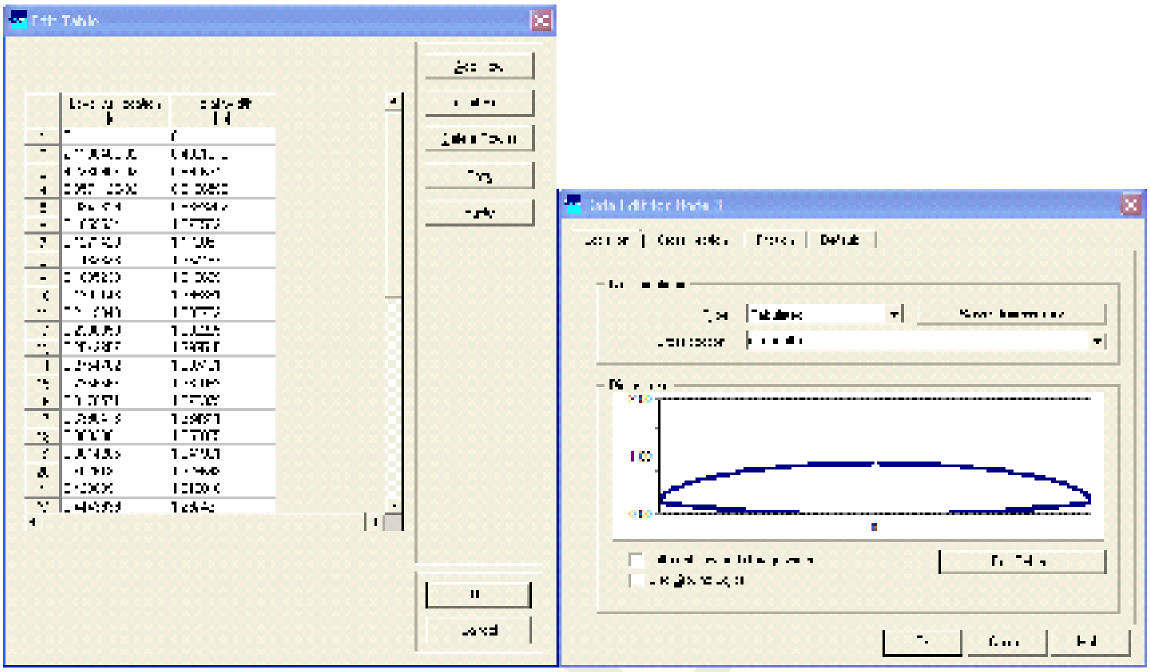


Figure 5.240: *The corresponding tabulated profile data of the cunette profile in Figure 5.239*

For more information, see Cross-Section type: Steel Cunette (section 5.6.15).

5.6.7 Flow - Cross Section node (Egg-shape type)

Cross-sections of the Egg-shape type are closed profiles using analytical formulae for computing lumped conveyance as function of water depth. An egg-shaped cross section is defined by its bed level and width. The height is by definition 3/2 times the width.

5.6.8 Flow - Cross Section node (Elliptical type)

Cross-sections of the Elliptical type are closed profiles using the tabulated lumped conveyance approach (see section 6.1.20.1). An Elliptical cross section is defined by its maximum width and maximum height.

5.6.9 Flow - Cross Section node (Open Lumped Y-Z type)

Cross-sections of the open Lumped Y-Z type use the tabulated lumped conveyance approach (see section 6.1.20.1). A ground-layer as well as total widths (i.e. storage) can not be defined for an open lumped Y-Z profile. Furthermore, only one roughness value and corresponding roughness formula can be defined, which are applied to the entire wetted perimeter. *Please note* that an Open Lumped Y-Z profile is obtained by selecting the item 'Open Lumped' in the 'Type of Y-Z profile' list-box (see Figure 5.241), accomodated on the 'Cross-section' Tab (Edit/Model data/ Flow - Cross Section).

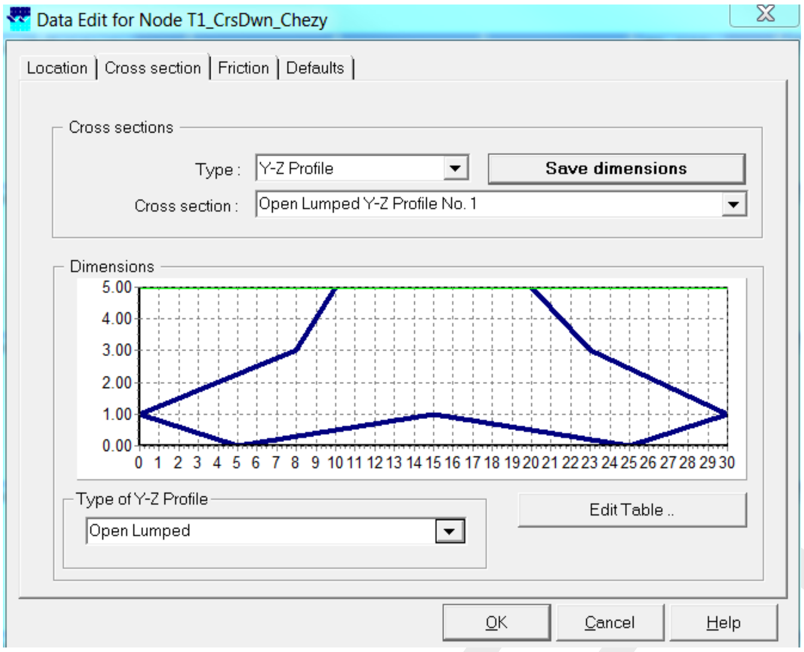


Figure 5.241: Example of an Open Lumped Y-Z profile defined in ModelEdt

The algorithm in SOBEK does not allow for open lumped Y-Z profiles, having 'Y-Z line-segments' that can be wetted from both sides. See for instance the 'Y-Z line-segment' from (Y,Z) points (25,3) to (5,4) in Figure 5.242, that is intersected by a water level of 3.5 metres. Hence a part of this 'Y-Z line-segment is at a water level of 3.5 metres wetted at its left-side as well as at its right-side. To avoid wetting from both sides. If the Z-value of the first user-defined (Y,Z) point is below the maximum Z-value (Zmax) in the user defined open lumped Y-Z profile, a (Y,Z) point is added at the beginning of the user-defined Y-Z table for which yields that its Y-value equals the Y-value of the first user-defined (Y,Z) point and its Z-value equals Zmax. The same applies when the Z-value of the last user-defined (Y,Z) point is below Zmax. In such case a (Y,Z) point is added at the end of the user-defined Y-Z table for which yields that its Y-value equals the Y-value of the last user-defined (Y,Z) point and its Z-value equals Zmax. In Figure 5.243 it can be seen that two additional (Y,Z) points are added by SOBEK to the Y-Z table of the open lumped Y-Z profile, that is depicted in Figure 5.242.

The computational procedure is as follows. Firstly, it is verified if a valid open lumped Y-Z profile is defined and an error message is given if this is not the case. Thereafter, a tabulated open lumped Y-Z table is constructed (see Figure 5.244). Next, based on the symmetrical tabulated open lumped Y-Z profile depicted in Figure 5.244, for a particular water level the corresponding flow area and wetted perimeter is determined. Finally, using this information and the defined roughness,the corresponding conveyance is computed (see section 6.1.20.1).

User-defined Open Lumped Y-Z profile		
No	Y	Z
-	m	m
1	25.0	3.00
2	5.0	4.00
3	5.0	0.50
4	30.0	1.00
5	55.0	0.50
6	55.0	4.50
7	35.0	3.50

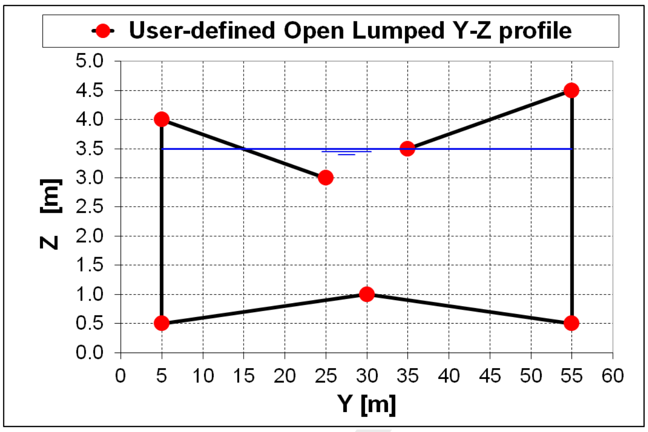


Figure 5.242: *User defined open Lumped Y-Z profile*

SOBEK Adapted Open Lumped Y-Z profile		
No	Y	Z
-	m	m
0	25.0	4.50
1	25.0	3.00
2	5.0	4.00
3	5.0	0.50
4	30.0	1.00
5	55.0	0.50
6	55.0	4.50
7	35.0	3.50
8	35.0	4.50

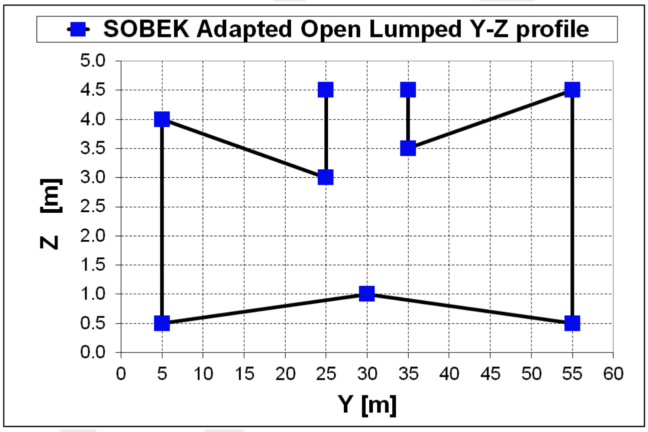


Figure 5.243: *SOBEK adapted open Lumped Y-Z profile*

Tabulated Open Lumped Y-Z table	
Elevation	Width
m	m
0.50	0.00
1.00	50.00
3.00	50.00
3.50	40.00
4.00	20.00
4.50	10.00

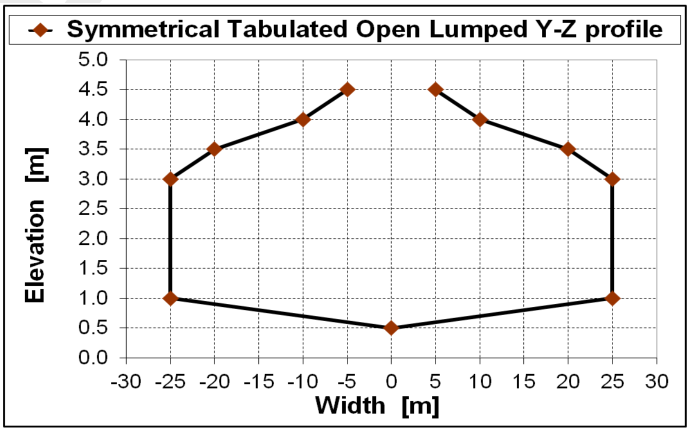


Figure 5.244: *Symmetrical tabulated open lumped Y-Z profile*

5.6.10 Flow - Cross Section node (Open Tabulated type)

Cross-sections of the tabulated type use the tabulated lumped conveyance approach (see section 6.1.20.1). A tabulated cross-section is open if the flow width at the heighest defined elevation is greater or equal than 20 millimetres, else it is a closed tabulated cross-section (see section 5.6.5). Total widths (i.e. storage) can be defined for open tabulated cross-sections.

5.6.11 Flow - Cross Section node (Open vertically segmented Y-Z type)

Cross-sections of the open vertically segmented Y-Z profile type use the vertically segmented conveyance approach (see section 6.1.20.2). *Please note* that an Open vertically segmented Y-Z profile is obtained by selecting the item 'Open Vertically Segmented' in the 'Type of Y-Z profile' list-box (see Figure 5.245), accomodated on the 'Cross-section' Tab (Edit/Model data/ Flow - Cross Section).

Open vertically segmented Y-Z profiles are often used when river bed loadings (bathymetry measurements) are available. The Y-Z table corresponding to Figure 5.245 is given in Table 5.5.

Table 5.5: *Cross Section - Y-Z table*

Y **distance along measuring-section**	**Z** **level of the river bed**
0	16.4
15	15.2
30	13.8
45	11.4
60	10.6
75	11.0
90	12.5
105	14.0
120	14.8
135	16.4

Cross section tab:

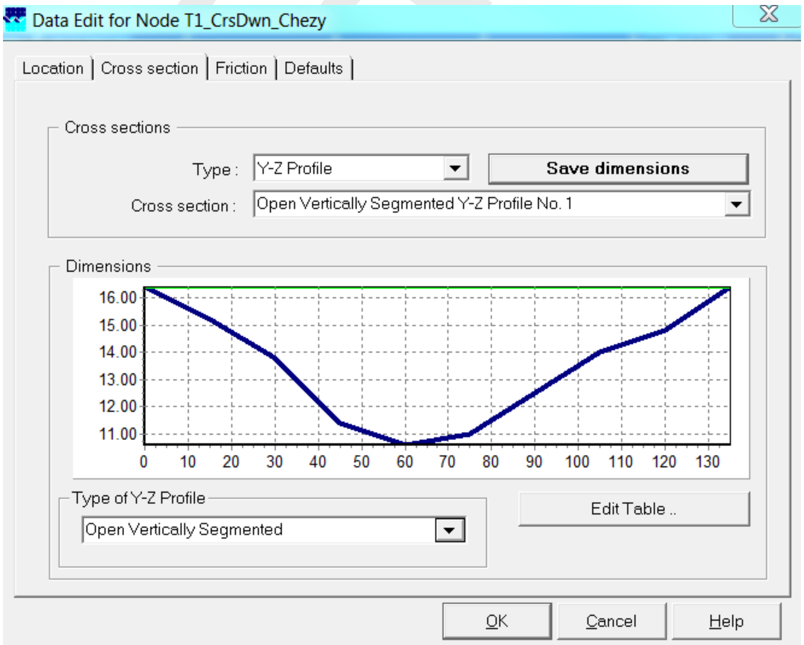


Figure 5.245: *Example of an Open Vertically Segmented Y-Z profile defined in ModelEdt*

Create a cross section definition:

To start creating a cross section definition, do the following:

1 Choose the cross section of the "Y-Z profile" type in the 'Choose Type' box;
2 Create a new definition by typing a name for it in the "select cross section" field
3 Select option "Open Vertically Segmented" in the "Type of Y-Z profile" frame
4 Click the button *Define Dimensions* and subsequently *Edit Table*;
5 Enter the desired values in the parameter columns Y and Z;
6 Leave the table editor again by clicking *OK*
7 Click the button *Save Dimensions*.

Friction tab:

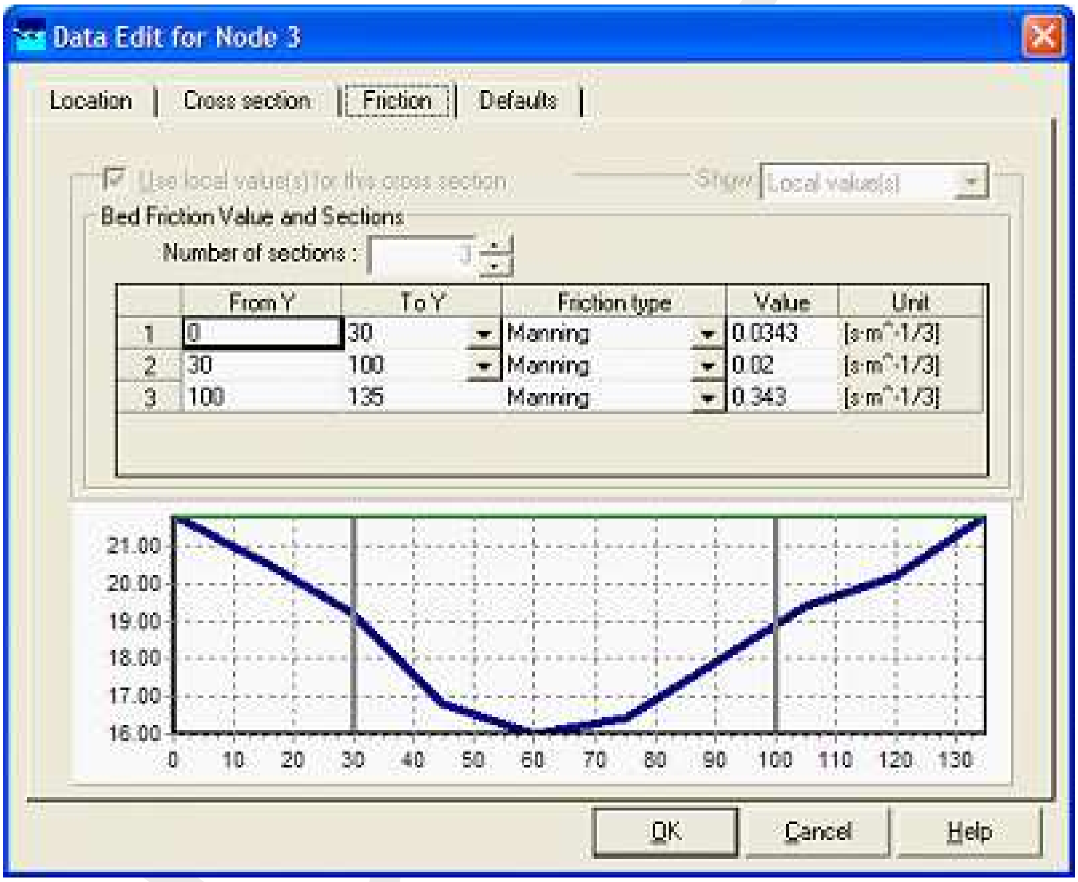


***Figure 5.246: Data Edit**, Friction tab, Open vertically segmented YZ profile*

The conveyance of an open vertically segmented Y-Z profile is computed based on the vertically segmented conveyance approach. You can define subsections along the Y-Z profile, having different friction values. For instance, the main gully may have a lower friction value than the area near the embankments. In the above figure, this principle is shown.

You can change the number of subsections by clicking the arrow keys next to the 'Number of sections' field. Subsequently, you'll have to assign the distances from the shore where the change in friction will occur. Do that in the 'to Y' fields.

5.6.12 Flow - Cross Section node (Rectangle type)

Cross-sections of the Rectangular type are closed profiles using the tabulated lumped conveyance approach (see section 6.1.20.1). The rectangular cross-section is defined by its bed level, its width and its height.

5.6.13 Flow - Cross Section node (River Profile type) (beta functionality)

Cross-sections of the River Profile type are open profiles using the tabulated lumped conveyance approach (see section 6.1.20.1).

The "River Profile" cross section type becomes available when selecting the River Flow module in the Settings task block. In this chapter the options for this cross section type are discussed in more detail.

The Cross Section tab

The river profile is a cross section type that can be used to model river sections. It comes available by selecting the 'River Profile' cross section type in the second tab of the properties of a cross section. Once selected, the user can start to define the properties. Figure 5.247 shows an example of a profile:

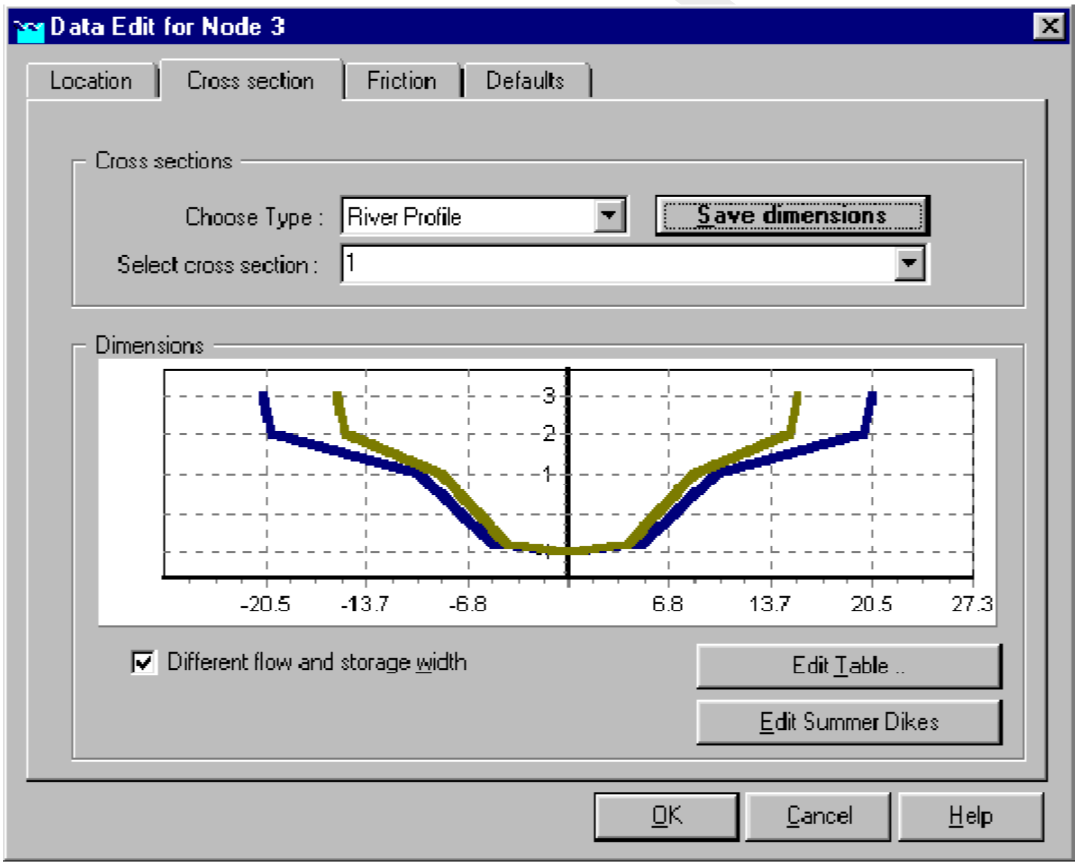


Figure 5.247: *Example of a 'river profile' type cross section*

The user has the option to differentiate between flow width and storage width. This is a useful option when part of the cross section, i.e. the sections between groynes, does not participate in the conveyance of the water, but does contribute to the river's storage capacity. The picture of the cross section will show the flow width as a green line, and the total width with a blue line.

The level-width table is entered in the table window, which is opened by pressing the *edit table* button:

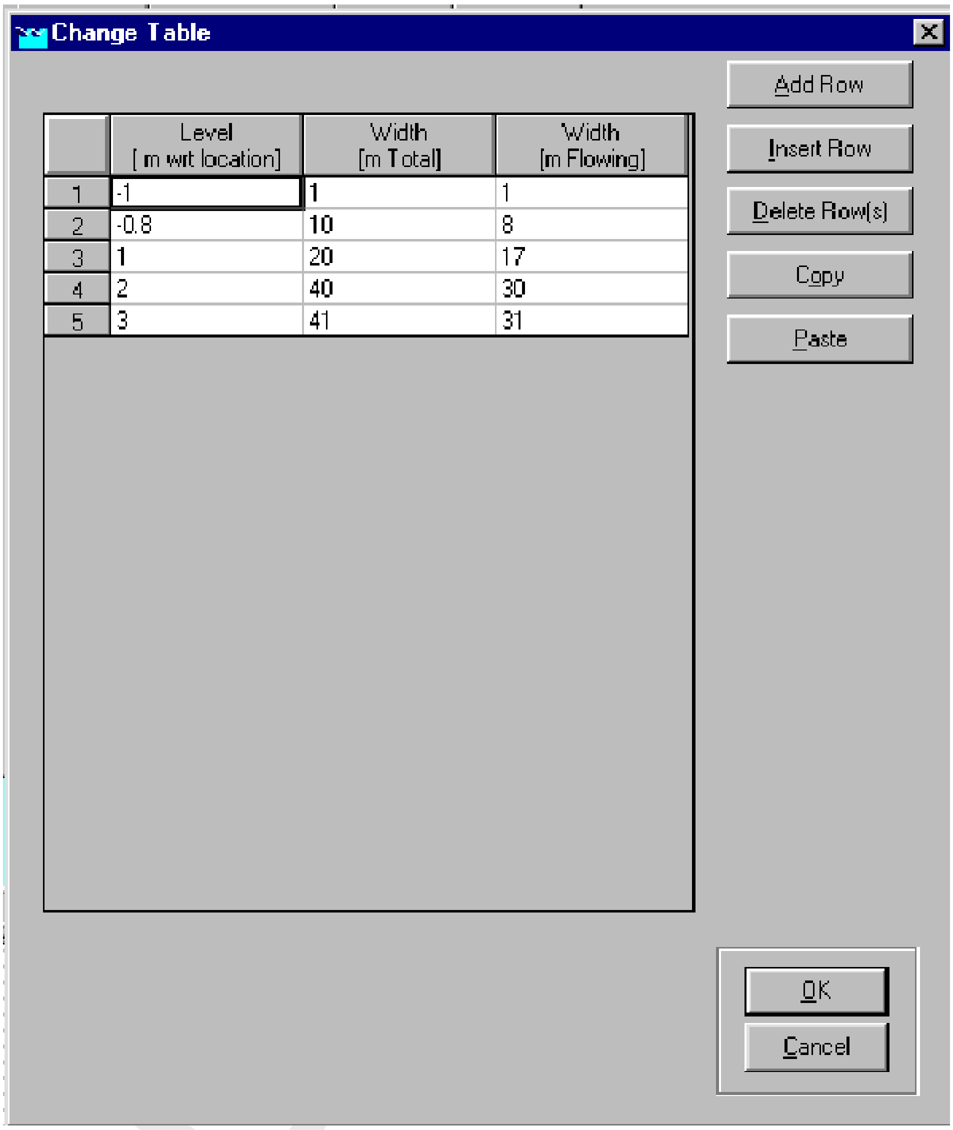


Figure 5.248: *Table where a cross section's dimensions are defined.*

The levels in the cross-section should be entered in ascending order. At each next level both the flow width and the total width (flow width) are to be specified. The entire (total) cross-section is divided into a flow part and a storage part. As a consequence, the terms storage width and storage area refer to the storage part of the cross-section and the terms total width and total area are used when the entire cross-section is meant. In view of the applied numerical solution method, it is important not to specify widths that change too rapidly with increasing level. Remark that such rapidly varying widths should also be avoided because of reasons of accuracy. A maximum relative increase of about a factor of 5 over 1 cm (or a factor of 50 over 10 cm, etc.) is recommended, i.e.

$$\frac{W_{i+1}/W_i}{z_{i+1} - z_i} = O(5/cm) \tag{5.2}$$

with W_i and W_{i+1} the cross-sectional widths specified at the two consecutive levels Z_i and Z_{i+1} (this applies to both flow width and storage width). So for example, when the storage width at a certain level measures 85 m, the storage width to be specified at a level that is 1 cm (5 cm) higher should not be much larger than 425 m (2 125 m), i.e., the bank slope should not be smaller than about 0.0025 m/100 m in this particular case.

Both the total width and the flowing width can be entered here as a function of the vertical level. The total width should always be equal to or larger than the flow width. When you choose to use a summer dike, you can enter the necessary information by selecting the 'edit summer dikes' option and switching on the 'use summer dike' option. See Figure 5.249.

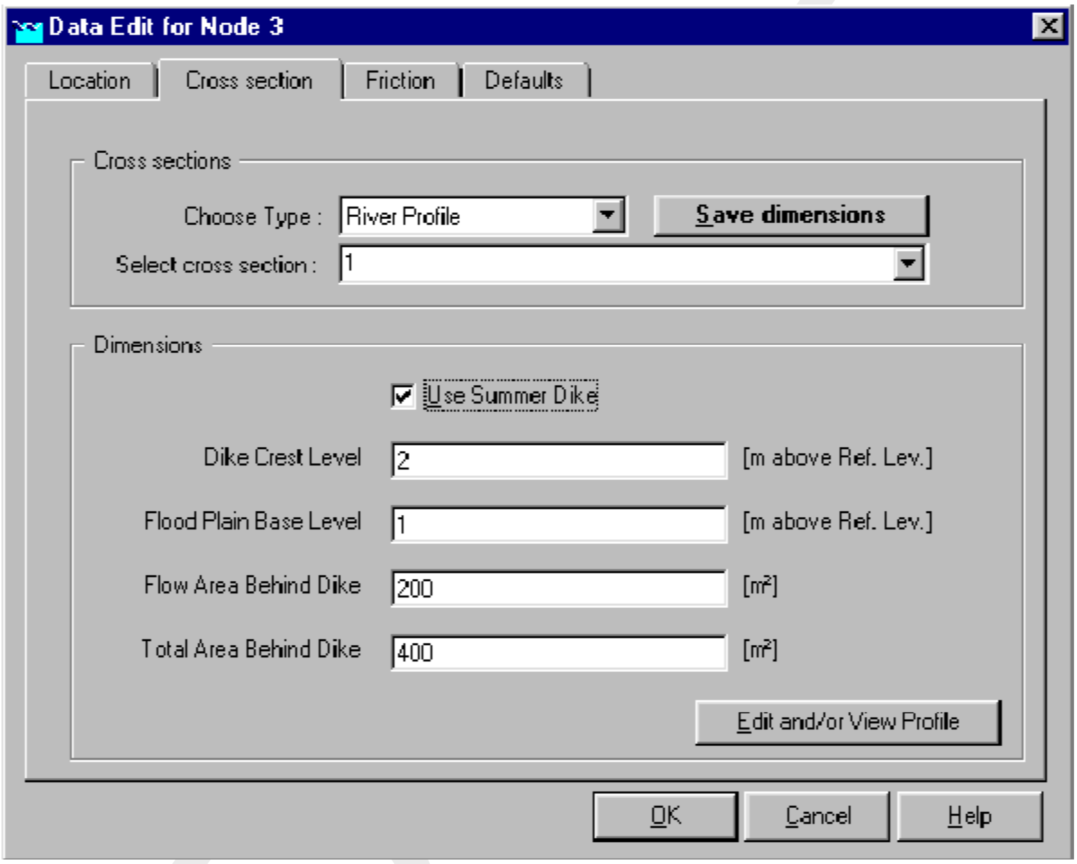


Figure 5.249: *Activating the 'summer dike' option*

The transition height for summer dikes should be defined in the 'Advanced Settings' Tab in the SOBEK-River section on the SETTINGS task block from the Case Manager. This transition height will act on all summer dikes defined in the model.

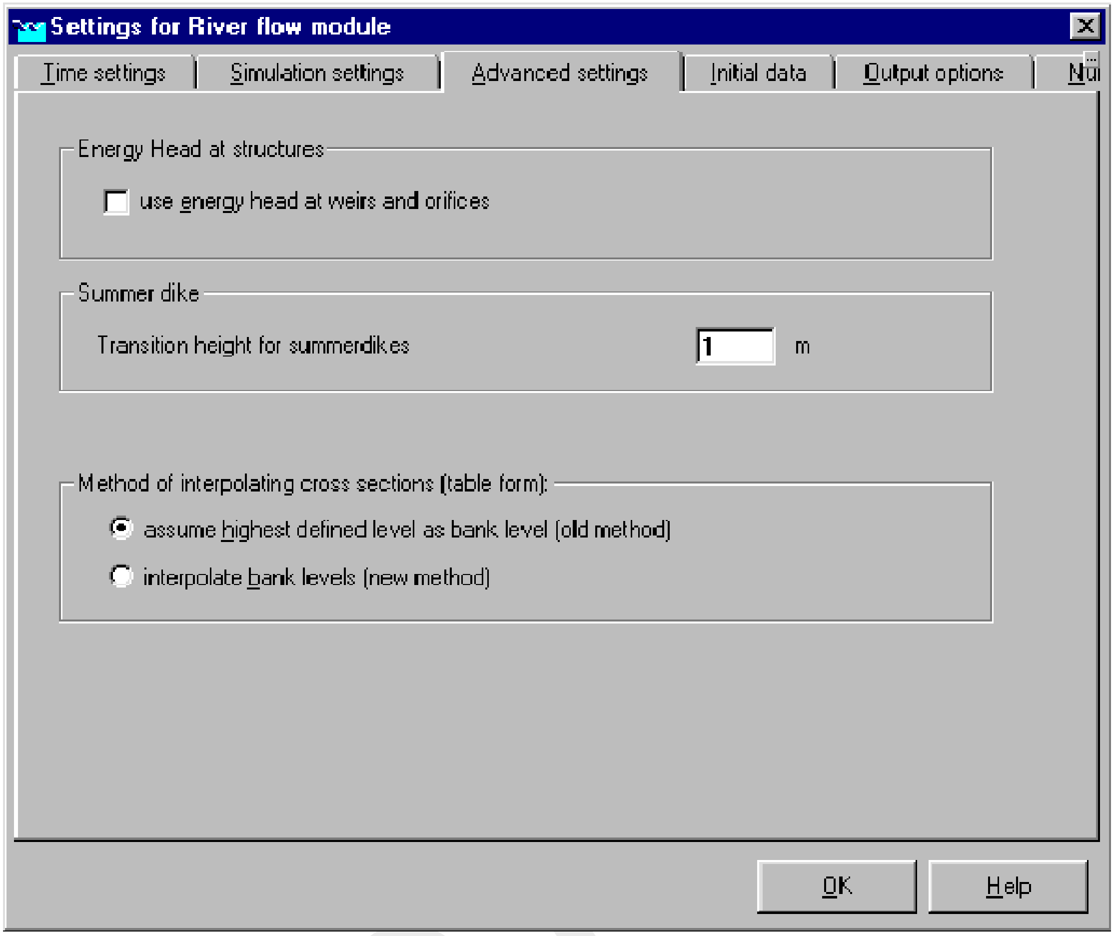


Figure 5.250: The advanced settings tab for the SOBEK-River settings.

When a flood plain is separated from the river by a summer dike, the cross section profile including the floodplain does not increase monotonously in height. In this case, it is possible to include summer dikes in the schematisation.

The Friction Tab

The friction type and value can be specified per subsection of the flowing part of the cross section. There are three different subsections: (i) the main river, (ii) floodplain 1 (fp_1) and (iii) floodplain 2 (fp_2). For every section the flow width can be chosen. The width of the three sections combined is always equal to the total flow width specified in the level-width table.

The following friction types can be applied:

- Chézy
- Manning
- Strickler (k_n)
- Strickler (k_s)
- White-Colebrook
- Engelund

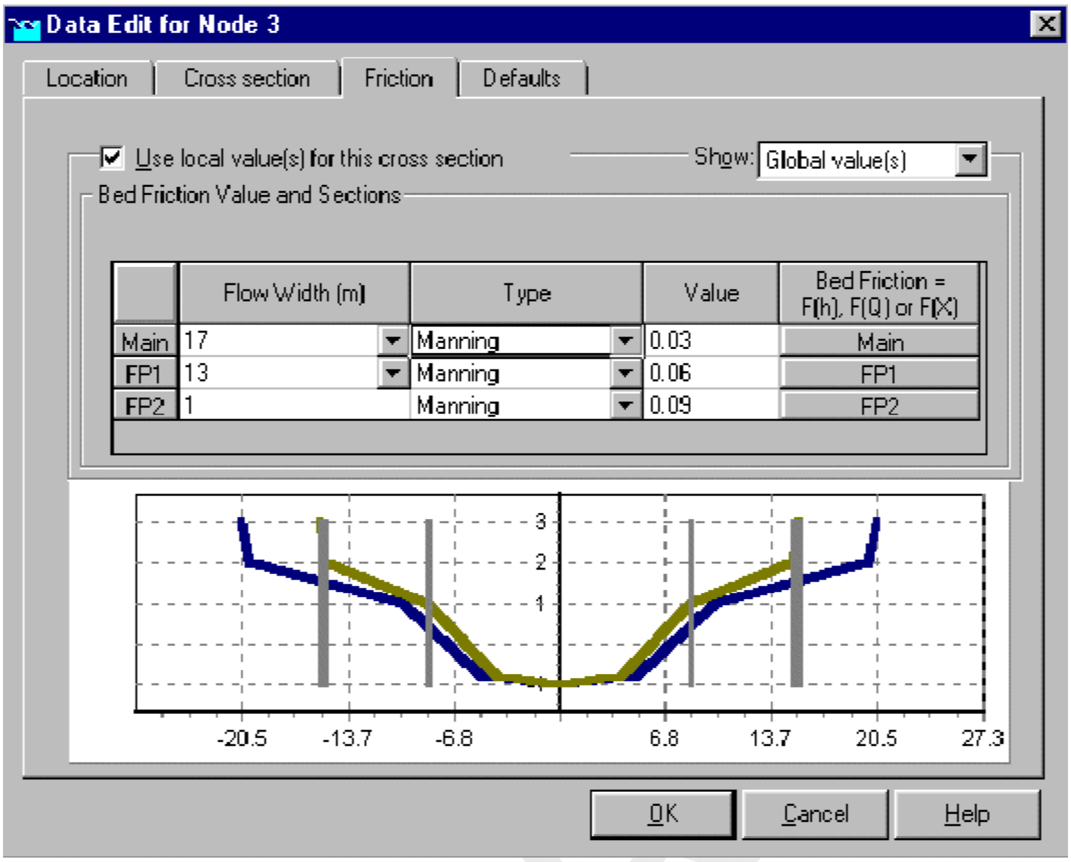


Figure 5.251: *Defining subsections for unique friction values on different sections*

By default, the friction value is the same for river flow in positive direction (defined branch direction) and in negative direction. The user can choose to change this by pressing the corresponding button in the ‘bed friction column in Figure 5.251. As a result, the form depicted in Figure 5.252 pops up.

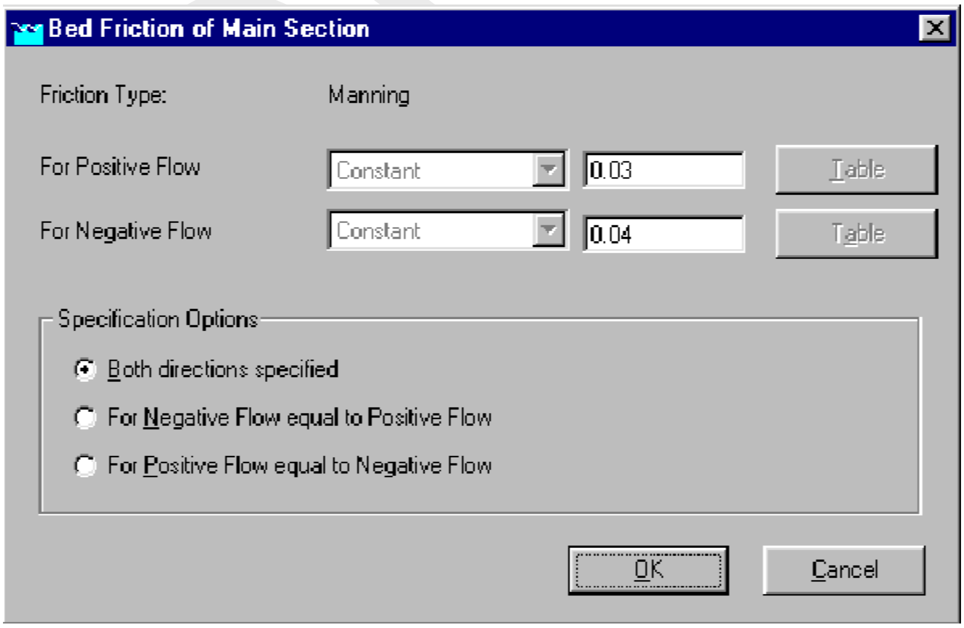


Figure 5.252: *Window for friction settings of the main profile section.*

Finally, the Chézy, Nikuradse, Manning or Strickler coefficients may be defined as

- ◇ a constant;
- ◇ spatially varying along the river branch $(F(X))$;
- ◇ a tabulated function of both the water level (h) and the x-axis $(F(h))$ or both the total discharge (Q) and the x-axis $(F(Q))$.

The last two options are only available for local friction definitions (when also local values have been selected), not for global definitions. All options are available for all three sections of the flowing part of the river. Figure 5.253 shows the empty table that can be filled in for friction values dependent on water levels and location.

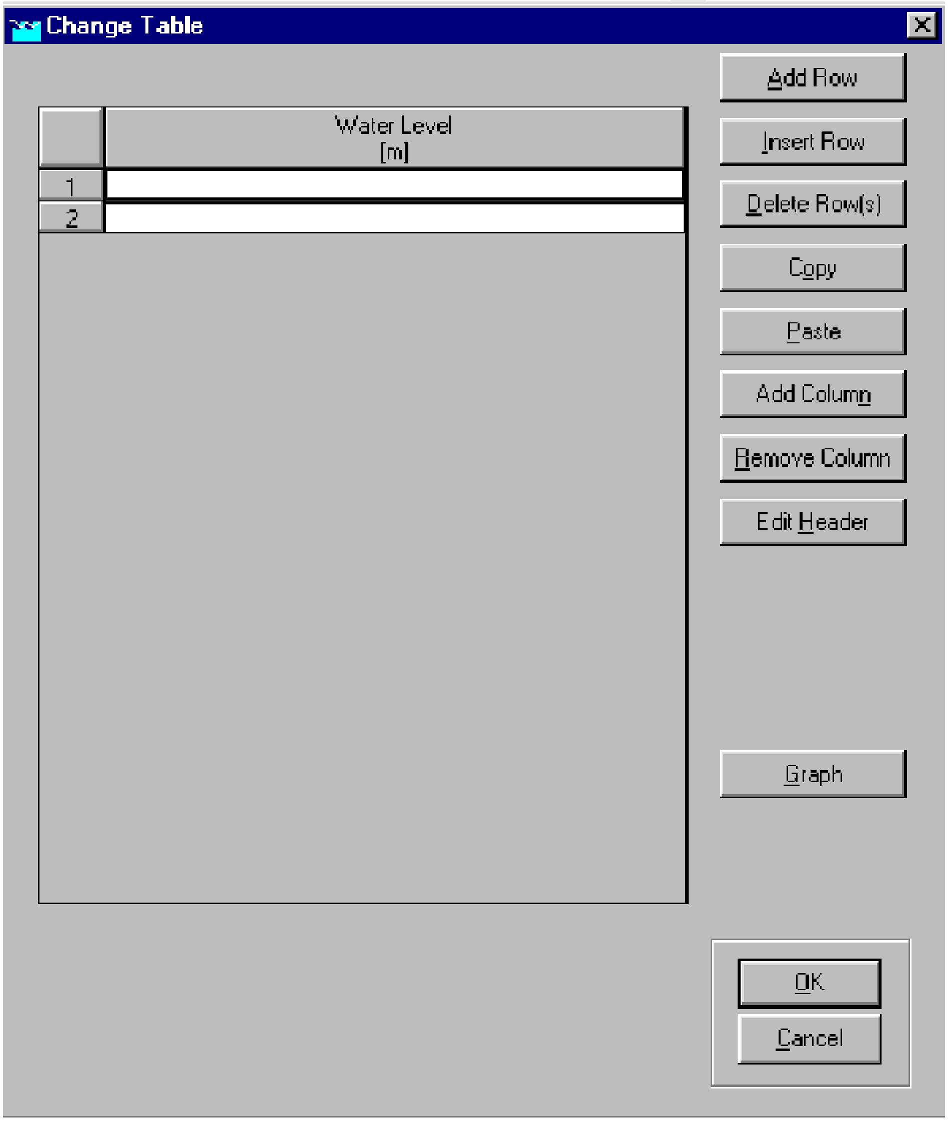


Figure 5.253: *Entering friction values as a function of water levels and location*

First of all, the locations for which the friction is a known variable should be filled in. This is done by adding columns (locations), and giving them the numerical value of the distance along the branch from the start of the branch to the location itself. In Figure 5.254, two locations have been added at 1 200 m and 2 400 m from the beginning, respectively. Next, the number of rows needs to match the number of water levels for which there is data. In the example below, two levels have been added to make a total of four levels available. Finally, the friction values as a function of $f(X,h)$ can be entered.

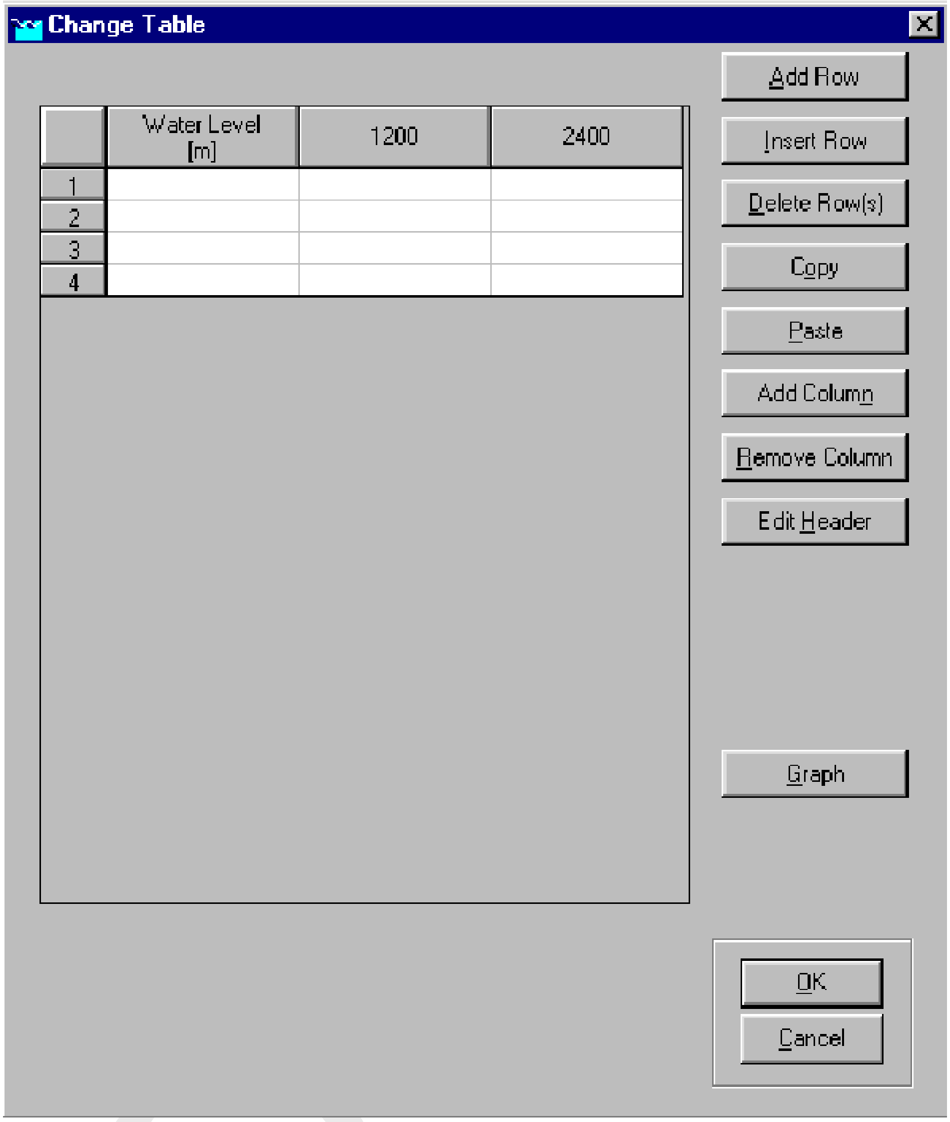


Figure 5.254: *Locations for which the friction values are known, added to the table.*

5.6.14 Flow - Cross Section node (Round type)

Cross-sections of the Round type are closed profiles using analytical formulae for computing lumped conveyance as function of water depth.

The input screen looks as follows:

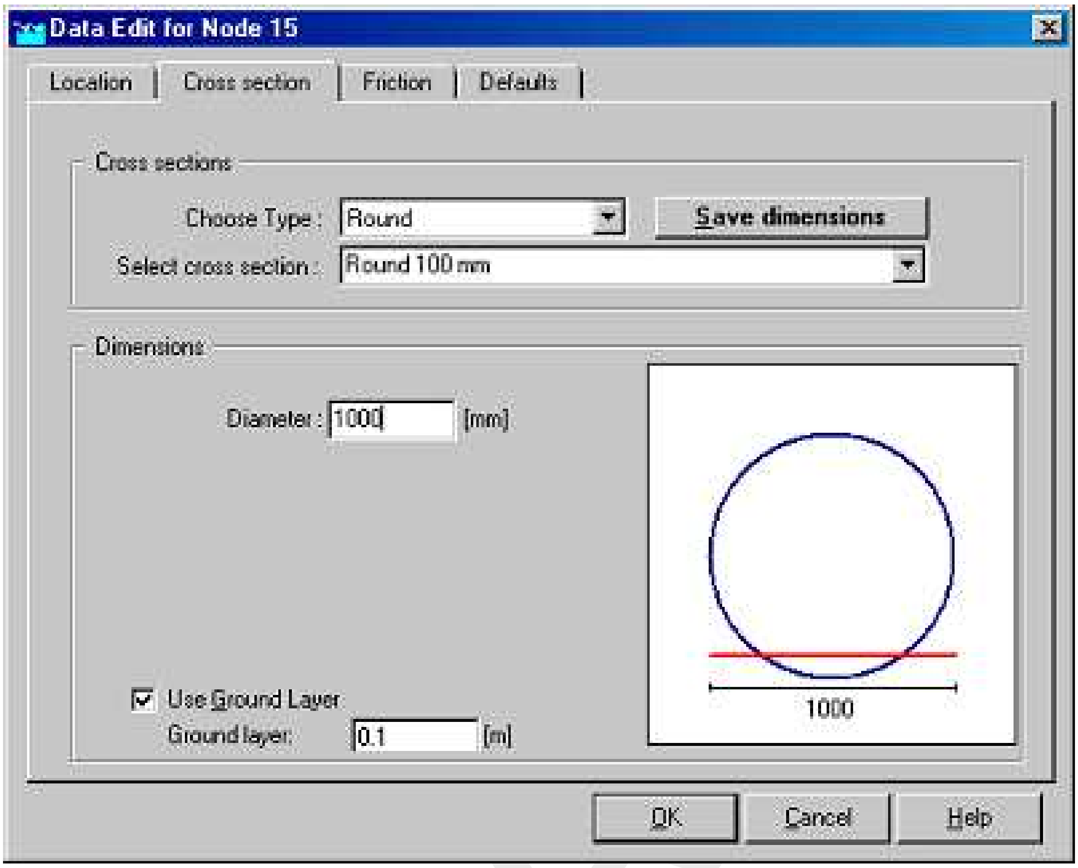


***Figure 5.255: Data Edit**, Cross section tab, Round type*

In the right-lower corner of the screen, an impression of the cross section shape that you enter will be plotted. The dimensions that you filled in will be plotted next to it.

Create a cross section definition:
To start creating a cross section definition, do the following:

1. choose the cross section of the "Round" type in the 'Choose Type' box;
2. Create a new definition by typing a name for it in the "select cross section" field
3. Click the button *Define Dimension*;
4. Enter the desired value in the parameter fields "diameter";
5. Optionally, turn on the "use ground layer option" and enter a value;
6. Click the button *Save Dimensions*.

Parameters:
This cross section type requires the following parameters:

- Diameter: represents the diameter the cross section.
- Use Ground Layer: switch this on if a layer of sediment lies in the cross section. An individual roughness value may be given for the ground layer. See the 'Friction' tab for this.
- Ground layer: enter the thickness of the sediment layer that lies within the cross section. This thickness will be made visible in the image on the right side through a red line.

5.6.15 Flow - Cross Section node (Steel Cunette type)

Cross sections of the Steel Cunette type are closed profiles using the tabulated lumped conveyance approach (see section 6.1.20.1).

The input screen looks as follows:

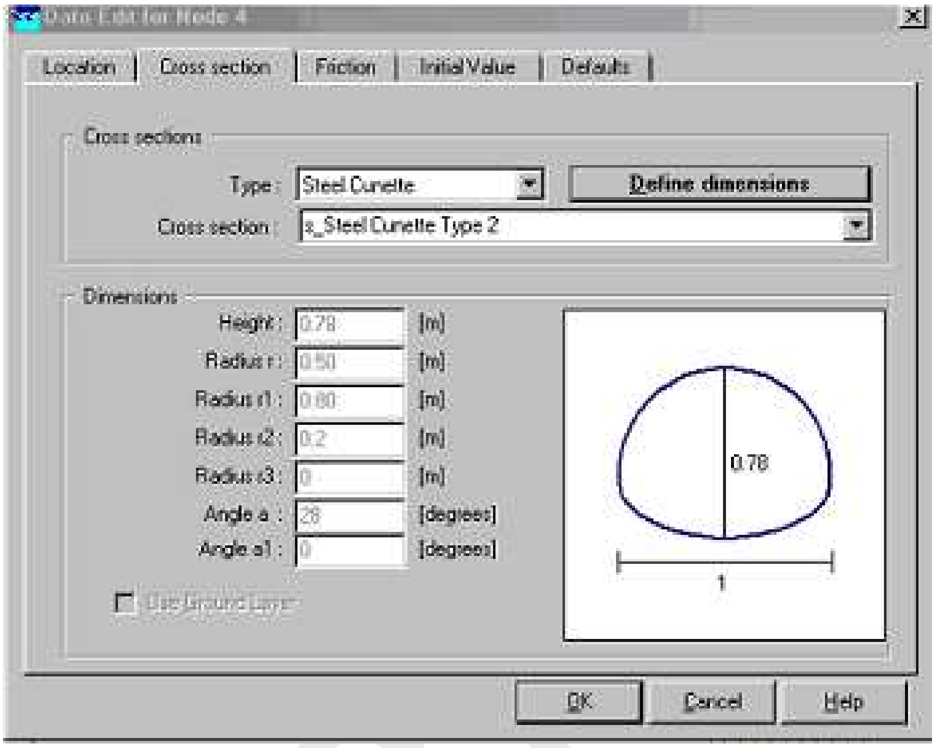


Figure 5.256: Steel Cunette type Cross section input screen.

In the right-lower corner of the screen, an impression of the cross-section shape that you enter will be plotted. The dimensions that you filled in will be plotted next to it.

Create a Steel Cunette cross section definition (see Figure 5.256):
To start creating a cross-section definition do the following:

- ◇ Choose the cross section of the “Steel Cunette” type in the ‘Choose Type’ box;
- ◇ Create a new definition by typing a name (e.g. Steel Cunette Type 2) for it in the “Cross section” field
- ◇ Click the button *Define Dimensions*;
- ◇ Enter the desired values in the parameter fields “Height, Radius r, Radius r1, Radius r2, Radius r3, Angle a, and Angle a1. To define a coherent set of these parameters, see the section Types of Steel Cunette Cross-section hereafter;
- ◇ Optionally, turn on the “use ground layer option” and enter a value;
- ◇ Click the button *Save Dimension*

Note:
That the prefix “s_” has been added in front of your defined Cross-section name, indicating that the Steel Cunette Cross-section is stored as Tabulated Cross-section, the entered parameters are not stored.

Editing a defined Steel Cunette cross-section (see Figure 5.257):

Once a Steel Cunette Cross-section has been saved, its dimensions can be changed in the following manner:

- ◇ Choose the cross section of the “Tabulated” type in the ‘Choose Type’ box;
- ◇ Open the scroll box available in the “Cross section” field and select the “s_Steel Cunette Type 2” cross-section;
- ◇ Click the button *Define Dimensions*.
- ◇ Optionally, turn on the “use ground layer option” and enter a value;
- ◇ Optionally, check the option “Different flow and storage width”;
- ◇ Click the button *Edit Table* for making changes in the [Level, Total width, Flow width] table,
- ◇ Click the button *Save Dimension* for saving your changes

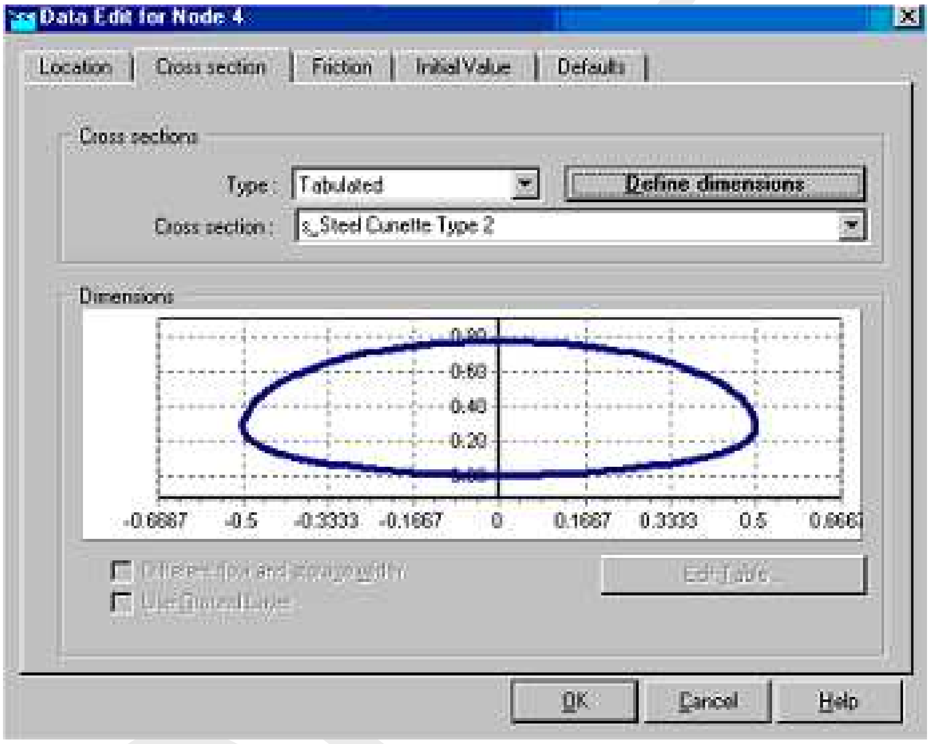


Figure 5.257: *Editing an existing Steel Cunette profile.*

Types of Steel Cunette Cross-sections:
Steel Cunette Cross-sections are symmetric w.r.t. the vertical axis through the centre of the profile. A Steel Cunette Cross-section may comprise of either three or four different circle segments.

Steel Cunette Cross-section comprising of three different circle segments:
A Steel Cunette comprising of three different circle segments is depicted in Figure 5.258. For obtaining a realistic cross-sectional profile the user-defined values for the parameters: Height, Angle a, Angle a1, Radius r, Radius r1, Radius r2 and Radius r3 should be mutually consistent (see Figure 5.258):

- ◇ Create the *first* circle segment (i.e. AB) with Radius r by starting at point A (located anywhere on the vertical symmetry axis) with centre point M (located on the vertical symmetry axis at a distance of Radius r below point A) and rotate over an angle of half the Angle a. In this way point B is obtained,
- ◇ Create the *second* circle segment (i.e. BC) with Radius r2 by starting at point B with centre point M2 (located at a distance of Radius r2 from point B on the line through points B and

M) and rotate over a particular angle in order to obtain point C,

- ◇ Create the *third* circle segment (i.e. CD) with Radius r1 by starting at point C with centre point M1 (located at the intersection of the vertical symmetry axis and the line through points C and M2) and rotate up to the intersection with the vertical symmetry axis and obtain point D.

Parameter Height is the vertical distance between points D and A. Parameters Angle a, Radius r, Radius r1 and Radius r2 follow from the three circle segments. Parameters Angle a1 and Radius r3 are equal to zero.

Remarks:

- ◇ In case the user-defined Angle a = 120 degrees; Angle a is used to compute the cross-sectional profile, including its actual height that is used instead of the user-defined height.
- ◇ In case Angle a $<$ 120 degrees, the user-defined Angle a is ignored and the actual applied Angle a is computed using the formula below (see Figure 5.258). Using this computed Angle a, the cross-sectional profile is computed including its actual height, that is used instead of the user-defined height.

$$\frac{1}{2}a = \arccos\left(\left(l_{M,M_1}^2 + l_{M,M_2}^2 - l_{M_1,M_2}^2\right)/\left(2l_{M,M_1}l_{M,M_2}\right)\right) \tag{5.3}$$

$$l_{M,M_1} = r_1 - H + r \tag{5.4}$$

$$l_{M,M_2} = r - r_2 \tag{5.5}$$

$$l_{M_1,M_2} = r_1 - r_2 \tag{5.6}$$

Steel Cunette Cross-section comprising of four different circle segments:

A Steel Cunette comprising of four different circle segments is depicted in Figure 5.259. For obtaining a realistic cross-sectional profile the user-defined values for the parameters: Height, Angle a, Angle a1, Radius r, Radius r1, Radius r2 and Radius r3 should be mutually consistent (see Figure 5.259):

- ◇ Create the *first* circle segment (i.e. AB) with Radius r by starting at point A (located anywhere on the vertical symmetry axis) with centre point M (located on the vertical symmetry axis at a distance of Radius r below point A) and rotate over an angle of half the Angle a. In this way point B is obtained,
- ◇ Create the *second* circle segment (i.e. BC) with Radius r3 by starting at point B with centre point M3 (located at a distance of Radius r3 from point B on the line through points B and M) and rotate over Angle a1 in order to obtain point C.
- ◇ Create the *third* circle segment (i.e. CD) with Radius r2 by starting at point C with centre point M2 (located at a distance of Radius r2 from point C on the line through points C and M3) and rotate over a particular angle in order to obtain point D,
- ◇ Create the *fourth* circle segment (i.e. DE) with Radius r1 by starting at point D with centre point M1 (located at the intersection of the vertical symmetry axis and the line through points D and M2) and rotate up to the intersection with the vertical symmetry axis and obtain point E.

Parameter Height is the vertical distance between points E and A. Parameters Angle a, Angle a1, Radius r, Radius r1, Radius r2 and Radius r3 follow from the four circle segments.

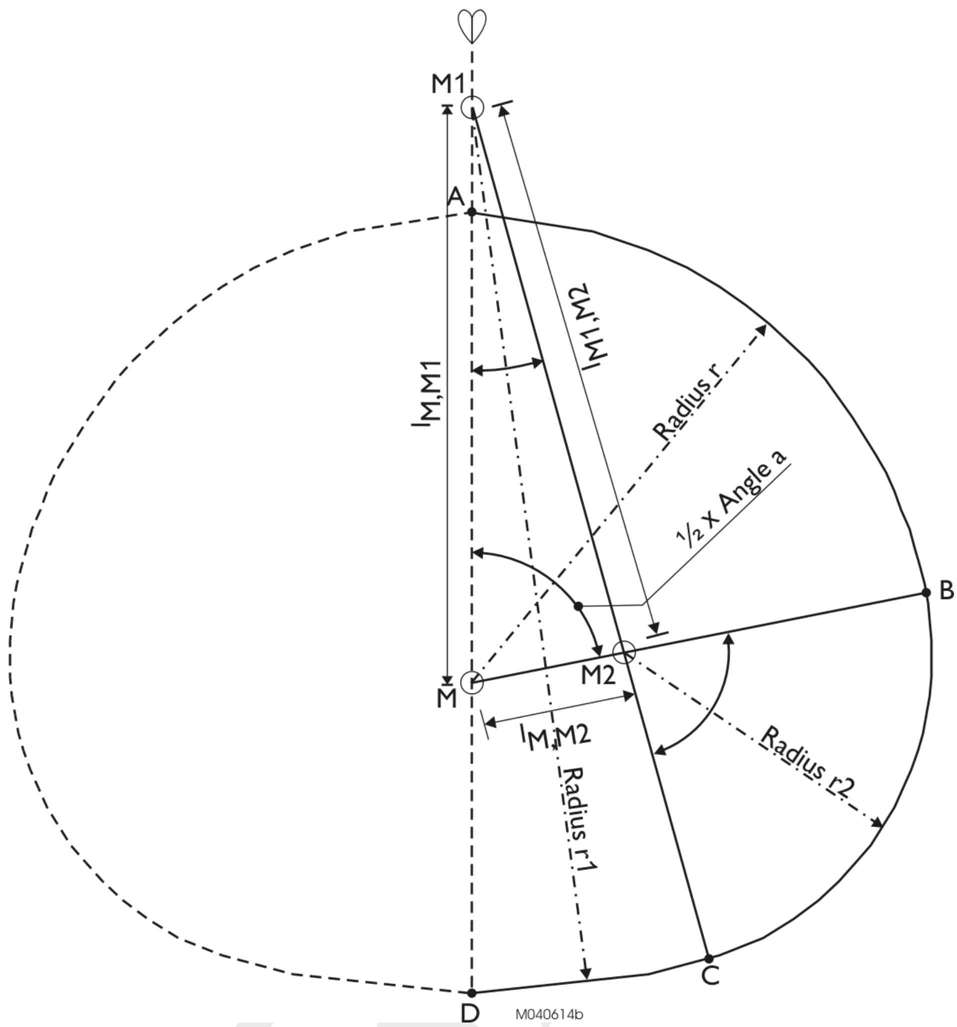


Figure 5.258: *Construction of a Steel Cunette Cross-section comprising of three circle Segments (i.e. Type 3, see Table 5.6 for examples of Steel Cunette Cross-sections)*

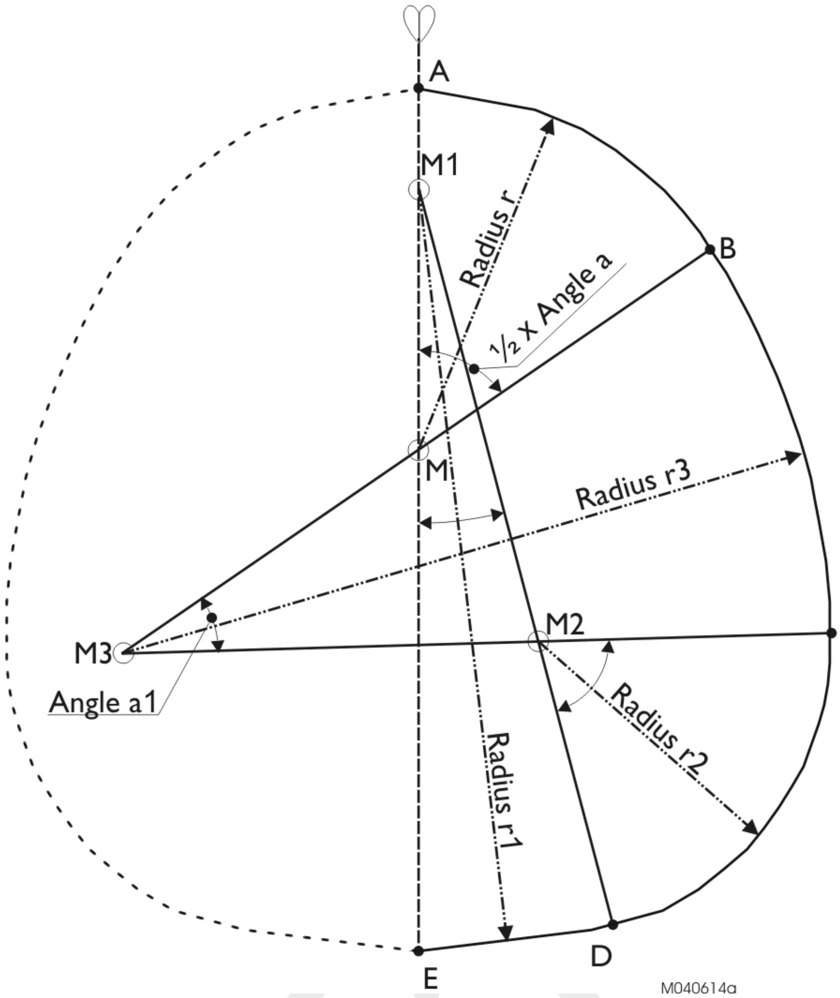


Figure 5.259: *Construction of a Steel Cunette Cross-section comprising of four circle Segments (i.e. Type 4, see Table 5.6 for examples of Steel Cunette Cross-sections)*

Examples of Steel Cunette Cross-sections:
In Table 5.6 four examples of Steel Cunette Cross-sections are given. Type 1 to 3 are based on three circle segments only, while Type 4 is based on four circle segments. For Type 1 the actual height is smaller than the user-defined height.

Table 5.6: *Examples of Steel Cunette Cross-sections*

Parameter	**Type 1** *(three circles)*	**Type 2** *(three circles)*	**Type 3** *(three circles)*	**Type 4** *(four circles)*
Height $[m]$	0.55	0.78	1.55	2.01
Radius r $[m]$	0.35	0.5	0.93	0.84
Radius r1 $[m]$	0.98	0.8	1.72	1.79
Radius r2 $[m]$	0.11	0.2	0.63	0.7
Radius r3 $[m]$	0	0	0	1.69

Angle a (deg)	180	28	159	112
Angle a1 (deg)	0	0	0	32

5.6.16 Flow - Cross Section node (Trapezium type)

Cross-sections of the Trapezium type are open profiles using the tabulated lumped conveyance approach (see section 6.1.20.1).

The input screen looks as follows:

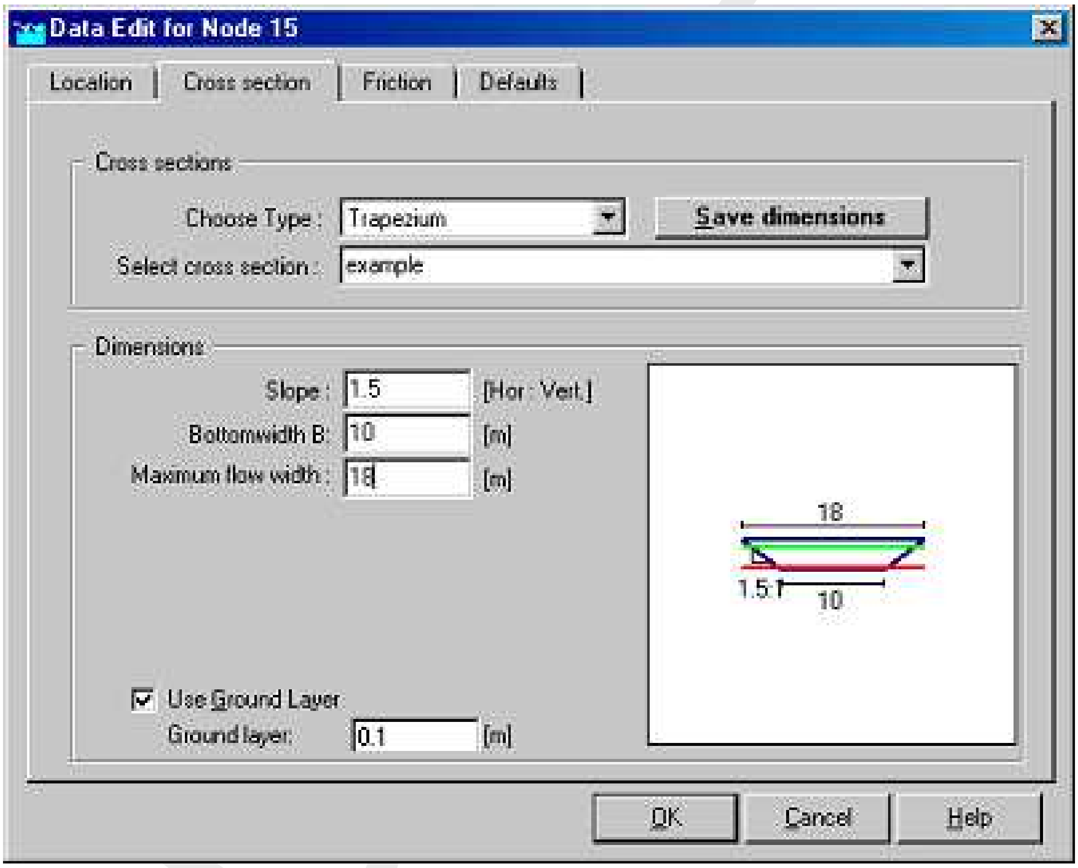


***Figure 5.260: Data Edit**, Cross section tab, Trapezium*

In the right-lower corner of the screen, an impression of the cross section shape that you enter will be plotted. The dimensions that you filled in will be plotted next to it.

Create a cross section definition:
To start creating a cross section definition, do the following:

- ◇ Choose the cross section of the "Trapezium" type in the ’Choose Type’ box.
- ◇ Create a new definition by typing a name for it in the "select cross section" field.
- ◇ Click the button *Define Dimensions*.
- ◇ Enter the desired values in the parameter fields "Slope", "Bottom Width" and "Maximum Flow width".
- ◇ Optionally, turn on the "use ground layer option" and enter a value.
- ◇ Click the button *Save Dimensions*.

Parameters:
This cross section type requires the following parameters:

Slope	represents the side slopes of the channel or pipe. Note that this value represents Horizontal/Vertical. Thus values $<$ 0 represent **steep** side slopes.
Bottom width B	represents the with at the bed level of the cross section;
Maximum Flow width	the final term that is needed to define the shape of a trapezium, though not really a physical realistic parameter in case of Cross Sections. Make sure that you choose a maximum flow width so that the channel's shape is correctly described from bed level to surface level. The surface level that you filled in on the "location" tab is visible here as a green line. It is best if the top of the trapezium lies higher than that line.
Use Ground Layer	switch on if a layer of sediment lies in the cross section. An individual roughness value may be given for the ground layer. See the 'Friction' tab for this.
Ground layer	enter the thickness of the sediment layer that lies within the cross section. This thickness will be made visible in the image on the right side through a red line.

Connecting the Rainfall-Runoff module to the 1DFLOW module
You can connect a SOBEK Rainfall-Runoff module to a SOBEK Channel Flow or River Flow schematisation by using either of the following node types:

The Flow-RR Connection on Channel is an object that should be placed on an existing Flow branch. A Flow-RR Connection on Flow Connection node is an equivalent for a normal Flow Connection node. That node type should therefore form the start or the end of a branch.

To both node types, an RR schematisation can be connected. The nodes will then form the boundary for the Rainfall-Runoff schematisation.

Tips 'n Tricks
Calculate water levels on other model objects, such as cross sections or structures
Besides the node type "calculation point" it is also possible to use other model objects for the calculation of water levels. One can, for instance, use cross sections as if they were calculation points. For every cross section in the model, the resulting water level will then be written to the result-files.

Here's how to include model objects such as cross sections in the calculation grid:

- ◇ Go to the Edit Network mode of SOBEK;
- ◇ Open the "Branch" toolbar and click the "Calculation Grid All branches" button
- ◇ The following screen will then pop up:

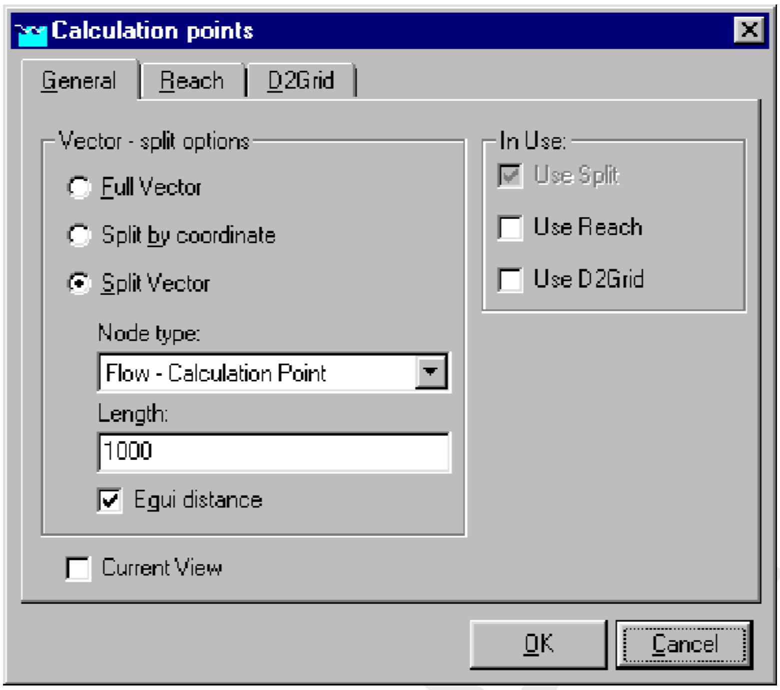


***Figure 5.261: Calculation points**, General tab*

◇ Activate the "Use Branch" option and go to the "Branch" tab:

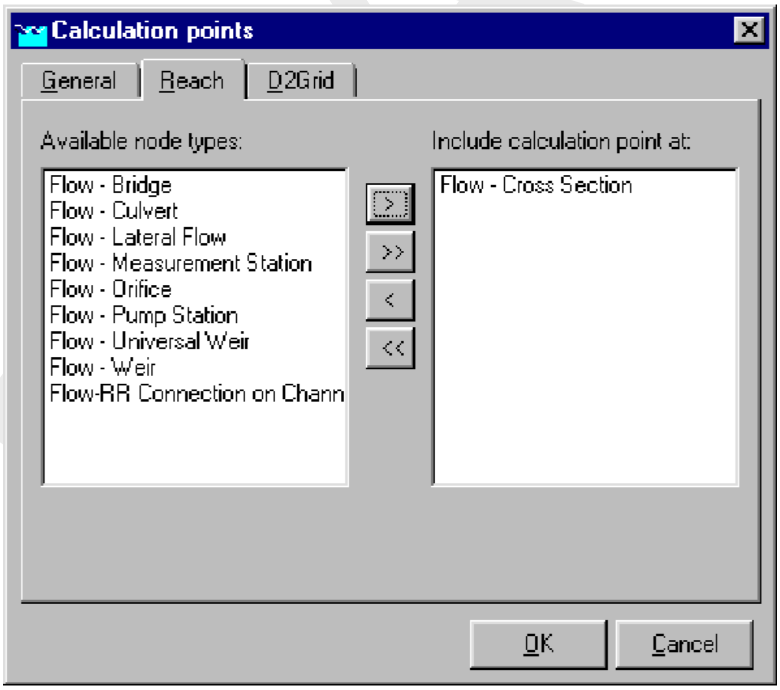


***Figure 5.262: Calculation points**, Branch tab*

◇ Move the objects that you want to use as calculation points to the right side by using the buttons in the middle of the screen.

◇ Click *OK*

◇ After you run the model, output data will also be available on the type of objects you selected above.

IMPORTANT NOTE: the type of objects you chose to use as calculation points will probably NOT be visible in the "Results in Maps" yet. Make them visible by clicking "Options" - "Network Options" - "Nodes" , selecting the node type, and activating the "visible" checkbox".

5.7 SOBEK-Urban 1DFLOW (Sewer Flow)

5.7.1 Features SOBEK-Urban 1DFLOW

- ◇ Uses the complete de Saint Venant Equations, thus including backwater and transient flow phenomena
- ◇ Models a wide variety of cross sections and manhole shapes (including user-defined ones) and allows you to build up your own cross section and manhole database
- ◇ Specially designed to handle large and complex sewer networks on an ordinary PC, where the computation time is only linear with the size of the network and independent of its complexity
- ◇ Has an automatic drying and pressurised procedure and handles real super critical flow and is always 100% mass conservative
- ◇ Self-selecting time step so your computer won't crash and accuracy is guaranteed
- ◇ All possible boundary conditions can be specified by you or are automatically applied
- ◇ You can specify virtually any type of hydraulic structure, such as single or multiple stage pumps, weirs of any shape, rectangular and circular gates, culverts and basins. All structures handle free, submerged and transient flow conditions
- ◇ Real-time control options, including PID control, are available for all structures and is ideal for complex centralised control systems using rainfall predictions
- ◇ Sediment transport computation shows where sediment might be deposited; interfaces completely with the SOBEK-Rural product line to provide an integrated model of the urban water system and its environment

Water flow

The Water Flow module of SOBEK-Urban allows you to analyse the hydrodynamic behaviour of a sewer network under different hydrological conditions and different management operation strategies. An example of an application is the analysis of the management of a sewer network discharging the inflow of runoff under high rainfall conditions. Such an application focuses on preventing high water levels at specific locations by taking measures such as creating additional storage, increasing the discharge capacity of the network, or adaptation of the operation rules.

In SOBEK-Urban Water Flow sewer systems can be modelled by the following branch elements:

- ◇ Flow-Pipe ;
- ◇ Flow-Pipe with Runoff ;
- ◇ Flow-Pipe with Infiltration ;
- ◇ Flow-Flow Measurement Pipe ;
- ◇ Flow-Internal Weir ;
- ◇ Flow-Internal Orifice ;
- ◇ Flow-Internal Pump Station ;

In SOBEK-Urban Water Flow sewer systems can be modelled by the following node elements:

- ◇ Flow-Manhole;
- ◇ Flow-Manhole with Level Measurement ;
- ◇ Flow-Manhole with Runoff;

◇ Flow-Manhole with Lateral Flow;
◇ Flow-External Weir ;
◇ Flow-External Orifice ;
◇ Flow-External Pump Station ;

With these node and branch types a wide range of networks can be represented. Consult the Technical Reference for detailed information about the topics mentioned.

5.7.2 Flow - Pipe with Infiltration

23, Flow - Pipe with Infiltration

A pipe with infiltration refers to a drainage pipe fully located inside a trench (see Figure 5.263). Although the drainage pipe is porous it still can get pressurized. The trench is filled with material (for instance gravel) having a certain porosity.

A pipe with infiltration allows for the exchange of water in the pipe towards the trench (positive sign) and vice versa as well as for the exchange of water in the trench towards the groundwater (negative sign) and vice versa. Hence, a pipe with infiltration can be seen as a pipe having a *diffusive lateral discharge* option (e.g. infiltration of water in the pipe towards the trench and exfiltration of water in the trench towards the pipe). For more information on the infiltration and exfiltration process, reference is made to Ellis and Bertrand-Krajewski (2010), Rutsch *et al.* (2008) and Karpf *et al.* (2008). For more information on the computational procedure of a Pipe with Infiltration reference is made to Section 6.1.11.5.

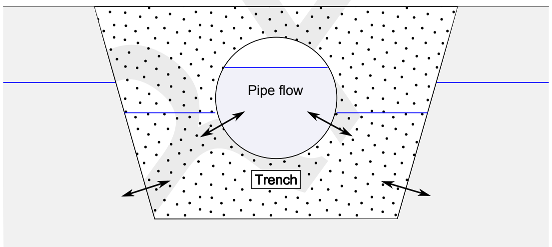


Figure 5.263: *Pipe with Infiltration (e.g. a drainage pipe located in a trench); Allowing for the exchange of water in the pipe towards the trench (positive sign) and vice versa; Allowing for the exchange of water from the trench towards the groundwater (negative sign) and vice versa*

5.7.2.1 Input screens of the Flow - Pipe with Infiltration

The Pipe with Infiltration input screens comprise of:

◇ **The Location Tab** (see Figure 5.264): On this tab, the ID and Name of the Pipe with Infiltration are shown. Furthermore, the invert level at the beginning of the pipe (e.g. upstream side) and the invert level at the end of the pipe (e.g. downstream side) can be defined.

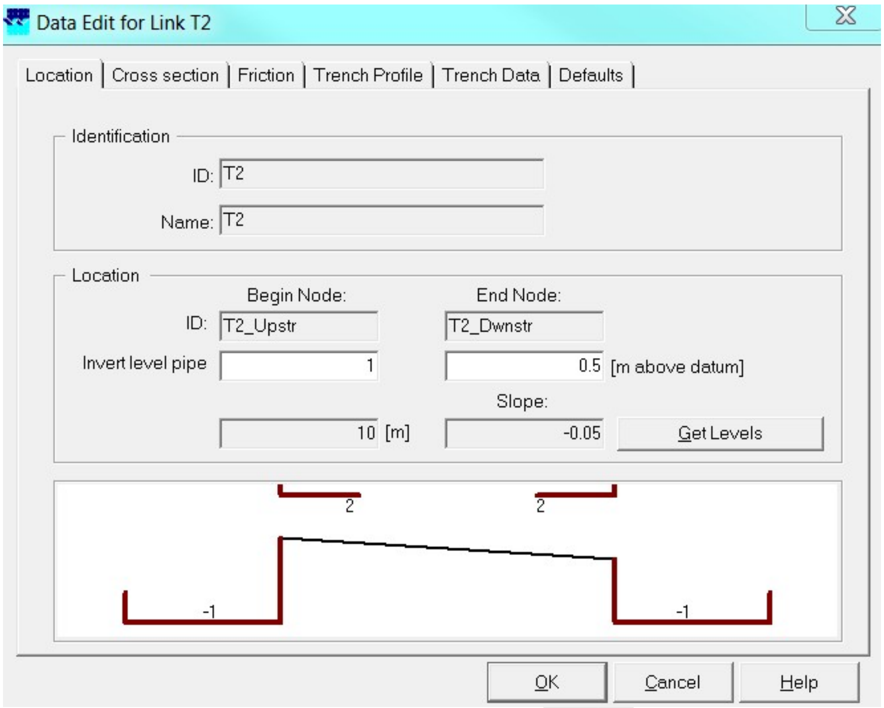


Figure 5.264: *The Location Tab of the Pipe with Infiltration*

◇ **The Cross-section Tab** (see Figure 5.265): On this tab, the pipe cross-section can be defined. Notice that in the "Type scroll box" only a Round, Egg-Shape or Rectangle **closed** cross-section type can be selected. Provide a name in the window, standing next to Cross sections, then click on the <Define dimensions> button and enter the cross-section dimensions. Now click on the <Save dimensions> button to save your cross-sectional profile.

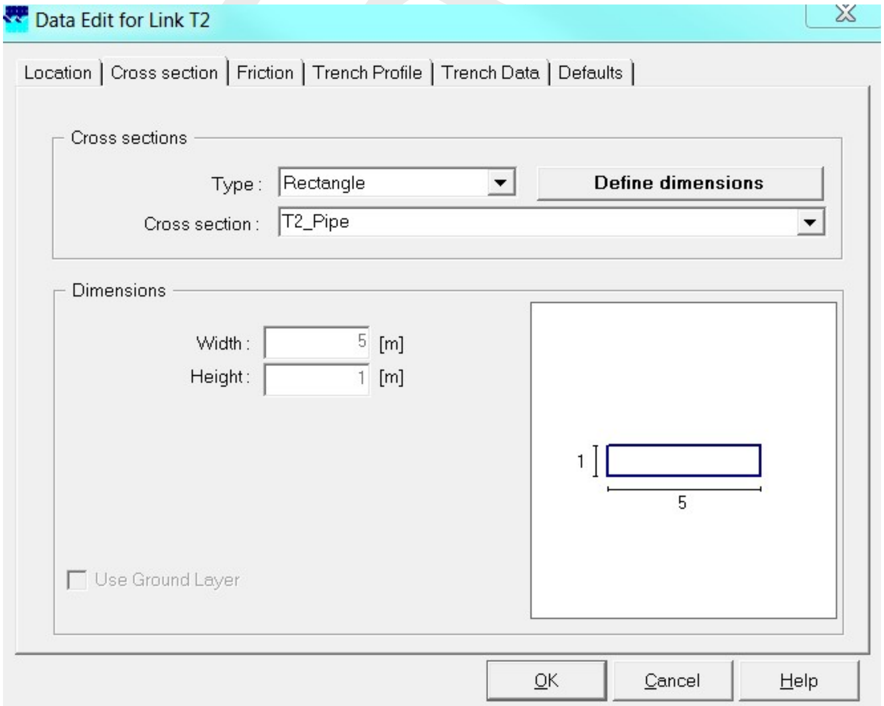


Figure 5.265: *The Cross Section Tab of the Pipe with Infiltration*

◇ **The Friction Tab** (see Figure 5.266): On this tab, the friction formula and friction value to applied for computing the 1D flow through the Pipe part of the Pipe with Infiltration can be defined.

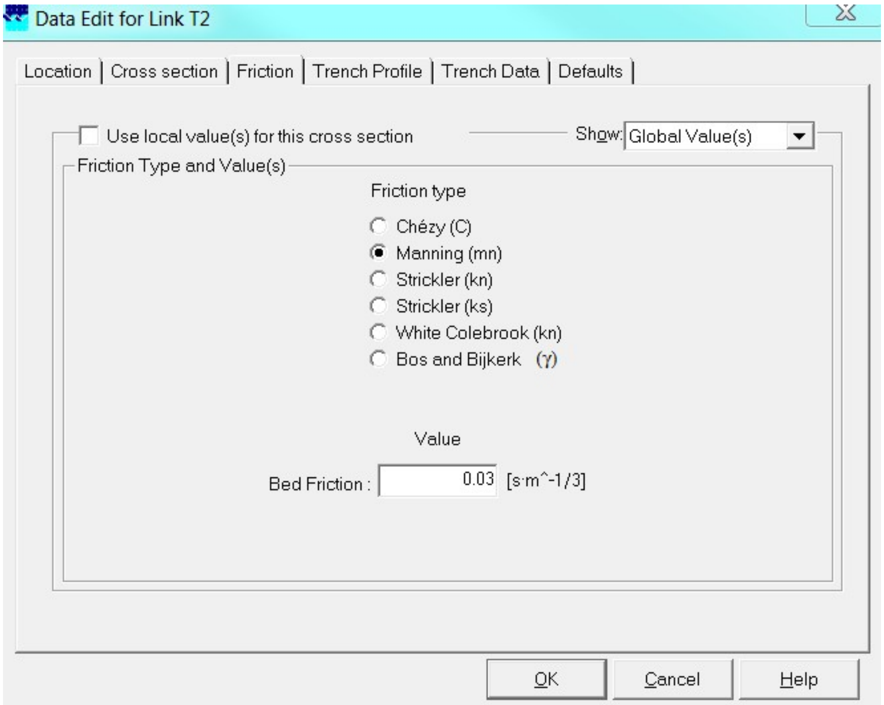


Figure 5.266: *The Friction Tab of the Pipe with Infiltration*

◇ **The Trench Profile Tab** (see Figure 5.267): On this tab, the trench profile can be defined. In the "Type scroll box" only a Trapezium or a Rectangle trench profile can be selected. Please notice that both the trapezium and rectangle trench profiles are **open** profiles. Hence, the water level in the trench may rise above the highest elevation, defined for such trench profile. Dimensions of the trench profile can be specified in the same manner as explained for the Cross-section Tab. In the picture, shown on the Trench profile Tab, it can be verified if the cross-sectional area of the pipe **is fully located inside** the trench profile. If this is not the case, a simulation will be terminated with a message in the SOBEK.log file.

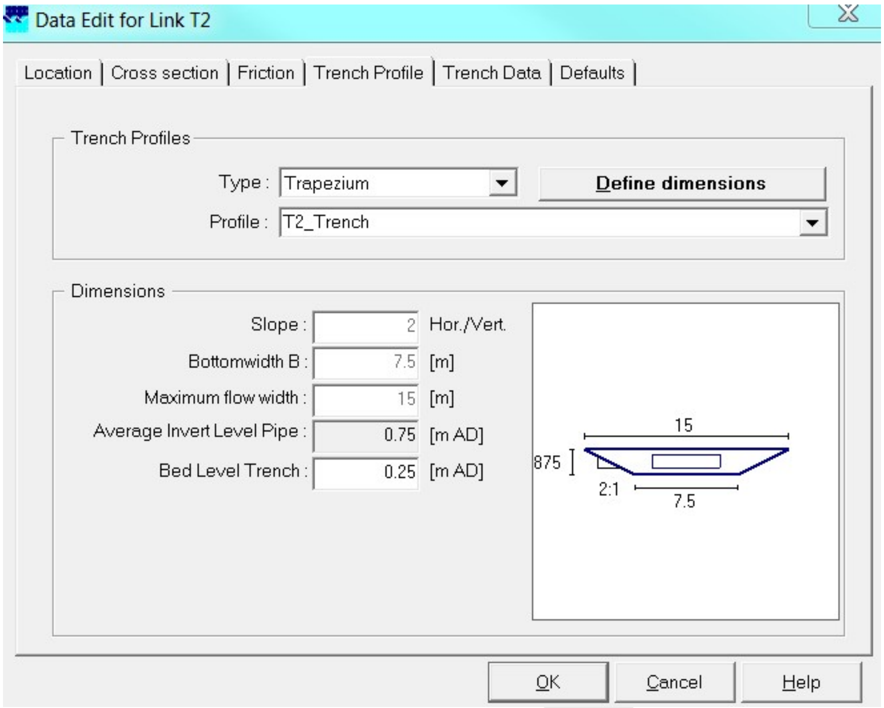


Figure 5.267: The Trench Profile Tab of the Pipe with Infiltration

◇ **The Trench Data Tab** (see Figure 5.267): On this tab in the trench window, the resistance of the interface between the Pipe and Trench, the porosity of the trench material (notice that porosity = 1, for trench water level above the top level of the trench or water on street) and the initial water level in the trench (e.g. user-defined value, equal to initial 1D water level or equal to groundwater level) can be defined. In the Groundwater window, the permeability of the ground/soil (e.g. permeable or impermeable), the resistance of the interface between the Trench and the Groundwater (only possible if the option permeable is selected) and groundwater levels (only possible if the option permeable is selected) can be defined. For more information on the background of the various input parameters, reference is made to Section 6.1.11.5.

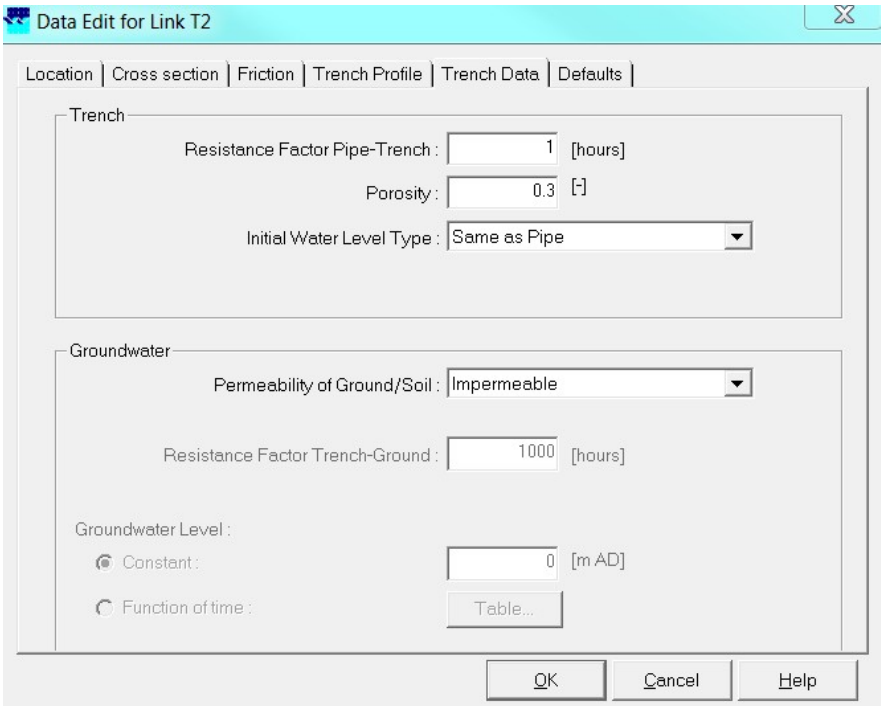


Figure 5.268: *The Trench Data Tab of the Pipe with Infiltration*

5.7.2.2 Additional Output available for Flow - Pipe with Infiltration

The "Settings for 1DFLOW module" Form (see Figure 5.269) becomes available after clicking on the 1D Flow <Edit> buttton in the Settings Task block. Additional output parameters for a Flow - Pipe with Infiltration become available by checking "Results from Infiltration Pipes" in the output parameter-box, located on the Output options Tab (see Figure 5.269). After a computation, these additional output parameters can be viewed at a Flow - Pipe with Infiltration in the "Results in Maps" Task block or under "Results at Branch Segments" in the "Results in Charts" Task block. The additional "Flow - Pipe with Infiltration" output parameters comprise of:

- ◇ **Groundwater level at trench**: The groundwater level boundary condition applied for the Flow - Pipe with Infiltration. [m above datum]
- ◇ **Groundw. to trench lat. flow**: Exchange of water between the external groundwater reservoir and the trench. Positive if water flows from the groundwater reservoir towards the trench. Negative if water flows from the trench towards the groundwater reservoir. $[m^3/s]$
- ◇ **Pipe to trench lat. flow**: Exchange of water between the pipe and the trench. Positive if water flows from the pipe towards the trench. Negative if water flows from the trench towards the pipe. $[m^3/s]$
- ◇ **Volume in trench**: The volume of water stored in the trench $[m^3]$.
- ◇ **Water depth in trench**: The water depth in the trench of the Flow - Pipe with Infiltration. $[m]$
- ◇ **Water level in trench**: The water level in the trench of the Flow - Pipe with Infiltration. [m above datum]

Note: If there is no water in the trench, a water level equal to the bed level of the trench is provided as output.

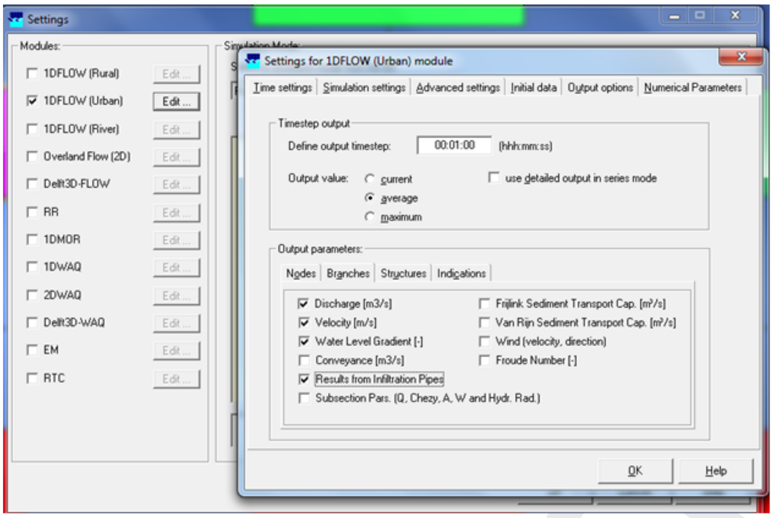


Figure 5.269: *The "Settings for 1DFLOW module Form" available in the Settings Taskblock*

5.7.3 Storage graph

Especially for sewer systems it is useful to determine the storage in the pipe system. Therefore you can use the Storage Graph tool. This tool can be used from the Tools menu in the network editor.

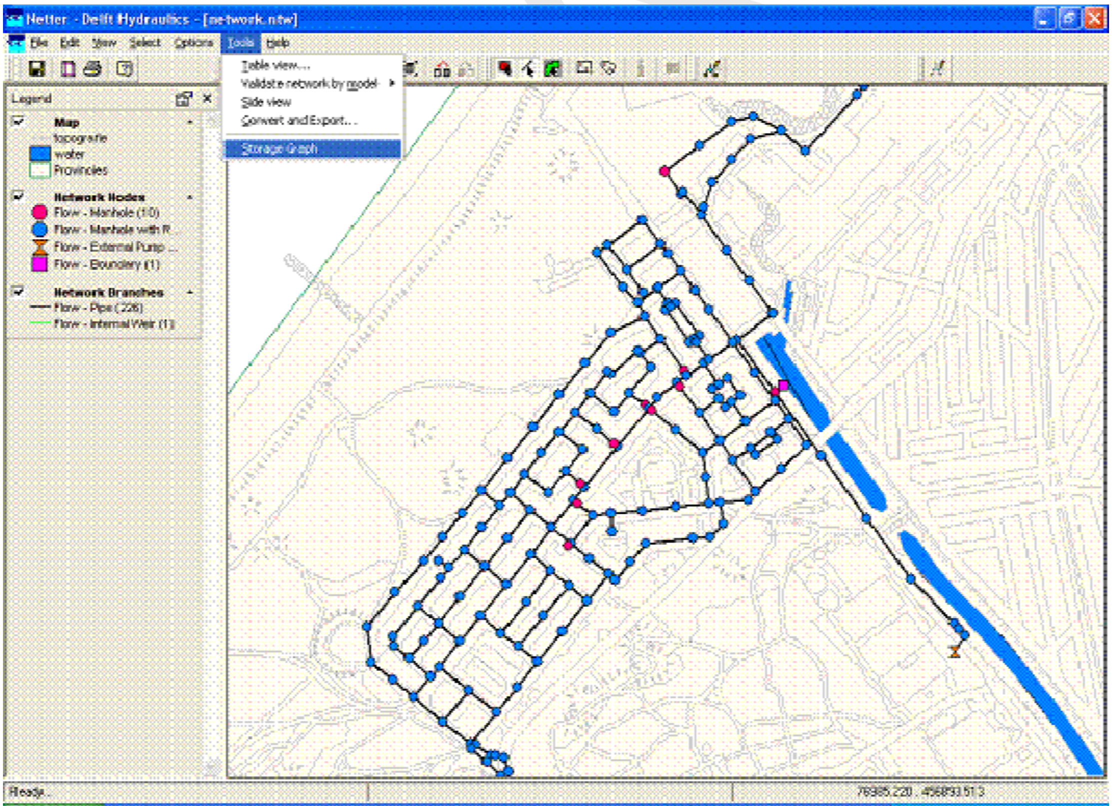


Figure 5.270: *To start the storage graph*

Once the program has been started the user has the option to determine the storage for the whole network, or for a part of the network, that must be selected already.

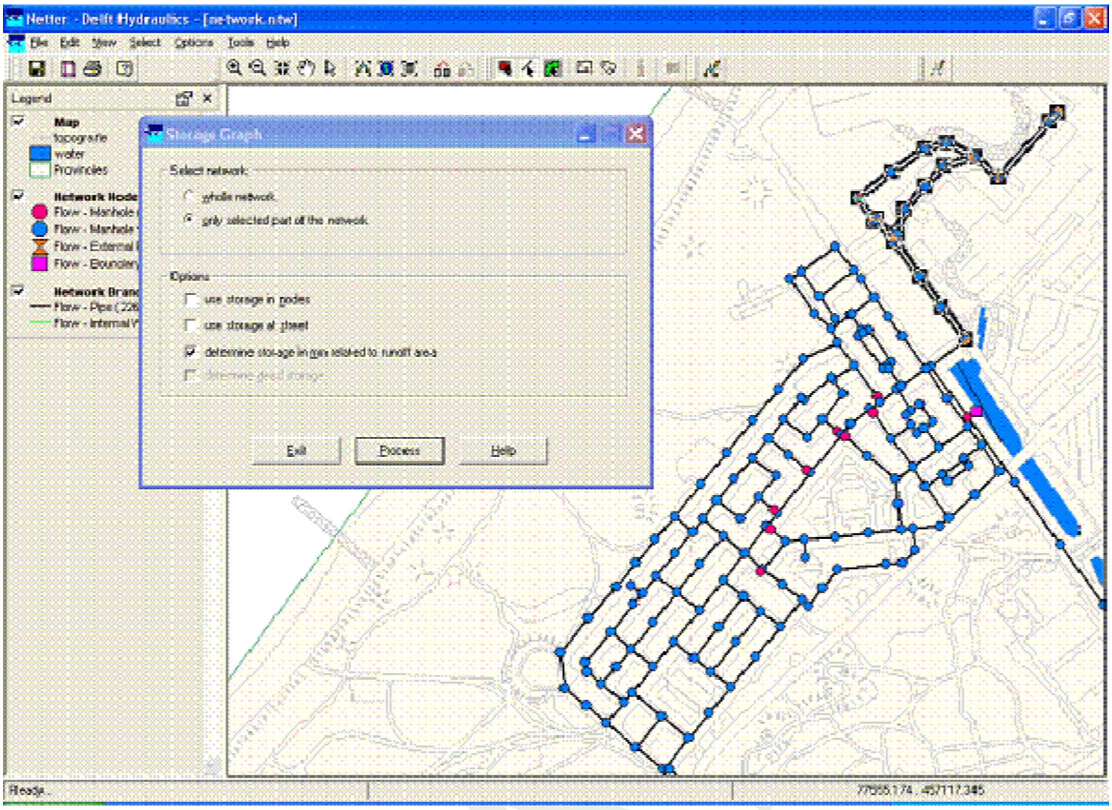


Figure 5.271: *Definition of storage graph options*

The next options are available (see Figure 5.271:

- *Use storage in nodes* If selected, both the storage in the pipes and the nodes are taken into account. If not selected, only the storage in the pipes are included.
- *Use the storage at the street*
- *Determine storage in mm related to the defined runoff area* If you select this option, the storage is not only determined in cubic metres, but also related to the runoff area that is defined in the selected network.

To perform the calculation of the storage, you have to press the *Process* button. The calculation might take a few seconds and finally the graph pops up as shown in Figure 5.272

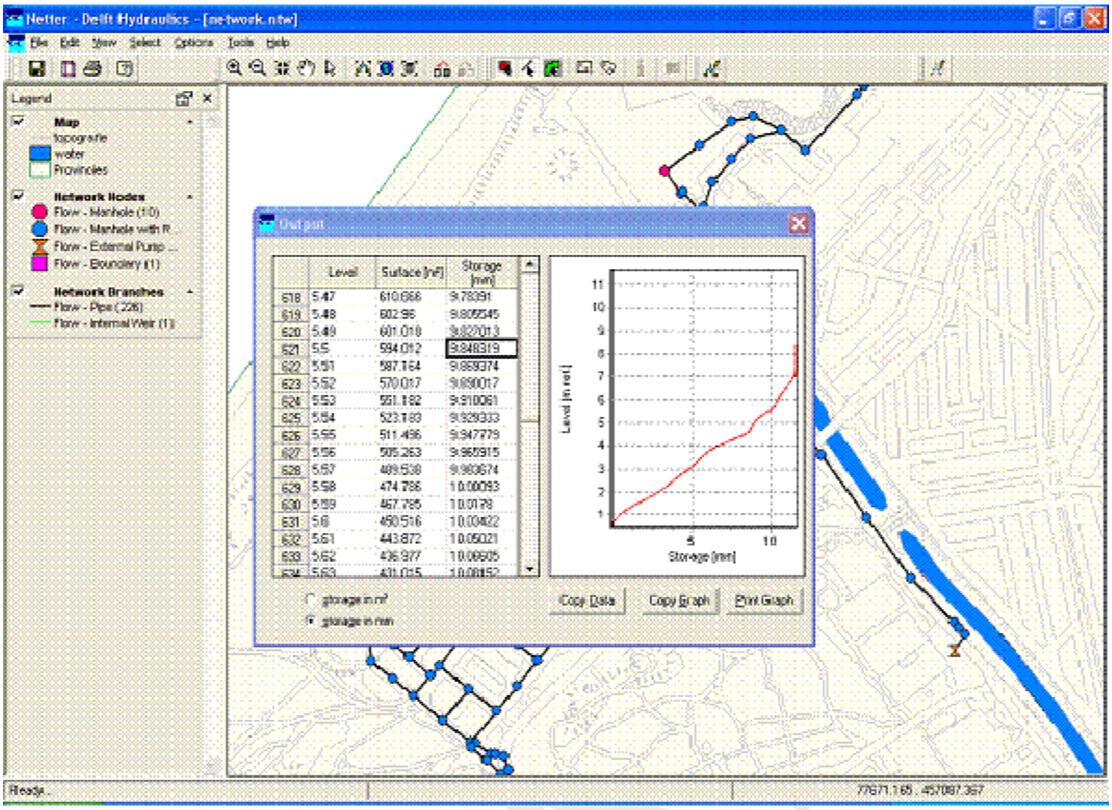


Figure 5.272: *Storage graph result*

The left part of the window contains a table with the surface area $[m^2]$ and storage (m^3) as function of the level. The table starts at the lowest part in the network and runs with a step of 1 cm to the highest level.

The right part of the window shows the graph. It is possible to print the graph and to copy the graph to the clipboard. Next to that also the contents of the table can be copied to the clipboard by pressing the button *Copy Data*.

5.7.4 Coupling with other modules

Coupling a Sewer Flow manhole with the Overland Flow module (2D grid)
If you desire to create an interaction between the Sewer Flow module and the Overland Flow module, you may want to couple a Flow - Manhole with the 2D-grid. You can either choose to use a manhole of the type "reservoir" or the type "loss".

Manhole of the type "closed":
If a Flow-manhole of the type "closed" is placed on a 2D grid cell, there will be **no** exchange of water between the 2D grid and the manhole.

Manhole of the type *loss*:
If you choose place a Flow-Manhole of the type *loss* on a 2D grid cell, the street level for that manhole is overruled by the level of the connected 2D grid cell:

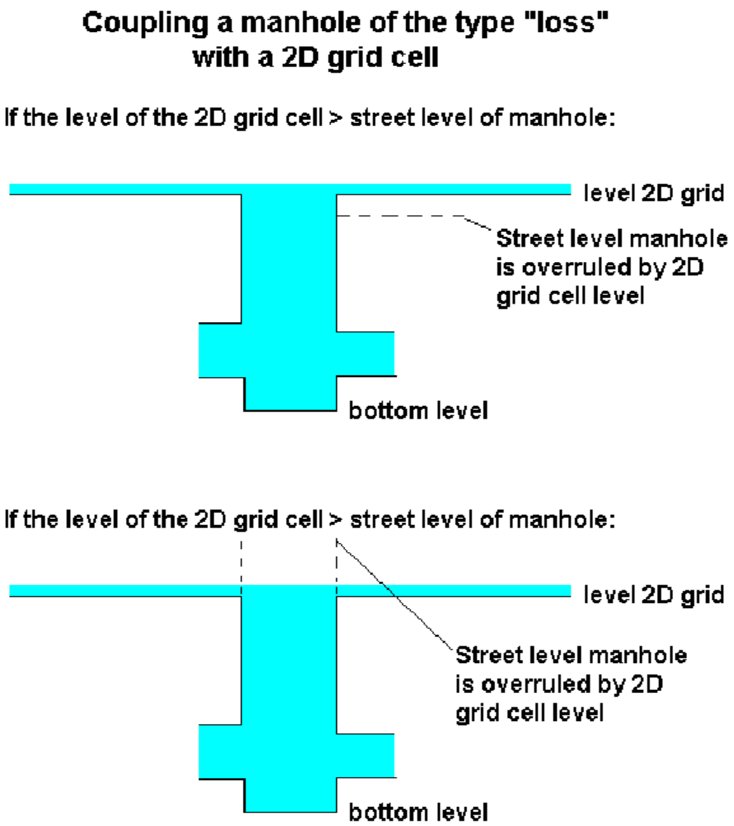


Figure 5.273: *Coupling a manhole ot the type* loss *with a 2D grid cell*

Manhole of the type *reservoir*:
If you choose a Flow-manhole of the type *reservoir*, and place it on top of a 2D grid cell, two situations may occur. See the picture below.

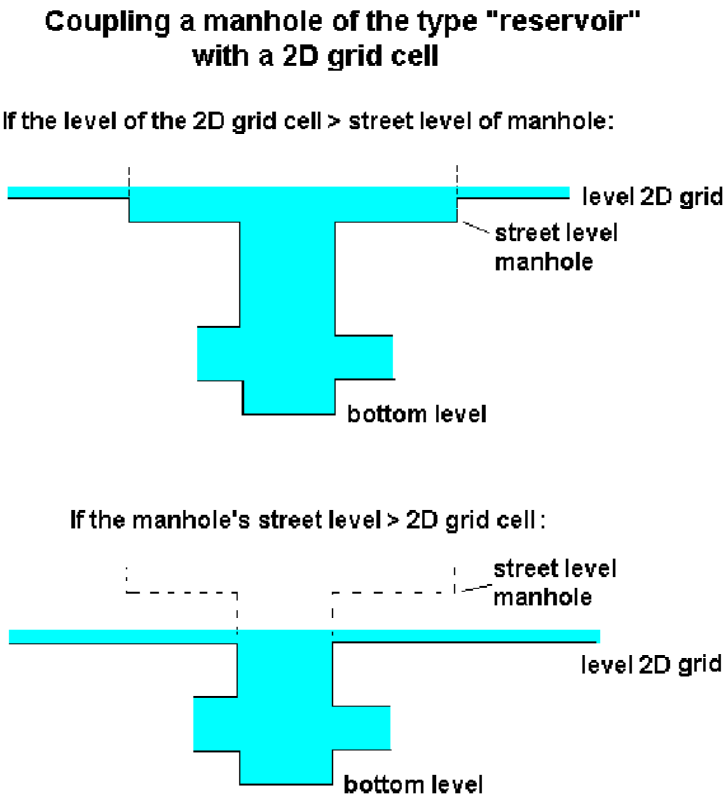


Figure 5.274: *Coupling a manhole ot the type* reservoir *with a 2D grid cell*

5.7.5 Connecting the Rainfall-Runoff module to the 1DFLOW module

You can connect a SOBEK Rainfall-Runoff module to a SOBEK Channel Flow or River Flow schematisation by using either of the following node types:

The Flow-RR Connection on Channel is an object that should be placed on an existing Flow branch. A Flow-RR Connection on Flow Connection node is an equivalent for a normal Flow Connection node. That node type should therefore form the start or the end of a branch.

To both node types, an RR schematisation can be connected. The nodes will then form the boundary for the Rainfall-Runoff schematisation.

5.8 River Flow controllers and triggers

In this section the so called River Flow controllers and triggers are discussed. These controllers and triggers can be assigned to compound structure members (excluding River Pump), a Database structure, a General structure, an Advanced weir and a River weir.

The Controllers tab:

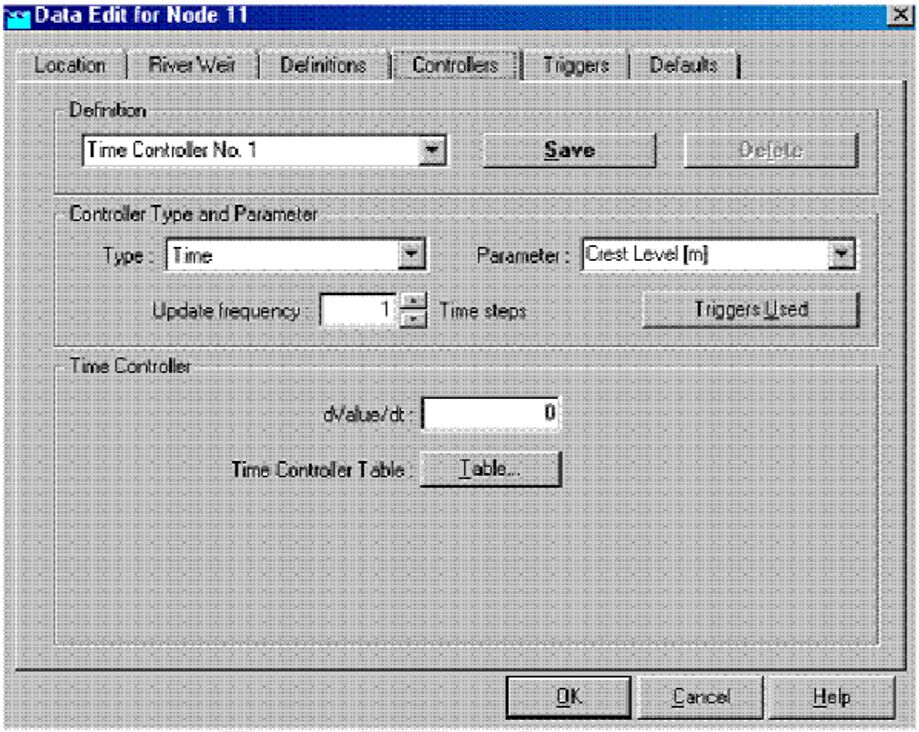


***Figure 5.275: Data Edit**, Controllers tab*

Each above mentioned structure can be operated by a combination of controllers (max. 4). There are six possible controllers that can be applied:

- ◇ Time Controller
- ◇ Hydraulic Controller
- ◇ Interval Controller
- ◇ PID Controller
- ◇ Relative from Time controller
- ◇ Relative from Value controller

A detailed description of the parameters for these controller types is given in the Technical Reference Manual. For further detailed explanation on controller functionalities, reference is made to Appendix River Flow controller options.

Defining a controller:A new controller can be defined as follows:

- ◇ Type an appropriate name for the controller in the "Definition" field.
- ◇ Click the button <Define>
- ◇ Choose the appropriate controller type from the "Type" drop-down box.
- ◇ Enter the parameter values for the chosen controller.

- ◇ If necessary, let the controller be activated and deactivated by a trigger. Do this by appointing the appropriate trigger on the <Triggers Used> button. Triggers can be defined on the "Triggers" tab.
- ◇ Don't forget to activate the controller. Do this on the tab "Structure"

Triggers:

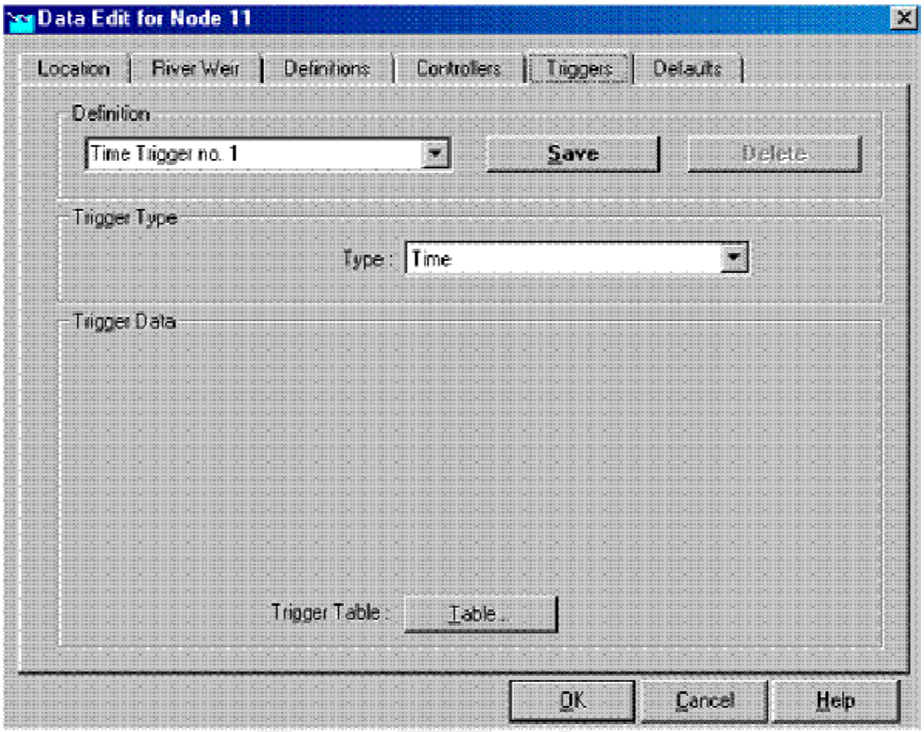


***Figure 5.276: Data Edit**, Triggers tab*

A certain controller can be activated or deactivated by using a trigger. A trigger can be programmed so that it is activated under certain circumstances. We distinguish three types of triggers:

- ◇ Time triggers: a time table defines when the trigger is active and when it is inactive.
- ◇ Hydraulic triggers: the trigger is activated or inactivated under specific hydraulic circumstances at a measurement location.
- ◇ Time & Hydraulic triggers: this refers to a combined of a Time trigger and Hydraulic trigger.

Defining a trigger:

- ◇ Type an appropriate name for the trigger in the "Definition" field.
- ◇ Click the button <Define>
- ◇ Choose the appropriate trigger type from the "Trigger Type" drop-down box.
- ◇ Enter the parameter values for the chosen trigger.

Don't forget to let your controller know that it should 'listen' to this trigger. Do this by clicking the <Triggers Used> button on the "Controllers" tab. For an example see Figure below, where you can see that Time Trigger no 1 is used to activate and de-activate Time Controller no 1. Note that clicking on the controller data button in the Figure below, provides information on the controller data of Time Controller no 1. Thereafter clicking one of the 'Trigger used' buttons (see two figures above) provides information on the triggers used (i.e. figure below is obtained again).

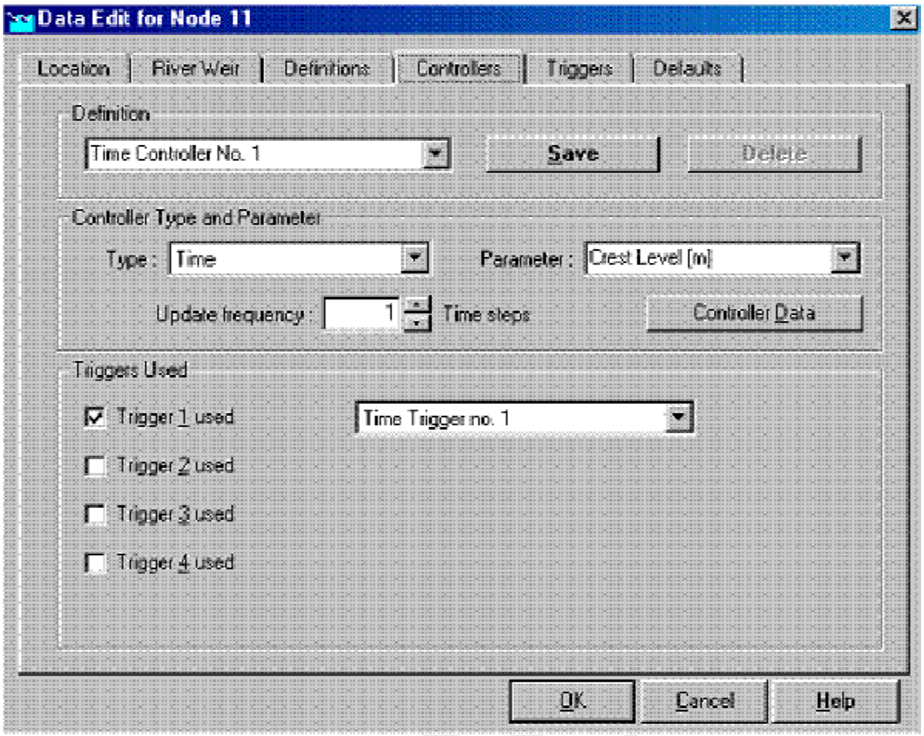


***Figure 5.277: Data Edit**, Controllers tab*

5.9 SOBEK-Rural/Urban/River Overland Flow (2D)

5.9.1 Introduction

In January 1998, RWS|DWW and WL | Delft Hydraulics branched an agreement to work together on a project to combine the functions of Delft-FLS and the SOBEK-Rural 1DFLOW module. The goal was to create a single computer model that could be used to model combined 1D and 2D flow for different scenarios, like for example a dike-bbranch. The agreement resulted in a number of projects which had as a goal to improve and expand the possibilities of the program. Many refinements were added, among which the concept of multiple grids.

The resulting functionality is known as '1D2D functionality. This functionality is available in case a 1DFLOW module and the Overland Flow (2D) module are selected.

Definitions
This chapter gives an overview of various terms used in this manual to describe the Overland Flow module.

ArcView
ArcView is a desktop-GIS application, that gives the user the possibility to solve GIS related problems in a very user-friendly environment.

Delft-FLS
Purely 2D hydrodynamical flood modelling system.

DEM
Digital Elevation Model. A DEM is a representation of terrain heights in grid format. Also called DTM (Digital Terrain Model)

GIS:
Geographic Information System. Geographically oriented database used to analyze and present spatially distributed data.

NETTER:
A GIS- tool from the Delft-Tools family. It is used within SOBEK to view and edit schematisations, and to visualize results from simulations.

Note: all units used in this manual and in SOBEK are metric (S.I. standard).

5.9.2 Viewing 2D Grid Info

Using the 2D Grid Info option, information can be obtained on:

- ◇ individual 2D grid cells, or
- ◇ on a path of 2D grid cells.

The user can add the information of various individual 2D grid cells to one and the same graph. A path of 2D grid cells can cross at any angle over a 2D grid. At present the path of 2D grid cells can not pass over nested grids (i.e. parent grid and child grid) in such way that only the active parent and child 2D grid cells are being selected.

Further on a distinction is to be made between static values and dynamic values. Static values refers to the values on the underlying 2D grid (for instance a bed elevation) or underlying map file (for instance maximum water depths). Dynamic values refers to time-dependent computation results (for instance incremental water depths).

How to make a 2D path:

Both in the ‘Schematisation’ and ‘Results in Maps’ Task blocks.

- ◇ In the main menu, click on Select/2D-Grid cell info
- ◇ Click on the 2D grid cell at which you like to start your 2D path (i.e. starting 2D grid cell).
- ◇ Note that in the ‘Path2D-Grid Info’ window, the column and row number of the starting 2D grid cell are given in the two boxes in front of the Path check-box.
- ◇ Click the Path check-box,
- ◇ While pressing on the Shift key, place the mouse-pointer on the 2D grid cell at which you like to end your 2D path (i.e. the end 2D grid cell). Releasing the Shift key will result in the selection of the end 2D grid cell. The column- and row number of the end 2D grid cell are now filled in the two boxes behind the Path check-box.
- ◇ Click on the <show> button in the ‘2D-Grid info’ window to visualise the 2D grid cells contained in your 2D Path.

Remark: In case you have more than 1(one) 2D grid (for instance in case of nested grids), the 2D grid cell of the 2D grid that lies on top will be selected. Using the *Next Grid* button the grid cell on the underlying 2D grid will be selected. Note that in this case the Grid ID will change.

How to retrieve 2D Grid Info:

- ◇ Under the ‘Schematisation’ Task block only information on static values of the 2D grid cells are available. *In case the 2D path option is not used*, the static value of the selected 2D grid cell is depicted on the one most last line in the '2D-Grid Info' window (Note that dynamic values are not available under the 'Schematisation Task' block). *In case a 2D path is selected*, the static values of the concerning 2D grid cells can be obtained by clicking the *Graph* button. Note that in the main menu of this graph the x-axis can be changed to either value (distances) or labels (column and row number of the 2D grid).

◇ Under the 'Results in Maps' Task block, information on both static- and dynamic values of the 2D grid cells are available. *In case the 2D path option is not used*, the static- and actual dynamic value of the selected 2D grid cell is depicted on the last two lines in the '2D-Grid Info' window. Please note that static values are displayed in grey. Clicking the most left *Graph* button, provides a graph of dynamic values of each selected 2D grid cell as function of time. Please note that by clicking on another 2D grid cell and sequentially clicking on this *Graph* button again, the user can add the dynamic results of various 2D grid cells in one and the same graph. *In case a 2D path was selected*, clicking on the most left *Graph* button results in a graph of the dynamic values of the starting 2D grid cell of the 2D path. Clicking the most right *Graph* button provides a graph, showing for all 2D grid cells lying on the 2D path, its static value as well as its dynamic values for a particular point-in-time.

Note:

1. the x-axis of this graph can be plotted as function of value (distance) or labels (column or row number);
2. dynamic values are only available in case they are selected beforehand (Click in main menu on File→Open Data→Depth Incremental (in the **Select Item** window));
3. using the 'View Data' window dynamic values can be added to the graph; and
4. using the *forward* and *backward* buttons in the 'View Data' window, the point-in-time for which the dynamic values are shown under the most right *Graph* button can be changed.

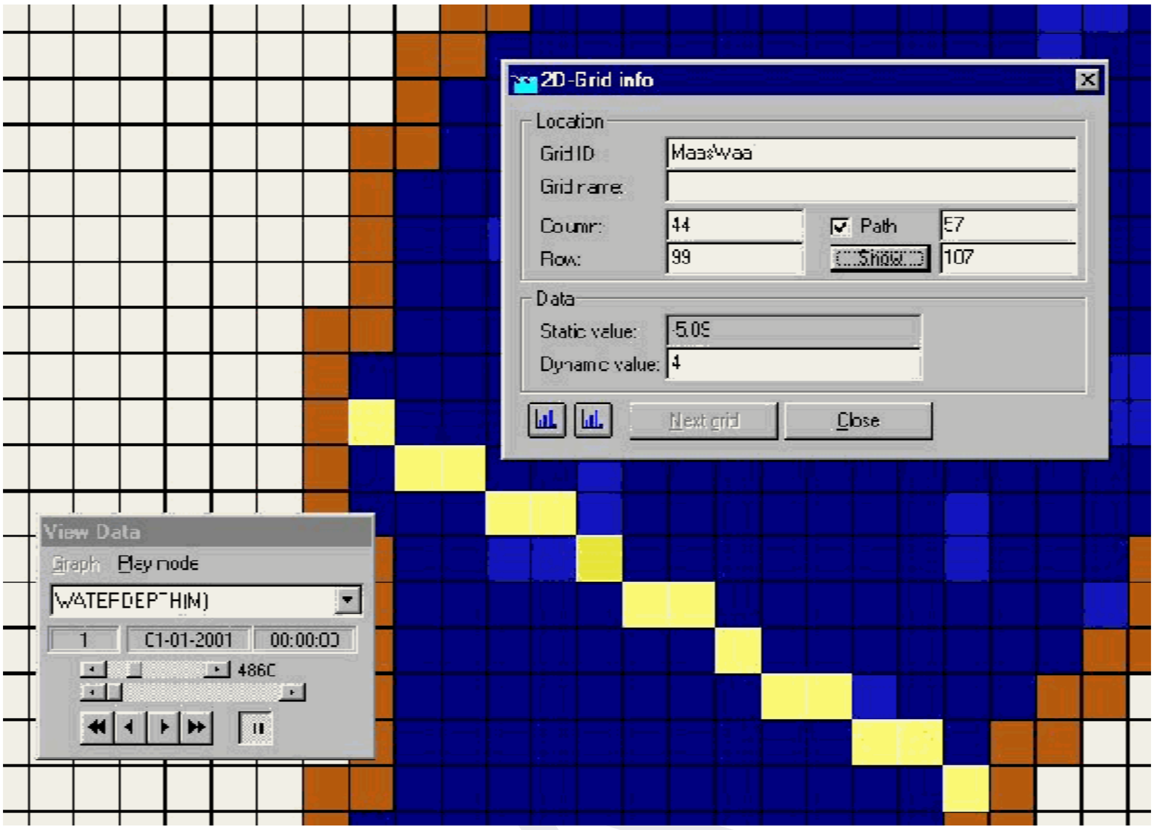


Figure 5.278: *Example of a 2D Path under a particular user defined line*

5.9.3 Coupling with other modules

General Aspects of a 1D2D model

The figure below depicts the currently available building blocks (called 'nodes' in SOBEK). For now, two groups of nodes are available, namely the flow nodes and the 2D nodes. The flow nodes are the ones used to build 1D schematisations, please refer to the SOBEK Channel-flow manual. The 2D nodes are the elements that have been added to SOBEK for the modelling of 2D systems. Together they make up the 1D-2D schematisation.

One important 2D building block that is not included in Figure 5.279 is the flow - dam break branch. This is a branch type, and not a node, which means that it can be found with the branches.

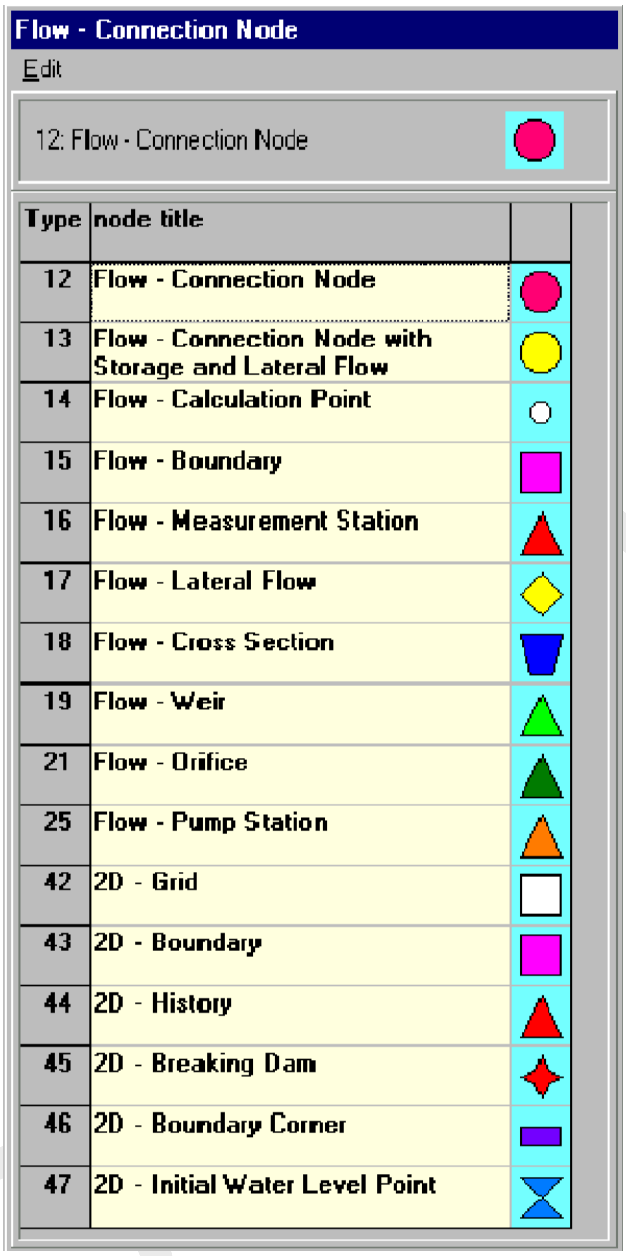


Figure 5.279: *Available node types for the Overland Flow & Channel Flow modules*

A very important aspect to consider when using this type of schematisation is the interaction between the 1D and the 2D system. The theory behind this interaction was explained in chapter 2, and the FAQ-section contains an example of a dike-bbranch modelled in 1D.

When building a 2D schematisation, there are a number of rules that need to be obeyed. One of the first rules is that every 2D-schematistion needs to contain al least one 1D-branch, even when the schematisation is purely 2D. In this case, the 1D-branch is referred to as a 'dummy-branch'. Usually, this branch would be placed somewhere outside of the grid, and it's properties would be as simple as possible. The figure below shows an example:

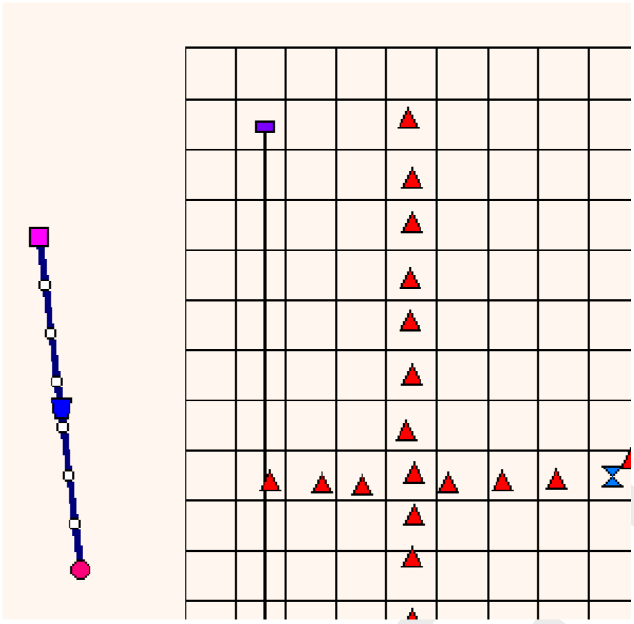


Figure 5.280: *Example of a 1D dummy branch*

The second rule is that it is not possible to put more than one object in one 2D grid cell. The only exception to this rule is the 2D- history node, which can be combined with any other type of 2D node.

One also needs to take care when combining 1D and 2D schematisations. Some combinations of elements will not be possible because of the (automatic) links made between the two models. As explained in chapter 2, links with the 2D grid are only possible starting from a 1D connection node or a 1D calculation point, and not from any other type of 1D node, like a 1D boundary. Because there can be only one 1D node connected to a 2D grid cell, it is a good idea to have exactly One, no more and no less, 1D calculation point (or connection node) defined per 2D grid cell.

Finally, the last important fact to remember is that SOBEK internally removes all outside grid cells from the grid, before starting the simulation! So the grid used for calculation will be 2 columns and 2 rows smaller. Knowing this, it is also not possible to define any 2D nodes in any of the outer grid cells. If you do so anyway, SOBEK can stop the simulation with an error message.

Coupling a Sewer Flow manhole with the Overland Flow module (2D grid)
If you desire to create an interaction between the Sewer Flow module and the Overland Flow module, you may want to couple a Flow - Manhole with the 2D-grid. You can either choose to use a manhole of the type "reservoir" or the type "loss".

Manhole of the type "closed":
If a Flow-manhole of the type "closed" is placed on a 2D grid cell, there will be **no** exchange of water between the 2D grid and the manhole.

Manhole of the type *loss*:
If you choose place a Flow-Manhole of the type *loss* on a 2D grid cell, the street level for that manhole is overruled by the level of the connected 2D grid cell:

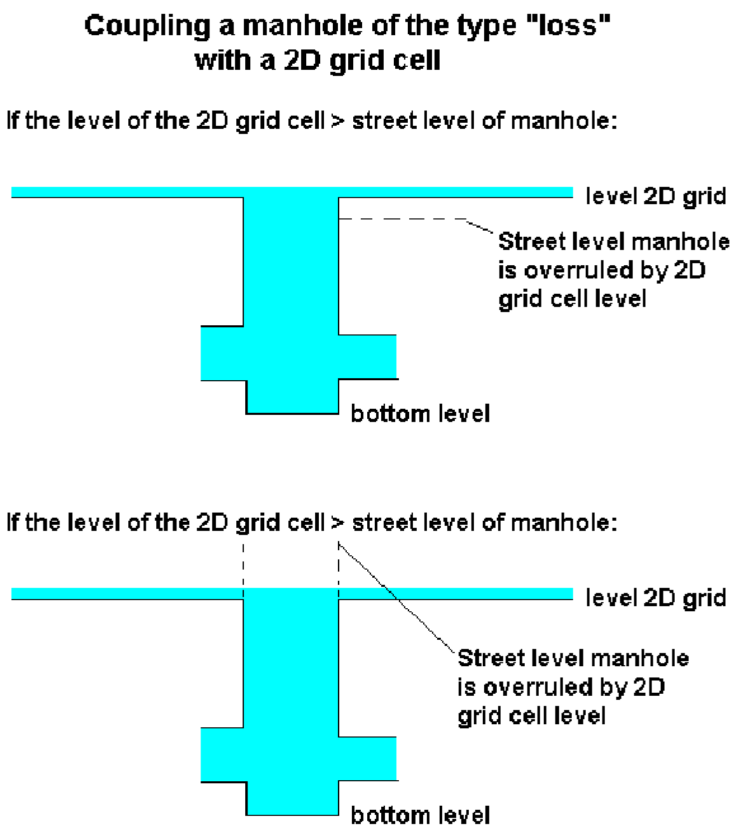


Figure 5.281: *Coupling a manhole ot the type* loss *with a 2D grid cell*

Manhole of the type *reservoir*:
If you choose a Flow-manhole of the type *reservoir*, and place it on top of a 2D grid cell, two situations may occur. See the picture below.

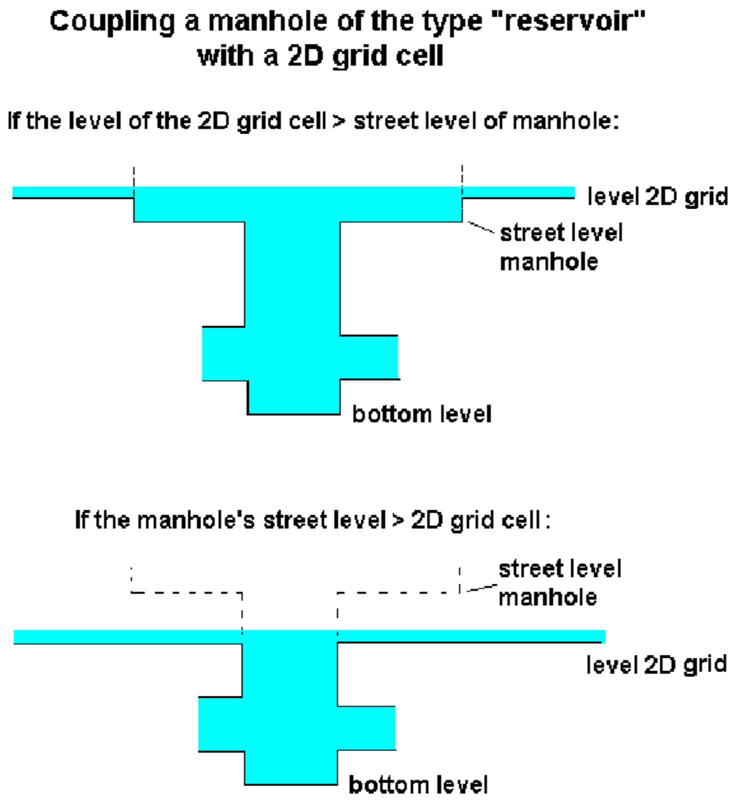


Figure 5.282: *Coupling a manhole ot the type* reservoir *with a 2D grid cell*

5.10 SOBEK-Rural RR (Rainfall-Runoff)

5.10.1 Features SOBEK-Rural Rainfall-Runoff

- ◇ Models rainfall run-off and other hydrological processes in rural areas and urban areas
- ◇ Catchment areas can easily be modelled in a lumped or detail manner, with no restriction to the number of catchment areas
- ◇ Catchment areas can be modelled in any detail using land elevation curves, soil characteristics, land cultivation, drainage characteristics etc.
- ◇ Distinguishes between various rainfall run-off processes such as surface run-off, sub-soil drainage and storage in saturated and unsaturated areas, taking into account crop evaporation and capillary rise
- ◇ Uses separate storm events or long time series of meteorological data for statistical analysis
- ◇ You can input your own rainfall patterns or use historical data, and model any number of rainfall gauges taking into account the spatial variation
- ◇ Model both flood events and dry spells
- ◇ Module can be used in combination with the flow module and real-time control module
- ◇ Can also be used as a stand-alone, using reservoir approach for open water flow

5.10.2 Connecting the Rainfall-Runoff module to the 1DFLOW module

You can connect a SOBEK Rainfall-Runoff module to a SOBEK Channel Flow or River Flow schematisation by using either of the following node types:

The Flow-RR Connection on Channel is an object that should be placed on an existing Flow branch. A Flow-RR Connection on Flow Connection node is an equivalent for a normal Flow Connection node. That node type should therefore form the start or the end of a branch.

To both node types, an RR schematisation can be connected. The nodes will then form the boundary for the Rainfall-Runoff schematisation.

5.10.3 Selecting a subset period of a rainfall event

This page explains how to select a subset period of a rainfall event for running the Flow module, while running the full period for the rainfall-runoff module.

For combined Rainfall-Runoff and Flow schematisations, it is possible to run the Flow simulations for a subset of the Rainfall-Runoff simulation period. Usually, the simulation period of the Rainfall-Runoff model is determined by the rainfall file as indicated in the following figure.

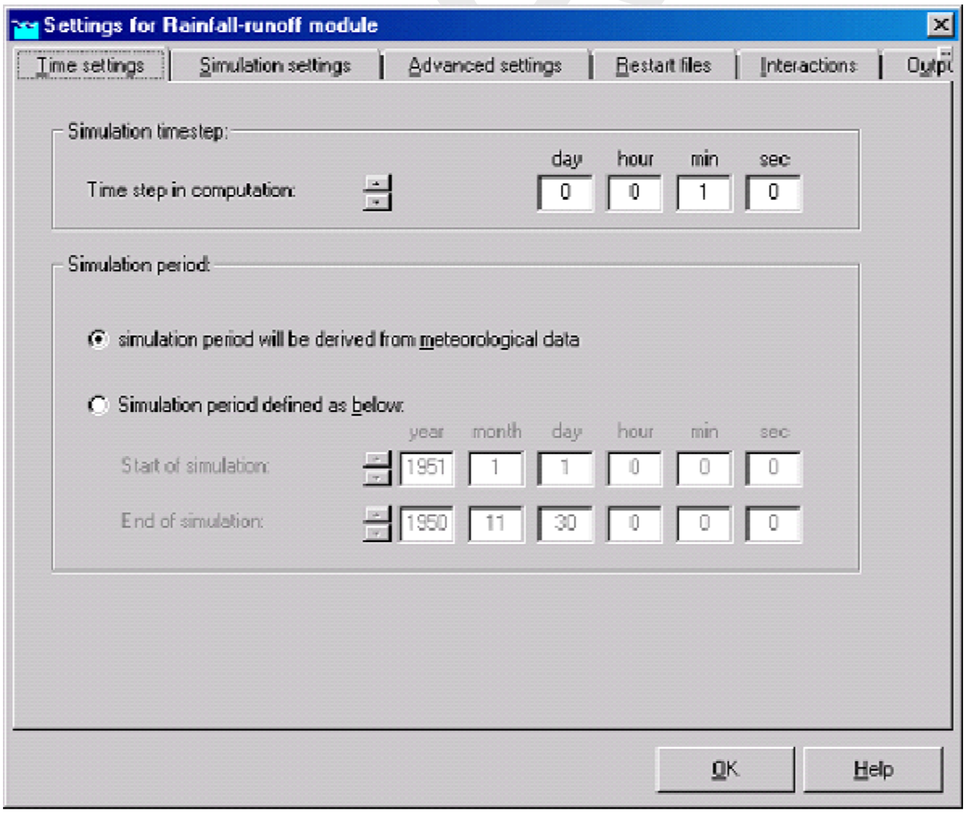


Figure 5.283: *Definition of simulation period for Rainfall-Runoff model.*

The Channel or Sewer Flow module simulation period is specified using a similar screen.

As an additional option, when both the RR and CF/SF simulation periods are derived from the meteorological data, there is the option to specify the interaction period for the RR and CF/SF modules. This is done in the Settings screens of the Rainfall-Runoff module. Of course, the interaction period should be a subset of the period covered by the meteorological data.

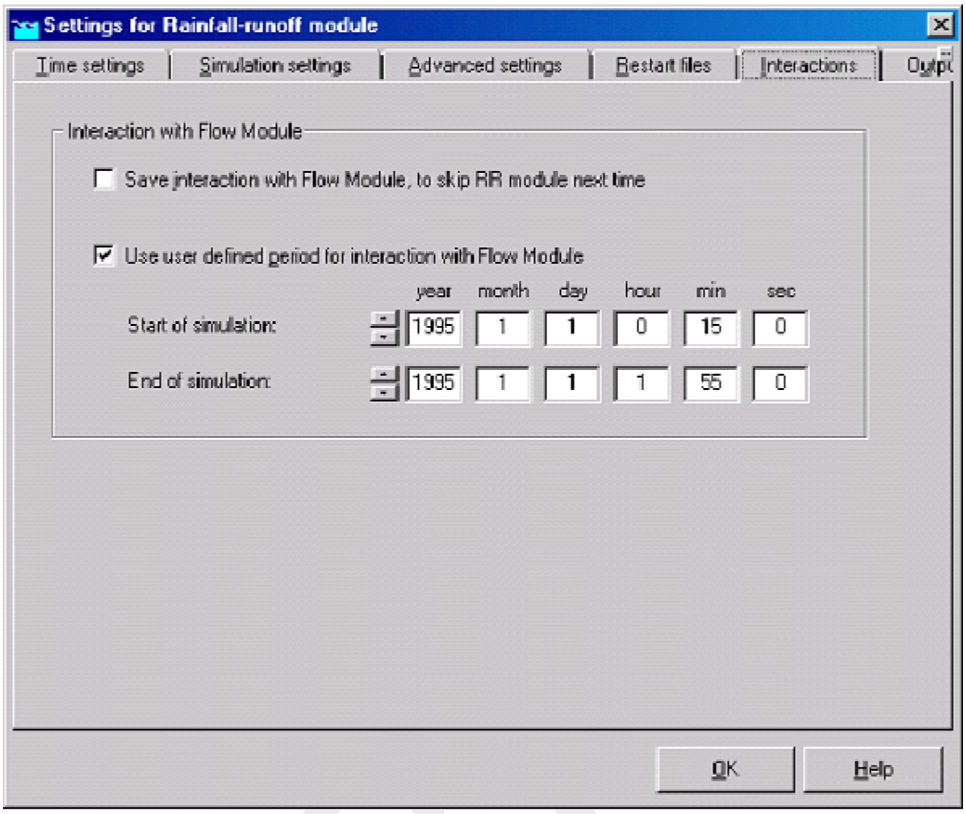


Figure 5.284: *Interactions of Rainfall-Runoff module with Flow module.*

In the example shown in the figure, the Rainfall-runoff module will use the full precipitation event (from 1995-01-01 0 o'clock to 2 o'clock) while the Flow module covers the period from 00:15 to 01:55 o'clock.

5.11 SOBEK-Urban RR (Rainfall-Runoff)

5.11.1 Features SOBEK-Urban RR (Rainfall-Runoff) module

- ◇ Models dry weather flows and the rainfall run-off processes for various types of paved areas, such as streets, roofs and parking lots
- ◇ Extended possibilities for unpaved areas and groundwater using the fully integrated link with the Rainfall Run-off module of the SOBEK-Rural product line
- ◇ Urban catchment areas can be easily modelled in a lumped or detailed manner with no limit to the number of catchments
- ◇ You can input your own time and spatially varied rainfall pattern or use historical data of storm events and long time series with or without dry periods
- ◇ Links two directionally to the hydrological database program HYMOS
- ◇ Infiltration is specified as a time-dependent process following the HORTON eqation

5.11.2 The SOBEK-Urban RR (Rainfall-Runoff) concept

The inflow towards the sewer system consists of runoff from rainfall and dry weather flow. The Runoff model of Flow-Manhole and Flow-Pipe, also called NWRW model, Nationale Werkgroep Riolering en Waterkwaliteit (NLingenieurs, 1978), describes the dry weather flow and the transformation in time of rainfall into runoff entering the sewer system. The Runoff model is based on the guidelines. The processes included are:

- ◇ moistening and puddle forming;
- ◇ infiltration;
- ◇ runoff delay.

It is illustrated that the rainfall-runoff process with netto rainfall is the same as the runoff towards the sewer system and is equal to the rainfall minus evaporation minus infiltration minus the change of storage.

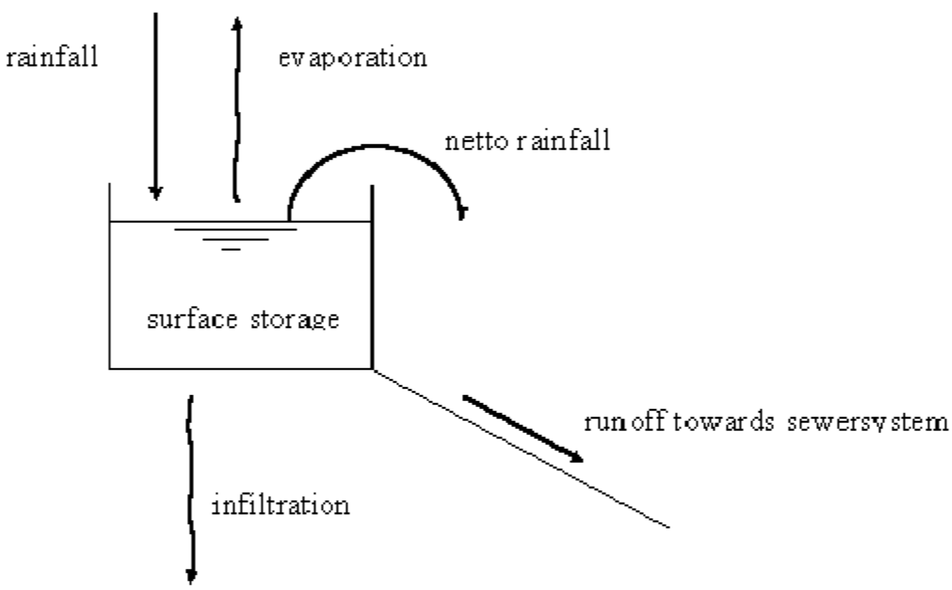


Figure 5.285: *Illustration of rainfall-runoff process*

The rainfall-runoff process
As a result of moistening and puddle forming, part of the rainfall will be stored temporarily on the surface. This storage is called surface storage. This storage is reduced by evaporation as well as infiltration. Different types of surfaces can be distinguished, depending on surface characteristics and slope. The model distinguishes four types of surfaces (closed paved, open paved, roof, unpaved) and three types of slopes (area with a slope, flat, stretched flat), thus twelve different area types. The slope of the surface and the infiltration capacity largely influence the rainfall-runoff process.

The infiltration of rainfall takes place in the open paved areas and unpaved areas. The infiltration capacity depends mostly on the type of surface and moisture condition. Other factors may also play an important role. For example, the infiltration capacity of brick paths depends on the condition of the openings between the bricks. The infiltration capacity of the unpaved areas depends on the vegetation, the kind of soil and the percentage of moisture in the subsurface. The description of the infiltration of the Runoff model is based on the formula of Horton (See also Technical Reference).

The delay of runoff depends on the average distance to the inflow location in sewer system, the slope and the geometry of the catchment. The formula which describes the runoff to the sewer system is the formula of the rational method (See also section 6.5.4).

5.11.3 SOBEK-Urban RR (Rainfall-Runoff) input screens

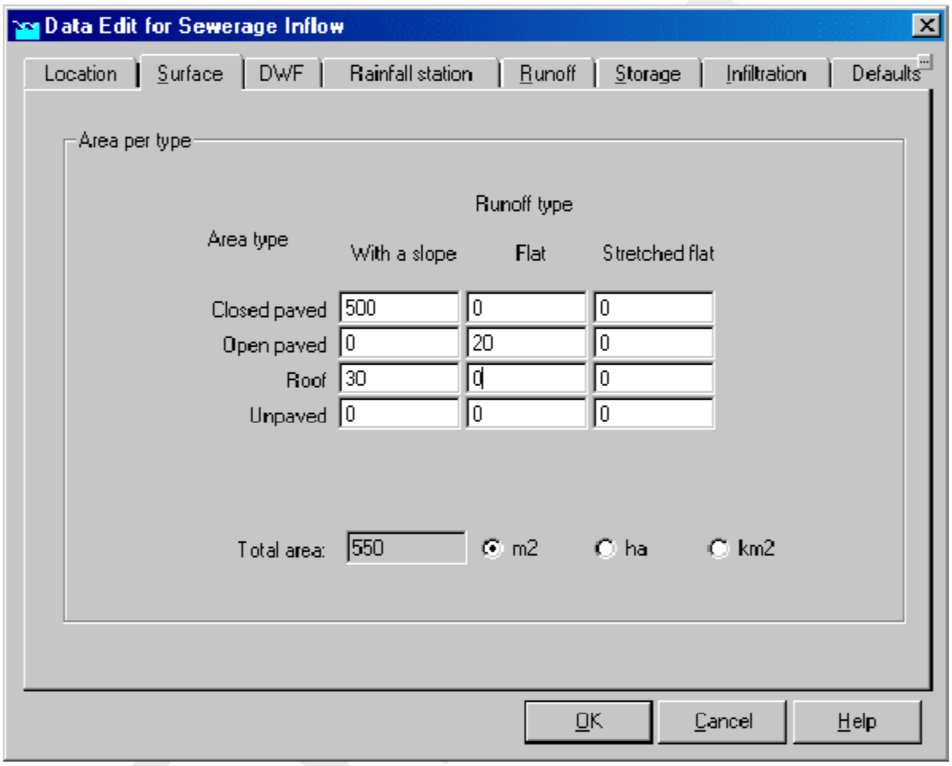


***Figure 5.286: Data Edit for Sewerage Inflow**, Surface tab*

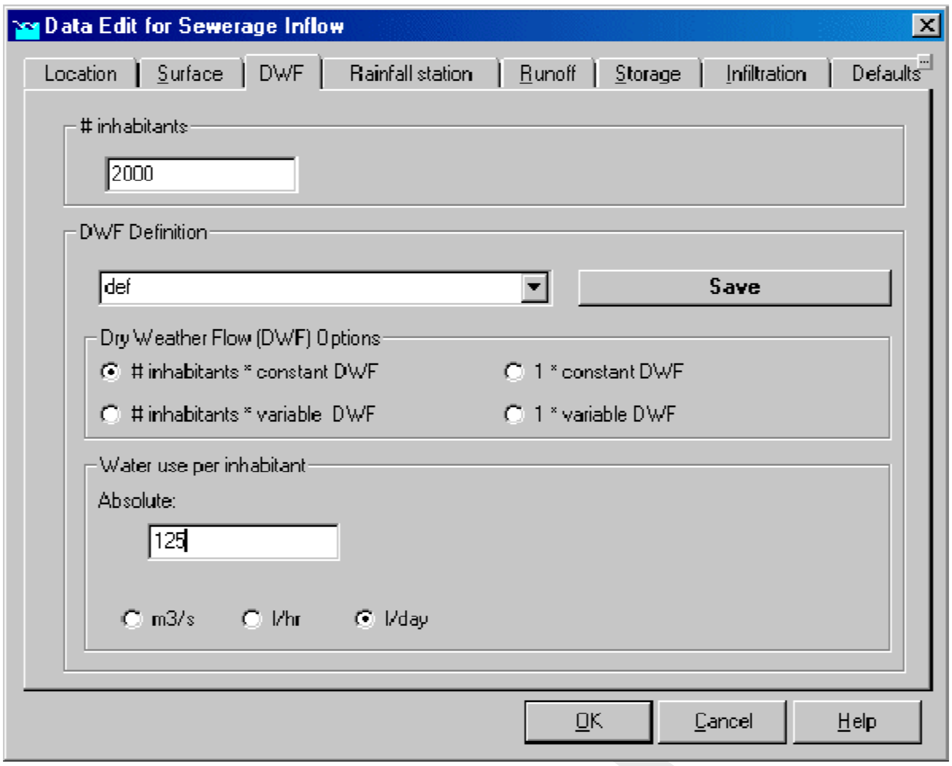


***Figure 5.287: Data Edit for Sewerage Inflow**, DWF tab*

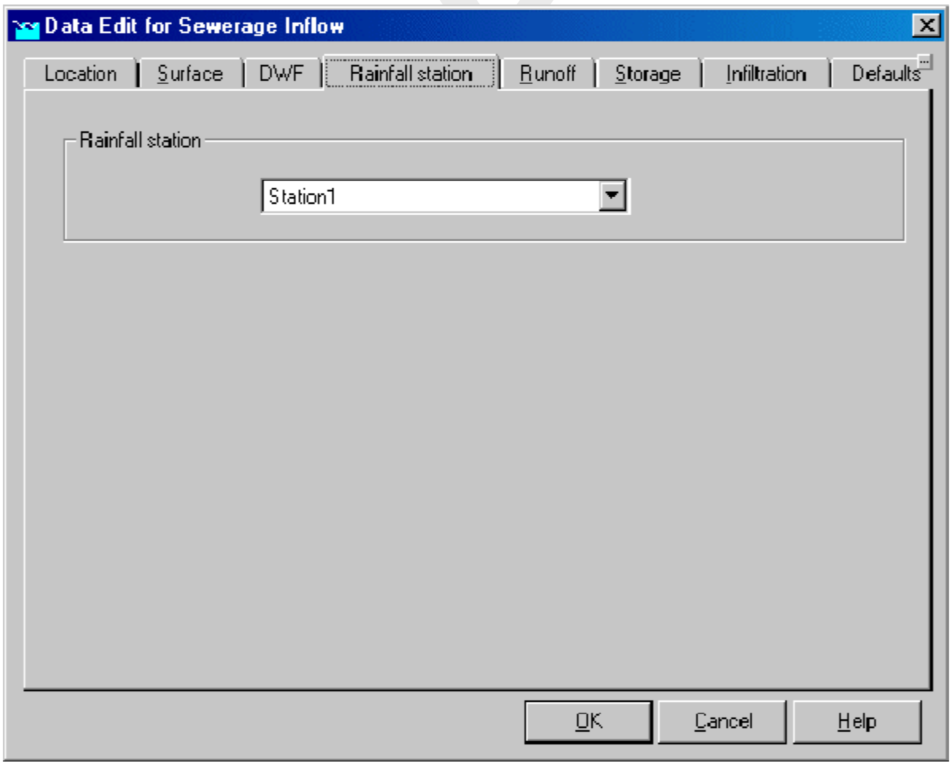


***Figure 5.288: Data Edit for Sewerage Inflow**, Rainfall station tab*

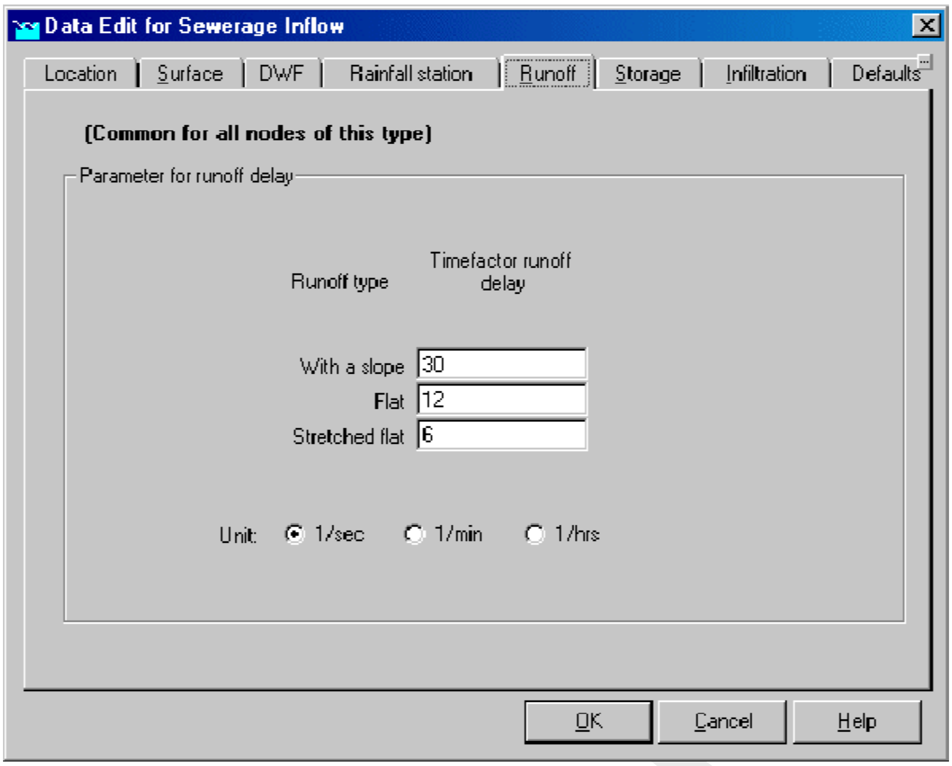


***Figure 5.289: Data Edit for Sewerage Inflow**, Runoff tab*

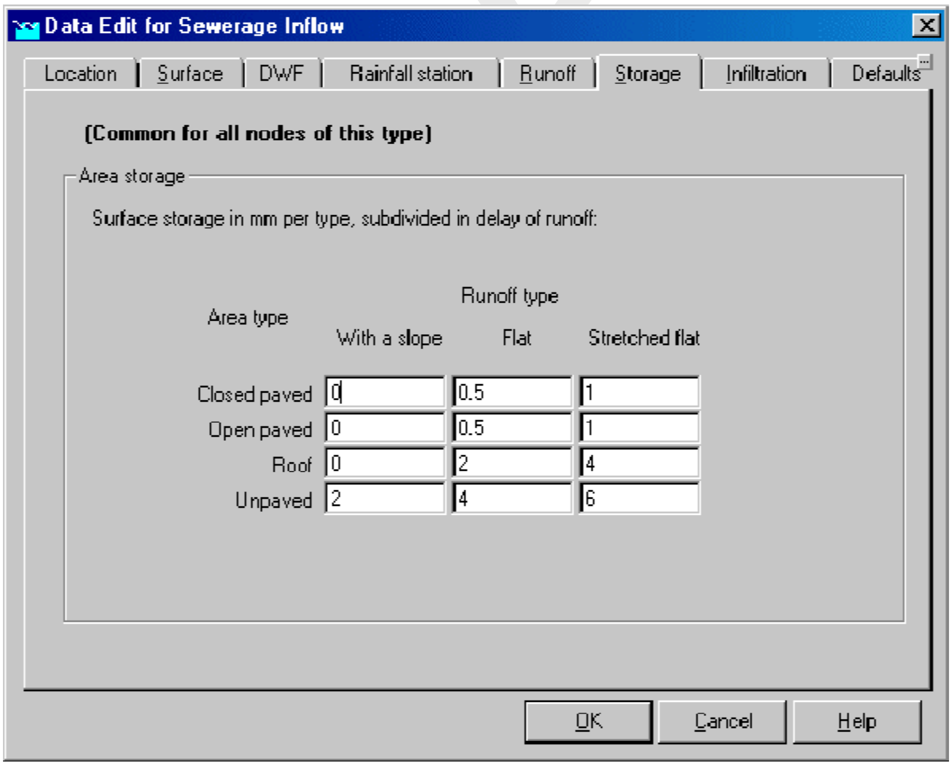


***Figure 5.290: Data Edit for Sewerage Inflow**, Storage tab*

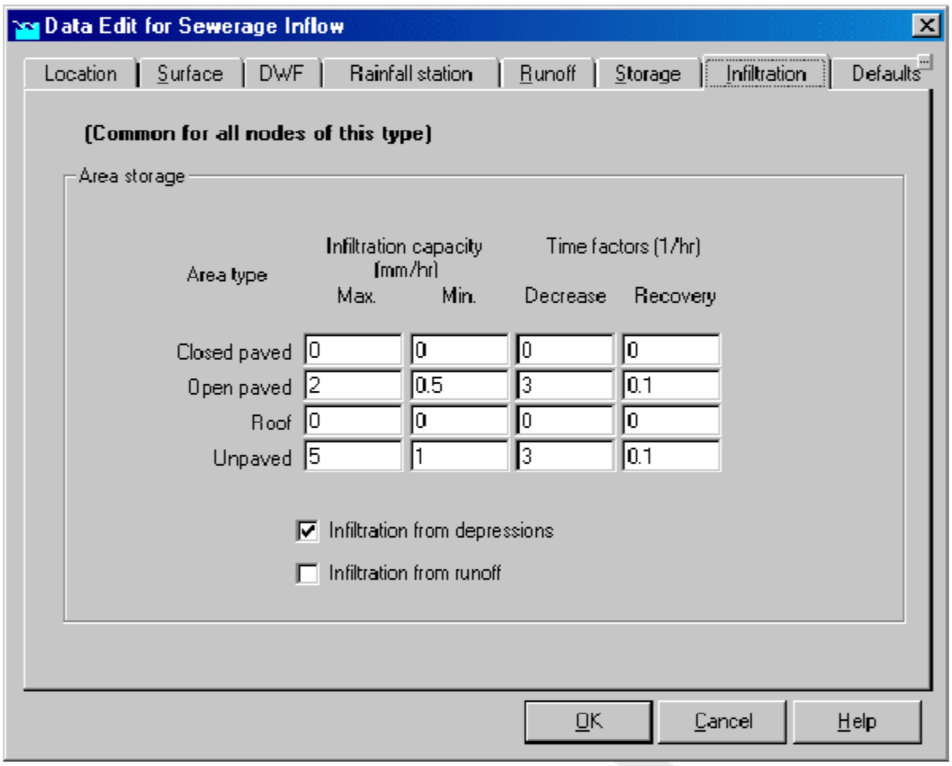


***Figure 5.291: Data Edit for Sewerage Inflow**, Infiltration tab*

For NWRW nodes (Urban rainfall-runoff model), there are two infiltration options available in the user interface:

◇ Infiltration from depressions;
◇ Infiltration from runoff;

Figure 5.298 illustrates the NWRW modelling concept.

The infiltration from depressions indicates the infiltration from water that is stored in depressions. The surface storage in depressions is typically a few mm. When the surface (depression) storage exceeds the defined threshold, runoff is computed.

However, the runoff may take some time to branch the outflow point (for urban sewer systems: the point where the runoff flows into the sewer system). The option 'infiltration from runoff' can be used to simulate infiltration from the runoff dynamic storage. The same set of Horton infiltration parameters are used for both infiltration from depressions and infiltration from runoff.

Default only the infiltration from depressions is switched on. Switching on the infiltration from runoff will result in some additional infiltration losses and less inflow in the sewer system.

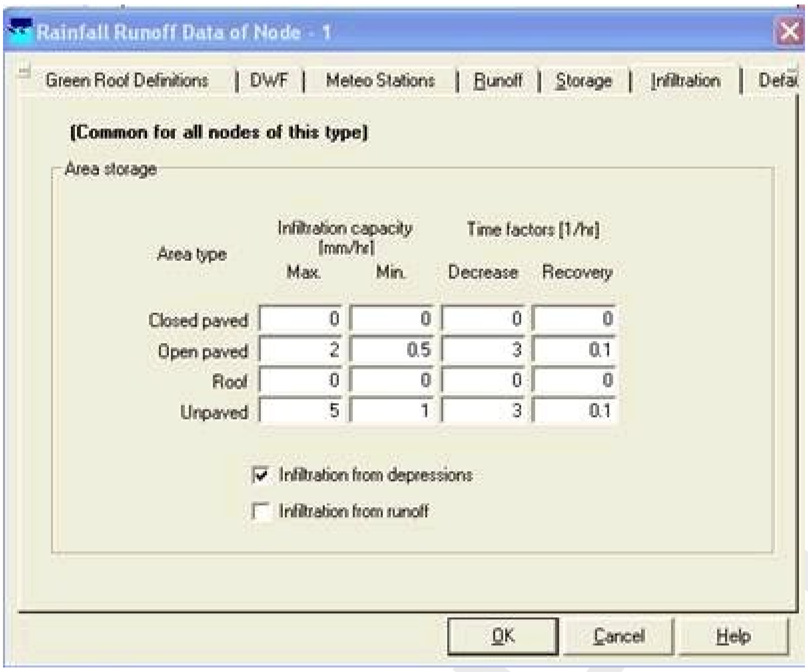


Figure 5.292: Concept of Urban runoff model

5.12 SOBEK-Rural/Urban/River RTC (Real Time Control)

5.12.1 Why a separate RTC (Real-time Control) module

Real-time Control is a separate module within SOBEK. One could ask for the reason for this. In a SOBEK-Urban 1DFLOW, a SOBEK-Rural 1DFLOW, a SOBEK-River 1DFLOW and a RR schematisation, controllers can be defined for operating the available structures. So real-time control options are already available within these modules. The reasons for having a separate Real-time Control (RTC) module are:

◇ the controllers in the SOBEK-Urban 1DFLOW, SOBEK-Rural 1DFLOW and SOBEK-River 1DFLOW modules are local controllers, looking at 1 (one) water level or flow in the hydrodynamic schematisation. Using Real-time Control, it is possible to take into account more than 1 location: you can control the structure looking at any linear combination of values at different locations;
◇ in Real-time Control, it is possible to define a measure (or RTC controller) for a (previously defined) controller in the Flow or Rainfall-Runoff schematisation. Such RTC measure can consist of several different type of decision rules, where each decision rule has its own priority. If defined active, such RTC measure will overrule its associated active controller in the Flow or Rainfall-Runoff schematisation. In case the associated controller is not-active (i.e. not triggered), the setpoint provided by the RTC measure will be neglected (see for more details RTC does not overrule Flow Triggers).
◇ most important, using Real-time Control it is possible to take into account information from different SOBEK modules. One can define the operation of a pump in Rainfall-Runoff based on results of the Flow modules (or hydrodynamic computational results), or define the operation of a weir or pump in a Flow module based on computed runoff by the rainfall-runoff module, or based on available rainfall and wind data, or on other data (e.g. water quality). This gives many more possibilities than just using local controllers in a Flow or Rainfall-Runoff schematisation.
◇ SOBEK RTC is equiped with a special reservoir module, allowing operation of reservoirs or ponds with multiple outlets using a set of rule curves.
◇ Further on the Real-time Control (RTC) module can be online coupled with a Matlab com-

putation, allowing for any type of controller to be applied.

5.12.2 Condition

A Real-time Control measure usually involves a logical condition. Only if that condition is satisfied, the measure will be active.

A condition is of the following form:

'value of decision parameter' 'check' 'check value'

The check can be '$<$', '=' or '$>$'.

The check value can either be a constant input value, or the value of another decision parameter.

5.12.3 Data locations in RTC

Data locations in Real-time Control (RTC) represent, as the name suggests, those locations at which data can be defined. Using the data defined at data locations, so-called decision parameters can be defined. In RTC so-called measures can be defined. A measure consists of a number of decision rules. These decision rules can be formulated using the defined decision parameters. In case a measure is defined active, the corresponding decision rules will be evaluated and will result in a setpoint for the associated Flow or RR controller, that is assigned to a particular structure[s] located in either the Flow or Rainfall-Runoff schematisation.

In Real-time Control five different types of data locations can be defined:

- 1DFLOW Data; computed hydrodynamic parameters at specific locations in the SOBEK-Urban 1DFLOW, SOBEK-Rural 1DFLOW and SOBEK-River 1DFLOW schematisations
- 2DFLOW Data; computed hydrodynamic parameters at specific locations in the Overland Flow schematisation
- RR Data; computed parameters at specific locations in the Rainfall-Runoff schematisation
- Precipitation Data; user-defined rainfall data
- Wind Data; user-defined wind data (i.e. wind velocities and wind directions)
- External Data - His Files; user-defined External Data (can be data, that is not used or computed by any SOBEK module)

5.12.4 Data measurement location

The data measurement locations represent, as the name suggests, those locations at which data is available for Real-time Control. At the moment, 4 types of data measurement locations are available. These are:

- locations in the Water Flow module (Water Flow locations)
- locations in the Rainfall-Runoff module (Rainfall-Runoff locations)
- rainfall prediction locations
- other locations (also called external locations); at the moment these locations are used for wind predictions and for data from external HIS files.

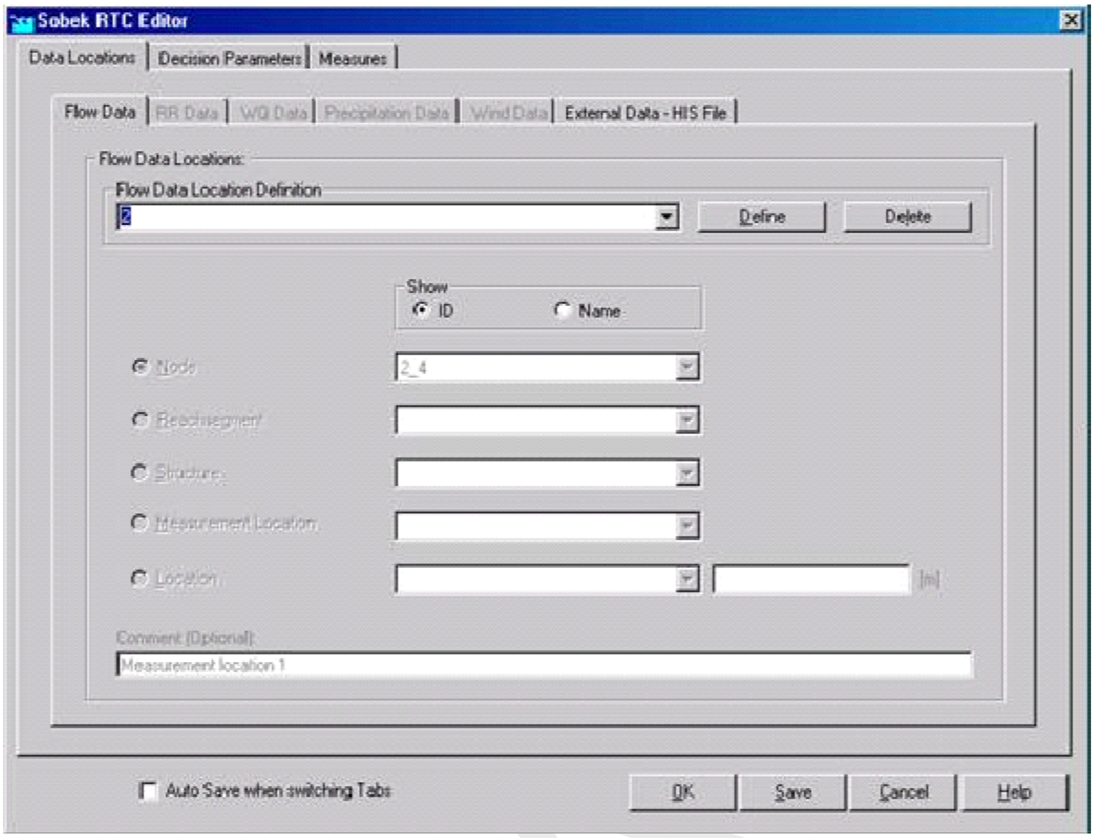


Figure 5.293: *RTC Flow data locations input screen*

The available data (measurements) at Water Flow locations are given in the following list:

- water level + RefL [m];
- flow [m^3/s];
- surface area [m^2];
- water depth [m];
- crest level [m];
- crest width [m];
- gate lower edge [m];
- gate opening height [m];
- structure flow area [m^2];
- discharge at the structure [m^3/s];
- flow velocity at the structure [m/s];
- water level up [m];
- water level down [m];
- head over structure [m];
- pressure difference over structure;
- pump capacity [m^3/s].

Data availability depends on the type of location (a node, a branchsegment, a flow measurement location, or a structure) and type of structure. For instance, for a weir, no pump capacity is available.

The available data at Rainfall-Runoff locations are:

- open water level (m + RefL);

◇ groundwater level (m + RefL);

The available data at rainfall prediction locations are:

◇ rainfall prediction at time step t+1 (mm/time step);
◇ rainfall prediction at time step t+2 (mm/time step);
◇ rainfall prediction at time step t+3, etc.

The available data for wind prediction are:

◇ wind direction prediction at time step t+1 (degrees);
◇ wind direction prediction at time step t+2; etc. and
◇ wind velocity prediction at time step t+1 $[m/s]$;
◇ wind velocity prediction at time step t+2, etc.

The available data for external locations related to a SOBEK HIS file are taken from the external HIS file and indicated by the user.

5.12.5 Decision Parameters in RTC

In Real-time Control (RTC), decision parameters can be defined on basis of the data available at earlier defined Data locations. Decision parameters are used in defining measures (or RTC controllers), that determine how a particular structure is to be operated.

Real-time Control allows to define a wide range of decision parameters. Examples of decision parameters are:

◇ The water level or discharge at a particular location in a 1DFLOW schematisation,
◇ The waterlevel or abs. velocity at particular location in an Overland Flow schematization,
◇ The outflow discharge from a Rainfall-Runoff schematisation,
◇ The predicted area weighted average rainfall of all rainfall stations,
◇ The predicted wind velocity

In RTC following type of decision parameters are discerned (see Figure below):

◇ Function (decision) parameters,
◇ Time (decision) parameters, and
◇ Reservoir (decision) parameters.

Please note that Interpolation Tables are not decision parameters. Interpolation Tables are used in defining so-called Interpolation Function decision parameter (see mathematical manipulations available for function decision parameters hereafter).

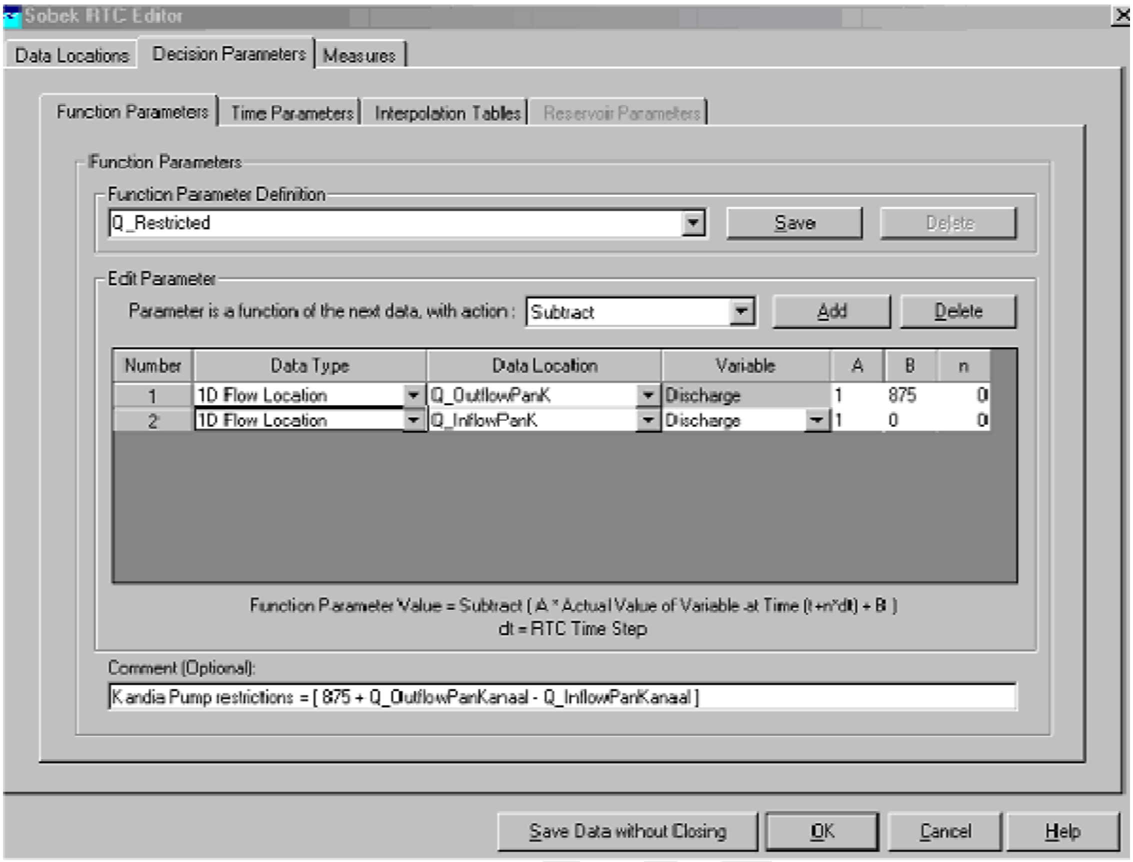


Figure 5.294: *Example of RTC input screen for Function (decision) parameters*

<u>Function (decision) parameters:</u>
The actual time-dependent value of a Function decision parameter can result from a mathematical manipulation:

◇ on the corresponding value of a single data series, or
◇ on the corresponding values of a set of data series.

Mathematical manipulations available for single data series are: None, Sin, Cos, Tan, ArcSin, ArcCos, ArcTan, Sinh, Cosh, Tanh, Interpolate, Floor, Ceil, Nint, Exp, Log, Log10, Square and Sqrt.

Remarks:

◇ None means that no mathematical manipulation is carried out on the data series,
◇ Interpolate; for each independent value, a corresponding dependent value (or Interpolate Function Decision parameter) is determined by linear interpolation in an Interpolation Table (first column: independent variable in ascending order; second column: dependent variable). If an independent variable is smaller than the value on the first row, the dependent variable will be set equal to its value on the first row. If an independent variable is larger than the value on the last row, the dependent variable will be set equal to its value on the last row.
◇ Floor(12.8)=12.0; Ceil(12.2)=13.0;
◇ Nint (Nearest INTeger), Nint(12.1)=12, Nint(12.5)=13, Nint(-12.5)=-13

Mathematical manipulations available for a set of data series are: Add, Subtract, Multiply, Divide, Max, Min, Average and Power (for more information, see Table 5.7)

Table 5.7: *Mathematical manipulations available for a set of data series*

Mathematical manipulation	**Values (V) of data series**	**Function Decision Parameter value**
Add	V1, V2, V3, ... Vn	V1 + V2 + V3 . . . + Vn
Subtract	V1, V2, V3, ... Vn	V1 - V2 - V3 . . . - Vn
Multiply	V1, V2, V3, ... Vn	V1 * V2 * V3 . . . * Vn
Divide	V1, V2, V3, ... Vn	V1 / V2 / V3 . . . / Vn
Max	V1, V2, V3, ... Vn	Max (V1, V2, V3, . . . , Vn)
Min	V1, V2, V3, ... Vn	Min (V1, V2, V3, . . . , Vn)
Average	V1, V2, V3, ... Vn	(V1 + V2 + V3 + . . . + Vn) / n
Power	V1, V2, V3, ... Vn	(((V1^V2)^V3)^V4) . . . ^Vn)

Note: V1 is the value of the data series defined on the first row and Vn is the value of the data series defined on the last row (see Figure above).

A data series is defined by its:

1 Data Type, Data location, and Variable Type,
2 Multiplication factor A, Off-set B, and RTC Time shift n.

Ad 1) Data Type, Data location, and Variable Type:

Presently the following data types are supported:

◇ Data Type=1D Flow location: Computed 1D Flow data at “1D Flow Data locations” previously defined on the “1D Flow Data” Tab (see 1D Flow Data in RTC),
◇ Data Type=2D Flow location: Computed 2D Flow data at “2D Flow Data locations” previously defined on the “2D Flow Data” Tab (see 2D Flow Data in RTC),
◇ Data Type=External: Available data locations are the “External Data locations” previously defined on the “External Data – His File” Tab (see External Data – His File in RTC),
◇ Data Type=Decision parameter: Available data locations are the previously defined Function/Time/Reservoir (decision) parameters,
◇ Data Type=Date/Time: Data source is the RTC computational point-in-time; Available data locations are: Year, Month, Day, Hour, Minute, Second, Date, Time, Date+Time, Day of Week and CompTimestep; Variable follows from the actual RTC computational point-in-time (for more information, see Table below)

Ad 2) Multiplication factor A, Off-set B, and RTC Time shift n:

Each data series can be a linear function of the actual value of a specific variable at a particular data location. The linear function is defined as: Data series value = A * Variable(t+n*dt) + B, where: A = Constant multiplication factor, t = actual RTC computational point-in-time, n=number of timesteps, dt= time step defined in RTC Settings, and B = Constant off-set value.

Resuming:
Say that a Function (decision) parameter should be the 1D water levels, computed at Rotterdam at a point-in-time that is 2 RTC time steps earlier than the actual RTC computational time. This Function (decision) parameter should comprise of only one data series with mathematical manipulation “None”. This data series is to be defined as: Data type=1D Flow location, Data location=Rotterdam (defined at the 1D Flow Data Tab), Variable=water level, A=1, B=0 and n=-2.

Table 5.8: *Data Type=Date/Time (yyyy-mm-dd; hh:mm:ss)*

Data Type	**Data location**	**Variable**
RTC “Date/Time”	Year	Integer value = “yyyy”
RTC “Date/Time”	Month	Integer value = “mm”
RTC “Date/Time”	Day	Integer value = “dd”
RTC “Date/Time”	Hour	Integer value = “hh”
RTC “Date/Time”	Minute	Integer value = “mm”
RTC “Date/Time”	Second	Integer value = “ss”
RTC “Date/Time”	Date	Integer value = “yyyymmdd”
RTC “Date/Time”	Time	Integer value = “hhmmss”
RTC “Date/Time”	Date+Time	Real value = “yyyymmdd.hhmmss”
RTC “Date/Time”	Day of week	Integer value = day no.
RTC “Date/Time”	CompTimestep	RTC timestep in seconds

Remarks:

- ◇ Say that: the RTC time-step=10 seconds; the actual RTC “Date/Time” is Tuesday, March 29th, 2005 at 12:10:50 (2005-03-29; 12:10:50); A=2; B=-4; and n=96. Than if:
  - ◇ Data location=Year; resulting value = 2*[2005-03-29; 12:10:50 – 96*10s]-4= 2*yyyy-4=4006
  - ◇ Data location=Minute; resulting value = 2*[2005-03-29; 12:10:50 – 96*10s]-4= 2*mm-4= 104
  - ◇ Data location=Date; resulting value = 2*[2005-03-29; 12:10:50 – 96*10s]-4= 2*hhmmss-4= 230896
  - ◇ Data location=Date+Time; resulting value = 2*[2005-03-29; 12:10:50 – 96*10s]-4= 2*yyyymmdd.hhmmss-4= 40100654.230900
  - ◇ Data location=Day of week; resulting value = 2*[2005-03-29; 12:10:50 – 96*10s]-4= 2*Day_of_week -4= 0
- ◇ Day_of_week(Sunday)=0; Day_of_week(Monday)=1; and Day_of_week(Saturday)=6
- ◇ Please note that resulting values follow from a mathematic expression only, and might become negative integer values or negative real values.

Time (decision) parameters:

A Time decision parameter can be a user-defined function of time.

5.12.6 Decision rules in RTC - General

In Real-time Control (RTC), Measures can be defined consisting of a set of so-called Decision rules. For each decision rule, the user has to define its priority and its decision rule type. A decision rule might consist of a number of criteria. Each criteria has the following form:

“Value of decision parameter” “Check” “Check value”

Note:

- ◇ The value of a decision parameter is the value that corresponds to the RTC computational point-in-time.
- ◇ The “Check” can be ‘<’, ‘=’ or ‘>’
- ◇ “Check value” can be a constant or a user-defined decision parameter

In case for a decision rule all its criteria are “True”, than the evaluation of such decision rule is “True”. Else the evaluation of such decision rule is “False”.

- ◇ In case the evaluation of more than one decision rule is “True”, than the setpoint for the Measure will be equal to the set-point of the decision rule having the highest priority (e.g. lowest integer number)
- ◇ In case the evaluation of more than 1 (one) decision rule having the highest defined priority are “True”, than the set-point for the Measure will be equal to the set-point of the decision rule that is the closest to end of the drop-down list.
- ◇ In case not one of the defined decision rules is “True”, than the set-point of the Measure will be kept equal to the set-point determined in the previous RTC time-step.

For more information on the type of decision rules available for RTC Flow Measures, see Flow Measures in RTC.

5.12.7 External Data - His File in RTC

On the External Data – His File Tab available under the Data locations Tab in Real-time Control (RTC), data contained in a so-called His file (<name>.his) can be selected. This His file can be placed at any location/folder at your computer and should be available at the start of the computation. A His file may contain any kind of data (for instance number of crocodiles lying on a river bank). In addition a His file can contain different type of data for a large number of locations.

5.12.8 Example of a MATLAB M-file

The example program below shows the use of some SOBEK output variables, the conversion of the SOBEKTime string to numerical values and the passing of setpoints computed by Matlab back to SOBEK.

```
if (SOBEKFirst == 1)
      SOBEKC_Matlab1=1;
      Sinitial=1;
end
SOBEKS_Matlab1=Sinitial;
Level_measurementlocation1=SOBEKH_2_4;
Level_measurementlocation2=SOBEKH_6;
Flow_measurementlocation1=SOBEKQ_2_4;
Flow_measurementlocation2=SOBEKQ_4;
Crestlevel=SOBEKCL_17;
Crestwidth=SOBEKCW_17;
DeltaHStructure=SOBEKDH_17;
TimeControllerSetpoint=1.;
temp_time=str2num(SOBEKTime);
Hour=round(floor(temp_time/1000000));
temp=temp_time-Hour*1 000 000;
Minute = round(floor(temp/10000));
if (temp_time < 6000000)
      TimeControllerSetpoint=1.;
elseif (temp_time < 10000000)
      TimeControllerSetpoint=0.9;
elseif (temp_time < 12000000)
      TimeControllerSetpoint=1.;
elseif (temp_time < 15000000)
      NrMinutes=(Hour-12)*60 + Minute
      TimeControllerSetpoint=1. + (NrMinutes)/180 * 0.4;
elseif (temp_time < 20000000)
      NrMinutes=(Hour-15)*60 + Minute
      TimeControllerSetpoint=1.4 - (NrMinutes)/300 * 0.4;
end
if (Flow_measurementlocation1 > 1.0)
      SOBEKS_Matlab1 = TimeControllerSetpoint;
end
```

Example m-file

Note:

1 In Matlab, you can on-line check the values of the variables in your program. You can request which variables are available by using the who command. The value of a variable can be requested by simply typing the name of the variable and pressing Enter. Note that Matlab is case-sensitive, so carefully check the variable names used in your m-file and the SOBEK-ids.
2 Matlab declarations must not be written to the screen, so must always end with a semicolon (;), e.g. SOBEKC_0_65=1;
3 When using Matlab 6.1., errors in the Matlab m-file (e.g. an undefined variable is used;

since Matlab is case-sensitive this may just be a typing error) , will be heard by regular beeps whenever the m-file is executed (each time step in the RTC-Matlab coupling). Older Matlab versions do not give this indication of error!

The M-FILE that is selected in the [Matlab Options] tab, must produce the following variables for the Flow controllers which are controlled by Matlab:

- ◇ SOBEKC_<controller> (optional)Indication if Matlab controls the SOBEK controller. For example with: SOBEKC_0_65 for the controller '0-65'. This identifier corresponds with the identifier following keyword `id` in the inputfile <SBK_MEAS.RTC>. The result has value 1 (=central control by Matlab) or 0 (=local control in the hydrodynamic model).
- ◇ SOBEKS_<controller>Setpoint for local controller or the controlled parameter of a controller. For example with: SOBEKS_0_65 for identifier '0-65'. This identifier corresponds with the identifier following keyword `id` in the input file <SBK_MEAS.RTC>.

The M-FILE that is selected in the [Matlab Options] tab, must produce the following variables for the RR structures which are controlled by Matlab:

- ◇ RRC_<RRstructure> (optional) Indication if Matlab controls the RR structure. For example with: RRC_0_65 for the structure '0-65'. This identifier corresponds with the identifier following keyword 'id' in the inputfile 3B_MEAS.RTC. The result has value 1 (=central control by Matlab) or 0 (=local control in the rainfall-runoff model).
- ◇ RRSlowon_<RRstructure> Switch on level low capacity pump for RR structure with id <RRstructure> as defined in the inputfile <3B_MEAS.RTC>.
- ◇ RRSlowoff_<RRstructure> Switch off level low capacity pump for RR structure with id <RRstructure> as defined in the inputfile <3B_MEAS.RTC>.
- ◇ RRShighon_<RRstructure> Switch on level low capacity pump for RR structure with id <RRstructure> as defined in the inputfile <3B_MEAS.RTC>.
- ◇ RRShighoff_<RRstructure> Switch off level low capacity pump for RR structure with id <RRstructure> as defined in the inputfile <3B_MEAS.RTC>.

Note:

1. The old variables with flow structure parameters: SOBEKP_<Structure>=X are not supported anymore. The old SOBEKP parameter contained different types of parameters as function of the type of corresponding structure.
2. The new version of RTC communicates structure parameters in more detail using different variables mentioned above, e.g. SOBEKCL_<Structure>=X to get the crest level of a weir, or SOBEKPC_<Structure>=X to get the pump capacity of a SOBEK pump.

5.12.9 Features Real-time Control

- ◇ Real-time control often saves money in the construction, operation and management of the water system infrastructure. The RTC module shows to what extent the existing infrastructure can be used in a better way
- ◇ Allows you to simulate complex real-time control of all structures in the canal network
- ◇ Allows the system to react optimally to actual water levels, discharges, (forecasted) rainfall, by controlling gates, weirs, sluices and pumps
- ◇ The Real-Time Control module can also be linked to Matlab, the industrial standard for control engineers, and even allows you to define your complete control system in Matlab
- ◇ Enables you to intervene in events taking place within your water system
- ◇ Helps you make informed choices about automation and the best water control strategy
- ◇ All standard irrigation automation concepts can be handled with this module

5.12.10 Flow Measures in RTC

On the Flow Measures Tab in Real-time Control (RTC), you can create a so-called "Flow Measure (or RTC Flow controller)" for any type of Controller defined in a Flow schematisation. This is done by clicking on the "Create RTC Controller" check-box. In case the "Create RTC Controller" check-box is not checked, there exists no Flow Measure for the selected Controller in the Flow schematisation. In other words, there will be no Flow Measure data what-so-ever stored in the SOBEK Model Database for the selected Controller in the Flow schematisation (see Figure below). A Flow-measure can be defined active by clicking on the "Active" check-box (see Figure below). A Flow Measure, when defined active, will overrule its associated active (see below) Flow Controller, that can be assigned to a structure located in a Flow schematisation. In the user-interface a Flow Measure can be linked to a particular Controller in the Flow schematisaton based on the Controller ID or the Controller Name (see Figure below). Internally, the Flow Measure identification is based on the Controller ID.

It is explicitly mentioned that RTC does not overrule Triggers, that might be assigned to Flow controllers. This means that in case a particular Flow controller is not triggered (i.e. not activated) at a certain point-in-time, this Flow controller will not be overruled by its associated Flow Measure (or RTC Flow controller). Hence, the setpoint of a not-triggered Flow controller will be equal to the setpoint of the last point-in-time for which it was triggered. Further on please note that in case no triggers are defined for a Flow controller, this means that such Flow controller will be activated during the entire computation.

Note:

1. For SOBEK-Urban 1DFLOW and SOBEK-Rural 1DFLOW structures; the Controller ID is identical to the structure ID given when the structure is defined. Later on the structure ID can be changed. However, the controller ID cannot be changed. Controller Names are user-defined and not related to the Structure Name,
2. For single and compound River structures (i.e. River Weir, Advanced Weir, General Structure, River Pump and Database Structure only) both Controller ID's and Names are not related to structure ID's and Names, and Controller Names are user-defined. The same applies for controllers assigned to 2D Breaking – Dams in the Overland Flow module.

Warnings:

- ◇ Suppose that you defined a Flow Measure for a Flow controller assigned to a particular structure, and that you later-on delete this structure in the Flow schematisation. The Real-time Control (RTC) module is not aware that the structure has been deleted and will determine a setpoint for the Flow Measure and provide this setpoint to the Flow controller. You should be aware that this setpoint will be ignored in the hydrodynamic (Flow) computation. Hence, to make things transparent, it is advised to either delete such Flow Measure or make it inactive. The same applies for Rainfall-Runoff RR measures.

◇ River Flow controllers may be assigned to various different type of single and compound River structures. You should be aware that a RTC Flow Measure associated to a River Flow controller might affect the operation of more that one structure. The same applies for controllers assigned to 2D Breaking – Dams in the Overland Flow module.

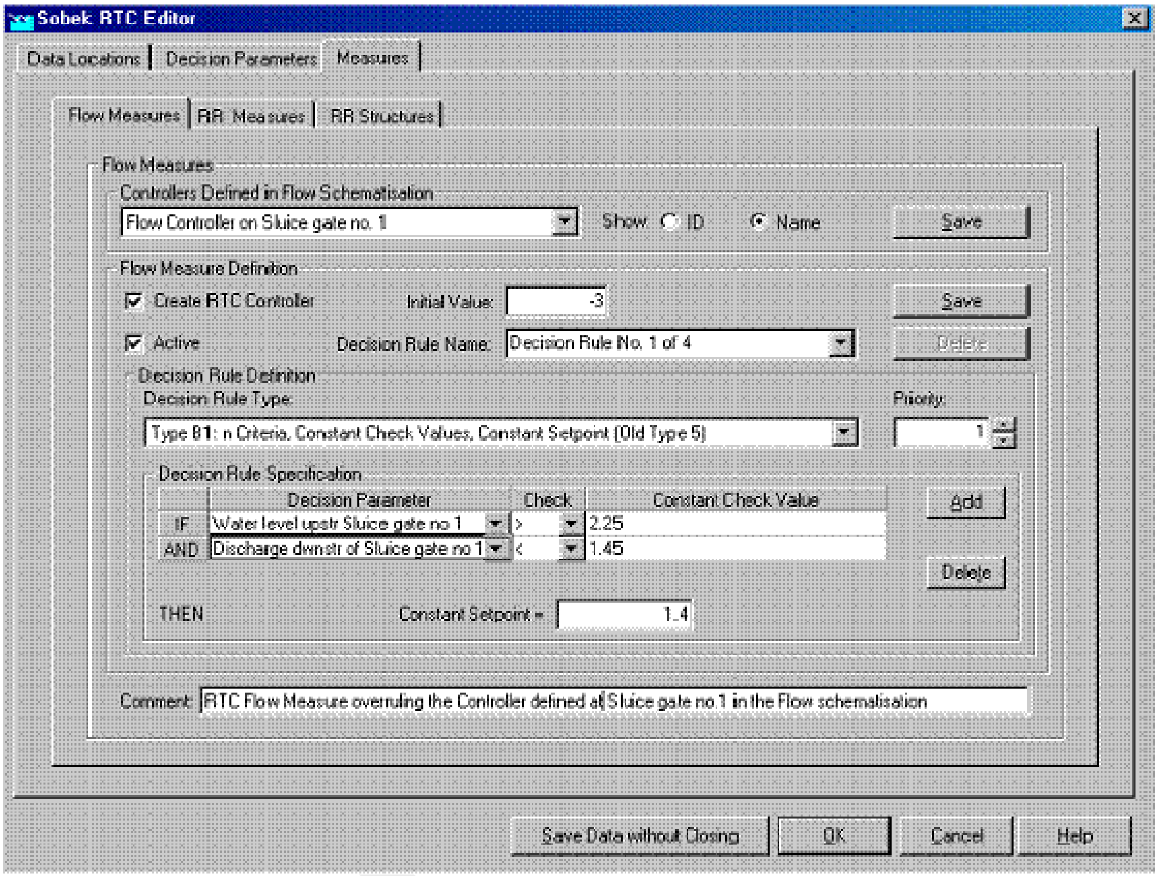


__Figure 5.295:__ Definition of RTC Flow Measures

RTC Flow measures can overrule all six different type of Flow Controllers. The output (or set-point) of a RTC Flow Measure depends on the type of overruled Flow Controller (see Table below).

__Table 5.9:__ Output (or set-point) of RTC Flow Measure as function of overruled Controller type

Flow Controller Type	Output of RTC Flow Measure
Time controller	set point for controlled structure parameter
Relative-time controller	set point for controlled structure parameter
Relative-from-value (time) controller	set point for controlled structure parameter
Hydraulic controller	set point for controlled structure parameter
Interval controller	set point for controlled water level or discharge
PID controller	set point for controlled water level or discharge

Remark:

- ◇ If the RTC Flow Measure is defined active, it means:
  - ◇ for a Time controller, a Relative-time controller, a Relative-from-value (time) controller, and a hydraulic controller, that the associated Flow controller is not active anymore, and
  - ◇ for a Interval controller and a PID controller, that only the set point (steer value) is provided by RTC, but that the controlling mechanisms of these controllers as such are still active.

For each Flow Measure, the user has to define its initial/default value. This initial/default value will be provided as setpoint in case:

1 an initial setpoint is not available for whatsoever reason;
2 in case the defined RTC decision rules do not result in the determination of a setpoint. A Flow Measure may be composed of several so-called "Decision rules".

For each Decision rule, you have to define its priority and its decision rule type. For decision rules comprising of criteria (see Decision rules in RTC – General):

- ◇ The evaluation of a decision rule can either be "true" or "false". In case the evaluation of more than one decision rule is "true", the set-point for the RTC Flow Measure will be equal to the set-point of the decision rule having the highest priority (e.g. lowest integer number).
- ◇ In case the evaluation of more than 1 (one) decision rule having the highest defined priority are "true", the set-point for the RTC Flow Measure will be equal to the set-point of the decision rule that is the closest to end of the drop-down list.
- ◇ In case not one of the defined "Decision rules is "True", than the set-point of the Flow Measure will be equal to the set-point determined in the previous RTC time-step.

Eight different type of decision rules are available for RTC Flow Measures:

- ◇ *Type A: Setpoint $f(t)$ from user-defined Interpolation Table (Internal Type 2)*The user has to select a previously defined decision parameter and a so-called Interpolation Table, with in the first column a value for the selected decision parameter and in the second column the corresponding set point. During the computation on basis of the actual decision parameter value, the set point value for the next computation RTC point-in-time will be determined from the Interpolation Table.
- ◇ *Type B1: n criteria, Constant Check Values, Constant Setpoint (Internal Type 5)*The user can define a number of criteria, that check if previously defined decision parameters are "$<$", "=" or "$>$" than an user-defined constant check-value. The evaluation of the decision rule is only "true" in case all defined criteria are "true", else the evaluation of the decision rule is "false". In case the evaluation of the decision rule is "true", the setpoint will become equal to the user-defined constant set point. In case the evaluation of the decision rule is "false", the set point will be equal to the set-point determined in the previous RTC time-step.
- ◇ *Type B2: n criteria, Constant Check Values, Set point f(t) (Internal Type 6)*Same as Type B1, with the exception that if all defined criteria are "true", the setpoint will be equal to the actual (time) value of a user-defined decision parameter.
- ◇ *Type B3: n criteria, Check Values f(t), Constant Set point (Internal Type 7)* Same at Type B1, with the exception that for check-values previously defined decision parameters can be applied, meaning that these check-values are a function of the actual computational time.
- ◇ *Type B4: n criteria, Check Values f(t), Set point f(t) (Internal Type 8)*Same at Type B3, with

the exception that if all defined criteria are “true”, the setpoint will be equal to the actual (time) value of a user-defined decision parameter.

- *Type B5: Unconditional Setpoint f(t) (Internal Type 10)*Set points will be the values of the specified decision parameter.
- *Type C1: Set point from Matlab simulation (Internal Type 9)*The value for the set point will be retrieved from a Matlab computation. The user has to define a Matlab ID, which will be linked to ID of the selected Controller in the Flow schematisation. In addition the user has to define a “Default value”. The RTC set point will be taken equal to this “Default value” in case during a computation no set-point is provided by Matlab (i.e. SOBEKC_<Matlab ID> = 0) For further information on the use of Matlab, reference is made to the topics: RTC Wind/Rain/Matlab/Reservoir Control options in Settings, RTC-Matlab communication – General, Flow structure parameters in RTC and Matlab and Example of a Matlab m-file.
- *Type C2: Set point from external TCN measure (Internal Type 12)*The value for the set point will be retrieved from a TCN (TeleControlNet) computation. The user has to define an ID, which will be linked to ID of the selected Controller in the Flow schematisation. In addition the user has to define a “Default value”. The RTC set point will be taken equal to this “Default value” in case during a computation no set-point is provided by TCN. For further information on the use of TCN, reference is made to the topics: RTC Wind/Rain/Matlab/Reservoir Control options in Settings (see section 5.12.24).

It is to be mentioned that from the v2.10 SOBEK version onwards, the old decision rule Types 1, 3 and 4 are not anymore available, since they are special cases of respectively Type B1 to B4. When opening a model created in a previous SOBEK version (i.e. < v2.10), the old decision rule Types 1, 3 and 4 will be automatically be updated to the new B1, B2, B3 and B4 ones. The same applies for the old decision rule Types 2, 5, 6, 7 and 8.

5.12.11 Flow structure parameters in RTC and Matlab

The type of data series, available at (Flow) Structures on the 1D Flow Data Tab and 2D Flow Data Tab under Data locations in RTC, depend on the type of structure. These data series can be transferred by RTC to Matlab. In the Table below an overview is given of available data per structure type, including its Matlab string (see also RTC Matlab Communication – General, Example of a MATLAB m-file).

Table 5.10: *Data per structure type for RTC Matlab communication*

Structure type	Available Parameters	String in Matlab m-file
“Flow - External Weir”	1. Crest level 2. Structure flow area 3. Discharge structure 4. Structure velocity 5. Water level Up 6. Water level Down 7. Head 8. Pressure difference	1. SOBEKCL_<SOBEK structure ID>=X 2. SOBEKFA_<SOBEK structure ID>=X 3. SOBEKQS_<SOBEK structure ID>=X 4. SOBEKVS_<SOBEK structure ID>=X 5. SOBEKHU_<SOBEK structure ID>=X 6. SOBEKHD_<SOBEK structure ID>=X 7. SOBEKDH_<SOBEK structure ID>=X 8. SOBEKPD_<SOBEK structure ID>=X

Table 5.10: *Data per structure type for RTC Matlab communication*

Structure type	**Available Parameters**	**String in Matlab m-file**
“Flow - External Orifice”	1. Opening height 2. Structure flow area 3. Discharge structure 4. Structure velocity 5. Water level Up 6. Water level Down 7. Head 8. Pressure difference	1. SOBEKGO_<SOBEK structure ID>=X 2. SOBEKFA_<SOBEK structure ID>=X 3. SOBEKQS_<SOBEK structure ID>=X 4. SOBEKVS_<SOBEK structure ID>=X 5. SOBEKHU_<SOBEK structure ID>=X 6. SOBEKHD_<SOBEK structure ID>=X 7. SOBEKDH_<SOBEK structure ID>=X 8. SOBEKPD_<SOBEK structure ID>=X
“Flow - External Culvert”	1. Gate lower edge level 2. Structure flow area 3. Discharge structure 4. Structure velocity 5. Water level Up 6. Water level Down 7. Head	1. SOBEKGL_<SOBEK structure ID>=X 2. SOBEKFA_<SOBEK structure ID>=X 3. SOBEKQS_<SOBEK structure ID>=X 4. SOBEKVS_<SOBEK structure ID>=X 5. SOBEKHU_<SOBEK structure ID>=X 6. SOBEKHD_<SOBEK structure ID>=X 7. SOBEKDH_<SOBEK structure ID>=X
“Flow - External Pump station”	1. Pump capacity 2. Pump structure 3. Water level Up 4. Water level Down 5. Head	1. SOBEKPC_<SOBEK structure ID>=X 2. SOBEKQS_<SOBEK structure ID>=X 3. SOBEKHU_<SOBEK structure ID>=X 4. SOBEKHD_<SOBEK structure ID>=X 5. SOBEKDH_<SOBEK structure ID>=X
“Flow - Internal Weir”	see External Weir	see External Weir
“Flow - Internal Orifice”	see External Orifice	see External Orifice
“Flow - Internal Culvert”	see External Culvert	see External Culvert
“Flow - Internal Pump station”	see External Pump station	see External Pump station
“Flow - Weir”	see External Weir	see External Weir
“Flow - Orifice”	see External Orifice	see External Orifice
“Flow - Culvert”	see External Culvert	see External Culvert
“Flow - Pump station”	see External Pump station	see External Pump station
“Flow - Universal weir”	1. Discharge structure 2. Water level Up 3. Water level Down 4. Head	1. SOBEKQS_<SOBEK structure ID>=X 2. SOBEKHU_<SOBEK structure ID>=X 3. SOBEKHD_<SOBEK structure ID>=X 4. SOBEKDH_<SOBEK structure ID>=X

Table 5.10: *Data per structure type for RTC Matlab communication*

Structure type	Available Parameters	String in Matlab m-file
“Flow - Bridge”	1. Discharge structure 2. Water level Up 3. Water level Down 4. Head	1. SOBEKQS_<SOBEK structure ID>=X 2. SOBEKHU_<SOBEK structure ID>=X 3. SOBEKHD_<SOBEK structure ID>=X 4. SOBEKDH_<SOBEK structure ID>=X
“Flow - River weir”	1. Crest level 2. Crest width 3. Structure flow area 4. Discharge structure 5. Structure velocity 6. Water level Up 7. Water level Down 8. Head 9. Pressure difference	1. SOBEKCL_<SOBEK structure ID>=X 2. SOBEKCW_<SOBEK structure ID>=X 3. SOBEKFA_<SOBEK structure ID>=X 4. SOBEKQS_<SOBEK structure ID>=X 5. SOBEKVS_<SOBEK structure ID>=X 6. SOBEKHU_<SOBEK structure ID>=X 7. SOBEKHD_<SOBEK structure ID>=X 8. SOBEKDH_<SOBEK structure ID>=X 9. SOBEKPD_<SOBEK structure ID>=X
“Flow - Advanced weir”	1. Crest level 2. Discharge structure 3. Water level Up 4. Water level Down 5. Head 6. Pressure difference	1. SOBEKCL_<SOBEK structure ID>=X 2. SOBEKQS_<SOBEK structure ID>=X 3. SOBEKHU_<SOBEK structure ID>=X 4. SOBEKHD_<SOBEK structure ID>=X 5. SOBEKDH_<SOBEK structure ID>=X 6. SOBEKPD_<SOBEK structure ID>=X
“Flow - General structure”	1. Crest level 2. Crest width 3. Gate lower edge level 4. Structure flow area 5. Discharge structure 6. Structure velocity 7. Water level Up 8. Water level Down 9. Head 10. Pressure difference	1. SOBEKCL_<SOBEK structure ID>=X 2. SOBEKCW_<SOBEK structure ID>=X 3. SOBEKGL_<SOBEK structure ID>=X 4. SOBEKFA_<SOBEK structure ID>=X 5. SOBEKQS_<SOBEK structure ID>=X 6. SOBEKVS_<SOBEK structure ID>=X 7. SOBEKHU_<SOBEK structure ID>=X 8. SOBEKHD_<SOBEK structure ID>=X 9. SOBEKDH_<SOBEK structure ID>=X 10. SOBEKPD_<SOBEK structure ID>=X
“Flow - Database structure”	1. Crest level 2. Discharge structure 3. Water level Up 4. Water level Down 5. Head 6. Pressure difference	1. SOBEKCL_<SOBEK structure ID>=X 2. SOBEKQS_<SOBEK structure ID>=X 3. SOBEKHU_<SOBEK structure ID>=X 4. SOBEKHD_<SOBEK structure ID>=X 5. SOBEKDH_<SOBEK structure ID>=X 6. SOBEKPD_<SOBEK structure ID>=X
“Flow - River pump”	1. Pump capacity 2. Pump structure 3. Water level Up 4. Water level Down 5. Head	1. SOBEKPC_<SOBEK structure ID>=X 2. SOBEKQS_<SOBEK structure ID>=X 3. SOBEKHU_<SOBEK structure ID>=X 4. SOBEKHD_<SOBEK structure ID>=X 5. SOBEKDH_<SOBEK structure ID>=X

Table 5.10: Data per structure type for RTC Matlab communication

Structure type	Available Parameters	String in Matlab m-file
Compound "Flow - River weir" member	see River weir	see River weir
Compound "Flow - Advanced weir" member	see Advanced weir	see Advanced weir
Compound "Flow - General structure" member	see General structure	see General structure
Compound "Flow - Database structure" member	see Database structure	see Database structure
Compound "Flow - River pump" member	see River pump	see River pump
2D Breaking - Dam	Water level Water depth Bottom(bed) level U-velocity V-velocity Abs. velocity [=$sqrt(U^2 + V^2)$]	1. SOBEK1D2DH_<SOBEK ID>=X 2. SOBEK1D2DWD_<SOBEK ID>=X 3. SOBEK1D2DBL_<SOBEK ID>=X 4. SOBEK1D2DU_<SOBEK ID>=X 5. SOBEK1D2DV_<SOBEK ID>=X 6. SOBEK1D2DC_<SOBEK ID>=X

5.12.12 Measures – general

Real-time Control allows to define measures for Water Flow and for the Rainfall-Runoff. A measure is a decision rule using one or more decision parameters. A measure can be active or inactive, based on the evaluation of a logical condition. When a measure is active, it can change the set point of a structure in Water Flow or change the availability of pumps in Rainfall-Runoff. Several measures can apply to one object (e.g. a structure in the hydrodynamic module Water Flow).

Measures can be complementary in the sense that one measure defines the set point of the structure for low flow conditions, one for medium flow conditions, and one for high flow conditions (in which the conditions are mutually exclusive, and cover all possible situations). In this case there is always only one active measure.

But also it is possible that several measures are active at the same time. In this case the measure with the highest priority determines the set point. In case of equal priorities, the first/last active measure determines the set point.

The Real-Time Control module in fact adjusts the settings of a Flow controller. RTC control can therefore only be applied on Flow-structures which are using a controller (a time controller, hydraulic controller, interval controller or PID controller). The interpretation of the RTC results in the Flow module is depending on the type of controller defined:

Type of local controller in Flow module	Interpretation of RTC result
Time controller	setting of control parameter
Hydraulic controller	setting of control parameter
Interval controller	set point
PID controller	set point

So for a time controller or hydraulic controller you can directly set the weir crest level, gate opening height or pump capacity using RTC.

For an interval controller or PID controller, you adjust the set point of the controller, i.e. the the desired water level or discharge at the specified measurement location of the local Flow controller. The Flow controller will than adjust the control parameter (weir crest level, gate opening height or pump capacity) according to the local controller rules.

Real-time Control measures for Rainfall-Runoff can only be defined for all structures in Rainfall-Runoff, but they are typically applied for RR-pump stations. With RTC, you can temporarily switch off the structure (i.e. set the flow to zero) in Rainfall-Runoff based on conditions outside Rainfall-Runoff. Using Matlab, for RR-pumps also the switch-on and -off levels can be adjusted.

5.12.13 Precipitation Data in RTC

On the Precipitation Data Tab available under the Data locations Tab in Real-time Control (RTC), you can select the precipitations stations available for use in RTC. It is possible to select Precipitation data on basis of rainfall data defined for individual rainfall stations.

5.12.14 Rainfall-Runoff measure

Real-time Control measures for Rainfall-Runoff can be defined for all structures in Rainfall-Runoff. They are typically applied for RR-pump stations. This can be either an inlet or an outlet pump. With RTC, you can temporarily switch off the structure (i.e. set the flow to zero) in Rainfall-Runoff based on conditions outside Rainfall-Runoff. Using Matlab, for RR-pumps also the switch-on and -off levels can be adjusted.

A typical application for a combined Water Flow and Rainfall-Runoff network in wet situations is the following: in order to prevent too high water levels at the Water Flow network, outlet pumps in the Rainfall-Runoff schematisation are switched off if the water level at certain locations in the Water Flow network is exceeding a critical maximum level.

A similar type of rule can be applied for dry situations: in that case the Rainfall-Runoff inlet pumps are switched off if the water level in the Channel Flow network is below a critical minimum level.

The measures for RR can be defined in the user-interface using screens like shown below.

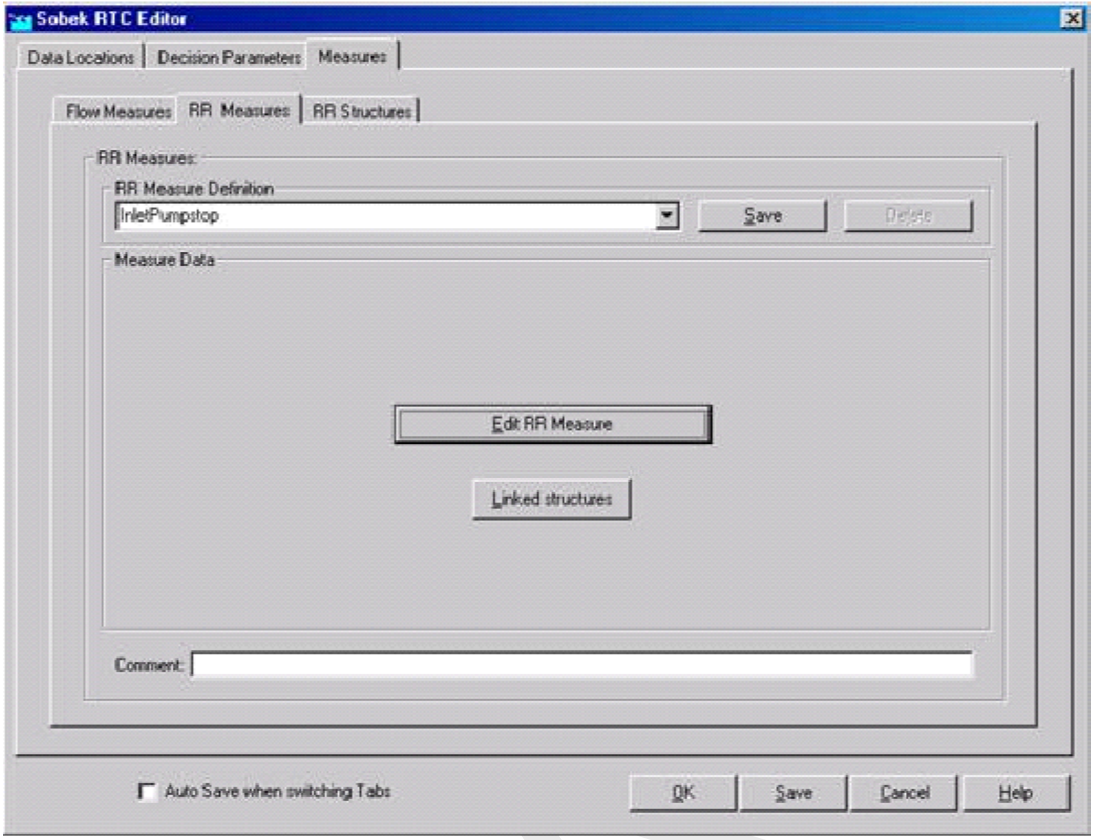


Figure 5.296: ***SOBEK RTC Editor*** *window, the tab Measure → RR Measure*

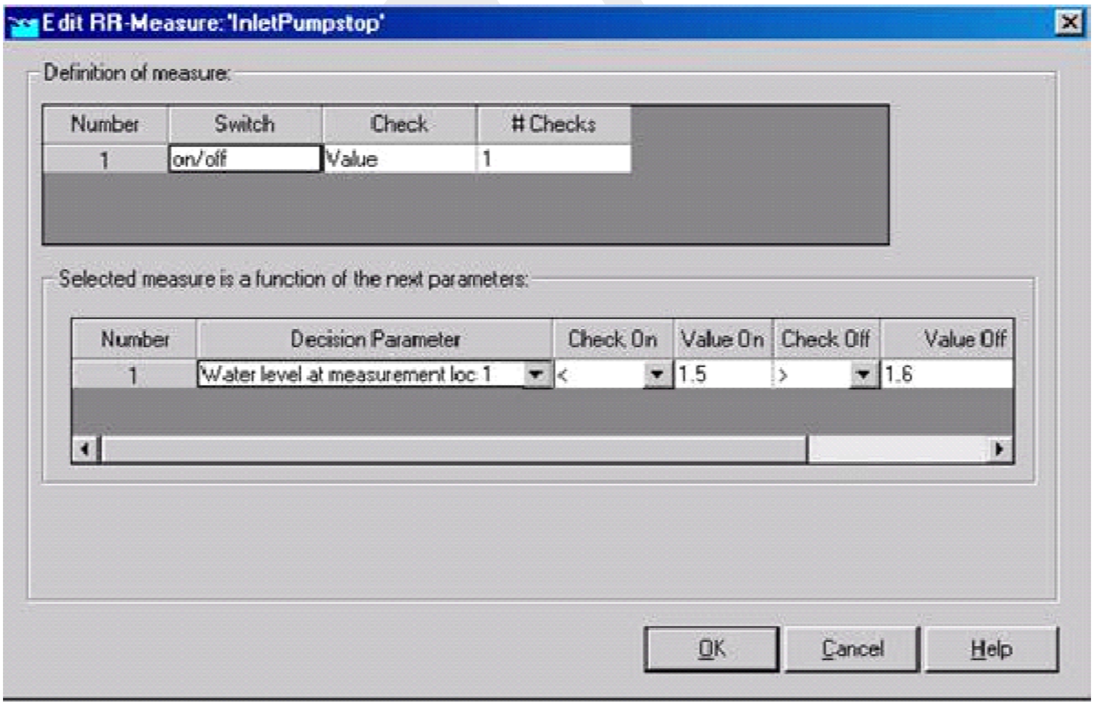


Figure 5.297: *Definition of RR measure*

5.12.15 Real-time Control concepts or elements

There are three important concepts (or elements) in Real-time Control. They are:

- ◇ Data locations;
- ◇ Decision parameters; and
- ◇ Measures;

5.12.16 RR Data in RTC

On the RR Data Tab, available under the Data locations Tab, you can define the Rainfall-Runoff (RR) locations for which RTC will receive data, computed by the Rainfall-Runoff module.

The following type of RR Data are available:

- ◇ open water level at RR Open water nodes [m AD];
- ◇ groundwater level at RR unpaved nodes [m AD];

5.12.17 RTC Communication-General

At present the Real-time Control (RTC) module can be used in combination with following SOBEK modules:

- ◇ The SOBEK-Urban 1DFLOW module,
- ◇ The SOBEK-Rural 1DFLOW module,
- ◇ The SOBEK-River 1DFLOW module,
- ◇ The SOBEK-Rural RR module, and
- ◇ The SOBEK-Rural 1DWAQ module (Note: 1DWAQ data can not yet be defined in the RTC Editor)

In the RTC module use can be made of the user-defined Meteo (rainfall & wind) data as well as user-defined external data.

Further on the RTC module can be linked to online Matlab computations, allowing for any kind of control mechanism to be incorporated into SOBEK.

In the above mentioned options (i.e. linkage with other SOBEK modules, the use of other data sources, and the use of Matlab, the RTC module take cares of the following data exchange:

RTC – SOBEK-Urban 1DFLOW, SOBEK-Rural 1DFLOW and SOBEK-River 1DFLOW modules:

- ◇ Exchange of Flow (computational) data towards RTC,
- ◇ Exchange of computed setpoints from RTC towards Flow controllers

RTC – SOBEK-Rural RR module:

- ◇ Exchange of Rainfall Runofff (computational) data towards RTC,
- ◇ Exchange of computed setpoints from RTC towards RR controllers

RTC – other data sources:

- ◇ Transfer of other (rain, wind, external) Data towards RTC

RTC – Matlab computation:

- Exchange of Flow (computational) data from RTC towards Matlab
- Exchange of Rainfall Runofff (computational) data from RTC towards Matlab,
- Exchange of other (rain, wind, external) data from RTC towards Matlab
- Exchange of by Matlab computed setpoints for RR controllers towards RTC
- Exchange of by Matlab computed setpoints for Flow controllers towards RTC

5.12.18 RTC definitions and options in Settings

The SOBEK Real-Time Control (RTC) module can be activated by checking the "Realtime control" check-box in Settings (see Figure below). The RTC module runs simultaneously with other SOBEK modules, since RTC may overrule controllers defined in other SOBEK modules. In addition you might like to use data, computed by other SOBEK modules for defining decision rules. These decision rules determine the setpoints for the controllers. At present SOBEK modules that can run simultaneously with the RTC module are:

- SOBEK-Rural RR module
- SOBEK-Rural 1DFLOW module
- SOBEK-Urban 1DFLOW module
- SOBEK-River 1DFLOW module
- SOBEK Overland Flow module
- 1DWAQ module (Note: 1DWAQ data can not yet be defined in the RTC Editor)

By pressing the Edit button behind the Realtime control module (see Figure 5.298), following RTC options can be specified:

- RTC Time settings
- RTC Wind/Rain/Matlab/Reservoir Control options
- RTC Output options

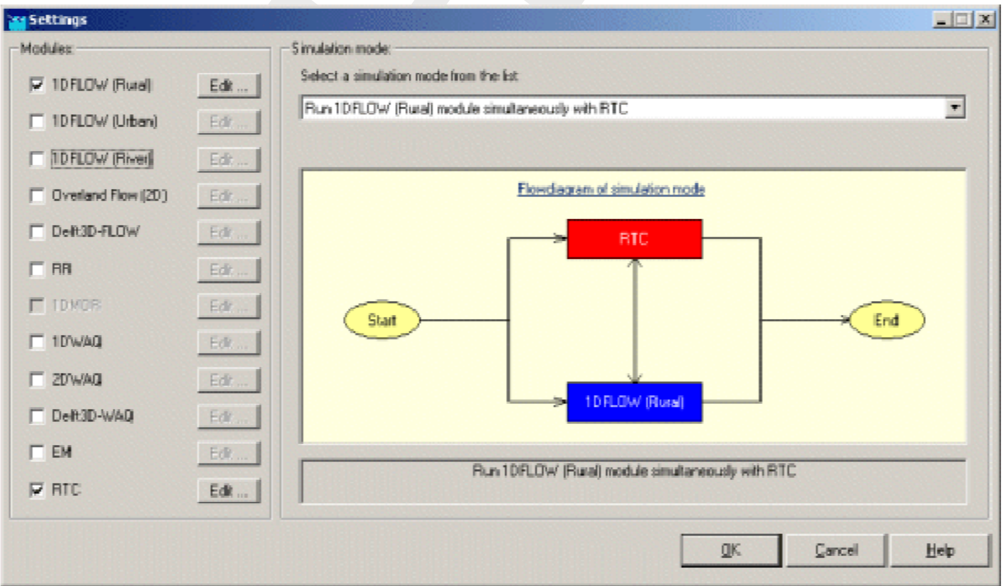


Figure 5.298: *Settings input screen for SOBEK-Rural 1DFLOW-RTC run*

5.12.19 RTC does not overrule Flow Triggers

In Real-time Control (RTC), RTC Measures (or RTC Flow controllers) can be defined. In case a RTC Measure is defined active, this Measure will provide a setpoint for its associated controller in either a Flow or Rainfall-Runoff schematisation. In case the associated controller is active, the setpoint provided by the RTC Measure will be used in either the Flow or Rainfall-Runoff computation. Hence in such situation the RTC Measure overrules the controller.

RTC does, however, not overrule *Triggers* defined for River Flow controllers. The evaluation of these triggers determine whether the controller is active or not active during the computation. In case the controller is in-active, a setpoint provided by a RTC Measure will be neglected and the setpoint of the in-active controller will be kept equal to the previously applied setpoint value.

Note: that in case no triggers are defined for controllers in a model schematisation, this means that these controllers are always active during a computation.

5.12.20 RTC - Matlab Coupling

You can use RTC together with Matlab for overruling a particular controller defined in your Flow or Rainfall-Runoff (RR) schematisation (see also: Example of a MATLAB m-file). More precisely this means that the setpoints of this controller are determined in a Matlab computation, and that these Matlab setpoints are transferred from Matlab to this Flow or RR controller by RTC. For determining the setpoints in Matlab, you can make use of the data series available at the data locations that you defined in the RTC module. Hence the coupling between RTC and Matlab consists of:

- ◇ Transfer of data series by RTC to Matlab,
- ◇ Transfer of computed setpoints by Matlab to RTC

Note:

1.
   - ◇ Flow controllers that are not actived by their associated triggers, will not be overruled by an active RTC-Matlab Measure (see section 5.12.19);
   - ◇ Flow controllers having no associated triggers will always be active and hence will always be overruled by RTC-Matlab.
2. In order to facilitate the coupling between Matlab and RTC, installing Matlab and SOBEK might not be enough. If RTC informs you that Matlab cannot be found, Matlab will need to manually registered for coupling through *com* files. This can be done by typing this command in your Matlab installation directory:
   - ◇ matlab.exe /regserver

   Please use the Windows option 'Run as Administrator' to run this command.

Ad 1) Transfer of data series by RTC to Matlab:

All data series available at the data locations (see section 5.12.3) that you defined in the RTC module, can be used in a Matlab computation. Furthermore the RTC computational point-in-time and the RTC computational time-step can be passed to Matlab. The corresponding "strings" to be used in a Matlab m-file are given in the Table below. An overview of the type of data series that are available for each Flow structure is given in section 5.12.11.

Table 5.11: *Table: Strings in a Matlab m-file for obtaining values of data series that are available at data locations defined in the RTC module*

String in Matlab m-file	**Explanation**
SOBEKFirst=X	Indication whether this is the first simulation timestep (X=1) or not (X=0)
SOBEKDate=X	Date of simulation step. The date is given as a string of the form yyyymmdd (e.g. 20041231 is December 31st, 2004)
SOBEKTime=X	Time of simulation step. The time is given as a string of the form HHmmsshh, where HH are “hours” and hh are “seconds/100”. So 22150000 is a quarter past ten in the evening.
SOBEKCompTimestepSize=X	RTC computational time-step (as defined in Settings) in seconds
SOBEKH_<SOBEK node id>=X	water level
SOBEKQ_<SOBEK branchsegment id>=X	discharge
SOBEKSA_<SOBEK node id>=X	surface area
SOBEKWD_<SOBEK node id>=X	water depth
SOBEKCL_<SOBEK structure id>=X	crest level of structure
SOBEKCW_<SOBEK structure id>=X	crest width of structure
SOBEKGL_<SOBEK structure id>=X	gate lower edge level (orifice)
SOBEKGO_<SOBEK structure id>=X	opening height (orifice)
SOBEKFA_<SOBEK structure id>=X	structure flow area
SOBEKQS_<SOBEK structure id>=X	discharge structure
SOBEKVS_<SOBEK structure id>=X	velocity at structure
SOBEKHU_<SOBEK structure id>=X	water level up
SOBEKHD_<SOBEK structure id>=X	water level down
SOBEKDH_<SOBEK structure id>=X	head over structure
SOBEKPD_<SOBEK structure id>=X	pressure difference over structure
SOBEKPC_<SOBEK structure id>=X	pump capacity (pump)
SOBEK1D2DH_<location id>=X	2D water level
SOBEK1D2DWD_<location id>=X	2D water depth
SOBEK1D2DBL_<location id >=X	2D bed level
SOBEK1D2DU_<location id >=X	2D U-velocity
SOBEK1D2DV_<location id >=X	2D V-velocity
SOBEK1D2DC_<location id >=X	2D Abs velocity $[= \sqrt{U^2 + V^2}]$
RRH_<RR id>=X	RR water level or groundwater level
RainH_<rain_id>=X	precipitation

Ad 2) Transfer of computed setpoints by Matlab to RTC

In a Matlab computation, the setpoints for a Flow or Rainfall-Runoff (RR) controller can be determined. These Matlab setpoints are transferred by the RTC module to the corresponding Flow or RR controller. The corresponding “strings” to be used in a Matlab m-file are given in the Table below. For Flow controllers, the id in the Matlab string is the “Matlab id” defined on the Flow Measures Tab in RTC (see decision rule Type C in Flow Measures in RTC). For RR controllers, the id in the Matlab string is the id of the structure in the RR schematisation.

Table 5.12: Table: Strings in a Matlab m-file for transferring computed setpoints for a particular Flow or RR controller towards the RTC module.

String in Matlab m-file	Explanation
SOBEKC_<Matlab id>=X	optional, controlled by RTC yes(=1)/no(=0)
SOBEKS_<Matlab id>=X	setpoint of SOBEK-Flow controller
RRSLowon<RRStructure id>=X	RR-pump switch on level, low capacity
RRSLowoff<RRStructure id>=X	RR-pump switch off level, low capacity
RRSHighon<RRStructure id>=X	RR-pump switch on level, high capacity
RRSHighoff<RRStructure id>=X	RR-pump switch off level, high capacity

Note: ID's in Matlab can only consist of letters, digits and underscores. Matlab ID's can not contain dots, plus signs, minus signs, equal or not-equal signs, etc., since these are interpreted by Matlab. However, SOBEK can handle these signs in ID's. Therefore, in sending data to Matlab all these characters are changed into underscores. Hence unique ID's in SOBEK might not be unique ID's in Matlab, in case SOBEK ID's contain the above mentioned characters that are interpreted by Matlab. Also Matlab will not accept these signs in the ID's for which data has to be sent to SOBEK.

Example: Suppose the water level at SOBEK node with id 0-64 is 0.55 m, and this node is specified as a data location for RTC. SOBEK-RTC will then pass the following string to Matlab: SOBEKH_0_64=0.55

Note that the − sign in the SOBEK-id is replaced by the _ sign in the string put to Matlab.

5.12.21 RTC - TCN (Telecontrolnet) coupling

In a SOBEK computation, the real time control (RTC) module can be coupled with the external Telecontrolnet software, which is developed by InterAct (www.telecontrolnet.nl). This coupling is, hereafter, shortly referred to as the RTC - TCN coupling.

Using the RTC - TCN coupling, any triggered (or active, see section 5.12.19) controller in a 1D Flow schematisation can be overruled by TCN. Overruling means that the setpoint for an overruled controller is determined by TCN. Such setpoint is determined by TCN on basis of values of state parameters (water levels, discharges etc.) in the Flow schematisation, that are provided by the RTC module.

Note: Overruling does not affect the algorithm applied by a controller in a Flow schematisation. Say that a PID controller has to main water levels upstream of a weir by adjusting the crest level of this weir. Overruling this PID controller means that TCN only provides the values for the water levels to be maintained upstream of this weir. The controller using its PID algorithm, still determines the weir crest levels required for maintaining these (by TCN provided) water levels upstream of the weir.

Specifying a RTC -TCN coupling comprises of:

1 In the Settings task block, click on <Edit> button next to RTC and open the Contol options tab (see Figure 5.299). On the Control options tab:

   1.1 Check the check-box "Use TCN-Coupling"

   1.2 In the box next to "TCN Run Command:", specify the path and name of the TCN run command (i.e. a <name>.bat batch file, see example in Figure 5.306), which calls

TCN at the beginning of each RTC computational time step. In a TCN call two or three command line arguments (%1 to %3) are included:

- ◇ **%1**: The RTC event number. A Flow - RTC computation comprises of one (1) event only with a duration, that follows from the begin-time and end-time of the simulation as specified in settings. For a rainfall-runoff(RR) - Flow - RTC computation the number of events is equal to the number of rainfall events (or storms) defined in the Meteo task block (either in the <name>.bui file or in the <name>.rks file). The duration of each event is equal to the duration of its rainfall event.
- ◇ **%2**: The RTC time step number within the event. This is an integer, varying from 1 up to the duration of the event divided by the RTC computational time step.
- ◇ **%3**: Firstrun. This text string is only given for the first RTC time step within a particular event.

1.3 In the box next to "SOBEK to TCN:", specify the path of the <name>.csv file in which the RTC module writes the current values of all specified decision parameters (for more information, see point 2 hereunder).

1.4 In the box next to "TCN to SOBEK:", specify the path of the <name>.csv file in which TCN writes the setpoints for the by TCN overruled controllers (for more information, see point 3 hereunder) and which setpoints are applied in the current RTC computational time step.

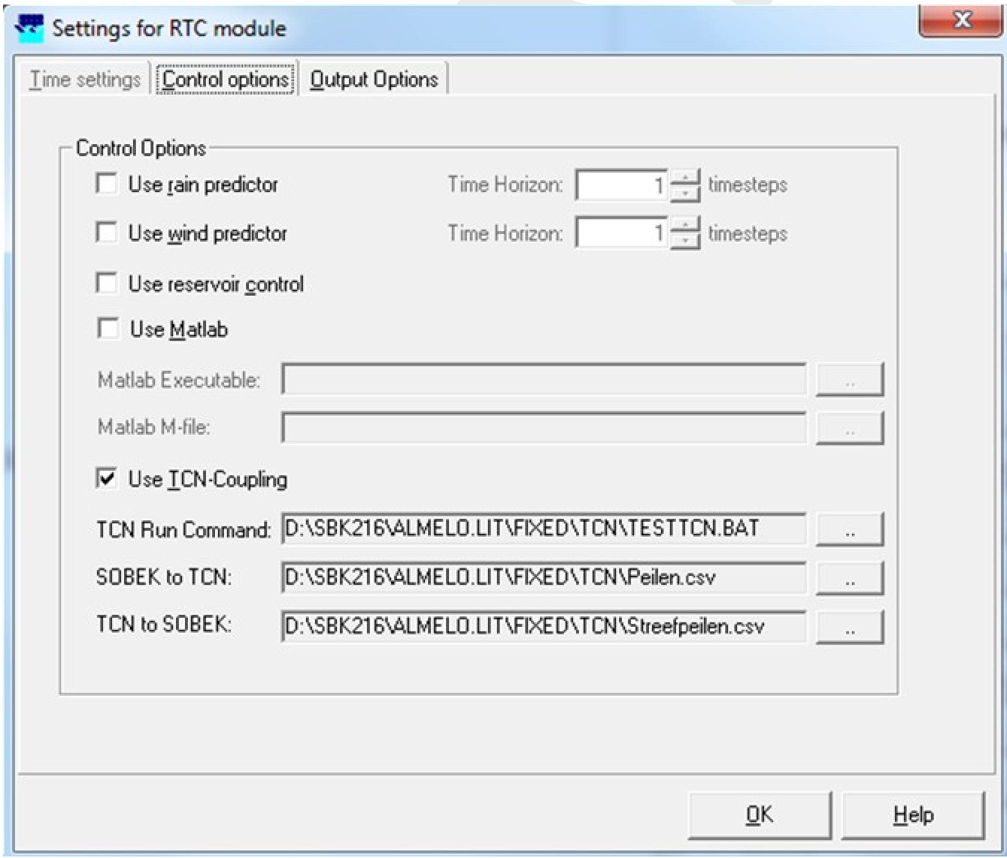


Figure 5.299: *Settings Task block, RTC settings for the RTC - TCN coupling*

2 In the RTC editor (see section 5.12.5), define those decision parameters which values are to be transferred towards TCN in a <name>.csv file (SOBEK to TCN, see Figure 5.299) at the beginning of each RTC computational time step. Please note that the values of all

decision parameters are transferred to TCN.
As an illustration Figure 5.300 shows that decision parameter with id "//11/N1_mwfrtc" is defined as the water level of the 1D water level point with id "10_292810". In Figure 5.301, file <Peilen>.csv is shown in which at the beginning of a particular RTC computational time step, the value of decision parameter with id "//11/N1_mwfrtc" is transferred towards TCN. The first line in file <Peilen>.csv is a header ("id","description","value") line. Each next line contains the information of a specific decision parameter. Please note that on such line the id and the description are between double quotes, so better make sure that the id and description (defined in the RTC editor) itself does not contain a double quote.

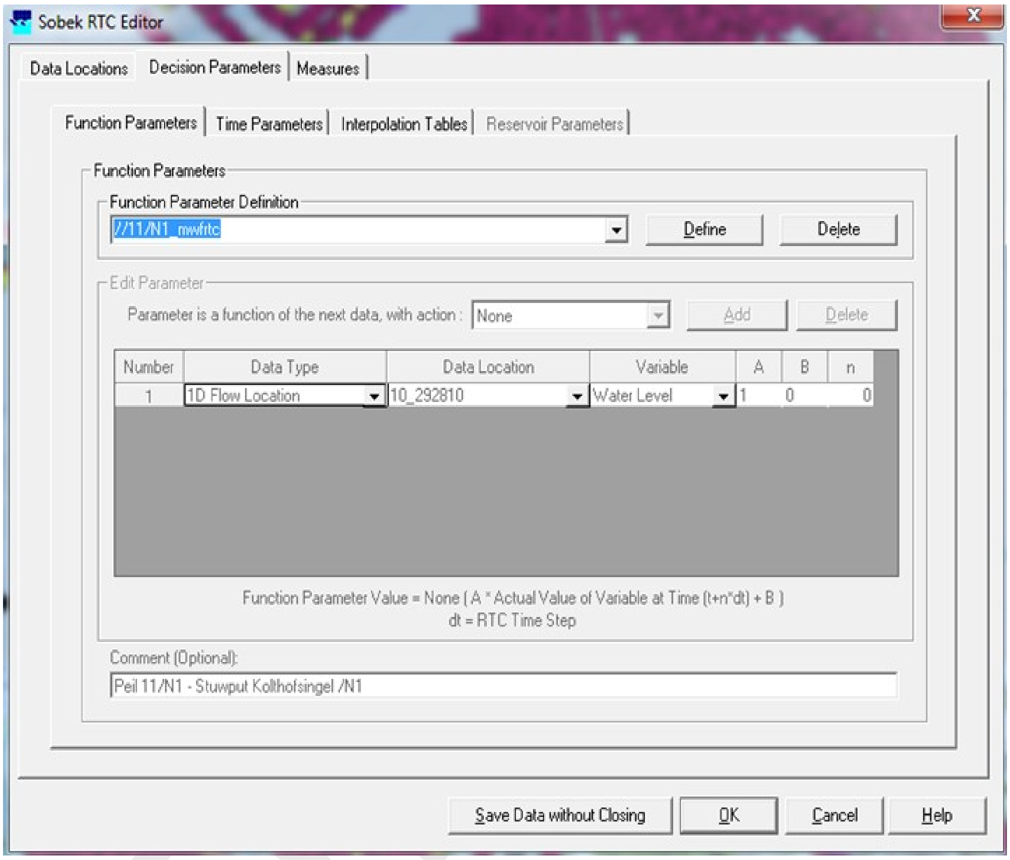


Figure 5.300: *RTC editor in the Schematisation Task block. Definition of RTC decision parameter with id "//11/N1_mwfrtc", which data is transferred towards TCN in a <name>.csv file (SOBEK to TCN, see Figure 5.299) at the beginning of each RTC computational time step (see Figure 5.301).*

```
"id", "description", "value"
"//11/N1_mwfrtc", "Peil 11/N1 – Stuwput Kolthofsingel /N1",    9.046
```

Figure 5.301: *Example file <Peilen.csv> in which RTC at the beginning of each RTC computational time step, transfers the value for decision parameter with id "//11/N1_mwfrtc" towards TCN.*

3 In the RTC editor, define a Flow Measure with decision rule type C2 (including its TCN ID) for each controller in the 1D flow schematisation that is to be overruled by TCN (see section 5.12.10). TCN will provide setpoints for these controllers in a <name>.csv file (TCN to SOBEK, see Figure 5.299) which are applied by RTC in the current RTC computational time step. How TCN determines these setpoints is beyond the scope of this manual.
As an illustration Figure 5.302 shows that 1D flow controller with id "1_751010-21_751010"

is overruled by the setpoint with TCN id "//26/X1_spstrrtc", that is provided by TCN. In Figure 5.303, file <Streefpeilen>.csv is shown in which TCN writes the setpoint with TCN id "//26/X1_spstrrtc", that is used to overrule the 1D flow controller with id "1_751010-21_751010". The first line in file <Streefpeilen>.csv is a header ("id","value") line. Each next line contains the information of a specific TCN setpoint. Please note that on such line the TCN id is between double quotes, so better make sure that the TCN id (defined in the RTC editor) itself does not contain a double quote.

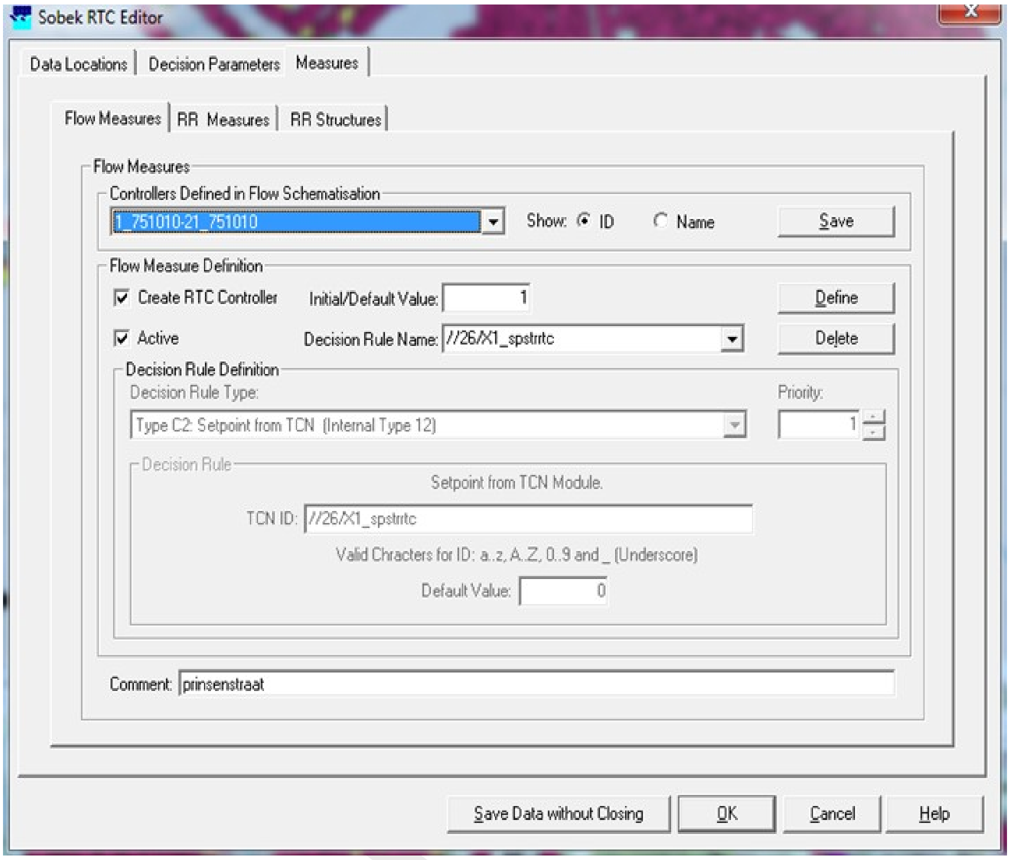


Figure 5.302: *RTC editor in the Schematisation Task block. Defining that 1D flow controller with id "1_751010-21_751010" is to be overruled by the TCN setpoint with TCN id "//26/X1_spstrrtc". TCN provides this setpoint in a <name>.csv file (TCN to SOBEK, see Figure 5.299) at the end of each RTC computational time step (see Figure 5.303).*

```
"id", "value"
"//26/X1_spstrrtc", 8.1
```

Figure 5.303: *Example file <Streefpeilen.csv> in which TCN at the end of each RTC computational time step, transfers the setpoint with TCN id "//26/X1_spstrrtc", which is used by RTC to overrule 1D flow controller with id "1_751010-21_751010".*

4 Specify the communication protocol between RTC and TCN by means of:

4.1 The <SOBEKTCN>.ini file:
Default the SOBEKTCN.ini file is located in the \SOBEK\Programs\RTC\TCN directory. The communication program SOBEKTCN.exe as well as the \Certifi subdirectory are also located in this directory. Please note that if you have several

projects using a RTC - TCN coupling, it is advised to copy the SOBEKTCN.exe and the SOBEKTCN.ini file into a sub-directory under the concerning project directory. The SOBEKTCN.ini (see Figure 5.304) is to be edited manually:

4.1.1 The data behind keywords, which are left empty (being keywords: tcndomain, user, password, client_id and client_secret) are to be filled in with the values supplied by Interact (company that developed the Telecontrolnet software).

4.1.2 The file names behind keyword input_file (default: Peilen.csv) and keyword output_file (default: Streefpeilen.csv) should respectively match with the file names specified in the box behind "SOBEK to TCN" and the box "TCN to SOBEK" in RTC settings (see Figure 5.299). Path names are to be omitted. Please note that on the TCN website, file names are case sensitive, while in SOBEK-RTC they are not case sensitive (since SOBEK runs under Windows).

4.1.3 Value RTC_SYS for keyword location_code means that TCN knows that it concerns a RTC - TCN coupling.

4.1.4 Please note that in case the duration of a TCN call lasts longer than the number of seconds specified with keyword time_out_seconds, the current RTC calculation is finalized using the setpoints that were provided by TCN in the previous RTC calculation time step. In such case, following message is given in the <RTC>.log file: value for parameter id not found in ReadCsvFile and set to previous value, where id refers to the TCN ID of an overruled controller.

4.1.5 An example of a sobektcn.log file or communication file of the RTC - TCN coupling is given in Figure 5.305

```
[general]
url=https://www.telecontrolnet.nl
tcndomain=
user=
password=
client_id=
client_secret=
input_file=Peilen.csv
output_file=Streefpeilen.csv
log_file=sobektcn.log
log_level=DEBUG
location_code=RTC_SYS
time_out_seconds=15
```

Figure 5.304: *Example of the SOBEKTCN.ini file, defining the communication protocol between RTC and TCN in a RTC - TCN coupling.*

```
2017/11/30  16:56:57  Starting SOBEK TCN Adapter
2017/11/30  16:56:57  Retrieving access token
2017/11/30  16:56:57  Starting new HTTPS connection (1): www.telecontrolnet.nl
2017/11/30  16:56:58  https://www.telecontrolnet.nl:443 "POST /oauth/token HTTP/1.1" 200 179
2017/11/30  16:56:58  Successfully retrieved token oGmQdns0bxdgUxMVCE7yejsnV1cI759c8bQH1Mrh
2017/11/30  16:56:58  Retrieving location id
2017/11/30  16:56:58  Starting new HTTPS connection (1): www.telecontrolnet.nl
2017/11/30  16:56:58  https://www.telecontrolnet.nl:443 "GET /api/v1/locations?code=RTC_SYS HTTP/1.1" 200 448
2017/11/30  16:56:58  Successfully retrieved location id 5254435f-5359-53
2017/11/30  16:56:58  Uploading input CSV file
2017/11/30  16:56:58  Starting new HTTPS connection (1): www.telecontrolnet.nl
2017/11/30  16:56:58  https://www.telecontrolnet.nl:443 "POST /api/v1/locations/5254435f-5359-53/files  HTTP/1.1" 201 21
2017/11/30  16:56:58  Successfully uploaded input file D:\SBK216\ALMELO.LIT\FIXED\TCN\Peilen.csv
2017/11/30  16:56:58  Downloading output CSV file
2017/11/30  16:56:58  Starting new HTTPS connection (1): www.telecontrolnet.nl
2017/11/30  16:56:59  https://www.telecontrolnet.nl:443 "GET /api/v1/locations/5254435f-5359-53/files  HTTP/1.1" 200 1900
2017/11/30  16:56:59  Successfully retrieved id of download file Streefpeilen.csv: 53747265-6566-7065-696c-656e2e637376
2017/11/30  16:56:59  Starting new HTTPS connection (1): www.telecontrolnet.nl
2017/11/30  16:56:59  https://www.telecontrolnet.nl:443 "GET /api/v1/locations/5254435f-5359-53/files/53747265-6566-7065-696c-656e2e637376?contents=1  HTTP/1.1" 200 686
2017/11/30  16:56:59  File for file_id 53747265-6566-7065-696c-656e2e637376  is available for download.
2017/11/30  16:56:59  Successfully downloaded output file to D:\SBK216\ALMELO.LIT\FIXED\TCN\Streefpeilen.csv
2017/11/30  16:56:59  Finished SOBEK TCN Adapter
```

Figure 5.305: *Example of a sobektcn.log file or communication file of the RTC - TCN coupling*

4.2 The <name>.bat file

As mentioned in point 1.2 above, TCN is called at the beginning of each RTC computational time step using a <name>.bat batch file. Such batch file not only calls TCN by starting SOBEKTCN.exe, but may as well contain some commands, that store the communication between SOBEK-RTC and TCN. In addition the batch file starts SOBEKTCN.exe with the proper command line arguments (%1 to %3, see point 1.2 above).

In Figure 5.299 TCN is called using batch file <TestTCN>.bat. This batch file is depicted in Figure 5.306 and does following:

4.2.1 The first line changes the working directory from the CMTWORK directory to the directory d:\SBK216\ALMELO.lit\FIXED\TCN

4.2.2 The second line start the SobekTCN.exe, which calls TCN (i.e. starts the RTC - TCN coupling program). The command line argument **%3** denotes if it is the first TCN call within a particular event or not (i.e. textstring is either "firstrun" or ""). Please note that the SOBEKTCN.exe should be located in directory d:\SBK216\ALMELO.lit\FIXED\TCN.

4.2.3 **Optional**. The third line stores all <sobektcn>.log files (see Figure 5.305) in directory d:\SBK216\ALMELO.lit\FIXED\TCN\test, including the RTC event number (**%1**) and the RTC time step number within the event (**%2**).

4.2.4 **Optional**. The fourth line stores all <Peilen>.csv files (see Figure 5.301) in directory d:\SBK216\ALMELO.lit\FIXED\TCN\test, including the RTC event number (**%1**) and the RTC time step number within the event (**%2**).

4.2.5 **Optional**. The fifth line stores all <Streefpeilen>.csv files (see Figure 5.303) in directory d:\SBK216\ALMELO.lit\FIXED\TCN\test, including the RTC event number (**%1**) and the RTC time step number within the event (**%2**).

4.2.6 The last row changes the working directory from d:\SBK216\ALMELO.lit\FIXED\TCN to d:\SBK216\ALMELO.lit\CMTWORK.

Note: The third line, fourth line and fifth line described above and shown in Figure 5.306) are optional (i.e. not essential) in defining a RTC - TCN coupling. These optional lines are, however, handy to verify if a RTC - TCN coupling is working to satisfaction.

```
cd d:\SBK216\ALMELO.LIT\FIXED\TCN
D:\SBK216\ALMELO.LIT\FIXED\TCN\SobekTCN.exe %3
copy sobektcn.log .\test\sobektcn.log%1_%2
copy peilen.csv .\test\peilen.%1_%2
copy streefpeilen.csv .\test\streefpeilen.%1_%2
cd d:\SBK216\ALMELO.LIT\CMTWORK
```

Figure 5.306: *Example <TestTCN>.bat, applied to call TCN and to store all communication files <Peilen>.csv (SOBEK to TCN) and <Streefpeilen>.csv (TCN to SOBEK) produced in a RTC - TCN coupling.*

5.12.22 RTC Output options in Settings

The RTC Output options Tab becomes available by clicking on the *Edit* button behind the RTC module in Settings (see Figure 5.307). On this Tab, the user can define the time-step for the RTC output data, available under the Result in Charts Task block and the Result in Maps Task block. The RTC output time-step should be a multiple of the defined RTC computational time-step.

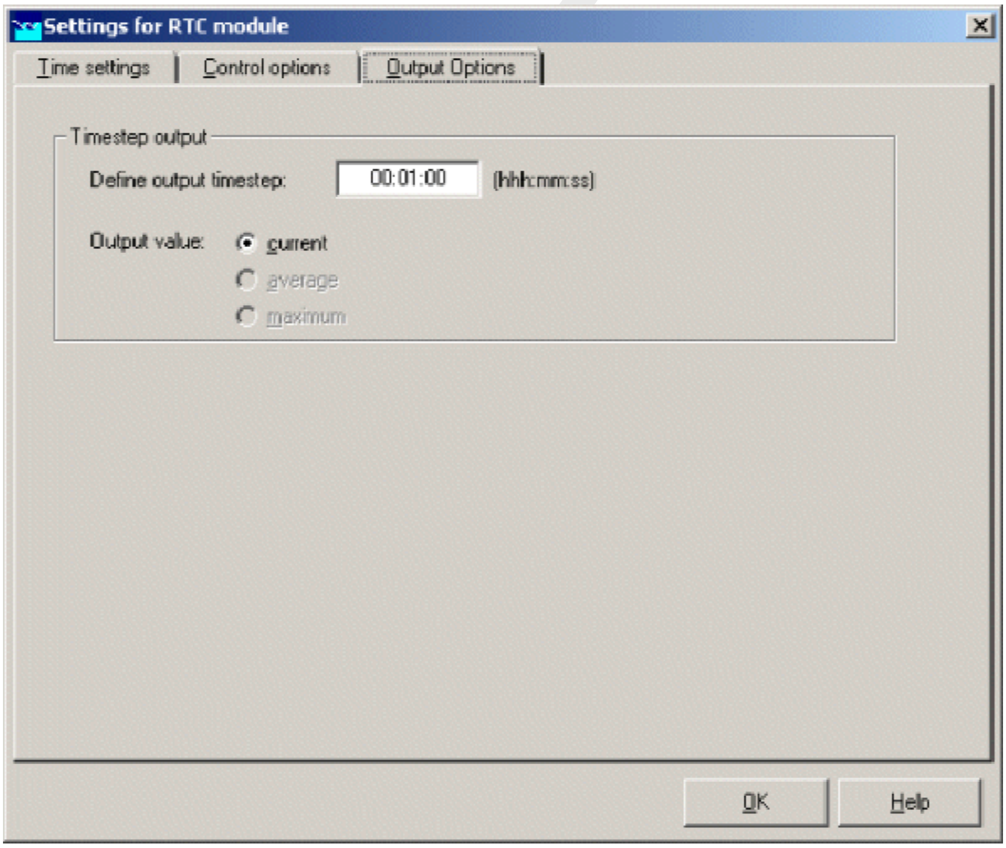


Figure 5.307: *RTC Output Options in Settings*

5.12.23 RTC Time settings (Time-step) in Settings

The RTC Time settings Tab becomes available by clicking on the *Edit* button behind the RTC module in Settings (see Figure below).

The time-step applied in Real-time Control (RTC) is defined in the Settings Task block (see Figure below).

Note:

- ◇ The defined RTC time-step is applied for each and every RTC measure, that might be overruling a controller defined in either the SOBEK-Urban 1DFLOW, SOBEK-Rural 1DFLOW, SOBEK-River 1DFLOW, SOBEK Overland Flow or SOBEK-Rural RR schematisation. This is also valid when the RTC setpoints are determined by a Matlab computation.
- ◇ In a combined Flow-RTC computation, the RTC time-step may be larger than the Flow time-step (i.e. time-step applied in the hydrodynamic computations), this means that the setpoints of the Flow controllers, that are overruled by a RTC Flow measure (or RTC Flow controller) are less frequently updated as the set points of Flow controllers that are not overruled by RTC Flow measures
- ◇ In a combined RR-RTC computation, the RTC time-step should be equal to the time-step applied in the RR computation. The same yields for a combined Flow-RR-RTC computation.

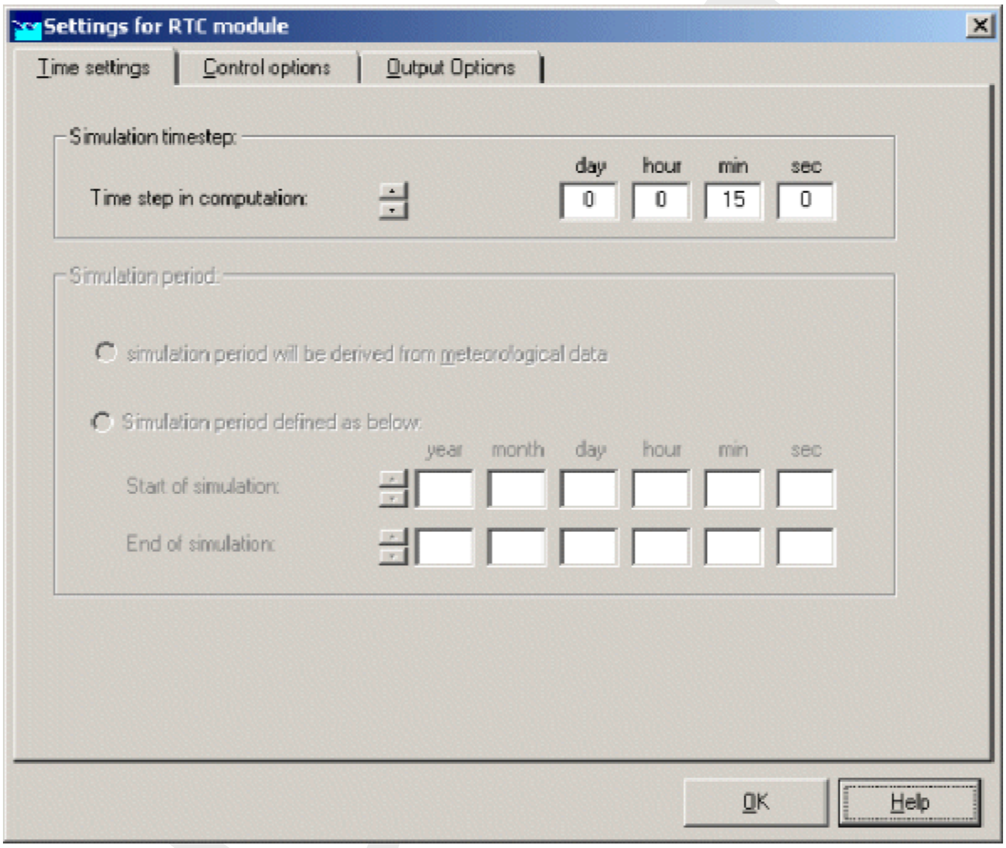


Figure 5.308:** **Settings for RTC module, the **Time Settings** ***tab***

5.12.24 RTC Wind/Rain/Matlab/Reservoir Control options in Settings

By clicking on the *Edit* button behind the RTC module in Settings, the RTC Control options Tab becomes available (see Figure 5.309). By clicking on the corresponding check-box, the following RTC options can be activated:

- ◇ Use rain predictor (rain data to be defined in Meteo Task block),
- ◇ Use wind predictor (rain data to be defined in Meteo Task block)
- ◇ Use Matlab
- ◇ Use reservoir control

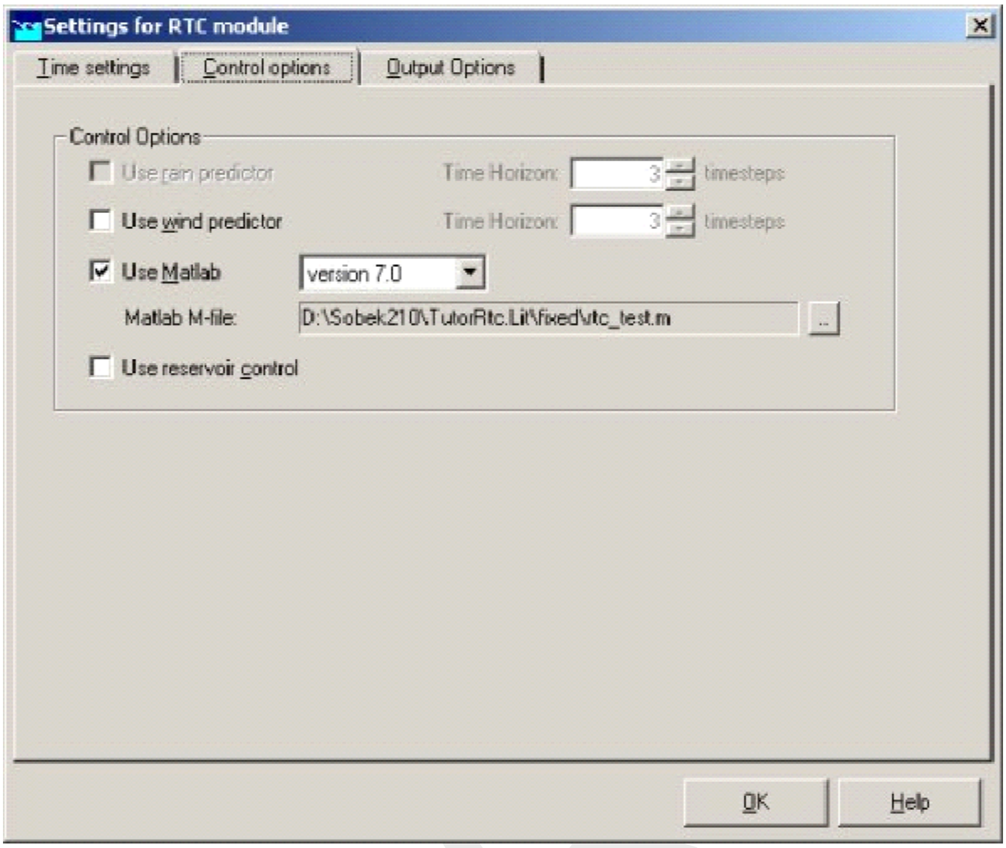


Figure 5.309: *RTC Control options*

Matlab is a general purpose software package, which allows you to write your own programs (so-called m-files) using e.g. 'fuzzy logic' or other control features which are available in Matlab. SOBEK RTC can make use of your Matlab m-files.

In order to use Matlab, it is required to have an installation of Matlab on your own PC or an accessible network drive. It is not necessary to specify to SOBEK where Matlab is located. If Matlab has been properly installed and used on the PC before, SOBEK will be able to communicate with it. However, if you specify a version of Matlab which is different from the version last used on your PC, the SOBEK RTC module will not be able to set up communication with Matlab properly. In that case you will get a message after starting the first timestep of the simulation, saying that there was an error starting Matlab.

To define the use of Matlab, go to the tab <Control Options>, switch on the use of Matlab, and specify which version of Matlab you want to use. SOBEK supports communication with Matlab versions 5.3, 6.0, 6.1, 6.5 and 7.0.

You also have to specify the location of your Matlab m-file.

5.12.25 Setpoints in RTC

If defined active a RTC Measure will overrule an active controller, that is defined in a Flow or a Rainfall-Runoff schematisation. With overruling is meant that the RTC Measure determines the setpoint for the controller (i.e. the gate-height, crest level, water level to be controlled etc.).

Note:

1 Flow controllers that are not actived by their associated triggers, will not be overruled by an active RTC Measure;
2 Flow controllers having no associated triggers will always be active and hence will always be overruled by RTC. For more information, see RTC does not overrule Flow Triggers.

For information on the type of set-points corresponding to each Flow controller, see Flow Measures in RTC.

5.12.26 Type of Measures available in Real-time Control

In Real-time Control following Type of Measures are available:

◇ Flow Measures,
◇ RR Measures,

Flow Measures (or RTC Flow controllers), when defined active, will overrule their associated active Flow controller in the SOBEK-Urban 1DFLOW, SOBEK-Rural 1DFLOW or SOBEK-River 1DFLOW schematisation (see Flow Measures in RTC and RTC does not overrule Flow Triggers).

RR Measures, when defined active, will overrule their associated controller in the Rainfall Runoff schematisation. At present RR Measures are to defined on the RR Measure Tab and the RR Structure Measure Tab.

5.12.27 Wind Data in RTC

On the Wind Data Tab available under the Data locations Tab in Real-time Control (RTC), you can select wind data (velocity and direction) available for use in RTC. At present only the complete set of wind data defined in the Meteo Task block can be selected at the Wind data Tab. Under Decision Parameters in RTC, however, it is possible to make a selection on basis of individual wind stations defined in the Meteo Task block.

5.12.28 1D Flow Data in RTC

On the 1D Flow Data Tab, available under the Data locations Tab, you can define the 1D Flow locations for which RTC will receive data, computed by the SOBEK 1DFLOW modules.

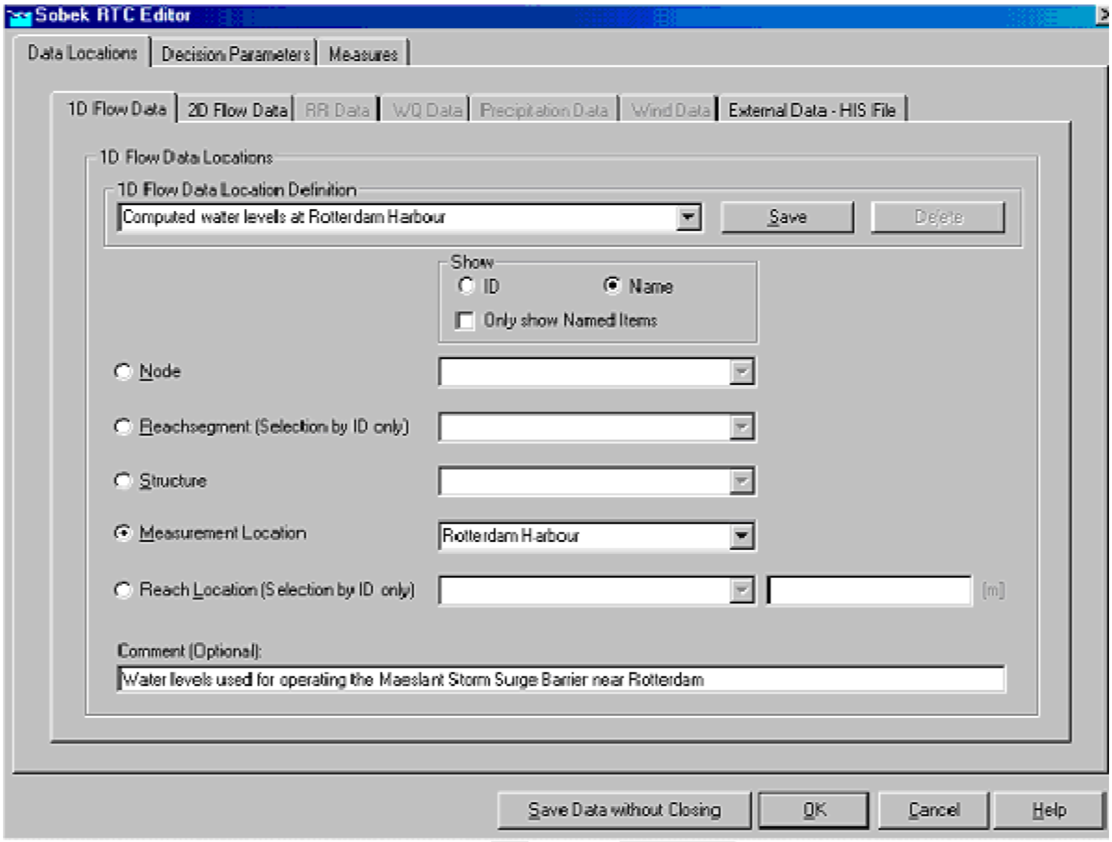


Figure 5.310: *RTC Flow Data input screen*

For each 1D Flow Data location, the user can define its ID as well as an optional comment string (see Figure above). Data locations can be selected on basis of their ID or user-defined Name. Five different type of 1D Flow Data locations are discerned, viz:

- Nodes:
- Branch segments
- Structures
- Measurement locations (or stations)
- Branch locations

Hint: If the “Show Name” option together with a checked “Only show Named Items” check-box is used (see Figure above), than only those objects in the drop-down lists are shown for which the user has defined a Name. In this way you can considerably reduce the length of the drop-down list (and hence the time needed for finding your object).

The data availability depends on the selected 1D Flow Data type. For instance: at Nodes, water levels and water depth are available; at Branch segment: velocities and discharges are available. Below an overview of the available 1D Flow Data (i.e, computed hydrodynamic parameters) is given

- water level [m AD];
- flow $[m^3/s]$;
- surface area $[m^2]$;
- water depth $[m]$;
- crest level $[m]$;
- crest width $[m]$;

- ◇ gate lower edge [m];
- ◇ gate opening height [m];
- ◇ structure flow area [m^2];
- ◇ discharge at the structure [m^3/s];
- ◇ flow velocity at the structure [m/s];
- ◇ water level up [m];
- ◇ water level down [m];
- ◇ head over structure [m];
- ◇ pressure difference over structure;
- ◇ pump capacity [m^3/s];

5.12.29 2D Flow Data in RTC

On the 2D Flow Data Tab, available under the Data locations Tab, you can define the 2D Flow locations for which RTC will receive data, computed by the SOBEK Overland Flow module.

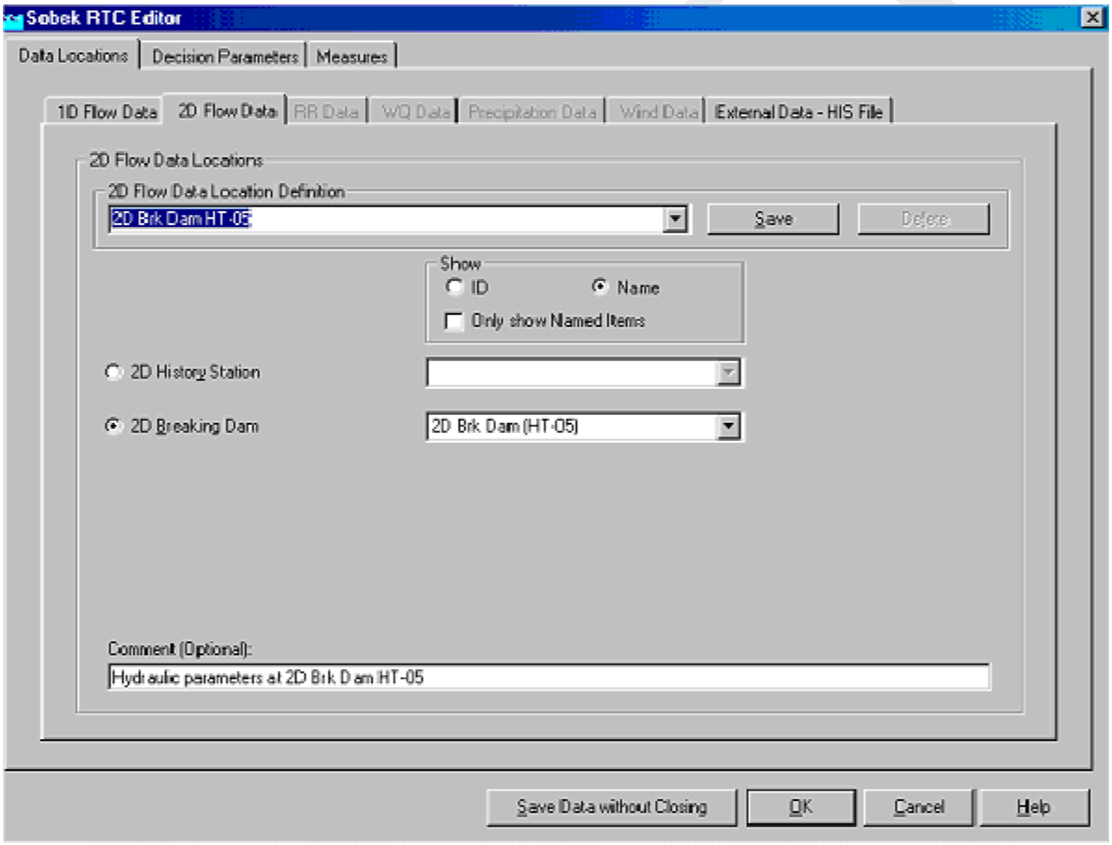


Figure 5.311: *RTC 2D Flow Data input screen*

For each 2D Flow Data location, the user can define its ID as well as an optional comment string (see Figure 5.311). Data locations can be selected on basis of their ID or user-defined Name. Two different type of 2D Flow Data locations are discerned, viz:

- ◇ 2D History stations
- ◇ 2D Breaking – Dams

Hint: If the “Show Name” option together with a checked “Only show Named Items” check-box is used (see Figure above), than only those objects in the drop-down lists are shown for which the user has defined a Name. In this way you can considerably reduce the length of the drop-down list (and hence the time needed for finding your object).

Following data is available at 2D History stations and 2D Breaking – Dams:

- water level [m AD];
- water depth [m];
- bottom(bed) level [m AD];
- U-velocity in [m/s];
- V-velocity in [m/s];
- Abs. velocity [$= \sqrt{U^2 + V^2}$] in [m/s].

5.13 SOBEK Tools

5.13.1 Calibration data editor

If you desire to compare **measured data** with **computed data**, you can create a "measured data file" according to the SOBEK Output format (.his files). This file can then be imported into the "Results in Tables" window or into the "Case Analysis Tool" for comparison with your simulation results.

Creating a custom HIS file (SOBEK results file)

- Go to the SOBEK Startup window
- Choose "Utilities" - "Calibration data editor" from the menu. The "Edit History file" window will appear.
- Choose "File" - "new". A spreadsheet-like environment will appear:

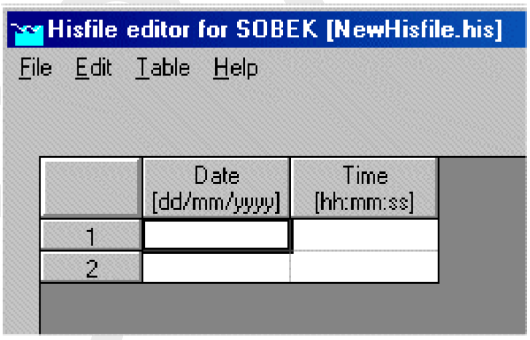


Figure 5.312: *Hisfile editor*

- Use the "Table" - "Add/Insert/Delete Row" options to customise the size of your table.
- Add measurement locations by choosing "Table" - "Add Location" and entering the name of the location.Note: you may decide to make this name equal to the ID of a node or branch segment in your schematisation!
- Enter the parameter name by choosing "Edit" - "Description". Choose for example the parameter name "water levels measured"
- You can copy and paste data from Excel into the fields. Important: the "date format" and "time format" of your cells in Excel **must** be equal to the format as indicated in the Hisfile editor: dd/mm/yyyy. You can change this format in Excel by selecting the cells, right-clicking your mouse, and selecting "format cells".
- When your file is ready, choose "File" - "Save as", and type a name for your HIS file.

Using your custom HIS file

◇ You can open this file in the Case Analysis Tool to compare it with your simulation results.
◇ You can also add this file to the "results in charts". In order to do so, open your simulation results and choose "File" - "add file" from the "Ods_view" window:

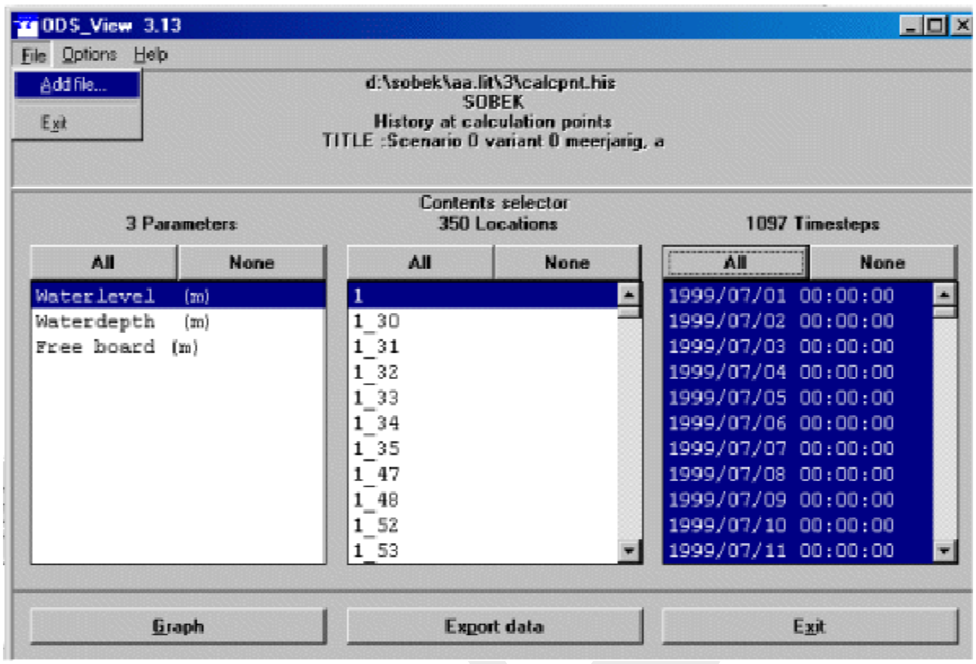


Figure 5.313: ODS view

Your measurement results will then appear as an extra parameter, but with the same location ID in the list of results:

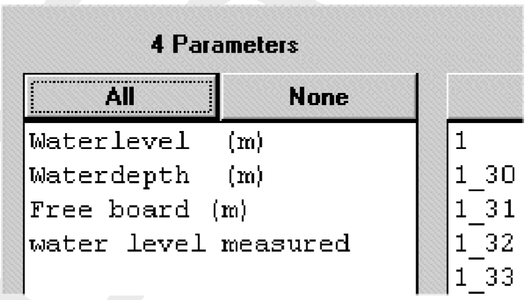


Figure 5.314: ODS View, parameter list

5.13.2 Online Visualisation (SOBEK-1D2D)

Sometimes it may be useful to visualise simulation results during computation on a detailed level. In Online Visualisation it is possible to click on waterlevel points (nodes) or on velocity points (links) and show the printed values of selected parameters.

To use Online Visualisation
Online Visualisation is available in SOBEK version 2.12.002 and upwards.

◇ After starting SOBEK, click on the menu item: "Options".
◇ Select the option: "SOBEK Options".
◇ Switch on the option"Online Visualisation" in the "Simulation" tab.

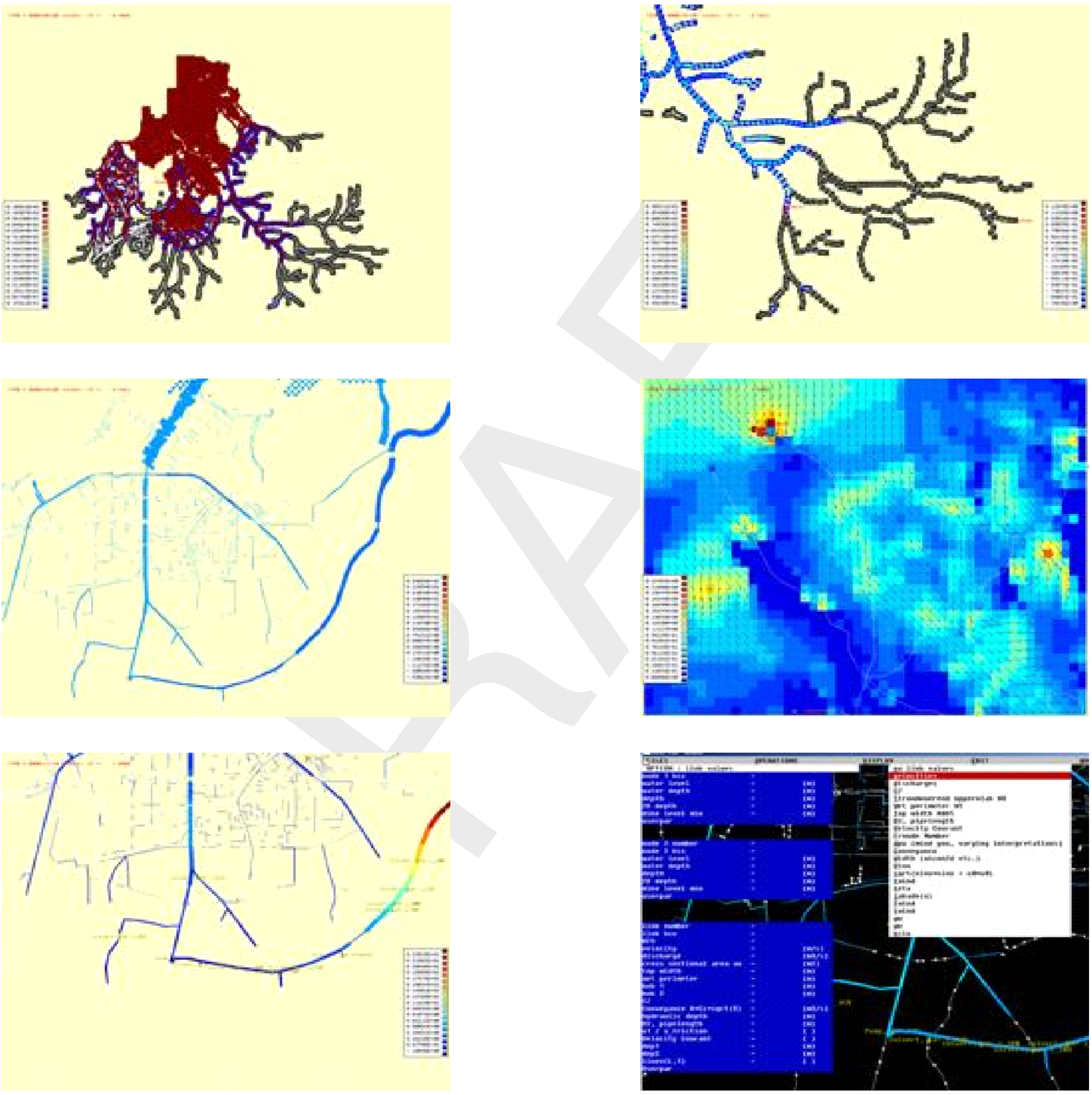


Figure 5.315: *Examples of the Online Visualisation.*

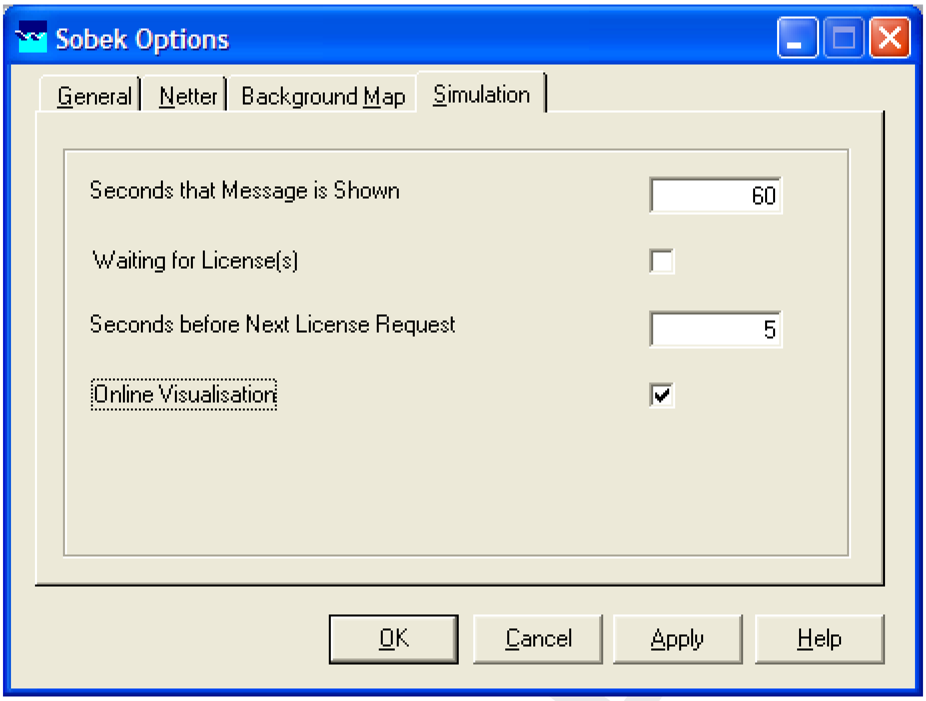


Figure 5.316: *Enabling the Online Visualisation in the SOBEK Options window.*

Continue or interrupt computation

- ◇ Double click the taskblock 'Simulation' to start a simulation.
- ◇ When the Online Visualisation Component appears, left mouse click on your screen.
- ◇ Click <Tab> to start the simulation, left mouse click to pause the simulation.
- ◇ Click spacebar if you want to proceed a single timestep.

Interrogate data on nodes and links

- ◇ Left mouse click either on nodes (square boxes) to display nodes values, or on links, (dots in 1D). Flow velocity, discharge, cross-sectional area etc will be shown.
- ◇ When clicking on a link, the items on both connected nodes will also be refreshed.
- ◇ In the top screen line, one can select a field parameter to be displayed: e.g.
  - ◻ 'Display' → 'Node values' → Waterlevel
  - ◻ When a node or link is clicked, this parameter is also printed as the last parameter in the list, see 'Userpar'

The left and right isoscales pertain to the requested node- and link-parameter fields.

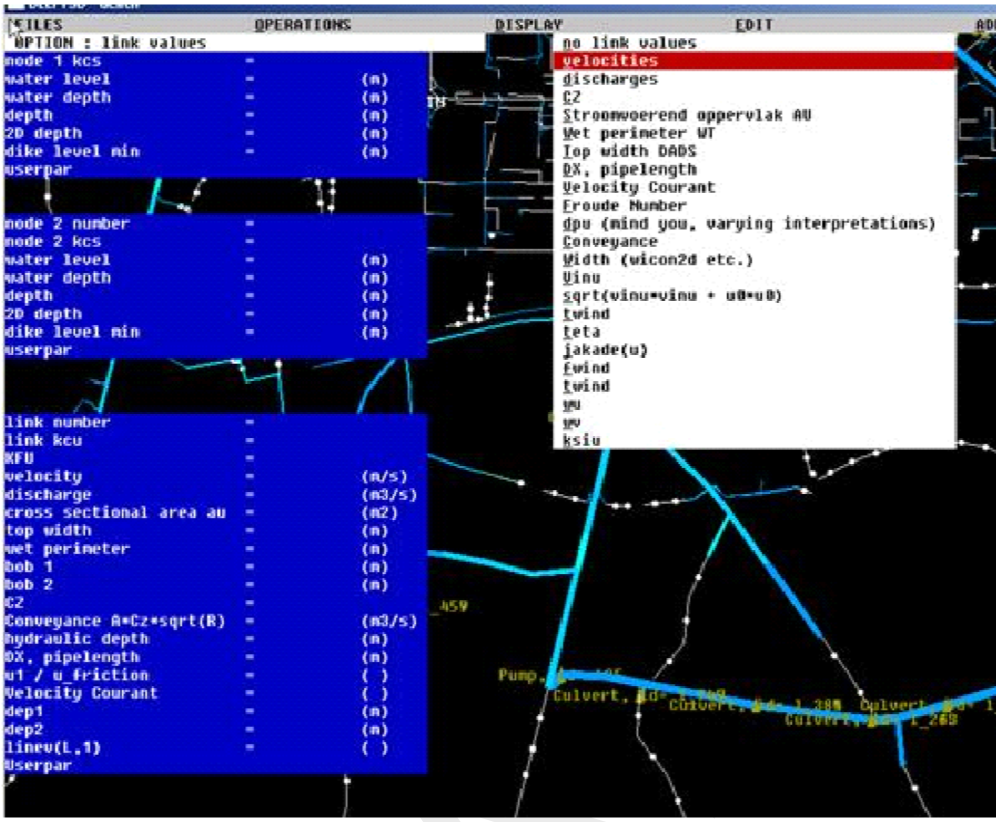


Figure 5.317: The Online Visualisation.

On the left side, from top tot bottom, after clicking on a link, we get the first node, the second node and the link in between. When more links exist on the same location, click more times.

For the first node:

- ◇ Node id: Corresponding to Netter id
- ◇ Node nr: Internal array nr
- ◇ Kcs code: 1 for 1D point, 100 for 2D point, 101 for 1D2D point
- ◇ Water level: Level in $[m]$ upward [1]
- ◇ Water depth: Difference between water level and bottom level [2]
- ◇ Depth: Negative value of bottom level, depth increases downward [3]
- ◇ 2D depth: Negative value of 2D bottom level [4]
- ◇ Dike level min: Lowest level for connecting 1D and 2D [5]
- ◇ Userpar: The value for the parameter specified in 'Display' $\rightarrow$ 'Node values'

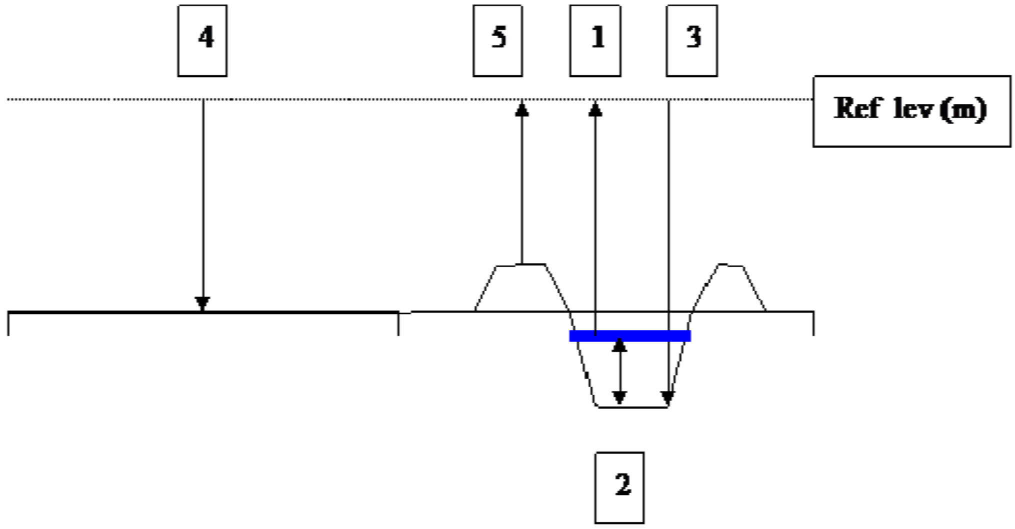


Figure 5.318: Explanation of cross-section for Onlie Visualisation

For the second node: same as for left node

For the connecting link:

- ◇ Link id: Corresponding to Netter id
- ◇ Link nr.
- ◇ Link kcu: 1=1D link, 100=2D link
- ◇ Kfu: 1=wet, 0=dry
- ◇ Velocity.
- ◇ Discharge.
- ◇ Cross sectional area: Flow carrying area
- ◇ Top width.
- ◇ Wet perimeter.
- ◇ Bob 1: Depth (pos down) at left side of pipe or link
- ◇ Bob 2: Depth (pos down) at right side of pipe or link
- ◇ Cz: Chézy parameter, if applicable
- ◇ Conveyance: given if applicable
- ◇ Hydraulic depth
- ◇ Dx, pipelength
- ◇ U1/Ufriction: Velocity vs equilibrium friction velocity
- ◇ Velocity Courant
- ◇ Dep1: Depth of manhole or waterlevel point on left side of link
- ◇ Dep2: Depth of manhole or waterlevel point on right side of link
- ◇ Linev(L,1) : Internal nr
- ◇ Userpar : The value for the parameter specified in 'Display' → 'Link values'

Display area control
Left mouse click on the schematisation, so that the crosshair cursor appears and the coordinates of the cursor are displayed in the top right corner of the screen.

Click <**z**>.

A window will appear that can be moved around. Now you can:

- ◇ Zoom in, with a left mouse click.
- ◇ Zoom out, with another <**z**>.
- ◇ Pan by moving the zoom window until it touches the screen border.
- ◇ Restore default viewing area with a right mouse click.

Press <a> to drop an anchor, to show the distance between the cursor and the anchor position in the upper right corner of the screen.

The size of 2D velocity vectors can be adjusted, Display, Display Parameters, Vectorsize.

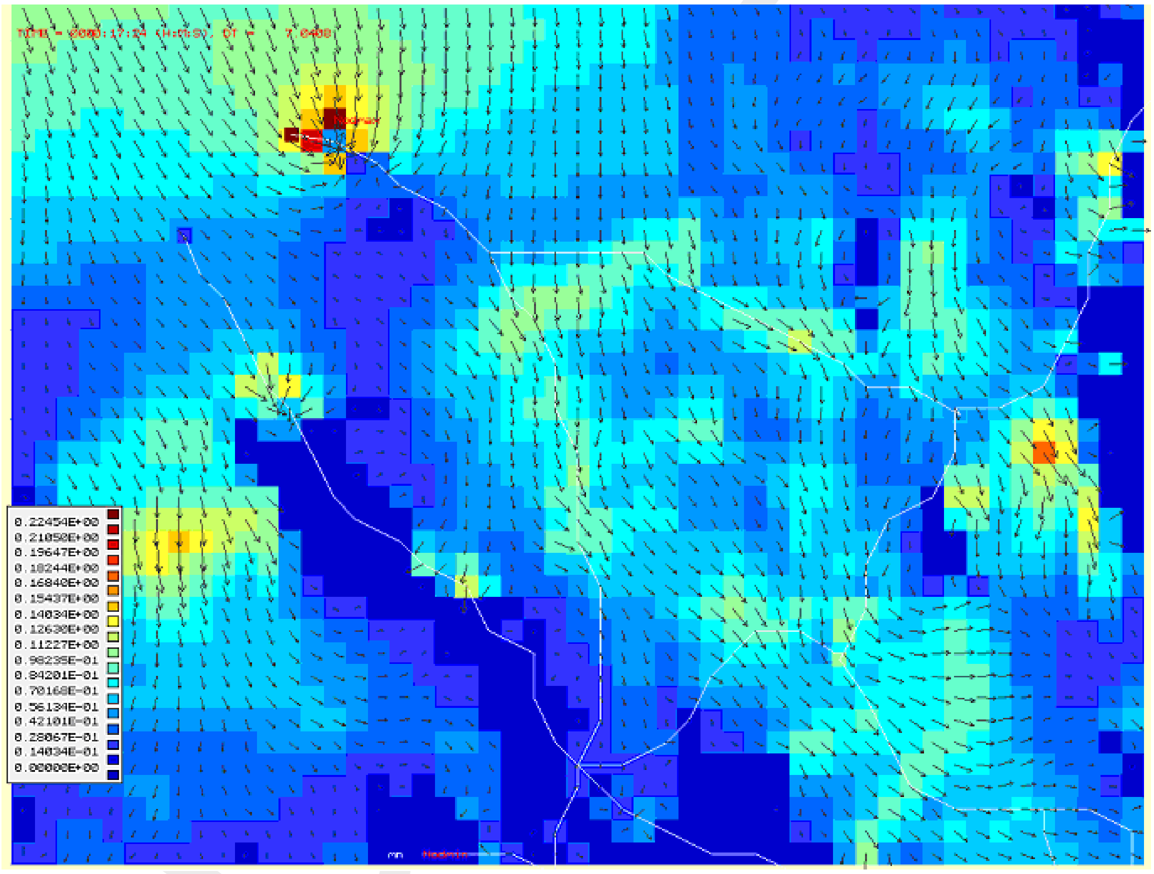


Figure 5.319: *Online Visualisation 2D velocity vectors.*

5.13.3 ReaHis (Convert HIS files to ASCII)

The ReaHis command-line tool can be used to easily convert .HIS files to ASCII format. After converting to ASCII, the output can be imported in other software packages or used in scripts.

The tool ReaHis.exe can be used after installing SOBEK. It is located in
.\SOBEK215\PROGRAMS\ReaHis\

Usage: ReaHis <*InputHis.his*> > <*Output.txt*>
For example: ReaHis flowhis.his > flowhis.txt

5.13.4 Time tables in SOBEK

Tables for parameters as a function of time

In many situations, one may encounter tables that have to be filled with time-dependent data. For example lateral inflows at a Flow - Lateral Flow node, or water levels at a Flow - Boundary node. In this chapter, the structure and the options of such tables is explained.

The general lay-out for the tables window looks like this:

Remark:

◇ The numbers in the table below do not correspond with the graphs later on.

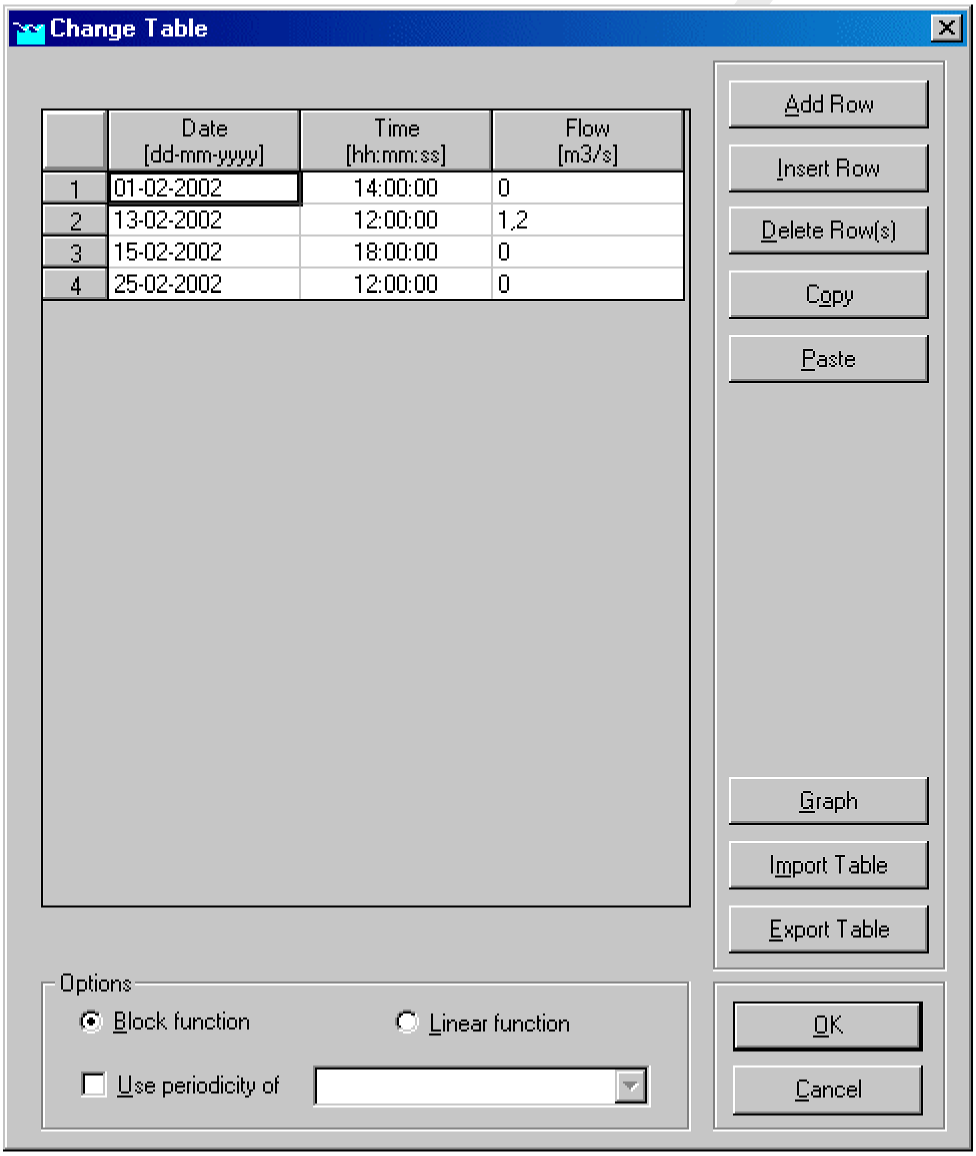


Figure 5.320: *A generic table.*

◇ **Left Column: date** Here, the dates for which you want to change certain parameter values should be entered. It is not necessary to type all values by hand: you can copy them from a spreadsheet, such as Microsoft Excel. But note that the field format within the spreadsheet should correspond to the format indicated by SOBEK: dd-mm-yyyy. In Excel, the field format can be altered by right-clicking a field, choosing "format cell" - "number" - "custom" - and type the appropriate format: dd-mm-yyyy (or dd-mm-jjjj in the Dutch version)

- ◇ **Middle Column: time** Here, the times on which you want to change certain parameter values should be entered. It is not necessary to type all values by hand: you can copy them from a spreadsheet, such as Microsoft Excel. But note that the field format within the spreadsheet should correspond to the format indicated by SOBEK: hh:mm:ss. In Excel, the field format can be altered by right-clicking a field, choosing "format cell" - "number" - "custom" - and type the appropriate format: hh:mm:ss (or uu:mm:ss in the Dutch version)
- ◇ **Right Column: parameter value** This column represents the parameter that needs to be changed in time. For example: discharges, water levels, moist content, crest levels etc.
- ◇ **The buttons on the right side of the window** <Add Row> adds one or more rows to the downside of the table <Insert Row> adds a new row just above the one that's currently selected <Delete Row(s)> deletes the currently selected row
  <Copy> copies the data of the currently selected cells to the clipboard. An alternative to this button is to press `Ctrl+ C`.
  <Paste> pastes data from the clipboard to the currently selected cells. An alternative to this button is to press `Ctrl + P`. Important: make sure that the data that you want to paste has the correct format (dd-mm-yyyy for date and hh:mm:ss for time)!
  Graph plots a graph of the data in the table <Import Table> gives the option to import a previously stored table <Export Table> gives the option to save the current table. This table may then be used for other objects too.
- ◇ **Options**
  - Block function: interprets the data from the table as a block function, thus the parameter value will be constant during the period between two data entries.
  - Linear function: for the calculation period between two data entries the values of both surrounding entries will be linearly interpolated.
- ◇ **Periodicity:** - Use periodicity: let the values in the table repeat themselves with a certain interval (year, month, week, day, 12 hours, hour)
  If the periodicity interval you chose is **smaller** than the time span of your table, SOBEK will apply the periodicity procedure to the **last** section of the table that covers the periodicity interval. The picture below shows how this situation is handled within SOBEK:

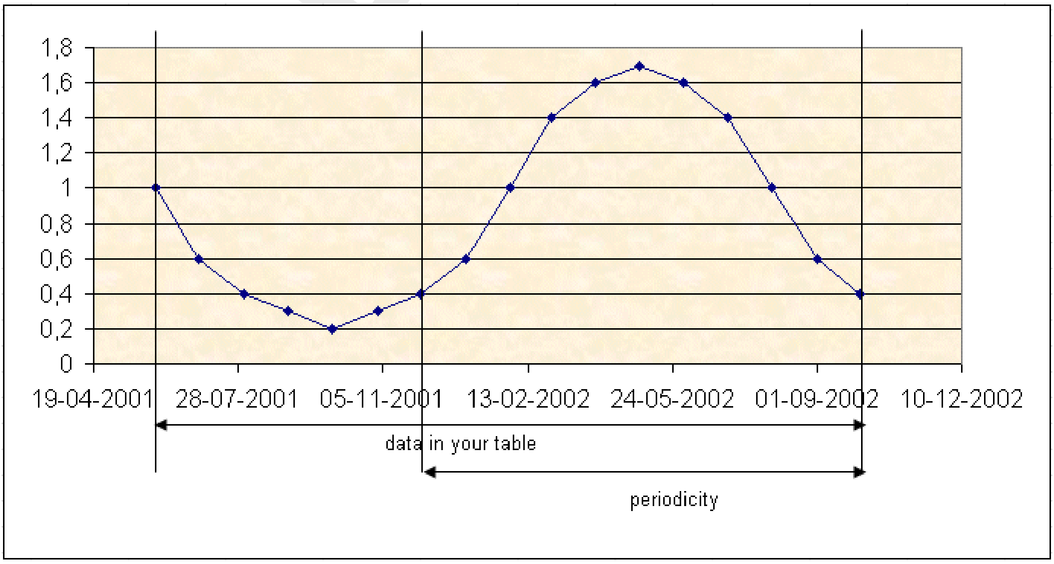


Figure 5.321: *This picture shows which part of your table will be continuously repeated if the periodicity timespan that you choose is shorter than the available data in your table.*

Note 2: If the periodicity interval you chose is **larger** than the time span of your table, SOBEK will linearly interpolate between the data in the last record of the table and the value from the first record of the table. See the picture below for a better understanding:

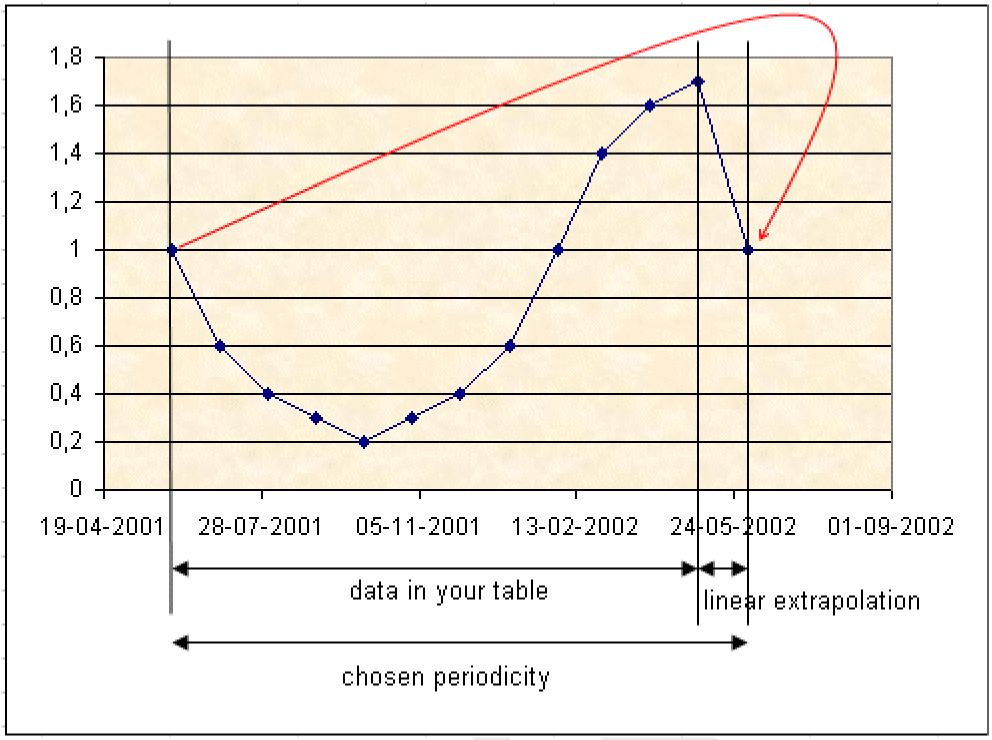


Figure 5.322: *In this graph you can see how SOBEK handles periodicities that exceed the time span in your table.*

The first value from the table is then assumed to occur at the end of the periodicity too, and all values between the last value of the table and that value are then interpolated blockwise or linearly, depending on the chosen interpolation option. The example assumes linear interpolation has been specified.

5.14 1D Hydraulic friction concepts

The following hydraulic friction concepts can be discerned in 1D flow:

- Global (or Model-wide) friction concept
- Local (or Branch-wise) friction concept
- Cross-section friction concept
- Culvert friction concept

5.14.1 Global (or Model-wide) friction concept

Global friction refers to defining (formula & value) for *Bed* friction (and optional *Groundlayer* friction):

- Use of Global friction:
  If Global friction definition is selected at cross-section X, this means that the Global friction definition is applied for all cross-sections, that are located on the same branch as cross-section X.
- Restrictions for Global friction:
  - Not available at “Y-Z” and “Asymetrical Trapezium” cross-sections.
  - At “River profiles” only Global *Bed* friction can be defined.

◇ Warning for Global friction:
Changing the Global friction at cross-section X, has consequences for all branches using Global friction.
For "River profiles" two situations are to be discerned:
There are only River profiles in the model: Different global values for positive flow and negative flow as well as friction formulations can be specified for Main section, Floodplain 1 and Floodplain 2.
There is also another type of cross-section: In case a global value and friction formulation is defined at a non-River profile (i.e. another type of cross-section), all the information regarding different global values and friction formulations for positive and negative flow at Main section, Floodplain 1 and Floodplain 2 is overruled by the global value and friction formulation defined at the non-River profile.

5.14.2 Local (or Branch-wise) friction concept

Local friction refers to defining (formula & value) for *Bed* friction (and optional *Groundlayer* friction):

◇ Use of Local Friction
If local friction definition is selected at cross-section X, this means that this specific Local friction definition is applied for all cross-sections, located on the same branch as cross-section X
◇ Restrictions for Local friction:
  - Not available at "Y-Z" and "Asymetrical Trapezium" cross-sections.
  - At "River profiles" only Local *Bed* friction can be defined.

◇ Exception for Local friction:
Only at River Profiles, Local *Bed* friction can be defined as function of the branch x_coordinate ($f(x)$, $f(Q, x)$ and $f(h, x)$)
◇ Warning for Local friction:
Changing local friction at cross-section X, has consequence for all cross-sections, that are located on the same branch as cross-section X.

5.14.3 Cross-section friction concept

Cross-section friction means that *Bed* friction (formula & value) can be defined per cross-section:

◇ Use of Cross-section friction
  - Only available for "Y-Z" and "Asymetrical Trapezium" cross-sections.
  - The cross-sectional area can be divided in (vertical) roughness sections for which a different *Bed* friction (formulae & value) can be defined.

◇ Restriction for Cross-section friction
Groundlayer friction cannot be defined

5.14.4 Culvert friction concept

Culvert friction means that *Bed* and optional *Groundlayer* friction (formula & value) can be defined for the closed cross-section available at a Culvert:

◇ Use of Culvert friction:
Only available for Culverts, Inverted Siphons and Siphons

6 Conceptual description

6.1 Hydrodynamics D-Flow 1D

6.1.1 Model equations

The water flow is computed by solving the complete De Saint Venant (1871) equations for unstaedy flow are based upon the the following series of assumptions (Cunge *et al.*, 1980):

- ◇ The flow is one-dimensional i.e. the velocity can be representated by a uniform flow over the cross-section and the water level can be assumed to be horizontal across the section.
- ◇ The streamline curvature is small and the vertical accelerations are negiglible, hence the pressure is hydrostatic.
- ◇ The effects of boundary friction and turbulence can be accounted for through resistance laws analogous to those used for steady flow.
- ◇ The average channel bed slope is small so that the cosine of the angle it makes with the horizontal may be replace by unity.

For one dimensional flow (Channel Flow and Sewer Flow modules) the following equations are solved

- ◇ continuity equation 1D
- ◇ momentum equation 1D

For two dimensional flow (Overland Flow module), three equations are solved:

- ◇ continuity equation 2D
- ◇ momentum equation 2D for the x-direction
- ◇ momentum equation 2D for the y-direction

These equations are solved numerically using the Delft-scheme.

Note: on the 2D equations
As opposed to the shallow water equations, the described equations do not incorporate the turbulent stress terms, accounting for the sub grid transfer of momentum in between grid cells. These terms have been omitted because they are relatively unimportant for flood flow computations, in order to save computational effort.

The wall friction terms have been introduced to account for the added resistance that is caused by vertical obstacles, like houses or trees. The wall friction coefficient is based on the average number and diameter of the obstacles per unit area and the average obstacle drag coefficient (C_d coefficient).

6.1.1.1 Continuity equation (1D)

The 1D continuity equation reads:

$$\frac{\partial A_T}{\partial t} + \frac{\partial Q}{\partial x} = q_{lat} \tag{6.1}$$

where:

A_T	Total area (sum of flow area and storage area) $[m^2]$
Q	Discharge $[m^3/s]$

q_{lat} Lateral discharge per unit length $[m^2/s]$. Positive value refers to inflow. Negative value refers to outflow.

6.1.1.2 Momentum equation (1D)

The 1D momentum equation reads:

$$\frac{\partial Q}{\partial t} + \frac{\partial}{\partial x}\left(\frac{Q^2}{A_F}\right) + gA_F\frac{\partial \zeta}{\partial x} + \frac{gQ|Q|}{C^2RA_F} - w_f\frac{\tau_{wind}}{\rho_w} + gA_F\frac{\xi Q|Q|}{L_x} = 0 \qquad (6.2)$$

The first term describes the inertia
The second term describes the convection
The third term describes the water level gradient
The fourth term describes the bed friction
The fifth term describes the influence of the wind force
The sixth term describes the influence of extra resistance

where:

A_F	Flow area $[m^2]$
C	Chézy value $[m^{1/2}/s]$
g	Acceleration due to gravity $[m/s^2]$
ζ	Water level $[m]$
L_x	Length of branchsegment, accomodating an Extra Resistance Node $[m]$
Q	Discharge $[m^3/s]$
R	Hydraulic radius $[m]$
t	Time $[s]$
w_f	Water surface width $[m]$
x	Distance along the channel axis $[m]$
ρ_w	Density of fresh water $[kg/m^3]$
τ_{wind}	Wind shear stress $[N/m^2]$ (see section 6.1.7)
ξ	Extra Resistance coefficient $[s^2/m^5]$

6.1.2 Hydrodynamic definitions

6.1.2.1 Model datum/reference level

The Model datum and reference level both refer to a horizontal plane from which elevations are defined (positive in upward direction). By default the model datum and reference level are equal to zero.

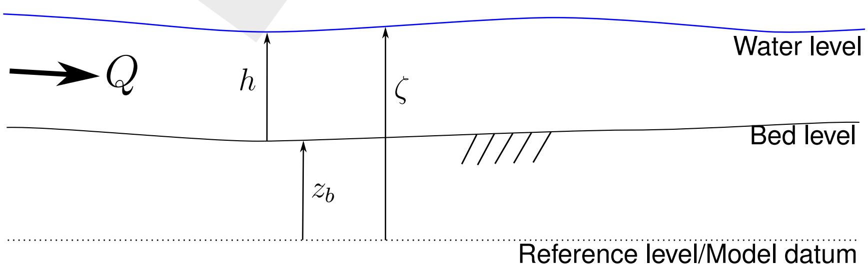


Figure 6.1: *Definition of model datum/reference level*

Note: All levels (quantities with a vertical coordinate) in SOBEK are defined with respect to the model datum or reference level.

6.1.2.2 Bed level

The bed level is defined as the lowest point in the cross section. In the definition of the cross section an example is given of the interpolation and extrapolation of the bed levels over a branch. The used symbol in the D-Flow 1D module is z_b.

It is given relative to a reference level, for example Mean Sea Level .

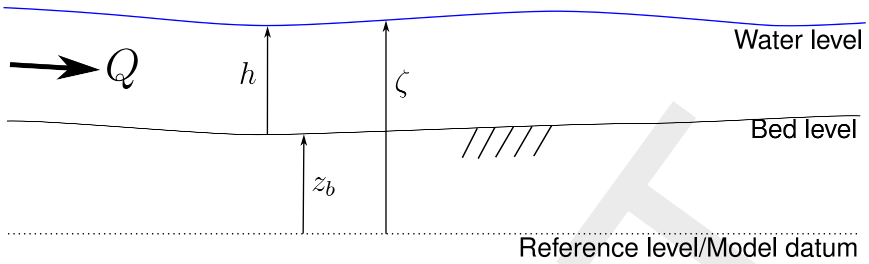


Figure 6.2: *Definition of bed level*

If the water level is lower than the bed level, drying occurs.

6.1.2.3 Water depth

In the D-Flow 1D module the water depth is the distance between the water level and the bed level. The symbol used for the water depth is h.

$$h = \zeta - z_b \tag{6.3}$$

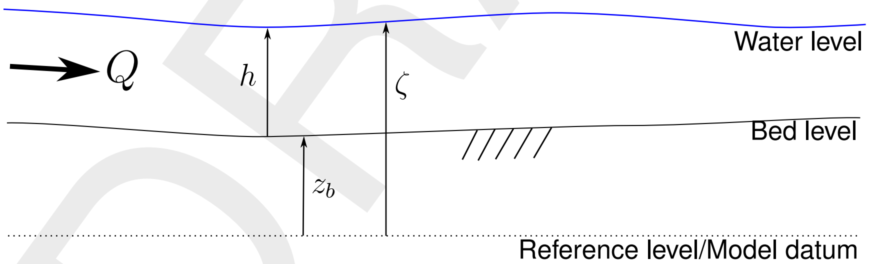


Figure 6.3: *Definition of water depth* $h = \zeta - z_b$

6.1.2.4 Water level

The water level (ζ) is the level of the water surface relative to the reference level or Model datum (= reference level).

In the one-dimensional D-Flow 1D module the water level perpendicular to the flow direction is assumed to be horizontal.

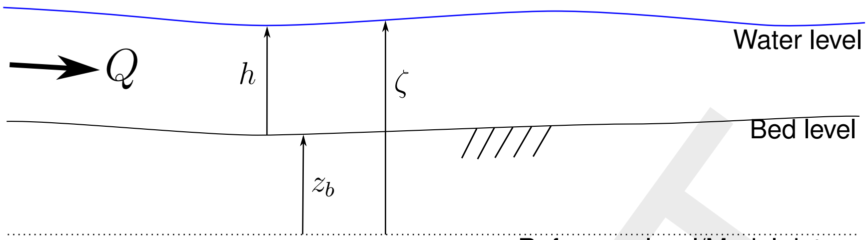


Figure 6.4: *Definition of water level*

The water level together with the discharge form the result of the D-Flow 1D module computations. Water levels are calculated at the connection nodes and the ζ-calculation points.

Note: In each D-Flow 1D, all levels are relative to the reference level

6.1.2.5 Flow area

The flow area A_F of a cross-section is the area through which water is actually flowing.

In a cross-section, distinction can be made between the flow area and the storage area. In the latter part, water is stored only, see Figure 6.5.

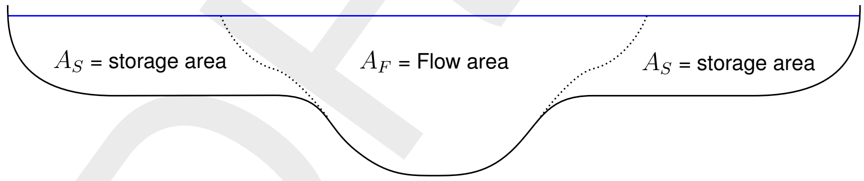


Figure 6.5: *Definiton of flow area (A_F) and storage area (A_S)*

The shape of the cross-section, and the distinction between the flow area and storage area are defined for each cross-section by user input.

6.1.2.6 Storage area

The storage area A_S of a cross-section is the area in which only water is stored i.e. the non-conveying part of the cross-section.

In a cross-section, distinction can be made between the flow area and the storage area, see Figure 6.5.

The shape of the cross-section, and the distinction between the flow area and storage area is defined for each cross-section by user input.

The total area A_T is defined as the flow area plus the storage area.

6.1.2.7 Wetted area

The wetted area (A_F [m^2]) is the part of the cross section that is filled with water. See Figure 6.6

6.1.2.8 Wetted perimeter

The wetted perimeter (O [m]) is the interface between the soil and the flowing water.

See Figure 6.6.

In the case of a main channel and one or two floodplains in the cross-section, a wetted perimeter is computed for each section. In that case the water interface between the sections is not included in the wetted perimeter. See also Figure 6.6 and Figure 6.7

The wetted perimeter is used to compute the hydraulic radius, which is important for the computation of the bed friction.

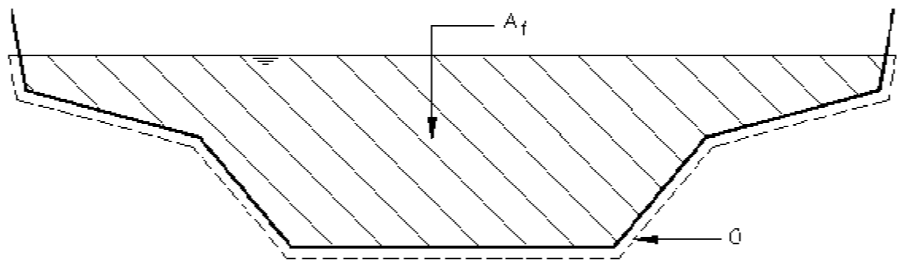


Figure 6.6: *Wetted perimeter*

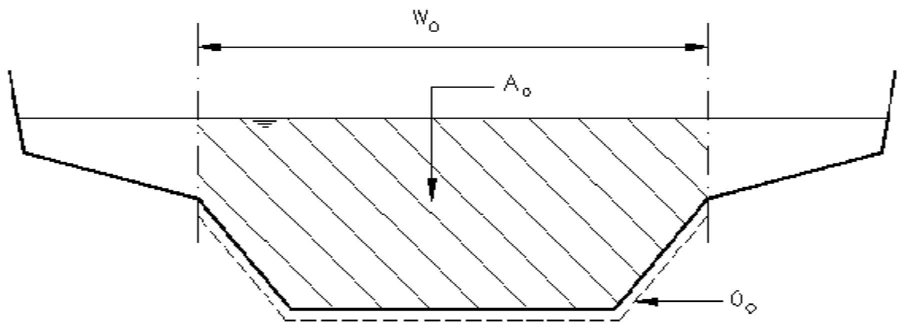


Figure 6.7: *Wetted perimeter for the main channel*

6.1.2.9 Flow velocity

The flow velocity (u) is defined as the average flow velocity in the flow section of the cross-section. It is by default given in [m/s].

The average flow velocity is derived by dividing the discharge [m^3/s] by the flow area [m^2]. Output of the overall average flow velocity, the flow velocity in the main channel, the flow velocity in floodplain 1 and the flow velocity in floodplain 2 is possible.

$$u = \frac{Q}{A_F}, \qquad u_0 = \frac{Q}{A_0}, \qquad u_1 = \frac{Q}{A_1}, \qquad u_2 = \frac{Q}{A_2}. \tag{6.4}$$

The indices 0, 1, 2 indicate the main channel, floodplain 1 and floodplain 2 respectively.

6.1.2.10 Velocity

The velocity (u) is defined as the average flow velocity in the wetted area of the cross section. It is given in [m/s].

$$u = \frac{Q}{A_F} \tag{6.5}$$

Note: Discharges and velocities are defined in branch segments, whereas cross sections are defined in ζ-calculation points. For the computation of the velocities according to the above formula. The D-Flow 1D module uses the upstream cross section

6.1.2.11 Hydraulic radius

The hydraulic radius is defined as the wetted area of the cross section divided by the wetted perimeter.

$$R = \frac{A_F}{O} \tag{6.6}$$

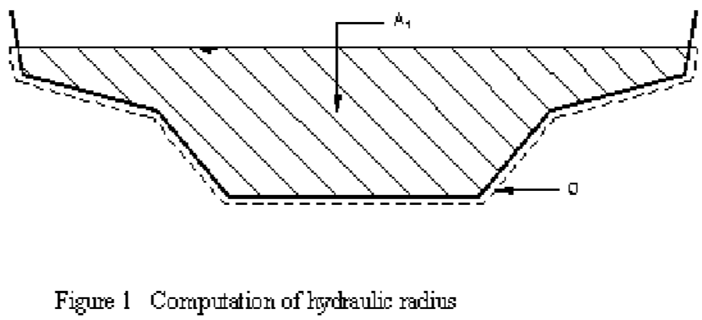


Figure 6.8: *Computation of hydraulic radius*

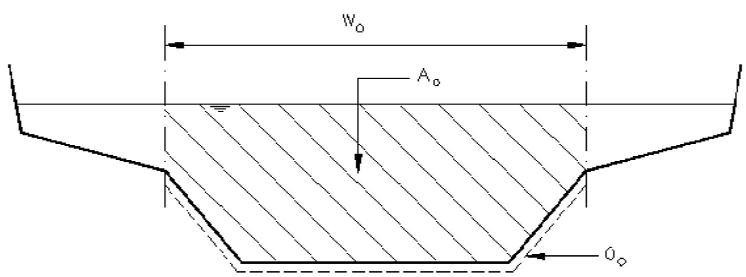


Figure 6.9: *Wetted perimeter for the main channel (Subdivision in main channel and floodplains only in SOBEK-River)*

6.1.3 Inertia

The flow in one dimension is described by two equations: the momentum equation and the continuity equation. The first term in the momentum equation is the inertia term:

$$\text{inertia} = \frac{\partial Q}{\partial t} \tag{6.7}$$

Q	Discharge $[m^3/s]$
t	Time $[s]$

6.1.4 Convection

In the D-Flow 1D module an convection term is used to guarantee the conservation of momentum in a branch (between branch segments). The same holds for a connection node when two branches are connected to this node. It that case the connection nodes could also have been modelled as a ζ-calculation point. If there are more than two branches connected to a connection node the convection term is set to zero.

When a branch has subsections, all subsections have an independent solution for the velocity. The velocity that is used for the conservation of momentum over the branch is the average of the velocities u_i of the subsections in the cross section.

6.1.5 Convection (1D)

The flow in one dimension is described by two equations: the momentum equation and the continuity equation. One of the terms in the momentum equation is the convection term:

$$\text{convection} = \frac{\partial}{\partial x}\left(\frac{Q^2}{A_F}\right) \tag{6.8}$$

Q	Discharge $[m^3/s]$
A_F	Flow area $[m^2]$
x	Distance $[m]$

6.1.6 Water level gradient

The third term in the momentum equation is the water level gradient.

$$\text{water level gradient} = gA_F\frac{\partial \zeta}{\partial x} \tag{6.9}$$

x	Distance $[m]$
A_F	Flow area $[m^2]$
g	Acceleration due to gravity $[m/s^2]$ (≈ 9.81)
ζ	Water level $[m]$ (with respect to the reference level)

This force tries to achieve a flat water surface under influence of the gravitational acceleration. This force together with the bed friction have the greatest effect on the water movements.

6.1.7 Wind friction

The wind friction term in the momentum equation is expressed as:

$$\tau_{wind} = -\rho_{air}C_{wind}u_{wind}^2\cos(\varphi_{wind} - \varphi_{channel}) \tag{6.10}$$

τ_{wind}	Wind shear stress (positive if acting along the positive x-axis of an open channel) $[N/m^2]$
ρ_{air}	Density of air, 1.205 kg/m^3
C_{wind}	$\alpha_{wind,1} + \alpha_{wind,2} u_{wind}$, the wind friction coefficient. The used values are:

$$a_{wind,1} = 0.50 \times 10^{-3} \qquad [-] \tag{6.11}$$

$$a_{wind,2} = 0.06 \times 10^{-3} \qquad [\text{s/m}] \tag{6.12}$$

u_{wind}	Wind velocity [m/s]
$\varphi_{channel}$	Clock-wise angle between the positive x-axis of the 1D open channel and the North (Nautical convention) [degrees]
φ_{wind}	Clock-wise angle between the wind direction and the North (Nautical convention) [degrees]

The wind direction that is specified in the D-Flow 1D module is the direction from which the wind is blowing with respect to the north. So, if the wind is blowing from the north, the wind direction is 0 degrees. Wind from the southeast has a wind direction of 135 degrees.

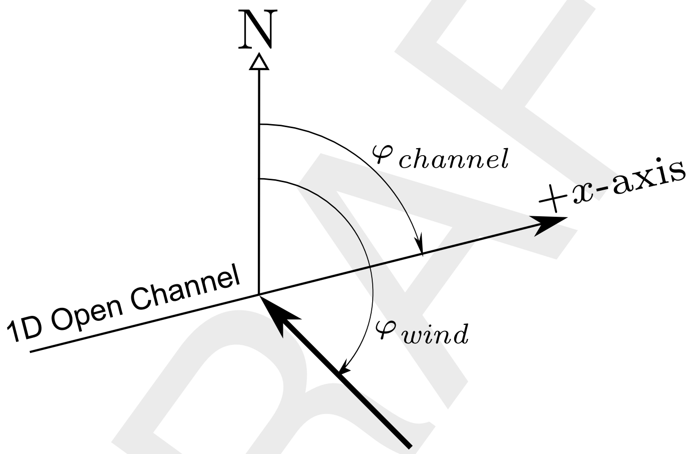


Figure 6.10: *Wind direction*

When a cross section has a top width smaller than 20 mm (circular, egg-shaped, or nearly closed tabulated cross section) than the cross section is considered closed. In that case the wind is neglected in the momentum equation

6.1.8 Initial conditions

The initial conditions are the water levels or depths and the discharges at the beginning of the simulation. The initial conditions are defined over a branch (water level and depth at the ζ-calculation points, discharges at the branch segments). Therefore the water level that initially should be taken at connection nodes is not strictly defined. This happens when, for example, branch 1 with initial water level 0.5 is connected by a node to branch 2 with initial water level 0.6. In that case the water level at the connection node is set to the lowest value of the connected branches (0.5). Because the ζ-calculation point at the end of branch 1, the connection node and the ζ-calculation point at the beginning of branch 2 have the same location, the water levels of these three points are set to 0.5.

6.1.9 Boundary

A boundary can be applied at the locations where the model network ends with a boundary node.

In order to solve the water flow equations (continuity equation and momentum equation), information about the water flow at the model boundaries must be supplied.

At each boundary node, one condition for the water flow must be specified. The following options are available:

- ◇ discharge (constant, tabulated function of time, tabulated function of the water level)
- ◇ water level (constant, tabulated function of time)

Because in the staggered grid, used by D-Flow 1D module, the discharges are defined for the branch segments, a discharge boundary is imposed on the first branch segment next to the boundary. Therefore the ζ-calculation point just before this first branch segment (first one in a branch) is undefined and will not be taken into consideration during a calculation.

A water level boundary is defined in the first calculation point next to the boundary. This ζ-calculation point actually has the same coordinates as the boundary node.

Note: In channel flow, usually, the discharge will be specified at boundaries where water is flowing into the model, and the water level where water is flowing out of the model. In both sewer and channel systems a dead end (or beginning) of a branch can be a connection node at the end of a pipe or channel. In contrast to a boundary node, this node has storage.

6.1.10 Discharge

The discharge Q is the amount of water passing a branch segment per unit of time. It is given in $[m^3/s]$.

The discharges at the branch segments together with the water levels at the ζ-calculation points, form the results of a water flow simulation.

The D-Flow 1D module gives a positive discharge when the water flows in positive direction with respect to the defined direction of a branch. If the water is flowing from branch end to branch beginning, the discharge is negative.

6.1.11 Lateral discharges

A distinction is to be made between a *Point* and *Diffusive* lateral discharge.

- ◇ **Point lateral discharge**: A point lateral discharge (Q_{lat}, $[m^3/s]$) refers to the inflow or outflow of water at a specific point/location in a model schematization.
- ◇ **Diffusive lateral discharge**: A diffusive lateral discharge (q_{lat}, $[m^2/s]$) refers to the inflow or outflow of water along a specific branchsegment in a model schematization. A branch segment is the distance (l_i, $[m]$) between two ζ-calculation points. A branch may consists of several branchsegments. The total diffusive lateral discharge at a branch segment equals $q_{lat} \times l_i$. A diffusive lateral discharge is uniform distributed along the axis of a branch (see Figure 6.11).

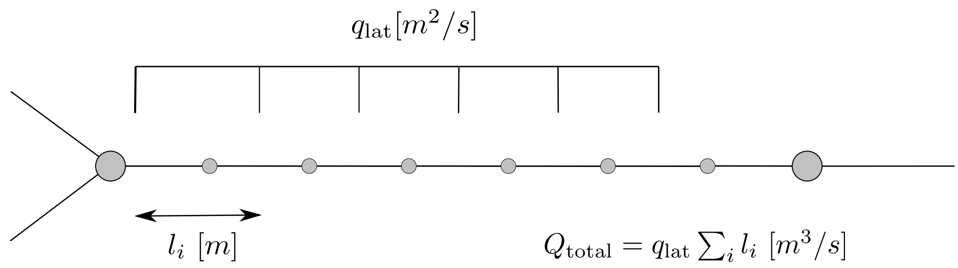


Figure 6.11: *Diffuse lateral discharge*

For both point lateral discharges and diffuse lateral discharges yields that they are assigned to a ζ*-calculation point(s)*. More precisely, their lateral inflow or outflow volume is accounted for in the local water balance of such ζ-calculation point (e.g. local discretization of the continuity equation, see section 6.1.11.1).

Two options are available for determining to which adjacent ζ-calculation point a lateral discharges is assigned (see section 6.1.11.2).

An overview of lateral discharges, available in SOBEK, is given in section 6.1.11.3. In general lateral discharges can be defined as a constant or as a function of time. Exceptions are:

- ◇ The Area Based Point lateral inflow (see section 6.1.11.4)
- ◇ The Pipe with Infiltration (see section 6.1.11.5)

6.1.11.1 Incorporating Lateral discharges in the Continuity Equation

Both point lateral discharges and diffusive lateral discharges (see Figure 6.12) are included in the third term (q_{lat}) of the continuity equation (see Equation (6.13)).

$$\frac{\partial A_F}{\partial t} + \frac{\partial Q}{\partial x} = q_{lat} \tag{6.13}$$

A_F	Flow area [m^2]
t	Time [s]
Q	Discharge [m^3/s]
x	Distance [m]
q_{lat}	Lateral discharge per unit length [m^2/s]

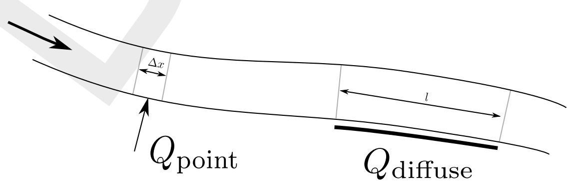


Figure 6.12: *Lateral discharges:*
Point lateral discharge: $q_{lat} = Q_{point}/\Delta x$; and
Diffuse lateral discharge: $q_{lat} = Q_{diffuse}/l$

Note: A large withdrawal of water (negative lateral discharge) can cause problems when the amount of outflowing water is larger than the storage available in a ζ-calculation point. As a result negative volumes could be computed in the continuity equation. For that reason the time step is temporarily reduced when the outflowing water volume is larger than half the available volume (safety factor) in the ζ-calculation point. The time step is set to a value that allows a withdrawal of half the available volume in one time step. If the time step resulting from this action is smaller than 0.01 second, the lateral withdrawal is set to zero.

6.1.11.2 Options for Assigning Lateral Discharges to ζ-calculation point(s)

Two different *lateral discharge assignment options* are available in Settings (see 1D Flow, Simulation Settings Tab), that determine to which ζ-calculation (or water level) point(s) a particular lateral discharge is to be assigned. The two different lateral discharge assignment options are:

Option 1: Lateral Assigned to Lowest Water Level Point,
Option 2: Lateral Assigned to Nearest Water Level Point.

Hereafter, the consequences of the two different lateral discharge assignment options are discussed for each type of lateral discharge (see section 6.1.11.3) separately.

- Lateral discharge on a Node [m^3/s]:
  Irrespective of the selected lateral discharge assignment option, a lateral discharge on a Node is assigned to its concerning Node (e.g. ζ-calculation point, Connection Node or Manhole).
- Point-lateral discharge on a Branch [m^3/s]:
  A point-lateral discharge on a Branch is either assigned to one or equally distributed over both of the ζ-calculation points, that are respectively located at the left-side and the right-side of the point-lateral discharge.
  - If **option 1** (Lateral Assigned to Lowest Water Level Point) is selected, the point-lateral discharge is assigned to the ζ-calculation point having the lowest bed level. If both calculation points have the same bed level and the point-lateral discharge is located at a u-velocity point (e.g. centre-point of the two adjacent ζ-calculation points), the point-lateral discharge is equally distributed over both adjacent calculation points. Else the point-lateral discharge is assigned to the nearest ζ-calculation point.
  - If **option 2** (Lateral Assigned to Nearest Water Level Point) is selected and the point-lateral discharge is located at a u-velocity point (e.g. centre-point of the two adjacent ζ-calculation points), the point-lateral discharge is equally distributed over both adjacent ζ-calculation points. Else the point-lateral discharge is assigned to the nearest ζ-calculation point.
- Diffusive lateral discharge along a Pipe or Branch [m^2/s]:
  A diffusive lateral discharge along a Pipe or Branch is either assigned to one or equally distributed over both of the ζ-calculation points, that are respectively located at the beginning and the end of a pipe or branchsegment, receiving a diffusive lateral discharge.
  - If **option 1** (Lateral Assigned to Lowest Water Level Point) is selected, the diffusive lateral discharge is assigned to the ζ-calculation point having the lowest bed level. Else the diffusive lateral discharge is equally distributed over both adjacent calculation points.
  - If **option 2** (Lateral Assigned to Nearest Water Level Point) is selected, the diffusive lateral discharge is equally distributed over both adjacent ζ-calculation points.

6.1.11.3 Examples of Lateral Discharges

Three different types of lateral discharges are discerned:

1 Lateral Discharge on a Node $[m^3/s]$:
Examples of lateral discharges on a Node are:

- ◇ *Point-Lateral Discharge, that is exactly located on a ζ-calculation point*,
- ◇ *Connection Node with Storage and Lateral Flow*,
- ◇ *Manhole with Lateral Flow*,
- ◇ *Manhole with Lateral Disch. and Runoff*
- ◇ *Manhole with Runoff*
- ◇ *Flow - RR Connection on Flow Connection Node*.

2 Point-Lateral Discharge on a Branch $[m^3/s]$:
Examples of a point-lateral discharge on a Branch are:

- ◇ *Flow - Lateral Flow Node*
- ◇ *Flow - RR Connection on Channel*.

3 Diffusive Lateral Discharge along a Pipe or Branch $[m^2/s]$:
Examples of a diffusive lateral discharge along a Pipe or Branch are:

- ◇ *Flow - Pipe with Runoff*,
- ◇ *Flow - Dry Weather Pipe*,
- ◇ *Flow - Rain Pipe*,
- ◇ *Flow - Pipe with Infiltration*
- ◇ *Flow - Channel with Lateral Discharge*.

6.1.11.4 Area Based Point Lateral Flow

Area based Point lateral flow is computed as:

$$Q_{lat} = A(f_d R + S) \tag{6.14}$$

Q_{lat}	Lateral discharge $[m^3/s]$
f_d	Design factor for area based lateral flow $[-]$.
R	Rainfall $[m/s]$.
A	The catchment runoff area $[m^2]$
S	Seepage (positive, inflow of groundwater) or Infiltration (negative, outflow to groundwater reservoir) $[m/s]$.

6.1.11.5 Pipe with infiltration (having a lateral diffusive discharge option)

A pipe with infiltration refers to a drainage pipe fully located inside a trench (see Figure 6.13). Although the drainage pipe is porous it still can get pressurized. The trench is filled with material (for instance gravel) having a certain porosity.

A pipe with infiltration allows for the exchange of water in the pipe towards the trench (positive sign) and vice versa as well as for the exchange of water in the trench towards the groundwater (negative sign) and vice versa. Hence, a pipe with infiltration can be seen as a pipe having a *diffusive lateral discharge* option (e.g. infiltration of water in the pipe towards the trench and exfiltration of water in the trench towards the pipe). For more information on the infiltration and exfiltration process, reference is made to Ellis and Bertrand-Krajewski (2010), Rutsch *et al.* (2008) and Karpf *et al.* (2008).

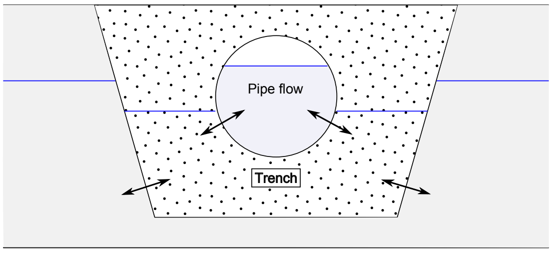


Figure 6.13: *Pipe with Infiltration (e.g. a drainage pipe located in a trench); Allowing for the exchange of water in the pipe towards the trench (positive sign) and vice versa; Allowing for the exchange of water from the trench towards the groundwater (negative sign) and vice versa*

For the computational procedure of a pipe with infiltration yields:

- ◇ The flow (water levels (ζ_p, see Figure 6.14), discharges and velocities) through a pipe with infiltration is computed in exactly the same manner as the flow in any pipe (e.g. by solving the one dimensional depth averaged St. Venant momentum and continuity equations), where the diffusive lateral flow along a pipe with infiltration is taken into account in accordance with the selected lateral discharge assignment option (see section 6.1.11.2).
- ◇ The exchange of water between the pipe and the trench is computed according to the "**Darcy leakage approach** (see Rutsch *et al.* (2008) and Karpf *et al.* (2008))" in which it is assumed that the rate of exchange of water between two media is a function of the resistance at the interface, the interface area and the hydraulic head over the interface. More precisely per meter pipe length (see Figure 6.14), the rate of exchange of water between the pipe and the trench (e.g. specific discharge $q_{p\leftrightarrow tr}$, see Equation (6.15)) depends on the resistance (c_{p-tr}), the wetted perimeter (P_{p-tr}), the averaged water level in the pipe (ζ_p) and the water level in the trench (ζ_{tr}).
- ◇ Per metre pipe length, the maximum volume of water that can be stored in the trench (e.g. fully saturated trench) equals the cross-sectional area of the trench that is filled with trench material ($A_{\text{filed with trench material}}$) times the porosity of the trench material (ϕ_{tr}, [-]). Once the trench is fully saturated, the water level in the trench (h_{tr}, see Figure 6.14) may rise above the top level of the trench ($z_{tr,top}$). In such case, per metre of pipe length an additional volume of water is stored, that is equal to the top width of the trench ($W_{tr,top}$) times the water depth on the trench ($= \zeta_{tr} - z_{tr,top}$). The top level of the trench is supposed to coincide with the local surface level. Therefore in urban terminology, water depth on the trench is referred to as **water on street**.
- ◇ The exchange of water between the groundwater and the trench per metre pipe length (e.g. specific discharge $q_{gw\leftrightarrow tr}$) is also computed according to the "Darcy leakage approach" (see Equation (6.16)).
- ◇ The groundwater reservoir as such is not modelled. The user-defined groundwater levels (h_{gw}, see Figure 6.14) are used as boundary conditions only.

Note: The local surface level is supposed to coincide with the top level of the trench. From a physical point of view, a groundwater level can not rise above the local surface level (see Figure 6.14). Therefore, at the start of a computation it is verified if user-defined ground water levels (h_{gw}) are above the top level of the trench ($z_{tr,top}$). If this is the case (e.g. if $h_{gw} > z_{tr,top}$), the computation is terminated with an error message denoting for which "Pipes with Infiltration" yields that groundwater level(s) are above the top level of the trench.

Note: The thickness of a possible ground-layer in a pipe does affect the flow through a pipe and is consequently taken into account in determining pipe flow. However, the thickness of a possible ground-layer is neglected in the resistance factor, applied for the interface between pipe and trench. In other words, irrespective of the thickness of a possible ground-layer, the resistance factor applied for the pipe-trench interface equals the user-defined resistance factor (c_{p-tr}) (see Equation (6.15)).

Note: Selecting the option "Impermeable" in the "Permeability of Ground/Soil" scroll-box, available on the "Trench Data" tab (see Section 5.7.2.1), means that the interface between trench and groundwater is considered to be impermeable (e.g. no exchange of water between trench and groundwater). More precisely, if impermeable is selected Equation (6.16) is not considered in the computational procedure.

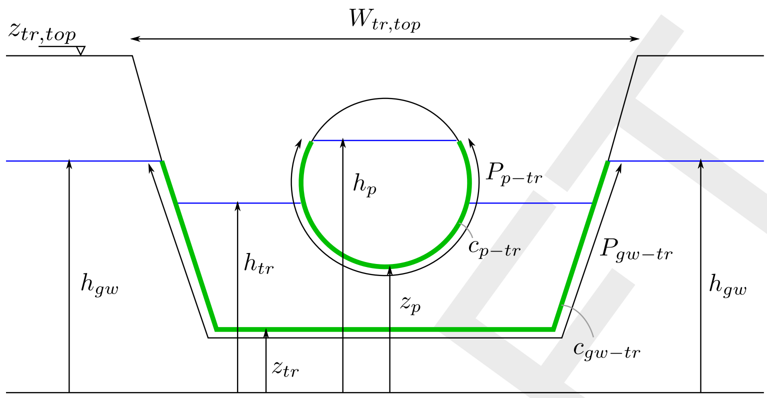


Figure 6.14: *Pipe with infiltration: definition of variables*

The specific discharge ($q_{p\leftrightarrow tr}$) between the pipe and the trench (see Figure 6.14) per metre pipe length is computed according to the "Darcy leakage approach" as follows:

$$q_{p\leftrightarrow tr} = \begin{cases} 0 & \text{if} \quad \zeta_p \leq z_p \quad \text{and} \quad \zeta_{tr} \leq z_p \\ P_{p-tr}(\zeta_p - z_p)/c_{p-tr} & \text{if} \quad \zeta_p > z_p \quad \text{and} \quad h_{tr} \leq z_p \\ P_{p-tr}(\zeta_p - \zeta_{tr})/c_{p-tr} & \text{if} \quad \zeta_p > z_p \quad \text{and} \quad h_{tr} > z_p \end{cases} \tag{6.15}$$

c_{p-tr} resistance factor of the interface between pipe and trench [h].

ζ_1 water level in the pipe (may become pressurized) at the beginning of the pipe (e.g. pipe-side with lowest x-coordinate) [m above datum].

ζ_2 water level in the pipe (may become pressurized) at the end of the pipe (e.g. pipe-side with highest x-coordinate) [m above datum].

ζ_p averaged water level in the pipe (e.g. $(\zeta_1 + \zeta_2)/2$, may become pressurized) [m above datum].

ζ_{tr} water level in the trench [m above datum].

P_{p-tr} wetted perimeter (e.g. length of the interface) between pipe and trench [m].
Note: If $\zeta_p \leq z_p$ and $\zeta_{tr} \leq z_p$, the wetted perimeter equals zero. Else the maximum water level (e.g. max($\zeta_p; \zeta_{tr}$)) determines the length of the wetted perimeter.

$q_{p\leftrightarrow tr}$ specific discharge per metre pipe length between trench and pipe. Positive if water flows from pipe towards the trench [m^2/h].

z_1 invert level at the beginning of the pipe (e.g. pipe-side with lowest x-coordinate) [m above datum].

z_2 invert level at the end of the pipe (e.g. pipe-side with highest x-coordinate) [m above datum].

z_p averaged pipe invert level (e.g. $(z_1 + z_2)/2$) [m above datum].

The specific discharge ($q_{gw\leftrightarrow tr}$) between the pipe and the trench (see Figure 6.14) per metre pipe length is computed according to the “Darcy leakage approach” as follows:

$$q_{gw\leftrightarrow tr} = \begin{cases} 0 & \text{if} \quad \zeta_{tr} \leq z_{tr} \quad \text{and} \quad h_{gw} \leq z_{tr} \\ P_{gw-tr}(\zeta_{tr} - z_{tr})/c_{gw-tr} & \text{if} \quad \zeta_{tr} > z_{tr} \quad \text{and} \quad h_{gw} \leq z_{tr} \\ P_{gw-tr}(\zeta_{tr} - h_{gw})/c_{gw-tr} & \text{if} \quad \zeta_{tr} > z_{tr} \quad \text{and} \quad h_{gw} > z_{tr} \end{cases} \tag{6.16}$$

c_{gw-tr}	resistance factor of the interface between groundwater and trench [h].
h_{gw}	groundwater level [m above datum].
ζ_{tr}	water level in the trench [m above datum].
P_{gw-tr}	wetted perimeter (e.g. length of the interface) between groundwater and trench [m]. **Note:** If $\zeta_{tr} \leq z_{tr}$ and $h_{gw} \leq z_{tr}$, the wetted perimeter equals zero. Else the water level (ζ_{pm}) determining the length of the wetted perimeter is computed as $\zeta_{pm} = \max(h^*_{tr}; h^*_{gw})$, where: if $\zeta_{tr} \geq z_{tr,top}$ then $h\zeta^*_{tr} = z_{tr,top}$ else $\zeta^*_{tr} = \zeta_{tr}$; and if $h_{gw} \geq z_{tr,top}$ then $h^*_{gw} = z_{tr,top}$ else $h^*_{gw} = h_{gw}$.
$q_{gw\leftrightarrow tr}$	specific discharge per metre pipe length between groundwater and trench. Positive if ground water flows into the trench [m^2/h].
$z_{tr,top}$	top level of the trench [m above datum]

6.1.12 Bed friction

The bed friction is the friction between the flowing water and the channel bed. As such, it exerts a force on the flowing water always in the direction opposite the water flow.

In water courses, this force together with the force caused by earth gravity usually determines the flow conditions: the other forces are far less important.

The fourth term of the momentum equation is the bed-friction term:

$$\text{bed friction} = \frac{gQ|Q|}{C^2 R A_F} \tag{6.17}$$

where:

g	Acceleration due to gravity [m/s^2] (≈ 9.81)
Q	Discharge [m^3/s]
C	Chézy coefficient [$m^{1/2}/s$]
R	Hydraulic radius [m]
A_F	Wetted area [m^2]

Note:
If the specified roughness parameter is equal to zero (Chézy=0, Manning=0, etc.), then the bed friction term is not taken into account in the momentum equation.

The following roughness definitions can be used:

- 6.1.12.1, Bos-Bijkerk;
- 6.1.12.2, Chézy;
- 6.1.12.3, Manning;
- 6.1.12.4, Nikuradse;
- 6.1.12.5, Strickler;
- 6.1.12.6, White-Colebrook.

6.1.12.1 Bos-Bijkerk

The bed friction formulation according to de Bos-Bijkerk describes the Manning coefficient as a function of the water depth and a parameter. This parameter can be used to shape the function to a certain empirical curve:

$$k_m = \gamma h^{1/3} \tag{6.18}$$

k_m	Manning roughness coefficient $[m^{1/3}/s]$
h	Water depth $[m]$
γ	Parameter $[1/s]$, normally between 20 and 40

This formula can be rewritten to a formulation for the Chézy coefficient as a function of the water depth

$$C = \gamma h^{1/3} R^{1/6} \tag{6.19}$$

C	Chézy coefficient $[m^{1/2}/s]$
R	Hydraulic radius $[m]$

For every computational time step the new friction value is calculated according to the Bos-Bijkerk formula and sequentially applied to the hydrodynamics calculations.

6.1.12.2 Chézy

The D-Flow 1D module uses the Chézy bed friction value in solving the water flow equations.

The following roughness formulations are possible:

- ◇ Chézy coefficient
- ◇ Bos-Bijkerk friction shape parameter g, resulting in a Chézy value according to:

$$C = \gamma h^{1/3} R^{1/6} \tag{6.20}$$

h	Water depth $[m]$
γ	Parameter $[1/s]$, normally between 20 and 40
C	Chézy coefficient $[m^{1/2}/s]$
R	Hydraulic radius $[m]$

- ◇ White-Colebrook, using the Nikuradse roughness coefficient k_n, results in a Chézy value according to:

$$C = 18\ {}^{10}\log\left(\frac{12R}{k_n}\right) \tag{6.21}$$

- ◇ Manning coefficient, n_m, resulting in a Chézy coefficient according to:

$$C = \frac{R^{1/6}}{n_m} \tag{6.22}$$

- ◇ Strickler, using the Nikuradse k_n roughness coefficient, results in a Chézy value according to:

$$C = 25\left(\frac{R}{k_n}\right)^{1/6} \tag{6.23}$$

◇ Strickler, using the Strickler k_s roughness coefficient, results in a Chézy value according to:

$$C = k_s R^{1/6} \tag{6.24}$$

◇ Engelund-like roughness predictor (for main sections of SOBEK-River profiles only) resulting in a Chézy value according to:

$$C = C_{90}\sqrt{\frac{\theta_{90}}{\theta_s}} \tag{6.25}$$

For SOBEK-River, the Chézy, Nikuradse, Manning or Strickler coefficients may be

◇ a constant
◇ spatially varying
◇ a tabulated function of the water level (h) or the total discharge (Q)

Different values or tables are possible for positive as well as negative flow. See also the chapter 1D hydraulic friction concepts for information on the different friction concepts present in the user interface.

6.1.12.3 Manning

One of the methods to define the bed roughness is using the Manning coefficient, symbol n_m. In the D-Flow 1D module, the Manning coefficient is used to compute the actual value of the Chézy coefficient, by:

$$C = \frac{R^{1/6}}{n_m} \tag{6.26}$$

C	Chézy coefficient [$m^{1/2}/s$]
R	Hydraulic radius [m]
n_m	Manning coefficient [$s/m^{1/3}$]

6.1.12.4 Nikuradse

One of the methods to define the bed friction is by entering an equivalent roughness according to Nikuradse, represented by the symbol k_n [m].

Values of k_n for open channels with bed forms are in the same order of magnitude as the height of the bed forms.

Using this option, the actual value of the Chézy coefficient will be computed according to the White-Colebrook formula.

6.1.12.5 Strickler

One of the methods to define the bed roughness is by using the Strickler formula. The actual value of the Chézy coefficient is computed using:

$$C = 25\left(\frac{R}{k_n}\right)^{1/6} \tag{6.27}$$

or:

$$C = k_s R^{1/6} \tag{6.28}$$

C	Chézy coefficient [$m^{1/2}/s$]
R	Hydraulic radius [m]
k_s	Strickler roughness coefficient [$m^{1/3}/s$]

in which k_n is the Nikuradse equivalent roughness, and k_s is the Strickler roughness coefficient. You may select either ks or kn as input value.

The value of the hydraulic radius R is taken from the last iteration loop.

It is possible to enter the coefficients varying in space and depending on the local flow direction.

6.1.12.6 White-Colebrook

One of the methods to define the bed friction is by specifying an equivalent roughness according to Nikuradse. Using this option, the value of the Chézy coefficient will be computed according to the White-Colebrook formula:

$$C = 18\ {}^{10}\log\left(\frac{12R}{k_n}\right) \tag{6.29}$$

C	Chézy coefficient
R	Hydraulic radius
k_n	Nikuradse equivalent roughness

Note:
Actually, this formula is a simplification of the complete White-Colebrook formula.

6.1.13 Froude number

The Froude number is defined by u/c, with u the average flow velocity and c the wave celerity.

Herein is c as follows:

$$c = \sqrt{\frac{gA_F}{W_f}} \tag{6.30}$$

Thus

$$\text{Froude number} = \frac{u}{\sqrt{gA_F/W_f}} \tag{6.31}$$

u	Velocity ($u = Q/A_F$) [m/s]
g	Acceleration due to gravity [m/s^2] (≈ 9.81)
A_F	Wetted area [m^2]
W_f	Flow width [m]

Remarks:

- ◇ When the Froude number is less than 1, the flow is sub-critical, when it is larger than 1, the flow is super-critical
- ◇ Transitions from sub-critical to super-critical flow causes an hydraulic jump.

6.1.14 Boussinesq

The Boussinesq coefficient is a parameter in the momentum equation for the water flow. It accounts for the non-uniform velocity distribution in a cross-section.

The Boussinesq constant α_B is multiplier to the convective term:

$$\frac{\partial}{\partial x}\left(\alpha_B \frac{Q^2}{A_F}\right) \tag{6.32}$$

The definition of the coefficient α_B is:

$$\alpha_B = \frac{A_F}{Q^2}\int_0^{W_f} \frac{q(y)^2}{h(y)}\, dy \tag{6.33}$$

The constant of Boussinesq is computed by D-Flow 1D. The computation is based upon the Engelund approach (Engelund and Hansen, 1967). In this approach the water level gradient and bed-friction term are assumed to be an order of magnitude larger than the other terms in the momentum equation. The resulting equation is:

$$gA_F\frac{\partial h}{\partial x} = \frac{gQ^2}{C^2RA_F} \tag{6.34}$$

in which locally a constant water level gradient is assumed. In an arbitrary cross-section the discharge can be expressed as:

$$Q = \int_0^{W_f} C(y)h(y)\sqrt{R(y)\frac{\partial h}{\partial x}}\, dy \tag{6.35}$$

in which h is the water depth. Furthermore, it is assumed that the water level and water level slope are the same for main and floodplains. Combining Equation (6.34) and Equation (6.35) leads to a more accurate estimate of the Chézy coefficient based upon the average of the local Chézy coefficients.

$$C = \frac{1}{A_F\sqrt{R}}\int_0^{W_f} C(y)h(y)\sqrt{R(y)}\, dy \tag{6.36}$$

Now Equation (6.33) gives combined with Equation (6.34) and Equation (6.35) the following expression for the Boussinesq coefficient:

$$\alpha_B = \frac{1}{A_FRC^2}\int_0^{W_f} C^2(y)h(y)R(y)\, dy \tag{6.37}$$

Cross-sections in D-Flow 1D may be divided in two or three parts. So the following three options are available.

option 1:
The cross-section is not divided into sections with different roughness (only a main channel is present). This gives for the Boussinesq coefficient:

$$\alpha_B = 1 \tag{6.38}$$

option 2:
The cross-section is divided into two sections with different roughness for the modeling of, for example, a cross-section with a main channel and a floodplain. During a computation this gives two possibilities:

possibility 1:
Actual water flow is in the main channel only:

$$\alpha_B = 1 \tag{6.39}$$

possibility 2:
Actual water flow is in the main channel and the floodplain:

$$\alpha_B = \frac{C_0^2 A_{f_0} R_0 + C_1^2 A_{f_1} R_1}{C^2 R A_F} \tag{6.40}$$

The indices 0 and 1 respectively indicate the main channel and floodplain.

option 3:
The cross-section is divided into three sections with different roughness (main channel, floodplain 1 and floodplain 2). During a computation this gives three possibilities:

possibility 1:
Actual water flow is in the main channel only:

$$\alpha_B = 1 \tag{6.41}$$

possibility 2:
Actual water flow is in the main channel and floodplain 1:

$$\alpha_B = \frac{C_0^2 A_{f_0} R_0 + C_1^2 A_{f_1} R_1}{C^2 R A_F(\zeta)} \tag{6.42}$$

The indices 0 and 1 respectively indicate the main channel and floodplain.
possibility 3:
Water is actually flowing in all sections, main channel, floodplain 1 and floodplain 2:

$$\alpha_B = \frac{C_0^2 A_{f_0} R_0 + C_1^2 A_{f_1} R_1 + C_2^2 A_{f_2} R_2}{C^2 R A_F} \tag{6.43}$$

The indices 0, 1 and 2 respectively indicate the main channel, floodplain 1 and floodplain 2.

6.1.15 Accuracy

Whether you are measuring in a prototype, studying water flow in a scale model or modelling with a software system, you should always make some considerations about the accuracy of your activities and of the results of your study.

As for the results of a mathematical model study, the following elements play a role in establishing the overall accuracy:

- ◇ The reliability of the available data describing the prototype;
- ◇ The accuracy of the available data describing the prototype;
- ◇ The violation of certain assumptions underlying the mathematical modelling concept being used;
- ◇ The experience and skill of the modeller(s);
- ◇ The overall accuracy that is required (perhaps more accurate results could be obtained but the aim of the study may not require such accuracy or due to a lack of time you may have to settle for less);
- ◇ The accuracy of the applied numerical modelling technique.

In general practice the first five items are the most important when you are using a one-dimensional mathematical model for hydrodynamic flow. The numerical accuracy is in that case normally of minor importance.

The D-Flow 1D module uses the Delft-scheme to solve the water flow equations. This scheme is developed with robustness as the most important design aspect. It can deal with phenomena such as drying/flooding, super-critical flow and it guarantees a solution for every time step.

The accuracy of the solution depends on the grid size; the smaller the grid sizes, the more accurate the solution is. The time step can be chosen arbitrarily. It is reduced internally when this is necessary to guarantee stability by means of a time step estimation procedure. Obviously small grid sizes result in a network with more elements and therefore in a longer simulation time.

The Delft-scheme is designed to produce a closed water balance.

6.1.16 Structures

In the D-Flow 1D-modules the following structure types are available:

The flow through structures is computed based on:

- Upstream water level or energy level (River weir, Advanced weir and General structure only)
- Downstream water level or energy level (River weir, Advanced weir and General structure only)
- Structure dimensions (some can be controlled: i.e. crest level, opening height, pump capacity, opening of valve)
- A number of user-defined parameters, depending on the structure type (contraction coefficient, reduction factor, etc.).

The discharges and wetted areas that are computed in the structure formulas are imposed in the branch segment where the structure is located. The formulas use the water levels of the ζ-calculation points on either side of the branch segment. For the River weir, Advanced weir, General structure, River pump, Database structure and Compound structure yields that

SOBEK by default places a computational point 0.5 m upstream and 0.5 m downstream of the structure location. Hence such structure is located in a branch having a default length of 1 m.

Note:
The dimensions of the structure do not contribute to the storage in the water system.

6.1.16.1 Advanced weir

Please note that the discharge through an Advanced weir is computed on basis of upstream and downstream energy levels. Further on please note that default a computational point is located 0.5 m in front and 0.5 m behind an Advanced weir.

Notation
The flow direction can be either positive or negative. A positive flow direction is a flow in the direction that the branches have been specified, i.e. with increasing x-coordinates. The upstream side, facing the beginning of the branch is denoted by the subscript 1, whereas the downstream location facing the end of the branch is denoted with 2. The subscript s is used for locations at the sill.

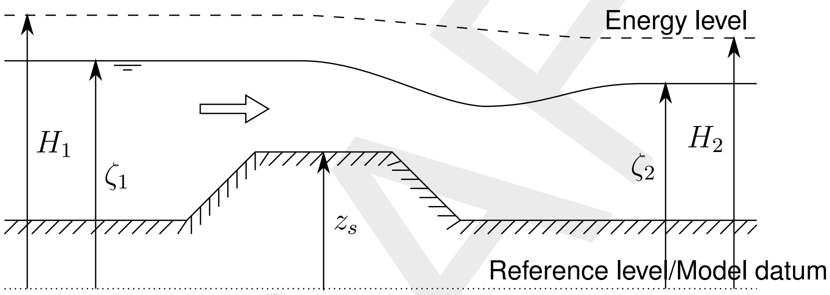


Figure 6.15: *Definition of energy and water level*

and the energy level H is defined as:

$$H = \zeta + \frac{u^2}{2g} \tag{6.44}$$

The user must enter the following input parameters for this type of hydraulic structure:

z_s	Level of crest
W_n	Total net width
N	Number of piers
P	Level of upstream face
H_0	Design head of the weir
K_p	Pier contraction coefficient
K_a	Abutment contraction coefficient

The following parameters can be controlled by a hydraulic structure controller:

W_s	Width across flow section
z_s	Crest level of weir

The discharge through the structure is computed with:

$$Q = C_1 C_2 W \sqrt{2g(H_1 - z_s)^3} \tag{6.45}$$

In which W is the active width and C_1 and C_2 are factors computed in the following way.

Note:
if $\zeta_1 < z_s$ then $Q = 0$.

$$W = W_k - 2(NK_p + K_a)(H_1 - z_s) \tag{6.46}$$

$$C_1 = C_0 C_k \tag{6.47}$$

The value of C_0 is computed depending on the value of the ratio between the upstream face and the design head:

$$\chi = \frac{P}{H_0} \tag{6.48}$$

if $\chi < 2$ then:

$$C_0 = (0.1256\chi^5 - 1.0178\chi^4 + 3\chi^3 - 3.94\chi^2 + 2.28\chi + 1.66)\frac{\xi}{\sqrt{2g}} \tag{6.49}$$

where ξ is a correction factor computed with:

$$\xi = -0.052\chi^3 + 0.145\chi^2 - 0.096\chi + 1.01 \tag{6.50}$$

if $\chi > 2$ then:

$$C_0 = \frac{2.1549008}{\sqrt{2g}} \tag{6.51}$$

The value of χ is computed depending on the value of the ratio between the energy level minus the sill level and the design head:

$$\chi_1 = \frac{H_1 - z_s}{H_0} \tag{6.52}$$

$$C_k = \begin{cases} 0.1394\chi_1^3 - 0.416\chi_1^2 + 0.488\chi_1 + 0.785 & \chi_1 < 1.6 \\ 1.0718224 & \chi_1 \geq 1.6 \end{cases} \tag{6.53}$$

The value of C_t is computed depending on the value of the ratio between the energy level minus the downstream water level and the energy level minus the sill level:

$$\chi_2 = \frac{H_1 - h_2}{H_1 - z_s} \tag{6.54}$$

$$C_t = \begin{cases} \sqrt{(1 - \frac{(\chi_2 - 0.7)^2}{0.49})} + 27(0.7 - \chi_2)^4\sqrt{\chi_2^3} & \chi_2 < 0.7 \\ 1 & \chi_2 \geq 0.7 \end{cases} \tag{6.55}$$

6.1.16.2 Bridge

A bridge is one of the structure types that can be included in the SOBEK-Flow-module. The following types of bridges can be modelled:

- ◇ Pillar bridge
- ◇ Abutment bridge
- ◇ Fixed bed bridge
- ◇ Soil bed bridge

The general description of the discharge through a bridge is given by:

$$Q = \mu A_f \sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.56}$$

Q	Discharge through bridge $[m^3/s]$
μ	Discharge coefficient derived from loss-coefficients $[-]$
A_f	Wetted area $[m^2]$ of flow through bridge at upstream side
g	Acceleration due to gravity $[m/s^2]$ (≈ 9.81)
ζ_1	Upstream water level $[m]$
ζ_2	Downstream water level $[m]$

Note: For numerical reasons (e.g. validity of structure Equation (6.56)) the discharge coefficient (μ) is limited to a maximum of 1.0. In other words if the discharge coefficient (μ) according to Equation (6.57) (Pilar bridge) or Equation (6.59) (Abutment bridge, Fixed bed bridge or Soil bed bridge) becomes larger than 1.0, the actual applied discharge coefficient in Equation (6.56) is limited to 1.0. This means that for a given discharge, the water level difference over such bridge might be larger than anticipated, when considering the defined friction loss coefficients. In such case it is advised to apply the Extra Resistance Node for modelling this particular bridge. An Extra Resistance Node (see section 5.3.8 and section 6.1.1.2) adds additional(extra) resistance to the St. Venant momemtum equation. Hence no structure equation is solved at an Extra Resistance Node.

Pillar bridge

A pillar bridge has one or more pillars that affect the discharge through the bridge with the following definition for the discharge coefficient (μ)

$$\mu = \frac{1}{\sqrt{\xi_y}} \tag{6.57}$$

ξ_y Pillar loss coefficient, defined as

$$\xi_y = \beta \frac{\alpha_y}{A_f} \tag{6.58}$$

β	Parameter $[-]$ depending on shape of pillar$[s]$ (shape factor). Normally between 0.22 and 1.56
A_f	Wetted area $[m^2]$ of flow through bridge at upstream side
α_y	Area $[m^2]$ of wetted part of pillar $[s]$ perpendicular to the flow direction, considered at upstream side

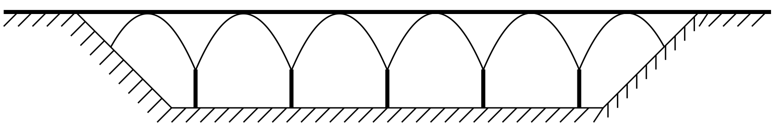


Figure 6.16: *Pillar bridge*

Note: For numerical reasons the discharge coefficient (μ) is limited to a maximum of 1.0. For more information, see the note below Equation (6.56).

The plate of the bridge is always so high that it does not effect the flow through the bridge. So the cross section is considered as open.

Note:
Pillar bridges are not allowed in closed cross-sections.

Abutment bridge

For the abutment bridge the overall loss coefficient is defined as:

$$\mu = \frac{1}{\sqrt{\xi_i + \xi_f + \xi_o}} \tag{6.59}$$

ξ_i	Entrance loss coefficient (in). Constant
ξ_f	Friction loss coefficient
ξ_o	Exit loss coefficient (out).

Note: For numerical reasons the discharge coefficient (μ) is limited to a maximum of 1.0. For more information, see the note below Equation (6.56).

The ξ_o coefficient is defined as:

$$\xi_o = k \left(1 - \frac{A_f}{A_{f_2}}\right)^2 \tag{6.60}$$

k	Constant exit loss coefficient
A_f	Wetted area [m^2] of flow through bridge at upstream side
A_{f_2}	Wetted area [m^2] of flow in branch at downstream side of bridge

The ξ_f coefficient is defined as:

$$\xi_f = \frac{2gL}{C^2 R} \tag{6.61}$$

g	Acceleration due to gravity [m/s^2] (≈ 9.81)
L	Length of bridge [m]
C	Chézy coefficient [$m^{1/2}/s$]
R	Hydraulic radius [m]

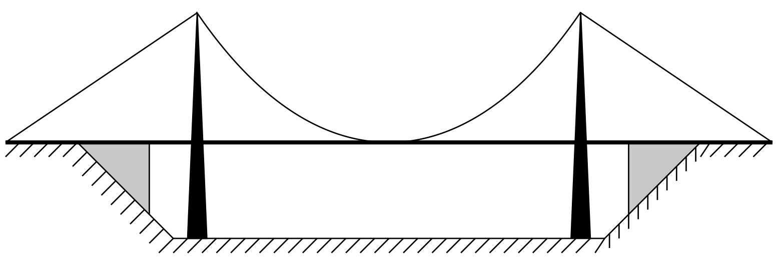


Figure 6.17: *A suspension bridge with abutments*

Here, the plate of the bridge can effect the flow through the bridge. The cross section is closed

Note:
The definition of this bridge is similar to the definition of a culvert.

Fixed bed bridge

This bridge has the same formulation as the abutment bridge except it has a rectangular profile.

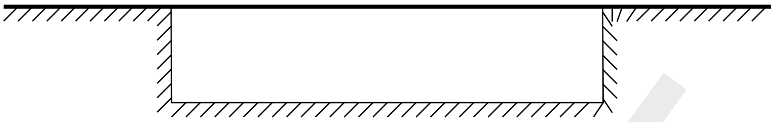


Figure 6.18: *Fixed bed bridge*

Soil bed bridge

This bridge has the same formulation as the fixed bed bridge including the rectangular profile. In addition to this it has a ground layer with a different friction formulation. This ground layer can have a zero thickness.

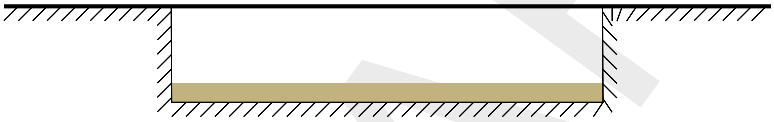


Figure 6.19: *Soil bed bridge*

6.1.16.3 Compound structure

A compound structure consists of several hydraulic structures parallel to each other at one location. These hydraulic structures may be of the same type or of different types. Presently following structure types can be placed as a member in a compound structure, viz.: General structure, Database structure, Advanced weir, River weir and River pump.

Please note that a value for structure inertia damping factor can be defined for each individual member of the compound structure.

Each member of the compound structure has its own triggers and controllers.

6.1.16.4 Culvert

A culvert is one of the structure types that can be included in the SOBEK-Flow-module. A culvert is an underground structure that normally connects two open channels. The flow through a culvert is affected by its upstream and downstream invert levels, the size and shape of its closed cross section, its ground layer thickness, its entrance loss, its friction loss, its valve loss and its exit loss.

Figure 6.20 shows a side view of a culvert.

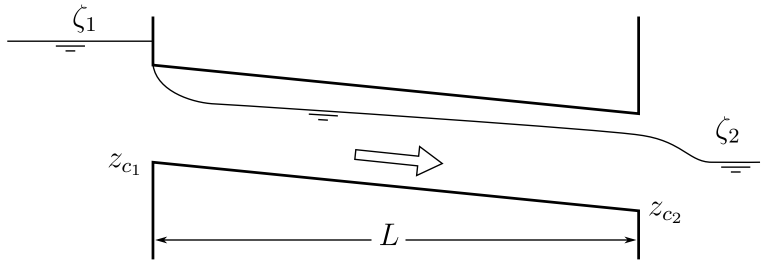


Figure 6.20: *Side view of a culvert*

Two flow conditions can occur:

Free flow when $\zeta_2 < z_{c2} + h_{c2}$

$$Q = \mu A_{fc}\sqrt{2g(\zeta_1 - (z_{c2} + h_{c2}))} \tag{6.62}$$

Submerged flow when $\zeta_2 \geq z_{c2} + h_{c2}$:

$$Q = \mu A_{fc}\sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.63}$$

Q	Discharge through culvert [m^3/s]
μ	Discharge coefficient, derived from loss-coefficients [$-$]
A_{fc}	Discharge culvert flow area $\min(A_{fc1}, A_{fcgate})$ [m^2]
	A_{fc1}: Flow area in the culvert at its upstream side [m^2]
	A_{fcgate}: Flow area under the culvert gate [m^2]
g	Acceleration due to gravity [m/s^2] (≈ 9.81)
ζ_1	Upstream water level [m]
ζ_2	Downstream water level [m]
z_{c2}	Downstream culvert invert level [m]
h_{c2}	Critical culvert depth at the downstream side, $\sqrt[3]{Q^2/(gT_2^2)}$ [m]
T_2	Surface width in the culvert at its downstream side [m]

For numerical reasons the discharge coefficient (μ) is limited to a maximum of 1.0. The discharge coefficient (μ) is computed as follows:

$$\mu = \frac{1}{\sqrt{\xi_i + \xi_f + \xi_v + \xi_o}} \tag{6.64}$$

ξ_i	Entrance loss coefficient [$-$]
ξ_f	Friction loss coefficient [$-$]
ξ_v	Valve loss coefficient [$-$]
ξ_o	Exit loss coefficient [$-$]

The entrance loss coefficient (ξ_i) can be defined as a constant value only.

The friction loss coefficient (ξ_f) is computed as follows:

$$\xi_f = \frac{2gL}{C_1^2 R} \tag{6.65}$$

L	Length of the culvert [m]
C_1	Chézy coefficient in the culvert at its upstream side [m$^{1/2}$/s]

R	Hydraulic radius [m]

$$R = \begin{cases} R_{c1} & \zeta_1 \geq z_{c1} + h_{gate} \\ R_{gate} & \zeta_1 < z_{c1} + h_{gate} \end{cases} \tag{6.66}$$

h_{gate}	Gate height opening [m]
R_{c1}	Hydraulic radius in the culvert at its upstream side [m]
R_{gate}	Hydraulic radius based on actual gate height opening [m]
z_{c1}	Upstream culvert invert level [m]

The valve loss coefficient (ξ_v) can be defined as a constant value or as a function of the ratio of the "Gate height opening" and the "maximum inner culvert height".

Note:

◇ In case the valve loss coefficient (ξ_v) is not a constant, in computations the actual valve loss coefficient (ξ_v) is derived from the user defined table, while using the ratio of the "actual gate height opening and the "actual maximum inner culvert height".
◇ In case the ground layer thickness is greater than zero, both the "actual gate height opening" and the "actual maximum inner culvert height" will differ from the values as defined in the user interface (see next paragraph)

The exit loss coefficient (ξ_o) is computed as follows:

Submerged flow ($\zeta_2 = z_{c2} + h_{c2}$):

$$\xi_o = k \left(1 - \frac{A_{fc}}{A_{fr2}}\right)^2 \tag{6.67}$$

k	User defined constant exit loss coefficient [$-$]
A_{fr2}	Flow area in the branch, adjacent to the downstream culvert side [m^2]
A_{fc}	Culvert flow area [m^2]

$$A_{fc} = \begin{cases} A_{f_{cgate}} & h_1 \geq z_{c1} + h_{gate} \\ A_{f_{c1}} & h1 < z_{c1} + h_{gate} \end{cases} \tag{6.68}$$

h_{gate}	Gate height opening [m]
z_{c1}	Upstream culvert invert level [m]
$A_{f_{cgate}}$	Flow area under the culvert gate [m^2]
$A_{f_{c1}}$	Flow area in the culvert at its upstream side [m^2]

Free flow ($\zeta_2 < z_{c2} + h_{c2}$):

$$\xi_o = 0 \tag{6.69}$$

Culvert cross-sections, bed friction and ground layer

For a culvert all available closed cross-section types can be used. In a culvert, a ground layer with constant thickness can be defined. Culvert friction and ground layer friction can be specified, using any of the available bed friction formulations.

Defining a ground layer thickness > 0 implies that in culvert computations:

◇ Defined invert levels are raised with the ground layer thickness,

◇ Gate height openings are reduced with the ground layer thickness,
◇ Maximum inner height of the culvert is reduced with the ground layer thickness,
◇ Cross-sectional parameters (such as: flow areas, hydraulic radius and so on) are computed based on a cross-sectional profile, that is reduced by the ground layer thickness.

Culvert, Good modelling practice aspects

It is advised to avoid that the bed level of a cross-section in front of a Culvert, Inverted Siphon or Siphon is above the ground-layer level (= invert level + ground-layer thickness), since such situation can result in very small computational time-steps (e.g. long required wall-clock times) or even in a termination of the simulation.

This is explained as follows. Consider the situation depicted in Figure 6.40 were the bed level in front of a Culvert (Inverted Siphon or Siphon) is 0.60 m above the ground-layer level. This means that at small upstream water depths, water will be sucked into the culvert (Inverted Siphon or Siphon), resulting in large flow velocities. For the computational time-step (Δt) yields that $\Delta t \leq \Delta x / U$, where U is the local flow velocity and Δx is distance between two water level computational points (or ζ-points). At very low discharges even negative water depths may be computed, leading to a termination of the simulation.

The situation explained above can be avoided by making the bed level in front of the Culvert (Inverted Siphon or Siphon) equal to the ground-layer level. In other words by defining a bed level slope from ζ-point ζ_2 to ζ_3 as depicted in Figure 6.40. Providing for the parameter "Maximum Lowering of Cross-section Bed Level at Culvert" a value greater or equal to 0.60 m means that before the computation starts, the bed level at ζ-point ζ_3 is set equal to the ground-layer level. In Figure 5.15 it is shown how to provide a value for parameter "Maximum Lowering of Cross-section Bed Level at Culvert".

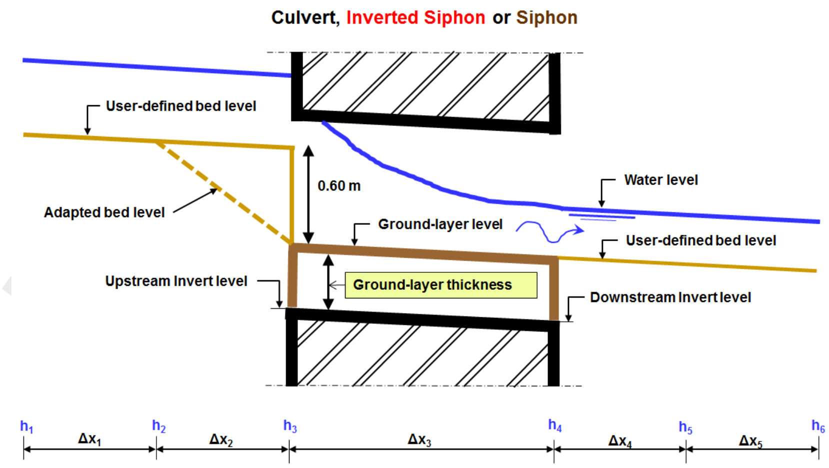


Figure 6.21: *Good modelling practice, Culvert, Inverted Siphon and Siphon*

6.1.16.5 Database structure

Please note that default a computational point is located 0.5 m before and 0.5 m behind a Database structure. In a hydraulic structure the discharge through the structure depends on upstream and downstream water levels and structure parameters that define the dimension of the structure etc. In other words the hydraulic behaviour of a structure can be defined in the structure equation as a relationship between upstream and dowstream water level. In the database structure however, this relationship is defined in a tabulated form which is stored in a database. The user has to define this database.

The database consists in fact of a matrix of discharges. At every point of the matrix a discharge (Q) is defined as a function of two corresponding water levels: one facing the beginning of the branch (ζ_1) and one facing the end (ζ_2), are defined. There are two ways to define the relation:

- ◇ a function of both water levels $Q = Q(\zeta_1 - z_s, \zeta_2 - z_s)$, to be used for structures with relatively high head differences; $z_s = crest\ level$ w.r.t. datum
- ◇ a function of the water level and the water level difference $Q = Q(\zeta_1 - z_s, \Delta\zeta = \zeta_1 - \zeta_2)$, to be used for structures with relatively small water level differences combined with large water level variations; $z_s = crest\ level$ w.r.t. datum

All discharges in a row correspond to the same water level (ζ_1) facing the beginning of the branch segment. All discharges in a column correspond either to the same water level (ζ_2) facing the end of the branch segment or to the water level difference ($\Delta\zeta = \zeta_1 - \zeta_2$). The water levels that are part of the database are defined with respect to a user defined datum.

During a simulation, discharges at any point in the domain of the matrix will be obtained by interpolation. In most cases this will be a linear interpolation in two directions. However when the discharge is requested at a water level that lies within 10 percent of a water level defined in the matrix, cubic interpolation takes place for that water level.

Warning:

- ◇ The crest level can be adjusted by a controller. This implies the assumption that the discharge head relation is independent of the level of the crest. This is not completely correct. The user should be aware of this.

6.1.16.6 General structure

Please note that default a computational point is located 0.5 m in front and 0.5 m behind a general structure. In the general structure type in sobek weir and gate flow is combined in one structure type. In addition, the general structure gives more freedom in defining the dimensions and the geometry of the hydraulic structure. The geometrical shape is given in Figure 6.22 and Figure 6.23. See section 5.3.10 for the definition of input parameters. Please note that the discharge through a General structure is computed on basis of upstream and downstream energy levels. Please note as well that a structural inertia damping factor can be defined for each individual General structure.

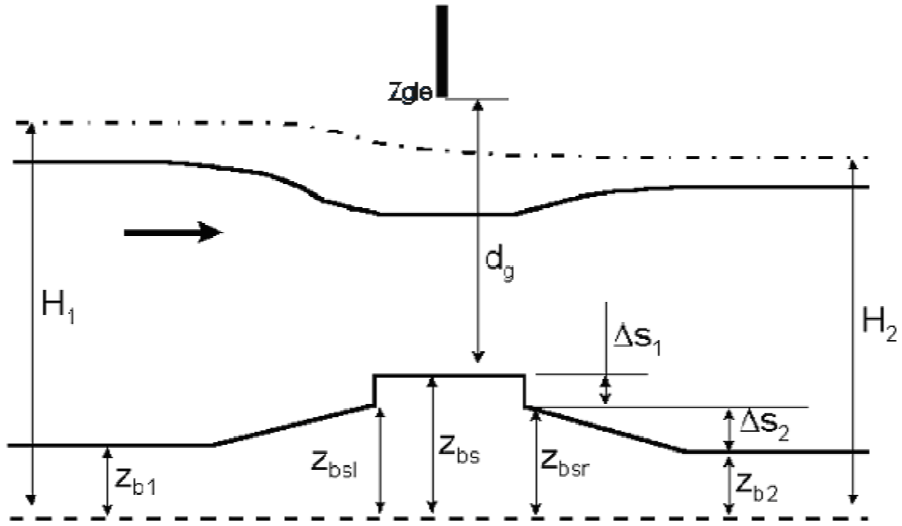


Figure 6.22: *General structure, side view*

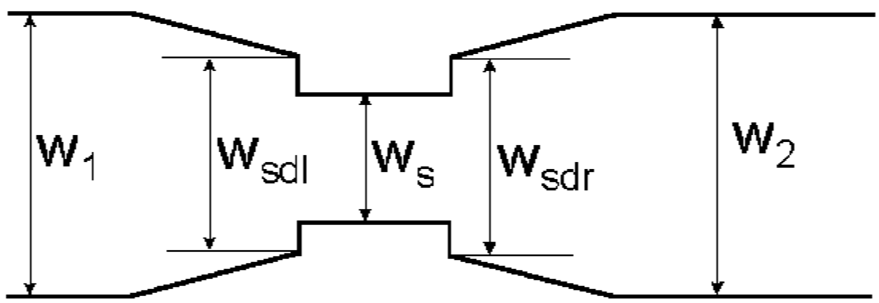


Figure 6.23: *General structure, top view*

Flow across the general structure can be of the following types: drowned weir flow, free weir flow, drowned gate flow, and free gate flow, depending on the dimensions of the structure and the flow conditions.

When salt intrusion is modelled, the density difference of the water over the structure is incorporated in the impulse balance (this is not the case with the other structure types).

In the solution method for the general structure particular attention has been given to the modelling of the transition between free- and submerged flow.

For this purpose, the water level at the sill (ζ_s) is computed by applying an impulse balance instead of taking it equal to the water level further downstream.

As a result the general structure is especially attractive for those who want to simulate the shifting conditions from free- to submerged gate flow accurately and of course in case of important density-differences over the structure.

Computation of downstream water level

In case of drowned gate flow or drowned weir flow the water level at the sill or downstream of the gate is required. This level is computed by application of the impulse balance.

◇ drowned gate-flow

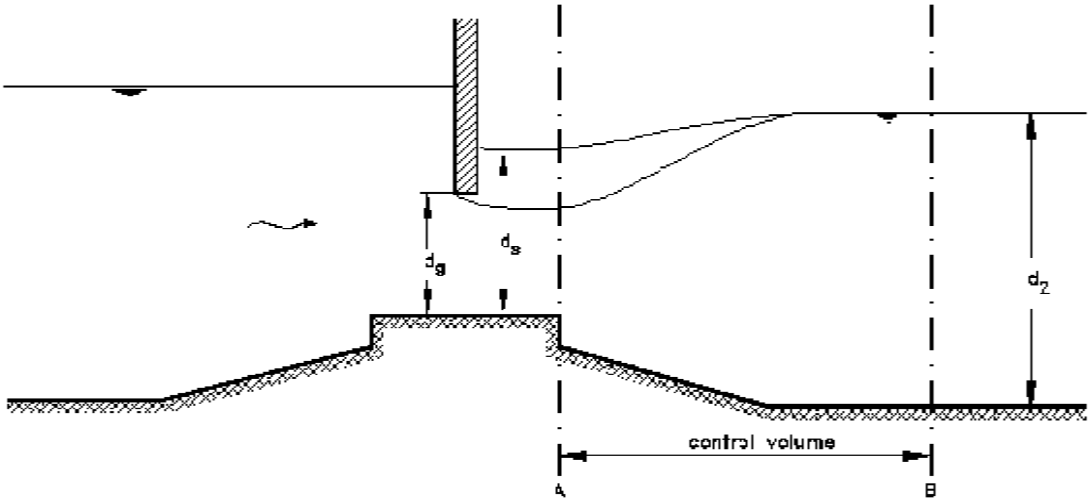


Figure 6.24: *Drowned gate flow*

The water depth h_s can be described by a second order algebraic equation:

$$A_w h_s^2 + B_w h_s + C_w = 0 \tag{6.70}$$

with:

$$A_g = (1+\rho^*)\left(\frac{W_{sd}}{3} + \frac{W_2}{6}\right) + (1-\rho^*)\left(\frac{W_{sd}}{4} + \frac{W_2}{12}\right) \tag{6.71}$$

$$\begin{aligned} B_g = {} & (1+\rho^*)\left((\Delta S_1 + \Delta S_2 - h_2)\left(\frac{W_{sd}}{3} + \frac{W_2}{6}\right) + (2\Delta S_1 + h_2)\frac{W_{sd}}{6} + \right. \\ & \left. (\Delta S_1 + 2h_2)\frac{W_2}{6}\right) + \\ & + (1-\rho^*)\left(\Delta S_1 \frac{W_{sd}}{3} + (\Delta S_1 + h_2)\frac{W_{sd}+W_2}{6}\right) + \\ & + \frac{4\rho^* c_{gd}^2 \mu_{gd}^2 d_g^2 W_s^2}{W_2 h_2}(1 + \frac{\lambda}{d_2}) - 4 c_{gd}\mu_{gd} d_g W_s \end{aligned} \tag{6.72}$$

$$\begin{aligned} C_g = {} & (1+\rho^*)(\Delta S_1 + \Delta S_2 - h_2)\left(2\Delta S_1 + h_2)\frac{W_{sd}}{6} + (\Delta S_1 + 2h_2)\frac{W_2}{6}\right) + \\ & + (1-\rho^*)\left((\Delta S_1)^2 \frac{W_{sd}}{6} + (\Delta S_1 + h_2)\frac{W_{sd}+W_2}{12}\right) + \\ & - \frac{4\rho^* c_{gd}^2 \mu_{gd}^2 d_g^2 W_s^2 H_{s_1}}{W_2 h_2}(1 + \frac{\lambda}{h_2}) + 4 c_{gd}\mu_{gd} d_g W_s H_{s_1} \end{aligned} \tag{6.73}$$

Equation (6.70) leads to:

$$h_s = \frac{-B_g + \sqrt{B_g^2 - 4A_gC_g}}{2A_g} \tag{6.74}$$

◇ drowned weir-flow

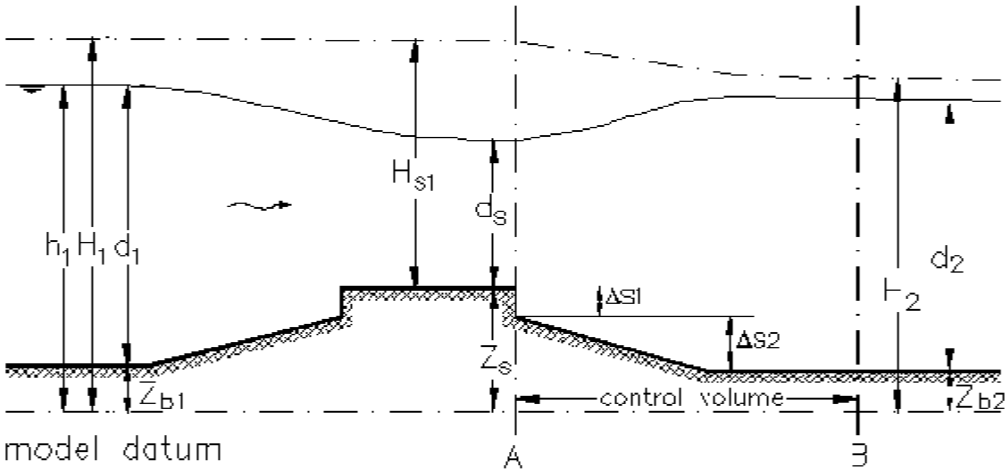


Figure 6.25: *Drowned weir-flow*

The water depth at the sill h_s is described by a third order algebraic equation:

$$D_w h_s^3 + A_w h_s^2 + B_w h_s + C_w = 0 \tag{6.75}$$

with:

$$D_w = \frac{4\rho^* c_{wd}^2 W_s^2}{W_2 h_2}(1 + \frac{\lambda}{h_2}) \tag{6.76}$$

$$A_w = (1+\rho^*)\left(\frac{W_{sd}}{3} + \frac{W_2}{6}\right) + (1-\rho^*)\left(\frac{W_{sd}}{4} + \frac{W_2}{12}\right) + \\ - \frac{4\rho^* c_{wd}^2 W_s^2 H_{s_1}}{W_2 h_2}(1 + \frac{\lambda}{h_2}) + 4c_{wd}W_s \tag{6.77}$$

$$B_w = (1+\rho^*)\left((\Delta S_1 + \Delta S_2 - h_2)\left(\frac{W_{sd}}{3} + \frac{W_2}{6}\right) + (2\Delta S_1 + h_2)\frac{W_{sd}}{6} + \right. \\ \left. (\Delta S_1 + 2h_2)W_2\frac{W_2}{6}\right) + \\ + (1-\rho^*)\left(\Delta S_1 \frac{W_{fd}}{3} + (\Delta S_1 + h_2)\frac{W_{fd} + W_2}{6}\right) + 4C_{wd}W_sH_{s_1} \tag{6.78}$$

$$C_w = (1+\rho^*)(\Delta S_1 + \Delta S_2 - h_2)\left((2\Delta S_1 + h_2)\frac{W_{sd}}{6} + (\Delta S_1 + 2h_2)\frac{W_2}{6}\right) + \\ + (1-\rho^*)\left((\Delta S_1)^2\frac{W_{sd}}{6} + (\Delta S_1 + h_2)\frac{W_{sd} + W_2}{12}\right) \tag{6.79}$$

A direct method is applied to calculate h_s from equation Equation (6.75).

Please note that:

The relative density ρ^* is given by the expression:

$$\rho^* = \frac{\rho_2}{\rho_1} \tag{6.80}$$

where:

ρ^*	relative density $[-]$
ρ_1	density of the water at the upstream side of the General structure $[kg/m^3]$
ρ_2	density of the water at the downstream side of the General structure $[kg/m^3]$

The extra resistance parameter λ relates to the friction force in the impulse balance, that in case of drowned gate flow or drowned weir flow is solved for the water movement at the downstream side of the General structure. The extra resistance parameter λ is given by the expression:

$$\lambda = \frac{gL}{C^2} \tag{6.81}$$

where:

λ	Extra resistance parameter $[m]$
L	Length of the expansion zone at the downstream side of the General structure $[m]$
g	Acceleration due to gravity $[m/s^2]$
C	Chézy value, representing the friction-roughness at the downstream side of the General structure $[m^{1/2}/s]$

Discharge equations

The following discharge equations are applied during the computations.

◇ Free gate flow:

$$u_s = \mu_{gf} c_{gf} \sqrt{2g(H_1 - (z_s + \mu_{gf} d_g))} \tag{6.82}$$

$$A_f = W_s d_g \tag{6.83}$$

$$Q = u_s A_f = \mu_{gf} c_{gf} W_s d_g \sqrt{2g(H_1 - (z_s + \mu_{gf} d_g))} \tag{6.84}$$

◇ Drowned gate flow:

$$u_s = \mu_{gd} c_{gd} \sqrt{2g(H_1 - (z_s + d_s))} \tag{6.85}$$

$$A_f = W_s d_g \tag{6.86}$$

$$Q = u_s A_f = \mu_{gd} c_{gd} W_s d_g \sqrt{2g(H_1 - (z_s + d_s))} \tag{6.87}$$

(6.88)

◇ Free weir flow:

$$u_s = c_{wf} \sqrt{\frac{2}{3} g(H_1 - z_s)} \tag{6.89}$$

$$A_f = W_s \; \frac{2}{3}(H_1 - z_s) \tag{6.90}$$

$$Q = u_s A_f = c_{wf} W_s \; \frac{2}{3} \sqrt{\frac{2}{3}} g(H_1 - z_s)^{3/2} \tag{6.91}$$

◇ Drowned weir flow:

$$u_s = c_{wd}\sqrt{2g(H_1 - (z_s + d_s))} \tag{6.92}$$

$$A_f = W_s d_s \tag{6.93}$$

$$Q = u_s A_f = c_{wd} W_s d_s \sqrt{2g(H_1 - (z_s + d_s))} \tag{6.94}$$

where:

μ_{gf}	contraction coefficient for free gate flow
μ_{gd}	contraction coefficient for drowned gate flow ($\mu_{gd} = \mu_{gf}$)
c_{gf}	correction coefficient for free gate flow
c_{gd}	correction coefficient for drowned gate flow
c_{wf}	correction coefficient for free weir flow
c_{wd}	correction coefficient for drowned weir flow

Note:
The contraction coefficient has a maximum value $\mu = 1.0$. In case the user specifies a higher value, a warning will be generated

Criteria for flow types

First it is assumed that it is weir flow. Then the water level at the sill ζ_s and the critical depth $h_c = \frac{2}{3}(H_1 - z_s)$ are calculated.

The criteria are:

◇ $\zeta_s > h_c$ and $d_g > \zeta_s$ drowned weir flow
◇ $\zeta_s < h_c$ and $d_g > \zeta_s$ free weir flow
◇ otherwise gate flow.

In the latter case the water level at the sill ζ_s is recalculated using the gate flow conditions. The critical depth h_c is now defined as $\mu_{gf} d_g$.

The criteria are.

◇ $\zeta_s > h_c$ drowned gate flow
◇ $\zeta_s < h_c$ free gate flow
◇ ζ_s imaginary free gate flow (i.e. downstream water level below crest level)

Note:
In case upstream water level is above gate lower edge level, there can still be **drowned weir flow** if $\zeta_s > h_c$ and $d_g > \zeta_s$ or **free weir flow** if $\zeta_s < h_c$ and $d_g > \zeta_s$.

6.1.16.7 Inverted siphon

An inverted siphon is one of the structure types, that can be included in the D-Flow 1D-module. An inverted siphon is a structure that normally connects two open channels, that are separated by a particular infrastructural work (e.g. dike, railroad). The inverted siphon makes an underground connection through such infrastructural work. An inverted siphon is assumed to be fully filled with water at its deepest point. In case this does not yield for your stucture, you can consider to use a culvert. The flow through an inverted siphon is affected by its upstream and downstream invert level, the size and shape of its closed cross section, its ground layer thickness, its entrance loss, friction loss, its valve loss, losses due to its bends (bend loss), and its exit loss.

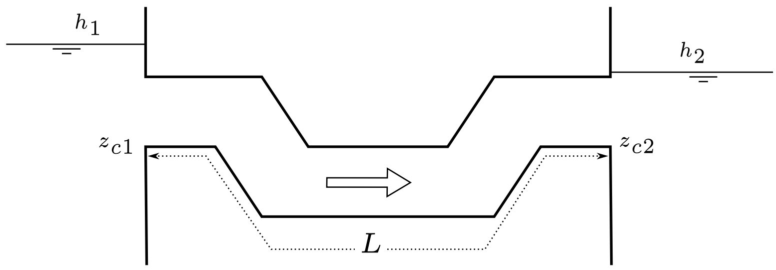


Figure 6.26: *Side view of an inverted siphon*

Two flow conditions can occur:

1 Free flow when $\zeta_2 < z_{c2} + d_{c2}$

$$Q = \mu A_{fis} \sqrt{2g(\zeta_1 - (z_{c2} + d_{c2})} \tag{6.95}$$

2 Submerged flow when $\zeta_2 \geq z_{c2} + d_{c2}$:

$$Q = \mu A_{fis} \sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.96}$$

Q	Discharge through inverted siphon $[m^3/s]$
μ	Discharge coefficient, derived from loss-coefficients $[-]$
A_{fis}	Discharge inverted siphon flow area ($= \min(A_{fcrs}, A_{fisgate})$) $[m^2]$
A_{fcrs}	Cross-sectional area of the inverted siphon. At its deepest point the inverted siphon is considered to be completely filled with water $[m^2]$
$A_{fisgate}$	Flow area under the inverted siphon gate $[m^2]$
g	Acceleration due to gravity $[m/s^2]$ (≈ 9.81)
ζ_1	Upstream water level $[m]$
ζ_2	Downstream water level $[m]$
z_{c2}	Downstream inverted siphon invert level $[m]$
d_{c2}	Critical inverted siphon depth at the downstream side

$$\left(d_{c2} = \sqrt[3]{Q^2/(gT_2^2)} \right) \qquad [m] \tag{6.97}$$

T_2	Surface width in the inverted siphon at its downstream side $[m]$

For numerical reasons the discharge coefficient (μ) is limited to a maximum of 1.0. The discharge coefficient (μ) is computed as follows:

$$\mu = \frac{1}{\sqrt{\xi_i + \xi_f + \xi_v + \xi_b + \xi_o}} \tag{6.98}$$

ξ_i	Entrance loss coefficient $[-]$
ξ_f	Friction loss coefficient $[-]$
ξ_v	Valve loss coefficient $[-]$
ξ_b	Bend loss coefficient $[-]$
ξ_o	Exit loss coefficient $[-]$

The entrance loss coefficient (ξ_i) can be defined as a constant value only.

The friction loss coefficient (ξ_f) is computed as follows:

$$\xi_f = \frac{2gL}{C^2 R} \tag{6.99}$$

L Length of the inverted siphon $[m]$
C Chézy coefficient for fuly water filled inverted siphon [m$^{1/2}$/s]
R Hydraulic radius $[m]$
If GHO $<$ MIISH; $R = R_{gate}$
If GHO $\geq$ MIISH; R$=$ $R_{inverted\ siphon}$
GHO Gate height opening $[m]$
MIISH Maximum inner inverted siphon height $[m]$
$R_{inverted\ siphon}$ Hydraulic radius based on a fully water filled inverted siphon $[m]$
R_{gate} Hydraulic radius based on actual gate height opening $[m]$

The valve loss coefficient (ξ_v) can be defined as a constant value or as a function of the ratio of the "Gate height opening" and the "maximum inner inverted siphon height".

Note:

- ◇ In case the valve loss coefficient (ξ_v) is not a constant, in computations the actual valve loss coefficient (ξ_v) is derived from the user defined table, while using the ratio of the "actual gate height opening and the "actual maximum inner inverted siphon height".
- ◇ In case the ground layer thickness is greater than zero, both the "actual gate height opening" and the "actual maximum inner inverted siphon height" will differ from the values as defined in the user interface.

The exit loss coefficient (ξ_o) is computed as follows:

Submerged flow ($\zeta_2 = z_{c2} + d_{c2}$):

$$\xi_o = k \left(1 - \frac{A_{fis}}{A_{fr2}}\right)^2 \tag{6.100}$$

k User defined constant exit loss coefficient $[-]$
A_{fr2} Flow area in the branch, adjacent to the downstream inverted siphon side $[m^2]$
A_{fis} Inverted siphon flow area $[m^2]$
If GHO $\leq$ MIISH; $A_{fis} = A_{fisgate}$
If GHO $>$ MIISH; $A_{fis} = A_{inverted\ siphon}$
GHO Gate height opening $[m]$
MIISH Maximum inner inverted siphon height $[m]$
$A_{fisgate}$ Flow area under the inverted siphon gate $[m^2]$
$A_{inverted\ siphon}$ Flow area based on a fully water filled inverted siphon $[m^2]$

Free flow ($\zeta_2 < z_{c2} + d_{c2}$):

$$\xi_o = 0 \tag{6.101}$$

Inverted siphon cross-sections, bed friction and ground layer:
For an inverted siphon all available closed cross-section types can be used. In an inverted siphon, a ground layer width constant thickness can be defined. Inverted siphon friction and ground layer friction can be specified, using any of the available bed friction formulations.

Defining a ground layer thickness > 0 implies that in inverted siphon computations:

- ◇ Defined invert levels are raised with the ground layer thickness,
- ◇ Gate height openings are reduced with the ground layer thickness,
- ◇ Maximum inner height of the inverted siphon is reduced with the ground layer thickness,

◇ Cross-sectional parameters (such as: flow areas, hydraulic radius and so on) are computed based on a cross-sectional profile, that is reduced by the ground layer thickness.

Inverted Siphon, Good modelling practice aspects

It is advised to avoid that the bed level of a cross-section in front of a Culvert, Inverted Siphon or Siphon is above the ground-layer level (= invert level + ground-layer thickness), since such situation can result in very small computational time-steps (e.g. long required wall-clock times) or even in a termination of the simulation.

This is explained as follows. Consider the situation depicted in Figure 6.40 were the bed level in front of a Culvert (Inverted Siphon or Siphon) is 0.60 m above the ground-layer level. This means that at small upstream water depths, water will be sucked into the culvert (Inverted Siphon or Siphon), resulting in large flow velocities. For the computational time-step (Δt) yields that $\Delta t \leq \Delta x/U$, where U is the local flow velocity and Δx is distance between two water level computational points (or ζ-points). At very low discharges even negative water depths may be computed, leading to a termination of the simulation.

The situation explained above can be avoided by making the bed level in front of the Culvert (Inverted Siphon or Siphon) equal to the ground-layer level. In other words by defining a bed level slope from ζ-point ζ_2 to ζ_3 as depicted in Figure 6.40. Providing for the parameter "Maximum Lowering of Cross-section Bed Level at Culvert" a value greater or equal to 0.60 m means that before the computation starts, the bed level at ζ-point ζ_3 is set equal to the ground-layer level. In Figure 5.15 it is shown how to provide a value for parameter "Maximum Lowering of Cross-section Bed Level at Culvert".

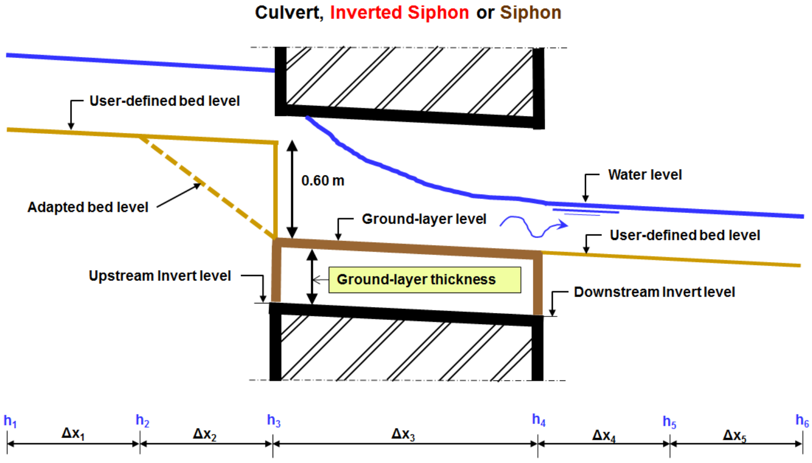


Figure 6.27: Good modelling practice, Culvert, Inverted Siphon and Siphon

6.1.16.8 Orifice

The geometrical shape of an orifice is given in Figure 6.28. Four different types of orifice flow can be discerned, viz: free weir flow, submerged weir flow, free gate flow and submerged gate flow. No discharge flows through the orifice if its gate is closed, if the upstream water level equals the downstream water level or if both the upstream water level and the downstream water level are below the crest level.

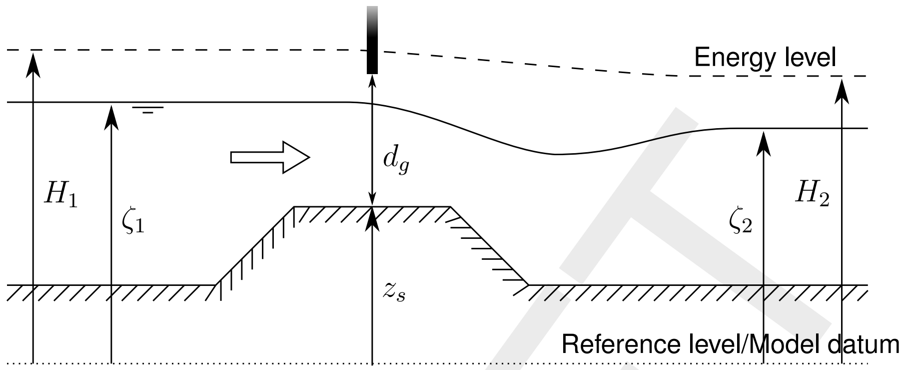


Figure 6.28: *Orifice*

The following four discharge equations are applied:

1 Free weir flow:

$$A_f = W_s \ \frac{2}{3}(\zeta_1 - z_s) \tag{6.102}$$

$$Q = c_w W_s \ \frac{2}{3}\sqrt{\frac{2}{3}g} \ \ (\zeta_1 - z_s)^{3/2} \tag{6.103}$$

2 Submerged weir flow:

$$A_f = W_s \left(\zeta_1 - z_s - \frac{{u_s}^2}{2g}\right) \tag{6.104}$$

$$Q = c_e c_w W_s \left(\zeta_1 - z_s - \frac{{u_s}^2}{2g}\right) \sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.105}$$

3 Free gate flow:

$$A_f = W_s \mu \, d_g \tag{6.106}$$

$$Q = c_w \, W_s \, \mu d_g \sqrt{2g(\zeta_1 - (z_s + \mu \, d_g))} \tag{6.107}$$

4 Submerged gate flow:

$$A_f = W_s \mu \, d_g \tag{6.108}$$

$$Q = c_w \, W_s \, \mu \, d_g \sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.109}$$

Q	Discharge across orifice $[m^3/s]$
A_F	Flow area $[m^2]$
μ	Contraction coefficient $[-]$ Normally 0.63
c_w	Lateral contraction coefficient $[-]$
c_e	Discharge coefficient $[-]$
W_s	Crest width $[m]$

d_g	Opening height $[m]$ (gate lower edge level minus crest level)
g	Gravity acceleration $[m/s^2]$ (≈ 9.81)
ζ_1	Upstream water level $[m]$
ζ_2	Downstream water level $[m]$
z_s	Crest level $[m]$
u_s	Velocity over crest $[m/s]$

The different formulas are applied when the following conditions are met

◇ Free weir flow:

$$\zeta_1 - z_s < \frac{3}{2}d_g \quad \text{and} \quad \zeta_1 - z_s > 3/2(\zeta_2 - z_s) \tag{6.110}$$

◇ Submerged weir flow:

$$\zeta_1 - z_s < \frac{3}{2}d_g \quad \text{and} \quad \zeta_1 - z_s \leq \frac{3}{2}(\zeta_2 - z_s) \tag{6.111}$$

◇ Free gate flow:

$$\zeta_1 - z_s \geq \frac{3}{2}d_g \quad \text{and} \quad \zeta_2 \leq z_s + d_g \tag{6.112}$$

◇ Submerged gate flow:

$$\zeta_1 - z_s \geq \frac{3}{2}d_g \quad \text{and} \quad \zeta_2 > z_s + d_g \tag{6.113}$$

6.1.16.9 Pump station and Internal Pump station

The functionality of a Pump station and an Internal Pump station is identical. The only difference comprises the fact that:

◇ Pump station: A Pump station is located on an open channel branch. The pump discharge is determined using the water levels at the nearest ζ-calculation points, respectively located at the upstream-side and downstream-side of the Pump station.
◇ Internal Pump station: An Internal Pump station is accommodated in a pipe. The pump discharge is determined using the water levels (or hydrostatic pressure heads) at the upstream side of the pipe and at the downstream side of the pipe.

Here after, both the Pump station and the Internal Pump station are referred to as the Pump station. Please note that:

◇ When activated, water is always pumped from the suction-side towards the delivery-side,
◇ A Pump station cannot be placed in a compound structure,
◇ A Time Controller, Hydraulic controller, a PID controller or an Interval controller can overrule the pump capacity of a Pump station.
◇ Pump station output parameters becomes available by checking the Pump Data check-box on the 1DFLOW/Output options/Structures Tab in Settings.

Pump direction

◇ Positive pump direction means that the pump discharge is flowing in the positive x-direction along a branch (see Figure 6.29). Hence, water is pumped in downstream direction towards the pump-side with the highest x-coordinate.
◇ Negative pump direction means that the pump discharge is flowing in the negative x-direction along a branch (see Figure 6.30). Hence, water is pumped in upstream direction towards the pump-side with the lowest x-coordinate.

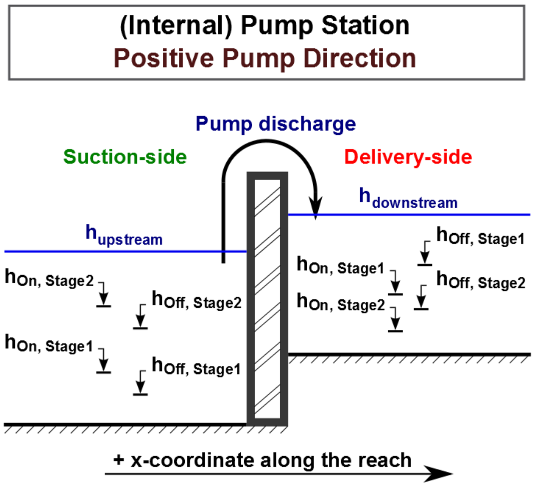


Figure 6.29: *Pump station with positive pump direction and two pump stages*

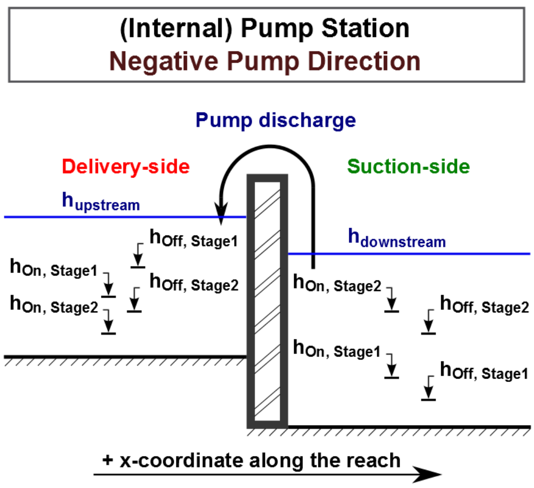


Figure 6.30: *Pump station with negative pump direction and two pump stages*

Six possible pump stages

A Pump station comprises of one (1) pump only, which can have up to six different pump stages (1, 2, 3, 4, 5 or 6). Each pump stage can have a different pump capacity. At each point-in-time, the pump can be in one (1) particular pump stage only.

The actual pump stage

The actual pump stage is determined using dead-band triggers, defined either at the suction-side only, at the delivery-side only or both at the suction-side and the delivery-side of the pump. At the beginning of each time-step for all pump stages, the status [On/Off] of their dead-band trigger at the suction-side (if defined) and their dead-band trigger at the delivery-side (if defined) are evaluated. This evaluation is done on basis of actual water levels and the Switch-on and Switch-off levels, defined for the concerning pump stage. For more details, see dead-band trigger algorithm. For more information on how to define Switch-on and Switch-off levels at the suction-side and the delivery-side of the pump, see Conventions for switch-on and switch-off levels. The actual pump stage, applied in the time-step from $t = t$ to $t = t + \Delta t$, is the number of the highest pump stage (1, 2, 3, 4, 5 or 6) that is triggered at $t = t$, meaning that all its dead-band triggers have the status [On]. If not any pump stage is triggered, the pump becomes inactive (pump discharge is zero) and the actual pump stage is set to 0. If the pump is overruled by a controller, the actual pump stage is set to -1.

Controllers

A Time-controller, a Hydraulic-controller, a PID controller or an Interval controller can be assigned to a pump station. A controller is only active in case the pump is triggered in accordance with the Switch-on and Switch-off levels, defined for stage 1. An active controller overrules the pump capacity of the triggered pump stage, while the actual pump stage is set to -1. Please note that capacity reduction factors are applied to the pump capacities set by a controller. Advice: In using a controller at a pump station, define only one (1) pump stage and take care that its dead-band trigger is always [On].

Capacity reduction table

A capacity reduction table can be defined, which is applied to all pump stages as well as to pump capacities, that are set by a controller. In the capacity reduction table a capacity reduction factor can be given as function of the pump head (e.g. water level at delivery-side minus water level at suction-side). Pump heads (first column) in a capacity reduction table should be in increasing order. Capacity reduction factors should be equal or larger than 0. The pump discharge equals the pump capacity times the capacity reduction factor. If no capacity reduction table is defined in effect a capacity reduction factor equal to 1 is applied. Please note that the pump head at $t = t$ is used to determine the capacity reduction factor to be applied in the time-step from $t = t$ to $t = t + \Delta t$.

Pump station output parameters

Pump station output parameters can be viewed in Result in Charts (Pump.his file) by checking the Pump Data check-box on the 1DFLOW/Output options/Structures Tab in Settings. Available pump station output parameters as function of time are:

- Suction-Side level: Water level at the suction-side of the pump.
- Delivery-Side level: Water level at the delivery-side of the pump.
- Pump Head: The water level at the delivery-side of the pump minus the water level at the suction-side of the pump.
- Actual Pump Stage: Equal to -1 if pump is overruled by a controller; equal to 0 if pump is inactive; or equal to 1, 2, 3, 4, 5 or 6 depending on the actual pump stage that is triggered (see Dead-band triggering algorithm).
- Pump Capacity: The pump capacity is either defined by its controller; equal to zero if pump is inactive; or equal to the pump capacity defined for the actual pump stage.
- Reduction Factor: The reduction factor follows from the pump head and the capacity reduction table.
- Pump Discharge: The pump discharge is equal to the pump capacity times the reduction factor.

Dead-band triggering algoritm

A distinction is to be made between evaluating the status [On/Off] of dead-band triggers defined at the suction-side of the pump and dead-band triggers defined at the delivery-side of the pump:

Dead-band triggering at the Suction-side of the pump

Parameters used in determining the status [On/Off] of a suction-side dead-band trigger are $h_{Suction}$ (water level at the suction-side), $h_{On,Suction}$ (switch-on-level) and $h_{Off,Suction}$ (switch-off-level), where $h_{On,Suction} > h_{Off,Suction}$. Two booleans are used (e.g. $S_{On,Suction}$ and $S_{Off,Suction}$):

- If $h_{Suction} > h_{On,Suction} \quad \Rightarrow \quad S_{On,Suction} = True;\ Else\ S_{On,Suction} = False$
- If $h_{Suction} < h_{Off,suction} \quad \Rightarrow \quad S_{Off,Suction} = True;\ Else\ S_{Off,Suction} = False$

If at the start of a computation yields that $h_{Off,Suction} \leq h_{Suction} \leq h_{On,Suction}$ the dead-band trigger is set [On]. During a computation the dead-band trigger is set [On] or [Off] according to the following rules:

- If $(S_{On,Suction} = True \quad and \quad S_{Off,Suction} = False) \quad \Rightarrow \quad Trigger = [On]$
- If $(S_{On,Suction} = False \quad and \quad S_{Off,Suction} = True) \quad \Rightarrow \quad Trigger = [Off]$
- If $(S_{On,Suction} = False \quad and \quad S_{Off,Suction} = False) \quad \Rightarrow \quad Trigger$ obtains the status determined in the previous computational time step.

Dead-band triggering at the Delivery-side of the pump

Parameters used in determining the status [On/Off] of a delivery-side dead-band trigger are $h_{Delivery}$ (water level at the delivery-side), $h_{On,Delivery}$ (switch-on-level), $h_{Off,Delivery}$ (switch-off-level), where $h_{Off,Delivery} > h_{On,Delivery}$. Two booleans are used (e.g. $S_{On,Delivery}$ and $S_{Off,delivery}$):

- ◇ If $h_{Delivery} < h_{On,Delivery} \quad \Rightarrow \quad S_{On,Delivery} = True; \; Else \; S_{On,Delivery} = False$
- ◇ If $h_{Delivery} > h_{Off,Delivery} \quad \Rightarrow \quad S_{Off,Delivery} = True; \; Else \; S_{Off,Delivery} = False$

If at the start of a computation yields that $h_{On,Delivery} \leq h_{Delivery} \leq h_{Off,Delivery}$ the dead-band trigger is set [On]. During a computation the dead-band trigger is set [On] or [Off] according to the following rules:

- ◇ If $(S_{On,Delivery} = True \;\; and \;\; S_{Off,Delivery} = False) \quad \Rightarrow \quad Trigger = [On]$
- ◇ If $(S_{On,Delivery} = False \;\; and \;\; S_{Off,Delivery} = True) \quad \Rightarrow \quad Trigger = [Off]$
- ◇ If $(S_{On,Delivery} = False \;\; and \;\; S_{Off,Delivery} = False) \quad \Rightarrow \quad Trigger$ obtains the status determined in the previous computational time step

Conventions for switch-on and switch-off levels

A distinction is to be made between switch-on-levels and switch-off-levels, defined at the suction-side and defined at the delivery-side of the pump (see Table 6.1).

- ◇ Suction-side: The switch-on level should increase with increasing stage number. For each stage, its switch-off-level should be lower than its switch-on-level. The switch-off-level should increase with increasing stage number.
- ◇ Delivery-side: The switch-on-level should decrease with increasing stage number. For each stage, its switch-off-level should be higher than its switch-on-level. The switch-off-level should decrease with increasing stage number.

Table 6.1: *Example of switch-on-levels and switch-off-levels at the suction-side and the delivery-side of a pump station*

Pump		**Suction-side**		**Delivery-side**	
Stage No	**Capacity** m^3/s	**Switch-on-level** m	**Switch-off-level** m	**Switch-on-level** m	**Switch-off-level** m
1	0.1	0.80	0.10	0.80	1.90
2	0.2	0.90	0.20	0.70	1.80
3	0.3	1.00	0.30	0.60	1.70
4	0.4	1.10	0.40	0.50	1.60
5	0.5	1.20	0.50	0.40	1.50
6	0.6	1.30	0.60	0.30	1.40

6.1.16.10 External Pump station

An External Pump station is to located at the end of an open channel branch or at the end of a (closed) pipe. The end of an open channel branch can either have the highest x-coordinate or an x-coordinate equal to zero. The same applies for a pipe. The pump discharge is either determined using the boundary water level and the water level at the branch-side of the pump or determined using the boundary water level and the water level (or hydrostatic pressure head) at the pipe-side of the pump. Please note that:

- When activated, water is always pumped from the suction-side towards the delivery-side.
- Water can be stored at an External Pump station, for more information see Storage options at an External Pump station.
- The "External Pump station" cannot be placed in a compound structure,
- A Time Controller, Hydraulic controller, a PID controller or an Interval controller can overrule the pump capacity of an External Pump station.
- External Pump station output parameters becomes available by checking the Pump Data check-box on the 1DFLOW/Output options/Structures Tab in Settings.

External Pump direction:

- Pump direction IN means that (see Figure 6.31):
  - Water is pumped into the model or flows towards the branch/pipe-side (or upstream-side) of the pump,
  - The pump discharge has a negative sign, this is irrespective of the x-direction along the branch (or pipe).
- Pump direction OUT means that (see Figure 6.32):
  - Water is pumped out off the model or flows towards the boundary-side (or downstream-side) of the pump ,
  - The pump discharge has a postive sign, this is irrespective of the x-direction along the branch (or pipe).

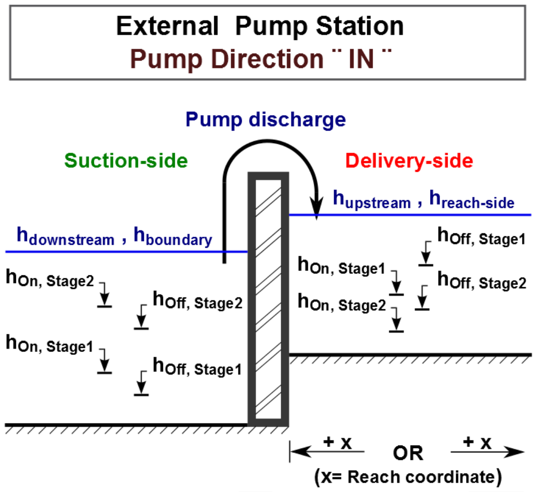


Figure 6.31: *External pump station with pump direction IN and two pump stages*

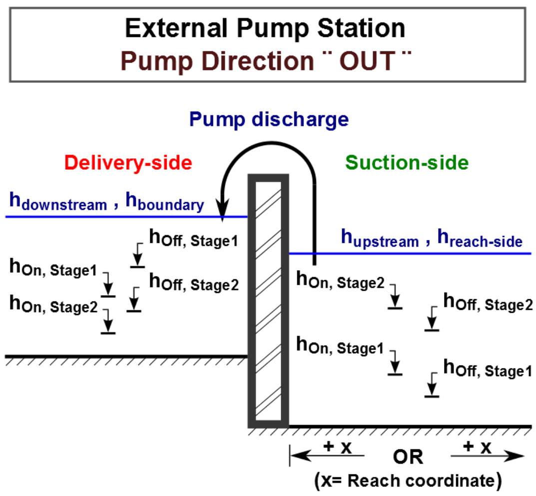


Figure 6.32: *External pump station with pump direction OUT and two pump stages*

Six possible External Pump stages

An External Pump station comprises of one (1) pump only, which can have up to six different pump stages (1, 2, 3, 4, 5 or 6). Each pump stage can have a different pump capacity. At each point-in-time, the pump can be in one (1) particular pump stage only.

The actual pump stage

The actual pump stage is determined using dead-band triggers, defined either at the suction-side only, at the delivery-side only or both at the suction-side and the delivery-side of the pump. At the beginning of each time-step for all pump stages, the status [On/Off] of their dead-band trigger at the suction-side (if defined) and their dead-band trigger at the delivery-side (if defined) are evaluated. This evaluation is done on basis of actual water levels and the Switch-on and Switch-off levels, defined for the concerning pump stage. For more details, see dead-band trigger algorithm. For more information on how to define Switch-on and Switch-off levels at the suction-side and the delivery-side of the External Pump station, see Conventions for switch-on and switch-off levels. The actual pump stage, applied in the time-step from t=t to t=t+dt, is the number of the highest pump stage (1, 2, 3, 4, 5 or 6) that is triggered at t=t, meaning that all its dead-band triggers have the status [On]. If not any pump stage is triggered, the pump becomes inactive (pump discharge is zero) and the actual pump stage is set to 0. If the pump is overruled by a controller, the actual pump stage is set to -1.

Controllers

A Time-controller, a Hydraulic-controller, a PID controller or an Interval controller can be assigned to an External Pump station. A controller is only active in case the pump is triggered in accordance with the Switch-on and Switch-off levels, defined for stage 1. An active controller overrules the pump capacity of the triggered pump stage, while the actual pump stage is set to -1. Please note that capacity reduction factors are applied to the pump capacities set by a controller. Advice: In using a controller at an External Pump station, define only one (1) pump stage and take care that its dead-band trigger is always [On].

Capacity reduction table

A capacity reduction table can be defined, which is applied to all pump stages as well as to pump capacities, that are set by a controller. In the capacity reduction table a capacity reduction factor can be given as function of the pump head (e.g. water level at delivery-side minus water level at suction-side). Pump heads (first column) in a capacity reduction table should be in increasing order. Capacity reduction factors should be equal or larger than 0. The pump discharge equals the pump capacity times the capacity reduction factor. If no capacity reduction table is defined in effect a capacity reduction factor equal to 1 is applied. Please note that the pump head at $t = t$ is used to determine the capacity reduction factor to be applied in the time-step from $t = t$ to $t = t + \Delta t$.

External Pump station output parameters

External Pump station output parameters can be viewed in Result in Charts (Pump.his file) by checking the Pump Data check-box on the 1DFLOW/Output options/Structures Tab in Settings. Available External pump station output parameters as function of time are:

◇ Suction-Side level: Water level at the suction-side of the pump.
◇ Delivery-Side level: Water level at the delivery-side of the pump.
◇ Pump Head: The water level at the delivery-side of the pump minus the water level at the suction-side of the pump.
◇ Actual Pump Stage: Equal to -1 if pump is overruled by a controller; equal to 0 if pump is inactive; or equal to 1, 2, 3, 4, 5 or 6 depending on the actual pump stage that is triggered (see Dead-band triggering algorithm).
◇ Pump Capacity: The pump capacity is either defined by its controller; equal to zero if pump is inactive; or equal to the pump capacity defined for the actual pump stage.
◇ Reduction Factor: The reduction factor follows from the pump head and the capacity reduction table.
◇ Pump Discharge: The pump discharge is equal to the pump capacity times the reduction factor.

Dead-band triggering algoritm

A distinction is to be made between evaluating the status [On/Off] of dead-band triggers defined at the suction-side of the External Pump station and dead-band triggers defined at the delivery-side of the External Pump station:

Dead-band triggering at the Suction-side of the External Pump station:

Parameters used in determining the status [On/Off] of a suction-side dead-band trigger are $h_{Suction}$ (water level at the suction-side), $h_{On,Suction}$ (switch-on-level) and $h_{Off,Suction}$ (switch-off-level), where $h_{On,Suction} > h_{Off,Suction}$. Two booleans are used (e.g. $S_{On,Suction}$ and $S_{Off,Suction}$):

◇ If $h_{Suction} > h_{On,Suction} \quad \Rightarrow \quad S_{On,Suction} = True;\ Else\ S_{On,Suction} = False$
◇ If $h_{Suction} < h_{Off,suction} \quad \Rightarrow \quad S_{Off,Suction} = True;\ Else\ S_{Off,Suction} = False$

If at the start of a computation yields that $h_{Off,Suction} \leq h_{Suction} \leq h_{On,Suction}$ the dead-band trigger is set [On]. During a computation the dead-band trigger is set [On] or [Off] according to the following rules:

◇ If $(S_{On,Suction} = True \quad and \quad S_{Off,Suction} = False) \quad \Rightarrow \quad Trigger = [On]$
◇ If $(S_{On,Suction} = False \quad and \quad S_{Off,Suction} = True) \quad \Rightarrow \quad Trigger = [Off]$
◇ If $(S_{On,Suction} = False \quad and \quad S_{Off,Suction} = False) \quad \Rightarrow \quad Trigger$ obtains the status determined in the previous computational time step.

Dead-band triggering at the Delivery-side of the External Pump station:

Parameters used in determining the status [On/Off] of a delivery-side dead-band trigger are $h_{Delivery}$ (water level at the delivery-side), $h_{On,Delivery}$ (switch-on-level), $h_{Off,Delivery}$ (switch-off-level), where $h_{Off,Delivery} > h_{On,Delivery}$. Two booleans are used (e.g. $S_{On,Delivery}$ and $S_{Off,delivery}$):

- If $h_{Delivery} < h_{On,Delivery} \Rightarrow S_{On,Delivery} = True;\ Else\ S_{On,Delivery} = False$
- If $h_{Delivery} > h_{Off,Delivery} \Rightarrow S_{Off,Delivery} = True;\ Else\ S_{Off,Delivery} = False$

If at the start of a computation yields that $h_{On,Delivery} \leq h_{Delivery} \leq h_{Off,Delivery}$ the dead-band trigger is set [On]. During a computation the dead-band trigger is set [On] or [Off] according to the following rules:

- If $(S_{On,Delivery} = True\ \ and\ \ S_{Off,Delivery} = False) \Rightarrow Trigger = [On]$
- If $(S_{On,Delivery} = False\ \ and\ \ S_{Off,Delivery} = True) \Rightarrow Trigger = [Off]$
- If $(S_{On,Delivery} = False\ \ and\ \ S_{Off,Delivery} = False) \Rightarrow Trigger$ obtains the status determined in the previous computational time step

Conventions for switch-on and switch-off levels

A distinction is to be made between switch-on-levels and switch-off-levels, defined at the suction-side and defined at the delivery-side of the External Pump station (see Table 6.2).

- Suction-side: The switch-on level should increase with increasing stage number. For each stage, its switch-off-level should be lower than its switch-on-level. The switch-off-level should increase with increasing stage number.
- Delivery-side: The switch-on-level should decrease with increasing stage number. For each stage, its switch-off-level should be higher than its switch-on-level. The switch-off-level should decrease with increasing stage number.

Table 6.2: *Example of switch-on-levels and switch-off-levels at the suction-side and the delivery-side of an external pump station*

Pump		Suction-side		Delivery-side	
Stage No	**Capacity** m^3/s	**Switch-on-level** m	**Switch-off-level** m	**Switch-on-level** m	**Switch-off-level** m
1	1.1	1.80	1.10	2.80	3.90
2	2.2	1.90	1.20	2.70	3.80
3	3.3	2.00	1.30	2.60	3.70
4	4.4	2.10	1.40	2.50	3.60
5	5.5	2.20	1.50	2.40	3.50
6	6.6	2.30	1.60	2.30	3.40

Storage options at an External Pump station

Water can be stored at the site, accommodating an external pump station. In SOBEK a distinction is made between water stored below and above a certain design level, hereafter referred to as street level.

- Below street level: At water levels below street level, water is considered to be stored in a so-called bottom-storage-reservoir.
- Above street level: For water levels above street level, there are three options:
    - Water is stored in a street-level-reservoir, having a bed level equal to street level and walls that cannot be overtopped;
    - Water cannot be stored (e.g. closed pump station), hence water levels inside the pump station may become pressurized; and
    - Water leaves the external pump station (e.g. loss of water) by flowing towards low lying areas, while the water level inside the pump station is considered to remain at street level.

6.1.16.11 River Pump

The River Pump is to be located on an open channel branch. Please note that:

- When activated, water is always pumped from the suction-side towards the delivery-side.
- The River – Pump can be placed in a compound structure.
- Controllers cannot be assigned to a River - Pump,
- Flow - ζ-calculation points are default located 0.5 m in front and 0.5 m behind a River - Pump.
- River pump output parameters becomes available by checking the Pump Data check-box on the 1DFLOW/Output options/Structures Tab in Settings.

Control direction determines location of trigger-water level (h_{trig}) of a River pump:

The control direction of the River pump defines the location of the so-called trigger-water level (h_{trig}):

- Upward control direction means that the trigger water level (h_{trig}) is located upward, e.g. at the pump-side with the lowest x-coordinate along the branch.
- Downward control direction means that the trigger water level (h_{trig}) is located downward, e.g. at the pump-side with the highest x-coordinate along the branch.

Start-level and stop-level determine location of the suction-side and the delivery-side:

The elevation of the start-level (h_{start}) and the elevation of the stop-level (h_{stop}) define the location of the suction-side and the delivery-side of a river pump as follows:

- If start-level is above stop-level: water is pumped away from the trigger-water level side of the pump (e.g. trigger-water level side coincides with the suction-side of the pump).
- If stop-level is above start-level: water is pumped towards the trigger-water level side of the pump (e.g. trigger-water level side coincides with the delivery-side of the pump).

Discharge direction of a River pump:

The discharge direction of a river pump with respect to the positive x-direction along the branch is defined by the control direction and the elevations of the start-level and the stop-level. Four different situations can be discerned:

◇ Upward control and start-level above stop-level (see Figure 6.33),
◇ Upward control and stop-level above start-level (see Figure 6.34),
◇ Downward control and start-level above stop-level (see Figure 6.35),
◇ Downward control and stop-level below start-level (see Figure 6.36),

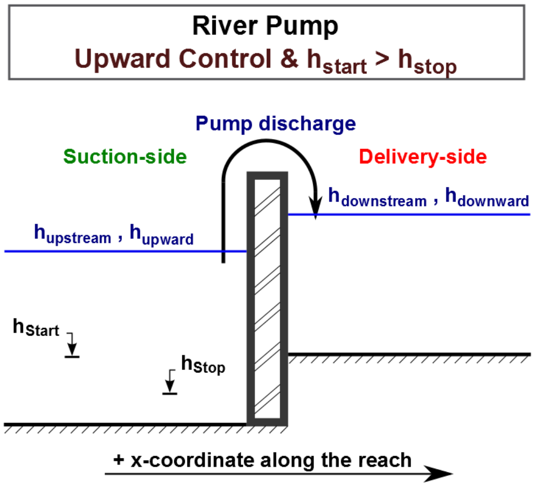


Figure 6.33: River pump with Upward control and start-level above stop-level

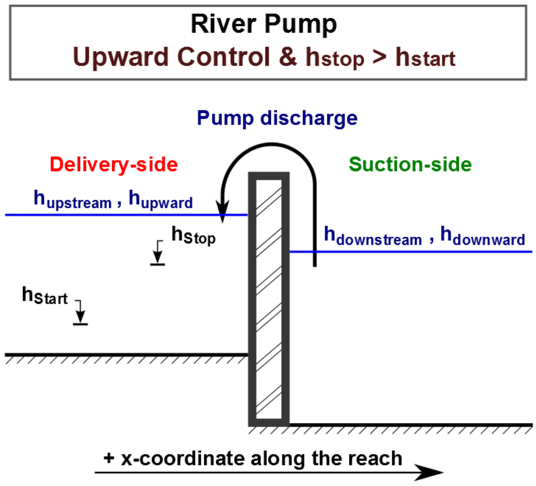


Figure 6.34: River pump with Upward control and stop-level above start-level

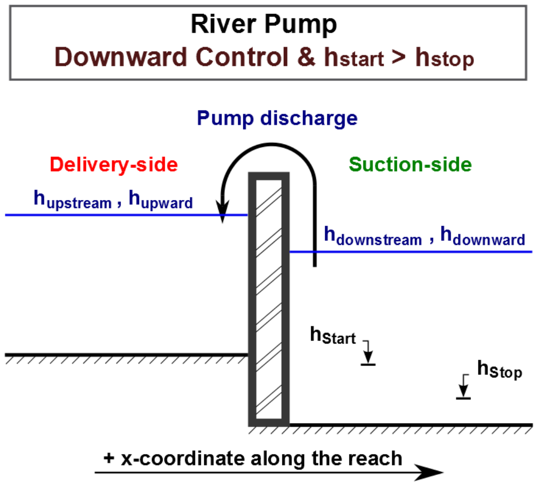


Figure 6.35: River pump with Downward control and start-level above stop-level

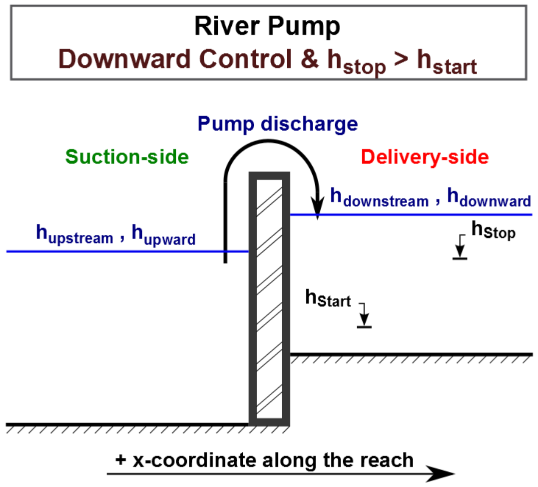


Figure 6.36: *River pump with Downward control and stop-level above start-level*

River pump stages

A River pump has only one pump stage, hence only one pump capacity can be defined.

Constant Reduction Factor or Reduction Factor F(Pump Head)

- ◇ Constant Reduction Factor: WARNING: This means that irrespective of the pump head, the actual pump capacity always equals the defined pump capacity times the constant reduction factor. Say you defined a pump capacity of 100 m^3/s and constant reduction factor of 0.7. This means that the pump will have a capacity of 70 (=0.7*100) m^3/s for both positive and negative pump heads.
- ◇ Reduction Factor F(Pump Head): With this option you define a capacity reduction factor as function of the pump head (e.g. water level at the delivery-side minus the water level at the suction-side). The capacity reduction factor may vary between 0 and 1. The first row in the $F(PumpHead)$ table should read 0,1 (pump head, capacity reduction factor). For positive pump-heads, the pump discharge equals the pump capacity times the capacity reduction factor. For negative pump-heads a capacity reduction factor equal to 1 is applied. Please note that the pump head at $t = t^n$ is used to determine the capacity reduction factor to be applied in the time-step from $t = t^n$ to $t = t^n + \Delta t$.

River pump output parameters

River pump output parameters can be viewed in Result in Charts (<Pump.his> file) by checking the Pump Data check-box on the 1DFLOW/Output options/Structures Tab in Settings. Available River pump output parameters as function of time are:

- ◇ Suction-Side level: Water level at the suction-side of the pump.
- ◇ Delivery-Side level: Water level at the delivery-side of the pump.
- ◇ Pump Head: The water level at the delivery-side of the pump minus the water level at the suction-side of the pump.
- ◇ Actual Pump Stage: Equal to 0 if pump is inactive; or equal to 1 if the pump is triggered (Please note that a river pump has one pump stage only).
- ◇ Pump Capacity: The pump capacity is either equal to zero if pump is inactive; or equal to its defined pump capacity.
- ◇ Reduction Factor: The reduction factor is either a constant or follows from the F(Pump Head) table
- ◇ Pump Discharge: The pump discharge is equal to the pump capacity times the reduction factor.

Dead-band triggering algorithm

A dead-band trigger [On/Off] determines if the River Pump should pump (e.g. actual pump stage = 1) or not pump (e.g. pump discharge is zero; actual pump stage = 0). The actual status [On/Off] of the dead-band trigger is determined as follows. Firstly, the value of two Booleans (S_{on} and S_{off}) is determined. In case of triggering at the suction-side of the pump, or in other words if the start-level (h_{Start}) is above the stop-level (h_{Stop}) yields (see Figure 6.33 and Figure 6.35):

- ◇ If $h_{trig} > h_{Start} \quad \Rightarrow S_{on} = True; \quad Else \quad S_{on} = False$
- ◇ If $h_{trig} < h_{Stop} \quad \Rightarrow S_{off} = True; \quad Else \quad S_{off} = False$

In case of triggering at the delivery-side of the pump, or in other words if the stop-level (hStop) is above the start-level (hStart) yields (see Figure 6.34 and Figure 6.36):

- ◇ If $h_{trig} < h_{Start} \quad \Rightarrow S_{on} = True; \quad Else \quad S_{on} = False$
- ◇ If $h_{trig} > h_{Stop} \quad \Rightarrow S_{off} = True; \quad Else \quad S_{off} = False$

If at the start of a computation yields that $h_{stop} \leq h_{trig} \leq h_{off}$, the dead-band trigger is set [On]. During a computation the dead-band trigger is set [On] or [Off] according to the following algorithm

- ◇ If $(S_{on} = True\ and\ S_{off} = False) \quad \Rightarrow \quad Trigger = [On]$
- ◇ If $(S_{on} = False\ and\ S_{off} = True) \quad \Rightarrow \quad Trigger = [Off]$
- ◇ If $(S_{on} = False\ and\ S_{off} = False) \quad \Rightarrow \quad Trigger$ obtains the status determined in the previous computational time step

Comparing a River Pump and a Pump station:

- ◇ From a pump direction point of view the Pump station covers the options available at a River Pump (see Table 6.3).
- ◇ Advantage of the River Pump over a Pump station is the fact that it can be placed in a compound structure
- ◇ Limitations of the River Pump with respect to the Pump station:
  - □ No controllers can be assigned,
  - □ Only one (1) pump stage can be defined,

- Only one (1) dead-band trigger can be defined at a River Pump, hence it is not possible to operate the pump on basis of water levels at both its suction-side and its delivery-side.

Table 6.3: *Flow-Pump station covers the options of a River pump with respect to the pump direction*

River pump		**Pump station**	
Control Direction	**Start-level** (h_{start}) **Stop-level** (h_{stop})	**Pump Direction**	**Triggering at**
Upward	$h_{start} > h_{stop}$	Positive	Suction-side
Upward	$h_{stop} > h_{start}$	Negative	Delivery-side
Downward	$h_{start} > h_{stop}$	Negative	Suction-side
Downward	$h_{stop} > h_{start}$	Positive	Delivery-side

6.1.16.12 River Weir

Please note that discharges are computed on basis of upstream and downstream energy levels. Further on please note that default a computational point is located 0.5 m in front and 0.5 m behind a River weir. A structure inertia damping factor can be defined for each individual River weir.

Two types of flow conditions can occur in the case of weir flow. These are free (modular) flow and drowned (submerged) flow. If high tail water conditions do affect the flow, the weir is said to be drowned. These conditions may then be computed by applying a reduction factor to the modular function, i.e. to the equation applied if the weir or flume is not drowned (Ackers *et al.*, 1978; Schmidt, 1955; Bos, 1989).

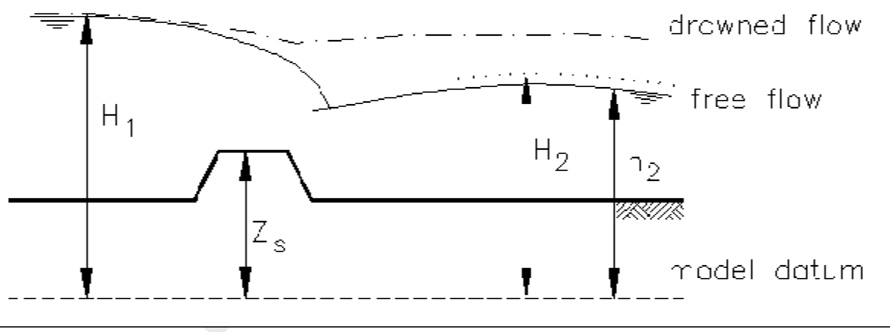


Figure 6.37: *Free and drowned weir flow*

The discharge through the weir is computed with:

$$Q = c_w f W_s \frac{2}{3}\sqrt{\frac{2}{3}g} \;\; (H_1 - z_s)^{3/2} \tag{6.114}$$

in which:

c_w	correction coefficient
W_s	width across flow section
H_1	upstream energy level

f_s	drowned flow reduction factor
z_s	crest or sill level of weir

The submergence factor of the weir is defined as:

$$S_f = \frac{H_2 - z_s}{H_1 - z_s} \tag{6.115}$$

in which:

S_f	submergence factor
H_2	downstream energy level

If the actual submergence factor exceeds the submergence limit, drowned flow occurs. In all other cases modular flow occurs and a drowned flow reduction factor equal to 1.0 should be applied. The submergence limit depends on the crest shape (see Table 6.4).

For a weir the user must specify:

- ◇ level of crest or crest height
- ◇ crest width across flow section
- ◇ crest shape (broad, triangular, round, or sharp)

Depending on the crest shape the following parameters are used (Table 6.4), which can be adjusted by the user:

- ◇ correction coefficient (default see Table 6.4)
- ◇ submergence limit (default see Table 6.4)
- ◇ drowned flow reduction curve (default see Table 6.4)

Table 6.4: *Crest shape and coefficients for simple weir structure (default values)*

Crest shape	**Correction coef.**	**Submergence limit**	**Reduction curve (Figure 6.38)**
broad	1.0	0.82	1
triangular	1.05	0.67	2
round	1.1	0.3	4
sharp	1.2	0.01	5

Notes:

- ◇ If only one flow direction has been specified, the coefficients for that flow direction are used as default coefficients for the other flow direction.
- ◇ The default drowned flow reduction curve depends on the submergence limit.

The drowned flow reduction curve is stored in tabulated format.

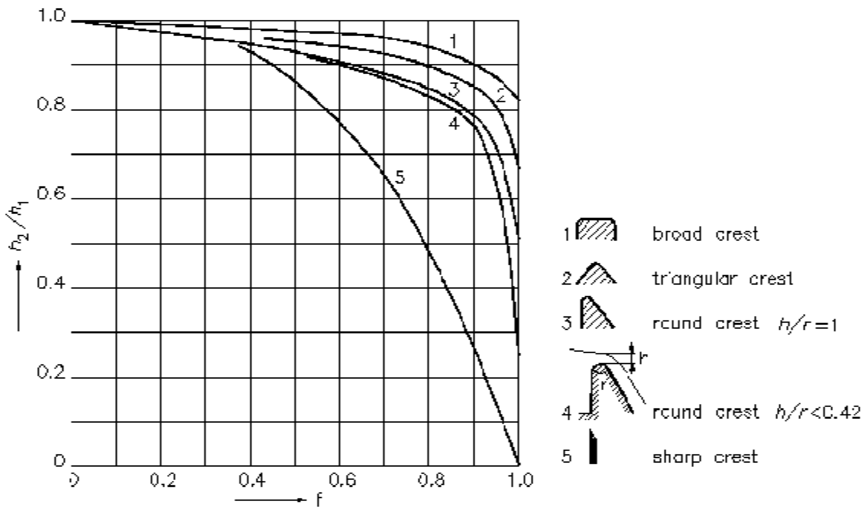


Figure 6.38: *Drowned flow reduction curves*

6.1.16.13 Siphon

A siphon (see Figure 6.39) can only transport water from its upstream defined side to its downstream defined side. ***Hence, only positive flow is possible.***

A switch-on-level and a switch-off-level at its downstream side determine if the siphon may or may not discharge water. If the downstream water level drops below the switch-on-level, the siphon start discharging water. If the downstream water level rises above the switch-off-level, the siphon stops discharging water. If the downstream water level is in between the switch-on-level and the switch-off-level (e.g. in between its dead-band), the siphon remains in its present state of operation. In case the downstream water level is in its dead-band at the onstart of a computation, the siphon will start discharging water.

Discharge can only flow through a siphon if its upstream cross-sectional area is fully filled with water, meaning that the upstream water level is above the obvert level (invert level + inner cross-sectional height) of the siphon. If the upstream cross-sectional area is not fully filled with water, air will be contained in het upper part of the siphon, this air will prevent the siphon from discharging water.

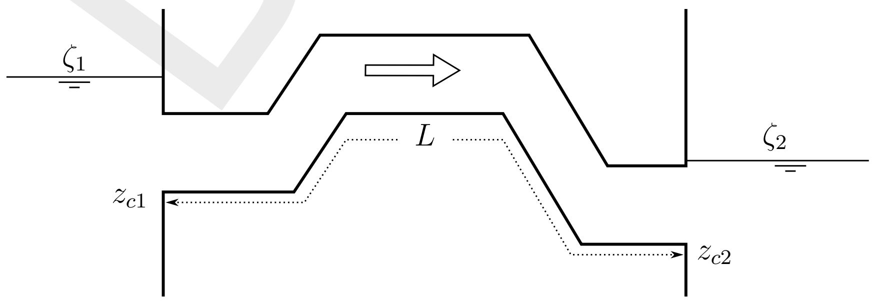


Figure 6.39: *Siphon*

Two flow conditions can occur:

Free flow when $\zeta_2 < z_{c2} + d_{c2}$

$$Q = \mu A_{fs} \sqrt{2g(\zeta_1 - (z_{c2} + d_{c2})} \tag{6.116}$$

Submerged flow when $\zeta_2 \geq z_{c2} + d_{c2}$:

$$Q = \mu A_{fs} \sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.117}$$

Q	Discharge through siphon $[m^3/s]$
μ	Discharge coefficient, derived from loss-coefficients $[-]$
A_{fs}	Discharge siphon flow area $(= \min(A_{fcrs}, A_{fsgate}))$ $[m^2]$
A_{fcrs}	Cross-sectional area of the siphon. At its highest point the siphon is considered to be completely filled with water $[m^2]$
A_{fsgate}	Flow area under the siphon gate $[m^2]$
g	Acceleration due to gravity $[m/s^2]$ (≈ 9.81)
ζ_1	Upstream water level $[m]$
ζ_2	Downstream water level $[m]$
z_{c2}	Downstream siphon invert level $[m]$
d_{c2}	Critical siphon depth at the downstream side

$$(= \sqrt[3]{Q^2/(gT_2^2)}) \qquad [m] \tag{6.118}$$

T_2	Surface width in the siphon at its downstream side $[m]$

For numerical reasons the discharge coefficient (μ) is limited to a maximum of 1.0. The discharge coefficient (μ) is computed as follows:

$$\mu = \frac{1}{\sqrt{\xi_i + \xi_f + \xi_v + \xi_b + \xi_o}} \tag{6.119}$$

ξ_i	Entrance loss coefficient $[-]$
ξ_f	Friction loss coefficient $[-]$
ξ_v	Valve loss coefficient $[-]$
ξ_b	Bend loss coefficient $[-]$
ξ_o	Exit loss coefficient $[-]$

The entrance loss coefficient (ξ_i) can be defined as a constant value only.

The friction loss coefficient (ξ_f) is computed as follows:

$$\xi_f = \frac{2gL}{C^2 R} \tag{6.120}$$

L	Length of the siphon $[m]$
C	Chézy coefficient for fuly water filled siphon [m$^{1/2}$/s]
R	Hydraulic radius $[m]$ If GHO $<$ MIISH; $R = R_{gate}$ If GHO $\geq$ MIISH; R$= R_{siphon}$
GHO	Gate height opening $[m]$
MIISH	Maximum inner inverted siphon height $[m]$
R_{siphon}	Hydraulic radius based on a fully water filled siphon $[m]$
R_{gate}	Hydraulic radius based on actual gate height opening $[m]$

The valve loss coefficient (ξ_v) can be defined as a constant value or as a function of the ratio of the “Gate height opening” and the “maximum inner siphon height”.

Note:

- ◇ In case the valve loss coefficient (ξ_v) is not a constant, in computations the actual valve loss coefficient (ξ_v) is derived from the user defined table, while using the ratio of the “actual gate height opening and the “actual maximum inner siphon height”.
- ◇ In case the ground layer thickness is greater than zero, both the “actual gate height opening” and the “actual maximum inner siphon height” will differ from the values as defined in the user interface (see the paragraph "Siphon cross-sections, bed friction and ground layer" below)

The exit loss coefficient (ξ_o) is computed as follows:

Submerged flow ($\zeta_2 = z_{c2} + d_{c2}$):

$$\xi_o = k\left(1 - \frac{A_{fs}}{A_{fr2}}\right)^2 \tag{6.121}$$

k	User defined constant exit loss coefficient $[-]$
A_{fr2}	Flow area in the branch, adjacent to the downstream siphon side $[m^2]$
A_{fis}	Siphon flow area $[m^2]$ If GHO $\leq$ MISH; $A_{fs} = A_{fsgate}$ If GHO $>$ MISH; $A_{fs} = A_{siphon}$
GHO	Gate height opening $[m]$
MISH	Maximum inner siphon height $[m]$
A_{fsgate}	Flow area under the siphon gate $[m^2]$
A_{siphon}	Flow area based on a fully water filled siphon $[m^2]$

Free flow ($\zeta_2 < z_{c2} + d_{c2}$):

$$\xi_o = 0 \tag{6.122}$$

Siphon cross-sections, bed friction and ground layer:

For a siphon all available closed cross-section types can be used. In a siphon, a ground layer with constant thickness can be defined. Siphon friction and ground layer friction can be specified, using any of the available bed friction formulations.

Defining a ground layer thickness $>$ 0 implies that in siphon computations:

- ◇ Defined invert levels are raised with the ground layer thickness,
- ◇ Gate height openings are reduced with the ground layer thickness,
- ◇ Maximum inner height of the siphon is reduced with the ground layer thickness,
- ◇ Cross-sectional parameters (such as: flow areas, hydraulic radius and so on) are computed based on a cross-sectional profile, that is reduced by the ground layer thickness.

Siphon, Good modelling practice aspects

It is advised to avoid that the bed level of a cross-section in front of a Culvert, Inverted Siphon or Siphon is above the ground-layer level (= invert level + ground-layer thickness), since such situation can result in very small computational time-steps (e.g. long required wall-clock times) or even in a termination of the simulation.

This is explained as follows. Consider the situation depicted in Figure 6.40 were the bed level in front of a Culvert (Inverted Siphon or Siphon) is 0.60 m above the ground-layer level. This means that at small upstream water depths, water will be sucked into the culvert (Inverted Siphon or Siphon), resulting in large flow velocities. For the computational time-step (Δt) yields that $\Delta t \leq \Delta x / U$, where U is the local flow velocity and Δx is distance between two water level computational points (or ζ-points). At very low discharges even negative water depths may be computed, leading to a termination of the simulation.

The situation explained above can be avoided by making the bed level in front of the Culvert (Inverted Siphon or Siphon) equal to the ground-layer level. In other words by defining a bed level slope from ζ-point ζ_2 to ζ_3 as depicted in Figure 6.40. Providing for the parameter "Maximum Lowering of Cross-section Bed Level at Culvert" a value greater or equal to 0.60 m means that before the computation starts, the bed level at ζ-point ζ_3 is set equal to the ground-layer level. In Figure 5.15 it is shown how to provide a value for parameter "Maximum Lowering of Cross-section Bed Level at Culvert".

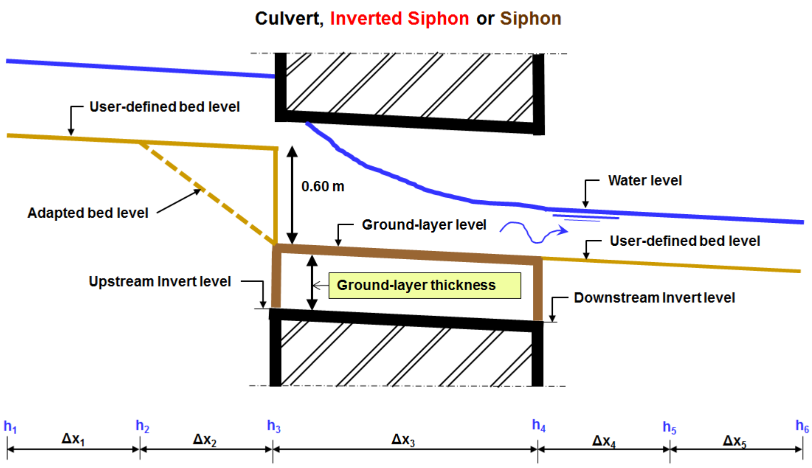


Figure 6.40: *Good modelling practice, Culvert, Inverted Siphon and Siphon*

6.1.16.14 Universal Weir

Crest levels of a universal weir (see Figure 6.41) can be defined as a Y-Z profile. The crest level profile of a universal weir is divided into *rectangular weir sections* having a horizontal bed and *triangular weir sections* having a sloping bed. It is assumed that the discharge over an universal weir is the summation of the discharges over each individual weir section. Rectangular weir sections are considered as a broad-crested weir. Triangular weir sections are considered as (the half of) a broad-crested weir with truncated triangular control section (Bos, 1989).

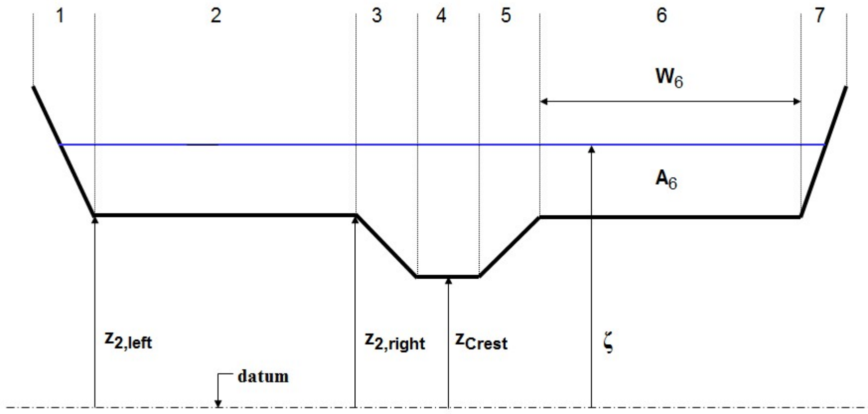


Figure 6.41: *Crest level (Y-Z) profile of a Universal weir, divided into three rectangular weir sections (2, 4 and 6) and four triangular weir sections (1, 3, 5 and 7)*

Parameters depicted in Figure 6.41 are:

A_i	Flow area of section i [m^2]
W_i	Width of weir section i [m]
$z_{i,left}$	Elevation at the left side of weir section i [mAD]
$z_{i,right}$	Elevation at the right side of weir section i [mAD]
z_{Crest}	Crest level of the Universal weir (output parameter only), equal to the lowest elevation of its crest level profile [mAD]
ζ	Water level [mAD]

Following equations yield for a universal weir:

$$Q = \sum_i^N (u_i A_i) = \sum_i^N Q_i \tag{6.123}$$

$$A = \sum_i^N A_i \tag{6.124}$$

$$U_{structure} = Q/A \tag{6.125}$$

where:

A	Flow area of the universal weir [m^2]
A_i	Flow area of weir section i [m^2]
N	Number of weir sections i [$-$]
Q	Discharge over the universal weir [m^3/s]
Q_i	Discharge over weir section i [m^3/s]
u_i	Flow velocity in weir section i [m/s]
$U_{structure}$	Average flow velocity over the universal weir [m/s]

Following equations yield for a rectangular weir section i:

$$z_{i,crest} = min(z_{i,left}\ ,\ z_{i,right}) \tag{6.126}$$

$$ml_{rectangular,i} = 2/3 \tag{6.127}$$

◇ Free rectangular weir flow if $\left(\frac{\zeta_2 - z_{i,crest}}{\zeta_1 - z_{i,crest}}\right) \leq ml_{rectangular,i}$

$$u_i = c_e\sqrt{\frac{2}{3}g}\sqrt{\zeta_1 - z_{i,crest}} \tag{6.128}$$

$$A_i = \frac{2}{3}W_i(\zeta_1 - z_{i,crest}) \tag{6.129}$$

$$Q_i = u_i\, A_i = c_e\, \frac{2}{3}\, \sqrt{\frac{2}{3}g} \quad W_i\, (\zeta_1 - z_{i,crest})^{3/2} \tag{6.130}$$

◇ Submerged rectangular weir flow if $\left(\frac{\zeta_2 - z_{i,crest}}{\zeta_1 - z_{i,crest}}\right) > ml_{rectangular,i}$

$$u_i = c_e\sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.131}$$

$$A_i = W_i(\zeta_2 - z_{i,crest}) \tag{6.132}$$

$$Q_i = u_i\, A_i = c_e\, W_i\, (\zeta_2 - z_{i,crest})\, \sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.133}$$

Following equations yield for a triangular weir section i:

$$z_{i,crest} = min(z_{i,left}\ ,\ z_{i,right}) \tag{6.134}$$

$$dz_i = |z_{i,left} - z_{i,right}| \tag{6.135}$$

$$ml_{triangular,i} = \begin{cases} \frac{4}{5} & \text{if } (\zeta_1 - z_{i,crest})) \leq 1.25\, dz_i \\ \frac{2}{3} + \frac{1}{6}\frac{dz_i}{(\zeta_1 - z_{i,crest})} & \text{if } (\zeta_1 - z_{i,crest})) > 1.25\, dz_i \end{cases} \tag{6.136}$$

◇ Free triangular weir flow if $\left(\frac{\zeta_2 - z_{i,crest}}{\zeta_1 - z_{i,crest}}\right) \leq ml_{triangular,i}$

$$u_i = c_e\sqrt{2g(1 - ml_{triangular,i})(\zeta_1 - z_{i,crest})} \tag{6.137}$$

$$A_i = \begin{cases} W_i\left(\frac{(ml_{triangular,i}\ (\zeta_1 - z_{i,crest}))^2}{2\ dz_i}\right) & \text{if } \frac{\zeta_1 - z_{i,crest}}{1/ml_{triangular,i}} \leq dz_i \\ W_i\left(\frac{\zeta_1 - z_{i,crest}}{1/ml_{triangular,i}} - \frac{dz_i}{2}\right) & \text{if } \frac{\zeta_1 - z_{i,crest}}{1/ml_{triangular,i}} > dz_i \end{cases} \tag{6.138}$$

$$Q_i = u_i\, A_i \tag{6.139}$$

◇ Submerged triangular weir flow if $\left(\frac{\zeta_2 - z_{i,crest}}{\zeta_1 - z_{i,crest}}\right) > ml_{triangular,i}$

$$u_i = c_e\sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.140}$$

$$A_i = \begin{cases} W_i\left(\frac{(\zeta_2 - z_{i,crest})^2}{2\ dz_i}\right) & \text{if } \zeta_2 - z_{i,crest} \leq dz_i \\ W_i\,(\zeta_2 - z_{i,crest} - \frac{dz_i}{2}) & \text{if } \zeta_2 - z_{i,crest} > dz_i \end{cases} \tag{6.141}$$

$$Q_i = u_i\, A_i \tag{6.142}$$

where:

A_i	Flow area of weir section i [m^2]
c_e	Discharge coefficient, applicable to all rectangular weir sections as well as to all triangular weir sections of a universal weir [$-$]
dz_i	Vertical distance between the elevation at the left side and the right side of a triangular weir section i [m]
g	Acceleration due to gravity [m/s^2] (≈ 9.814)
$ml_{rectangular,i}$	*Water-level-based* modular limit of a rectangular weir section i. Its value is always equal to 2/3 [$-$]
$ml_{triangular,i}$	*Water-level-based* modular limit of a triangular weir section i. Its value depends on the actual water depth. [$-$]
Q_i	Discharge over weir section i [m^3/s]
u_i	Velocity in weir section i [m/s]
W_i	Width of weir section i [m]
$z_{i,crest}$	Crest level of weir section i [mAD]
$z_{i,left}$	Elevation at the left side of weir section i [mAD]
$z_{i,right}$	Elevation at the right side of weir section i [mAD]
ζ_1	Water level at the upstream side of the universal weir [mAD]
ζ_2	Water level at the downstream side of the universal weir [mAD]

6.1.16.15 Vertical obstacle friction

The vertical obstacle friction terms have been introduced to account for the added resistance that is caused by vertical obstacles, like houses or trees. The vertical obstacle friction coefficient is based on the number of obstacles, their diameter and drag coefficient. This spatial coefficient can be prescribed in a ARC-INFO grid file, like the bathymetry.

The vertical obstacle friction coefficient a is specified as the summation of the product of obstacle width and drag coefficient over the number of obstacles per unit area.

$$a = \frac{1}{A}\sum_{i=1}^{N} D_i C_{d_i} \tag{6.143}$$

where:

a	Vertical obstacle friction coefficient [$1/m$]
N	Number of obstacles in unit area [$-$]
A	Unit area [m^2]
D_i	Diameter of obstacle i [m]
C_{d_i}	Drag coefficient of obstacle i [$-$]

For round pillars, Cd is usually 1.0. In this formulation it is assumed that the obstacle height will always be larger than the waterdepth. In case the obstacles do become submerged, a more advanced option is the Depth-Dependent Vegetation Roughness.

6.1.16.16 Weir

Three types of flow conditions can occur in the case of weir flow. These are free (modular) flow, submerged flow and no flow (water levels below crest level). If high tail water conditions do affect the flow, the weir is said to be submerged.

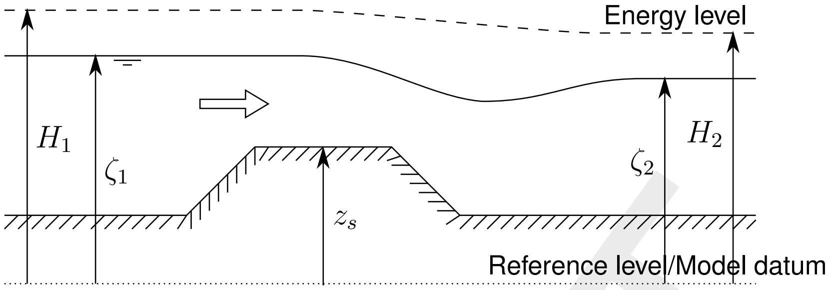


Figure 6.42: *Weir*

The discharge and wetted area through the weir is computed with the following formulas:

Free weir flow:

$$u_s = c_e\sqrt{\frac{2}{3}g}\sqrt{\zeta_1 - z_s} \tag{6.144}$$

$$A_f = c_w W_s \frac{2}{3}(\zeta_1 - z_s) \tag{6.145}$$

$$Q = u_s A_f = c_e c_w W_s \frac{2}{3}\sqrt{\frac{2}{3}g}(\zeta_1 - z_s)^{3/2} \tag{6.146}$$

Submerged weir flow:

$$u_s = c_e\sqrt{2g(\zeta_1 - \zeta_2)} \tag{6.147}$$

$$A_f = c_w W_s(\zeta_2 - z_s) \tag{6.148}$$

$$Q = u_s A_f = c_e c_w W_s\sqrt{2g(\zeta_1 - \zeta_2)}(\zeta_2 - z_s) \tag{6.149}$$

Q	Discharge across weir [m^3/s]
A_f	Wetted area [m^2]
c_e	Discharge coefficient [$-$]
c_w	Lateral contraction coefficient [$-$]
W_s	Crest width [m]
g	Acceleration due to gravity [m/s^2] (≈ 9.81)
ζ_1	Upstream water level [m]
ζ_2	Downstream water level [m]
z_s	Crest level [m]
u_s	Velocity over crest [m/s]

The different formulas are applied when the following conditions occur

Free weir flow:

$$\zeta_1 - z_s > \frac{3}{2}(\zeta_2 - z_s) \tag{6.150}$$

Submerged weir flow:

$$\zeta_1 - z_s \leq \frac{3}{2}(\zeta_2 - z_s) \tag{6.151}$$

Note:
Broad weirs can cause oscillations, because the discharge calculated with the broad crest move a lot of water at a time step. This large discharge can lower the upstream water level significantly resulting in a discharge with a reverse flow direction at the next time step. The following (rather conservative) rule-of-thumb can be used to avoid oscillations:

$$A_s = \frac{3}{2} W_s \tag{6.152}$$

A_s	Storage area upstream of the structure $[m^2]$
W_s	Crest width $[m]$

6.1.17 Staggered grid; ζ- and u-calculation points at different locations

A SOBEK-Flow-model consists of a network of branches connected to each other at connection nodes. The SOBEK-Flow-model uses a staggered grid. On a staggered grid the water level points (e.g. ζ-calculation points) and velocity points (e.g. u-calculation points) are at different locations (see Figure 6.43). Water levels are computed at the ζ-calculation points, while velocities are computed half way between the ζ-calculation points, on the so called u-calculation points. Please note that Connection Nodes and Boundary Nodes are ζ-calculation points.

In each branch a number of ζ-calculation points can be defined. These ζ-calculation points represent the spatial numerical grid used in the simulation. The momentum equation is solved between ζ-calculation points, while the continuity equation is solved between the u-calculation points.

The location of each ζ-calculation point should be selected on various criteria:

- ◇ the distance between two neighbouring ζ-calculation points should not be too large (for accuracy and proper representation of the physical processes)
- ◇ the distance between two neighbouring ζ-calculation points should not be too small because of increasing simulation time. The default minimum distance SOBEK uses during a simulation is 1 meter.
- ◇ the location of the ζ-calculation points may be non-equidistant.

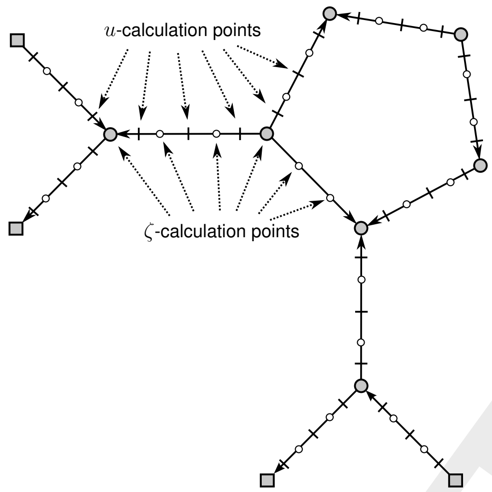


Figure 6.43: *Staggered grid, ζ- and u-calculation points at different locations*

6.1.18 Construction of the numerical bathymetry on basis of user-defined cross-sections

Cross-sections (e.g. bathymetrical data) may be defined at arbitrary locations in a model, provided that for each branch at least one cross-section is specified. An overview of the available type of cross-sections is given in section 5.6.1.

This section explains how the numerical bathymetry of a SOBEK model is constructed on basis of the user-defined cross-sections. With numerical bathymetry is referred to the bathymetrical information, that depending on the cross-section type is available at ζ-calculation points only or available at both ζ- and u-calcuation points. Water levels and discharges are computed using this numerical bathymetrical information. For more information on ζ- and u-calculation points, reference is made to section 6.1.17.

In constructing the numerical bathymetry a distinction is to be made between"'

◇ The Y-Z type of profiles (e.g. Open Vertical Segmented Y-Z profile, Asymmetrical Trapezium profile, Open Lumped Y-Z profile and Closed Lumped Y-Z profile).
◇ All cross-section types except for the Y-Z type of profiles.

6.1.18.1 The Y-Z type of profiles

With the Y-Z type of profiles is referred to the Open Vertical Segmented Y-Z profile, the Asymmetrical Trapezium profile, the Open Lumped Y-Z profile and the Closed Lumped Y-Z profile (for more information, see section 5.6.1).

For the Y-Z type of profiles yields that numerical bathymetrical information is made available at both ζ- and u-calculation points by interpolating (see section 6.1.19) between the user-defined cross-sectional profiles. The construction of the numerical bathymetrical information for Y-Z type of profiles is depicted in Figure 6.44.

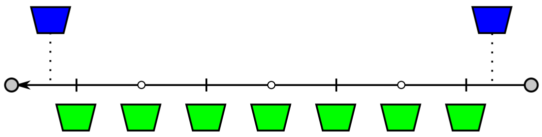


Figure 6.44: Construction of the numerical bathymetrical information for Y-Z type of profiles; Blue coloured objects are the user-defined cross-sections; Green coloured objects are the interpolated numerical bathymetrical information; Open circle are ζ-calculation points; Small vertical lines are u-calculation points

Note: If only one cross-section is specified for a branch, the bathymetrical information of each ζ-calculation point and each u-calculation point located on this branch will be identical to this specified cross-section.

6.1.18.2 All cross-section types except for the Y-Z type of profiles

With the Y-Z type of profiles is referred to the Open Vertical Segmented Y-Z profile, the Asymmetrical Trapezium profile, the Open Lumped Y-Z profile and the Closed Lumped Y-Z profile (for more information, see Section 5.6.1).

For all cross-section types except for the Y-Z type of profiles yields that numerical bathymetrical information is made available at ζ-calculation points only by interpolating (see section 6.1.19) between user-defined cross-sectional profiles. The construction of the numerical bathymetrical information for all cross-section types except the Y-Z type of profiles is depicted in Figure 6.45.

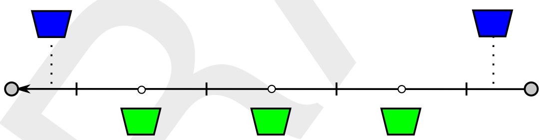


Figure 6.45: Construction of the numerical bathymetrical information for all cross-section types except for the Y-Z type of profiles; Blue coloured objects are the user-defined cross-sections; Green coloured objects are the interpolated numerical bathymetrical information; Open circle are ζ-calculation points; Small vertical lines are u-calculation points

Note: If only one cross-section is specified for a branch, the numerical bathymetrical information for each ζ-calculation point located on this branch will be identical to this specified cross-section.

6.1.19 Method of interpolating between user-defined cross-sections

This section explains the method of interpolating between user-defined cross-sections. An overview of the available type of cross-sections is given in section 5.6.1. Following distinction between cross-sections types is to be made:

- ◇ Round and Egg-shape (see section 6.1.19.1).
- ◇ Open Vertical Segmented Y-Z profile and Asymmetrical Trapezium profile (see section 6.1.19.2).
- ◇ Cross-sections not being a Round, Egg-shape, Open Vertical Segmented Y-Z profile or Asymmetrical Trapezium profile (see section 6.1.19.3).

6.1.19.1 Method of Interpolating between Round cross-sections and between Egg-shape cross-sections

A Round is specified by an invert level, a diameter and the elevation of a ground-layer surface. Interpolating between two Round cross-sections refers to the linear interpolation along the branch of invert level, diameter and the elevation of the ground-layer surface.

An Egg-shape is specified by an invert level, a height, a width and the elevation of a ground-layer surface. Interpolating between two Egg-shape cross-sections refers to the linear interpolation along the branch of invert level, height, width and the elevation of the ground-layer surface.

6.1.19.2 Method of Interpolating between Open Vertical Segmented Y-Z profiles and between Asymmetrical Trapezium profiles

For both an Open Vertical Segmented Y-Z profile and an Asymmetrical Trapezium profile yields that first from the user-defined cross-sectional information so-called conveyance tables are constructed. In a conveyance table the width, area and conveyance as function of water level are given. Interpolation between two of these type of profile refers to the linear interpolation along the branch of their respective conveyance tables.

6.1.19.3 Method of Interpolating between Cross-sections not being a Round, Egg-shape, Open Vertical Segmented Y-Z profile or Asymmetrical Trapezium profile

For all cross-sections not being a Round, Egg-shape, Open Vertical Segmented Y-Z profile or Asymmetrical Trapezium profile yields that the cross-sectional profile is defined in a Table.

Before interpolation starts it is taken care of that the concerning two cross-sections contain exactly the same vertical spacing for flow and total widths. In this context vertical spacing means the level at which a certain width in one of the two cross-sections is defined minus the bed level (lowest point) of the concerning cross-section. Afterwards, for each vertical spacing the flow width and total width of an intermediate constructed cross-section is determined by linear interpolation along the branch. The ground-layer surface of an intermediate constructed cross-section is determined by linear interpolation along the branch as well.

Note: In case of a River profile, the properties of a summer dike (e.g. Flow area, Total area, Dike crest level and Flood plain base level) are linear interpolated along the branch. The same applies for the width of the main section, the width of floodplain 1 and the width of flood plain 2.

Note: The height of a cross-section is the maximum level at which a width is defined minus the bed level (lowest point) of this cross-section. For open cross-sectional profiles yields that in case two cross-sections have different heights, the height of the lowest cross-section will be vertically extended with a width equal to the width defined at its maximum level. This is done to obtain in the lowest cross-section exactly the same vertical spacing as in the highest cross-section. Please note that the same procedure is applied for closed cross-sections, with the exception that the width of the lowest cross-section above its obvert level (maximum defined level) becomes equal to zero.

6.1.20 Methods for computing conveyance

Conveyance is computed for each u-calculation point (or velocity calculation point). A u-calculation point is located in the centre point of its two adjacent ζ-calculation points (or water level calculation points). A distinction is to be made between:

- ◇ The lumped conveyance approach (see section 6.1.20.1),
- ◇ Analytical formulae for computing lumped conveyance, and
- ◇ The vertically segmented conveyance approach (see section 6.1.20.2).

The vertically segmented conveyance approach is used for Y-Z profiles and Asymmetrical Trapezium cross-sections only. Analytical formulae for computing lumped conveyance are used for the Egg-shaped cross-section and the Round cross-section. For the remaining cross-section types, the lumped conveyance approach is used.

6.1.20.1 Lumped conveyance approach

In the lumped conveyance approach, it is assumed that there is an uniform flow velocity ($\overline{U}$) in the cross-sectional profile for a given water level (see Figure 6.46). Hence, it is assumed that differences in flow velocities across the cross-sectional profile can be neglected. This assumption generally does not yield for rivers. Therefore, in modelling rivers Y-Z profiles (or Asymmetrical Trapeziums) are preferred, where conveyance is computed using the vertically segmented conveyance approach (see section 6.1.20.2).

Figure 6.46: *Concept of the lumped conveyance approach*

First an asymmetrical cross-sectional profile as depicted in Figure 6.46 is transferred into a symmetrical cross-sectional profile. Thereafter the lumped conveyance is computed as:

$$K(h) = A(h)C(h)\sqrt{R(h)} \tag{6.153}$$

where:

$K(h)$	Lumped conveyance at water level h
$A(h)$	Cross-sectional area at water level h
$C(h)$	Chézy friction value at water level h
$P(h)$	Wetted Perimeter at water level h
$R(h)$	Hydraulic radius at water level h ($R(h) = A(h)/P(h)$)

If the water level in an open tabulated rises above the highest level in a cross-sectional profile, the lumped conveyance is computed taking the flow width at this water level equal to the flow width, defined at the highest cross-sectional level.

Note: Please note that in case of a River profile, the lumped conveyance of its main section, floodplain 1 and floodplain 2 are computed separately, and thereafter added to obtain the total lumped conveyance of a river profile at a particular water level.

6.1.20.2 Vertically segmented conveyance approach

The vertically segmented conveyance approach is suited for modelling flow in rivers, since in this approach it is assumed that flow velocities vary across the cross-sectional profile for a given water level. In line with this assumption, the cross-section is divided in vertical segments (see Figure 6.47).

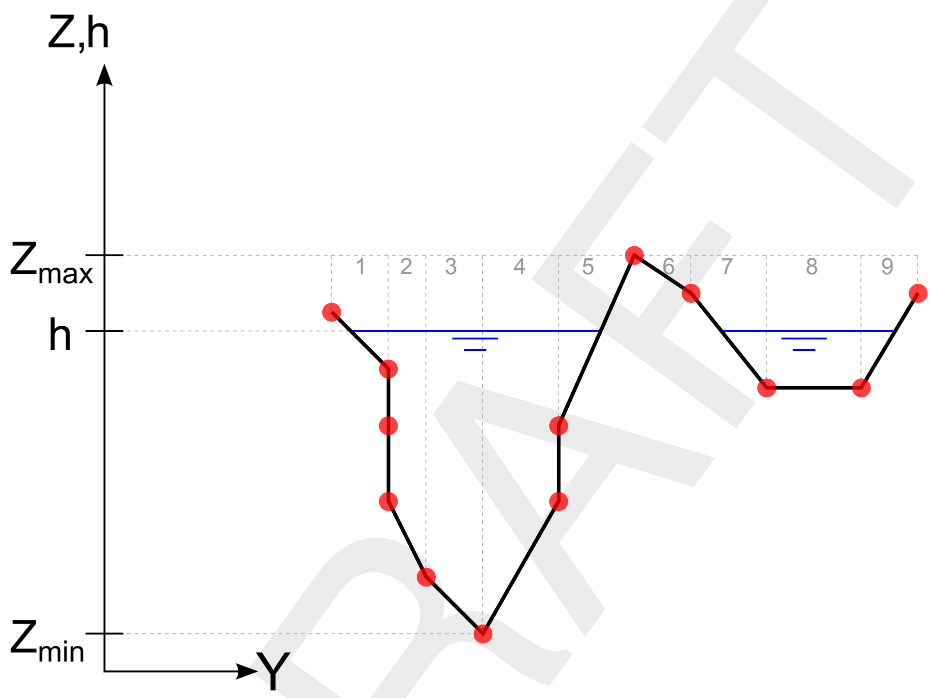


Figure 6.47: *Example of constructed vertical segments in a Y-Z profile*
13 (y_i, z_i) points red dots
9 vertical segments

Note: Conveyance table constructed from $Z_{\min}$ to $Z_{\max}$.

A Y-Z profile is defined by a number (n) of (y_i, z_i) points. For the total number $(nseg)$ of vertical segments yields: $n_{seg} = n - a - 1$, where a = number of succeeding (y_i, z_i) points for which $y_i = y_{i+1}$. So, *please note* that vertical walls in a Y-Z profile do not induce a vertical segment. *Moreover*, the length of vertical walls is not considered in computing the conveyance of a Y-Z profile. Omitting vertical walls in computing the conveyance of a Y-Z profile, means that for a canal having a rectangular profile with a small ratio of width and water depth only, substantial difference in discharge capacity can occur if such canal is modelled as a rectangular cross-section (i.e. lumped conveyance approach) or as a rectangle shaped Y-Z profile (i.e. vertically segmented conveyance approach). The surface level of a Y-Z profile is equal to its highest (y_i, z_i) point. In analogy, the same yields for an Asymmetrical Trapezium cross-section.

The conveyance (K_i) of each vertical segment is computed separately (for formulae, see conveyance per vertical segment further below). The conveyance of each inclined vertical segment is determined by solving an integral from y_i to y_{i+1}. The advantage of using an integral is that irrespective of the number of user-defined intermediate points, that lie on a straight line between (y_0, z_0) to (y_1, z_1), the same conveyance for the cross-sectional part from y_0 to y_1 is computed. The total conveyance (K_{tot}) at a particular water level is the summation of the conveyance of all the vertical segments ($K_{tot} = \sum K_i$).

Conveyance tables, applied in the vertically segmented conveyance approach

A conveyance table gives the conveyance, flow width and flow area of a cross-section as function of water level. Conveyance tables are used for Y-Z profiles and Asymmetrical Trapeziums only. These conveyance tables are made by a pre-processor before the onstart of a hydrodynamic calculation. In constructing the conveyance table of each user-defined cross-section, it is ensured that the difference in conveyance between two succeeding water levels is less than 1 %. Conveyance tables are used for reducing computational efforts, hence increasing SOBEK's computational performance.

Conveyance tables are made from the bed-level up to the highest (y_i, z_i) point, irrespective of the actual location of the highest (y_i, z_i) point within the cross-sectional profile. This implies that no distinction is made between the left and the right embankment height.

Conveyance tables for computational points located in between two user-defined cross-sections are obtained by linear interpolating the conveyance tables, constructed at these two user-defined cross-sections (see also cross-section).

Extrapolation of conveyance tables

Say the water level (h) rises above the highest level ($n_{\max}$) of a conveyance table. In this case the conveyance (K_h) at water level (h) is computed as

$$K_h = a(h - z_{\min})^b, \tag{6.154}$$

where: $z_{\min}$ is the lowest level in the cross-sectional profile; and a, b are constants derived from the conveyances at levels $n_{\max}$ and $n_{\max} - 1$.

Lumped hydraulic output for Y-Z profiles and Asymmetical Trapezium

As a courtesy at subsection branch output, lumped hydraulic output (Q, C_{lumped}, A_{flow}, W_{flow} and R) are provided for Y-Z profiles and Asymmetrical Trapeziums. The discharge (Q) is the discharge through the profile, having a flow area (A_{flow}) and flow width (W_{flow}). The hydraulic radius (R) is computed on basis of A_{flow} and W_{flow} assuming a symmetrical profile as function of water level. The latter assumption is made, since different Y-Z profiles may have different shapes, that might have to be interpolated. The lumped Chézy value (C_{lumped}) is computed as

$$C_{lumped} = \frac{K}{A_{flow}\sqrt{R}}, \tag{6.155}$$

where K is the conveyance.

Conveyance per vertical segment

A vertical segment is defined by two (y_0, z_0) and (y_1, z_1) points for which yields that y_0 is not equal to y_1 (see Figure 6.48).

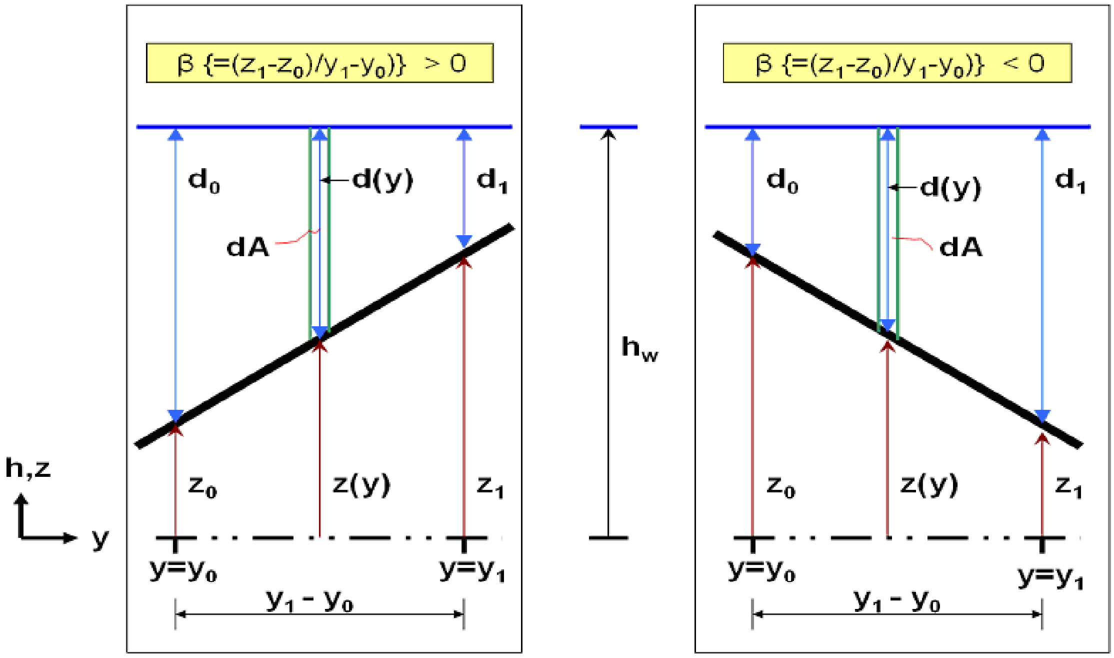


Figure 6.48: *Definition sketch of a vertical segment, considered in computing conveyance for a Y-Z profile and an Asymmetrical Trapezium cross-section*

Hereunder, the conveyance formulae for a vertical segment are given in case of:

- ◇ Chézy roughness,
- ◇ Manning roughness,
- ◇ Strickler (k_n) roughness,
- ◇ Strickler (k_s) roughness,
- ◇ White-Colebrook (k_n) roughness, and
- ◇ Bos&Bijkerk (γ) roughness.

Chézy roughness conveyance for a vertical segment (see Figure 6.48):

$d = \max(d_0, d_1)$

if $d \leq 0$ then

    $k = 0$

else

    if $d < |z_1 - z_0|$ then

        if $|\beta| \leq 0.01$ then

$$k_1 = \frac{2C}{|\beta|(1+\beta^2)^{1/4}} \left(\frac{d}{2}\right)^{5/2}$$

        elseif $|\beta| > 0.01$ then

$$k_2 = \frac{2C}{5|\beta|(1+\beta^2)^{1/4}} d^{5/2}$$

    elseif $d \geq |z_1 - z_0|$ then

        if $|\beta| \leq 0.01$ then

$$k_3 = \frac{C}{(1+\beta^2)^{1/4}} \left(\frac{d_0+d_1}{2}\right)^{3/2} (y_1 - y_0)$$

        elseif $|\beta| > 0.01$ then

$$k_4 = \frac{2C}{5|\beta|(1+\beta^2)^{1/4}} \left| d_0^{5/2} - d_1^{5/2} \right|$$

        endif

    endif

endif

Manning roughness conveyance for a vertical segment (see Figure 6.48):

$d = \max(d_0, d_1)$
if $d \leq 0$ then
$k = 0$
else
if $d < |z_1 - z_0|$ then
if $|\beta| \leq 0.01$ then
$k_1 = \frac{2}{n|\beta|(1+\beta^2)^{1/4}} \left(\frac{d}{2}\right)^{8/3}$
elseif $|\beta| > 0.01$ then
$k_2 = \frac{3}{8n|\beta|(1+\beta^2)^{1/4}} d^{8/3}$
elseif $d \geq |z_1 - z_0|$ then
if $|\beta| \leq 0.01$ then
$k_3 = \frac{1}{n(1+\beta^2)^{1/4}} \left(\frac{d_0+d_1}{2}\right)^{5/3} (y_1 - y_0)$
elseif $|\beta| > 0.01$ then
$k_4 = \frac{3}{8n|\beta|(1+\beta^2)^{1/4}} \left| d_0^{8/3} - d_1^{8/3} \right|$
endif
endif
endif

Strickler (k_n) roughness conveyance for a vertical segment (see Figure 6.48):

$d = \max(d_0, d_1)$
if $d \leq 0$ then
$k = 0$
else
if $d < |z_1 - z_0|$ then
if $|\beta| \leq 0.01$ then
$k_1 = \frac{50k_n^{-1/6}}{|\beta|(1+\beta^2)^{1/4}} \left(\frac{d}{2}\right)^{8/3}$
elseif $|\beta| > 0.01$ then
$k_2 = \frac{75k_n^{-1/6}}{8|\beta|(1+\beta^2)^{1/4}} d^{8/3}$
elseif $d \geq |z_1 - z_0|$ then
if $|\beta| \leq 0.01$ then
$k_3 = \frac{25k_n^{-1/6}}{(1+\beta^2)^{1/4}} \left(\frac{d_0+d_1}{2}\right)^{5/3} (y_1 - y_0)$
elseif $|\beta| > 0.01$ then
$k_4 = \frac{75k_n^{-1/6}}{8|\beta|(1+\beta^2)^{1/4}} \left| d_0^{8/3} - d_1^{8/3} \right|$
endif
endif
endif

Strickler (k_s) roughness conveyance for a vertical segment (see Figure 6.48):

$d = \max(d_0, d_1)$
if $d \leq 0$ then
$k = 0$

else
  if $d < |z_1 - z_0|$ then
    if $|\beta| \leq 0.01$ then
      $k_1 = \frac{2k_s}{|\beta|(1+\beta^2)^{1/4}} \left(\frac{d}{2}\right)^{8/3}$
    elseif $|\beta| > 0.01$ then
      $k_2 = \frac{3k_s}{8|\beta|(1+\beta^2)^{1/4}} d^{8/3}$
  elseif $d \geq |z_1 - z_0|$ then
    if $|\beta| \leq 0.01$ then
      $k_3 = \frac{k_s}{(1+\beta^2)^{1/4}} \left(\frac{d_0+d_1}{2}\right)^{5/3} (y_1 - y_0)$
    elseif $|\beta| > 0.01$ then
      $k_4 = \frac{3k_s}{8|\beta|(1+\beta^2)^{1/4}} \left| d_0^{8/3} - d_1^{8/3} \right|$
    endif
  endif
endif

White-Colebrook (k_n) roughness conveyance for a vertical segment (see Figure 6.48):

$d = \max(d_0, d_1)$
$c_1 = (1 + \beta^2)^{1/4} \ln(10)$
$c_2 = \frac{k_n}{12}$
if $d \leq 0$ then
  $k = 0$
else
  if $d < |z_1 - z_0|$ then
    if $|\beta| \leq 0.01$ then
      if $\frac{6d}{k_n} \leq 1.0005$ then
        $k_1 = \frac{36}{|\beta|(1+\beta^2)^{1/4}} 0.00022 \left(\frac{d}{2}\right)^{5/2}$
      else
        $k_1 = \frac{36}{|\beta|(1+\beta^2)^{1/4}} \, {}^{10}\log\left\{\frac{6d}{k_n}\right\} \left(\frac{d}{2}\right)^{5/2}$
      endif
    elseif $|\beta| > 0.01$ then
      if $\frac{d}{c_2} \leq 1.495$ then
        $k_2 = \frac{36}{5|\beta|c_1} d^{5/2} 0.00213$
      else
        $k_2 = \frac{36}{5|\beta|c_1} d^{5/2} \left(\ln\left\{\frac{d}{c_2}\right\} - \frac{2}{5}\right)$
      endif
  elseif $d \geq |z_1 - z_0|$ then
    if $|\beta| \leq 0.01$ then
      if $\frac{6(d_0+d_1)}{k_n} \leq 1.0005$ then
        $k_3 = \frac{18}{(1+\beta^2)^{1/4}} 0.00022 (y_1 - y_0) \left(\frac{d_0+d_1}{2}\right)^{3/2}$
      else
        $k_3 = \frac{18}{(1+\beta^2)^{1/4}} \, {}^{10}\log\left\{\frac{6(d_0+d_1)}{k_n}\right\} (y_1 - y_0) \left(\frac{d_0+d_1}{2}\right)^{3/2}$
      endif
    elseif $|\beta| > 0.01$ then
      if $\frac{d_0}{c_2} \leq 1.495$ and $\frac{d_1}{c_2} \leq 1.495$ then
        $k_4 = \frac{36}{5|\beta|c_1} \left| d_0^{5/2} 0.00213 - d_1^{5/2} 0.00213 \right|$

elseif $\frac{d_0}{c_2} \leq 1.495$ and $\frac{d_1}{c_2} > 1.495$ then

$k_4 = \frac{36}{5|\beta|c_1} \left| d_0^{5/2} 0.00213 - d_1^{5/2} \left(\ln \left\{ \frac{d_1}{c_2} \right\} - \frac{2}{5} \right) \right|$

elseif $\frac{d_0}{c_2} > 1.495$ and $\frac{d_1}{c_2} \leq 1.495$ then

$k_4 = \frac{36}{5|\beta|c_1} \left| d_0^{5/2} \left(\ln \left\{ \frac{d_0}{c_2} \right\} - \frac{2}{5} \right) - d_1^{5/2} 0.00213 \right|$

else

$k_4 = \frac{36}{5|\beta|c_1} \left| d_0^{5/2} \left(\ln \left\{ \frac{d_0}{c_2} \right\} - \frac{2}{5} \right) - d_1^{5/2} \left(\ln \left\{ \frac{d_1}{c_2} \right\} - \frac{2}{5} \right) \right|$

endif

endif

endif

endif

endif

Bos&Bijkerk (γ) roughness conveyance for a vertical segment (see Figure 6.48):

$d = \max(d_0, d_1)$

if $d \leq 0$ then

$k = 0$

else

if $d < |z_1 - z_0|$ then

if $|\beta| \leq 0.01$ then

$k_1 = \frac{2\gamma}{|\beta|(1+\beta^2)^{1/4}} \left(\frac{d}{2} \right)^3$

elseif $|\beta| > 0.01$ then

$k_2 = \frac{\gamma}{3|\beta|(1+\beta^2)^{1/4}} d^3$

elseif $d \geq |z_1 - z_0|$ then

if $|\beta| \leq 0.01$ then

$k_3 = \frac{\gamma}{(1+\beta^2)^{1/4}} \left(\frac{d_0+d_1}{2} \right)^2 (y_1 - y_0)$

elseif $|\beta| > 0.01$ then

$k_4 = \frac{\gamma}{3|\beta|(1+\beta^2)^{1/4}} \left| d_0^3 - d_1^3 \right|$

endif

endif

endif

6.1.21 Delft-scheme

The computation of the water levels and discharges in the SOBEK-flow-network is performed with the Delft-scheme. This scheme solves the De Saint-Venant equations (continuity and momentum equation) by means of a staggered grid. In this staggered grid the water levels are defined at the connection nodes and ζ-calculation points, while the discharges are defined at the intermediate branches or branch segments.

In general, numerical approximations must satisfy the following requirements:

◇ Robust, i.e. effective or capable of dealing with a wide range of practical problems, without producing numerical instabilities,
◇ Efficient, i.e. efficient use of computational resources such as processor time,
◇ Accurate, i.e. sufficient accuracy given the modelling purpose.

For the Delft-scheme robustness has been the most important design aspect. By the range of practical problems to be dealt with the following problems are included:

◇ Drying and flooding

◇ Super-critical flow including hydraulic jump.

The used procedure guarantees a solution. In certain flow conditions the time step is reduced temporarily by a time step estimation procedure to avoid numerical instability.

The Delft-scheme uses a so-called minimum degree algorithm with an iterative simulation technique. This is highly efficient in case of large networks and long time series.

The Delft-scheme is designed to produce a closed water balance, which makes it very suitable for water quality computations.

6.1.22 Drying/flooding

When a water level upstream of a branch segment is lower than 5 mm above the bed level of the ζ-calculation point at this upstream side, the branch segment is assumed to be dry; the discharge through this branch segment is said to be zero. When a branch segment is dry and upstream or downstream of the segment the water level rises higher than 10 mm above the bed level of the ζ-calculation point at that side, the branch segment starts flooding again.

The same deadband is used for the check on drying and flooding of structures. In case of a weir and orifice, the upstream water level is compared to the crest level. In case of culverts, siphons and inverted siphons, the water level is compared to the maximum of the bed level and the bottom of the structure at the upstream side.

6.1.23 Free board

Freeboard is the difference between surface level and water level at some location in the system. It is defined from surface level to water level, so the freeboard is positive when the water level is lower than the surface level. When the freeboard is negative, there is a flood at that location.

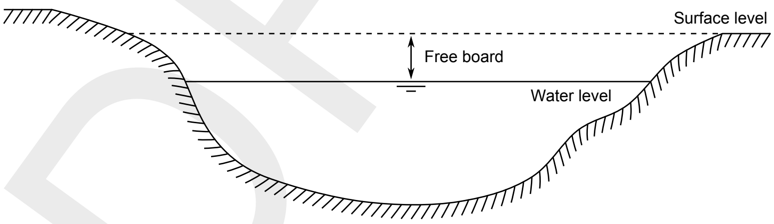


Figure 6.49: *Free board*

6.1.24 Ground layer

Except for a Y-Z profile, cross sections can have a fixed horizontal ground layer (or sediment layer). The ground layer level (top level of the ground layer) equals the bottom level (or invert level) plus the ground layer thickness. The part of the cross-section located beneath the ground layer level is omitted in the hydraulic computations. Hence, a ground layer reduces both the wetted area and the wetted perimeter for a given water level. In Figure 6.50 a circular cross section is shown with a ground layer with thickness D_{gr}

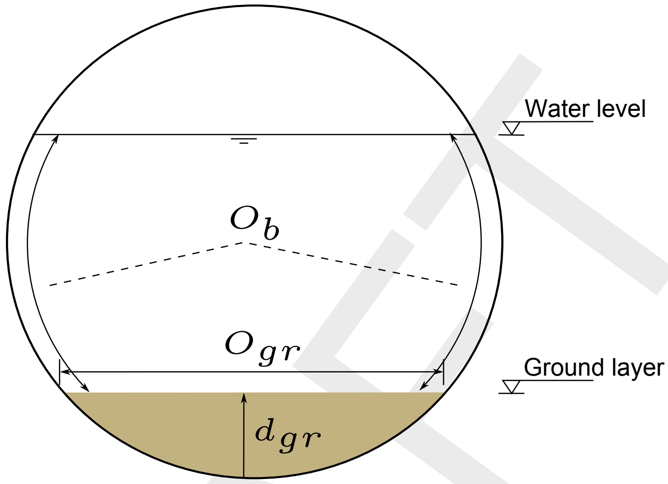


Figure 6.50: *Ground layer in circular cross section*

For a ground layer a friction value can be specified that differs from the bed friction value. The ground layer friction value is applied for the cross-sectional width at the ground layer level only. For the remaining wetted perimeter of the cross-section, the bed friction value is applied. The same friction formulation (i.e. Chézy, Manning etc.) is to be applied for both the ground layer and the remaining wetted perimeter (i.e. the bed) of the cross-sectional profile. The combined friction that is used is computed according to:

$$k_s = \sqrt{\frac{O_b k_{s_b}^2 + O_{gr} k_{s_{gr}}^2}{O_b + O_{gr}}} \tag{6.156}$$

k_s	Combined Strickler friction value
k_{s_b}	Strickler bed friction value
$k_{s_{gr}}$	Strickler friction value of the ground layer
O_b	Wetted perimeter of the bed
O_{gr}	Wetted perimeter of the ground layer

6.1.25 Measurement station

In channel flow a measurement station is placed on a branch. From a measurement station the actual (calculated) value of a water level or a discharge can be retrieved. This value can then be used in an algorithm of a controller.

The water level that a measurement station retrieves is the water level at the nearest calculation point. The retrieved discharge is the discharge at the branch segment where the measurement station is located.

In sewer flow the measurement station can be placed on a connection node or on a branch Water levels are taken at the nodes, discharges at the branches.

6.1.26 Network

The schematic basis of each model is the network of branches. These branches are connected to each other at connection nodes.

One can say that the network of branches and nodes is the modelled representation of the real geographic data of the modelled area.

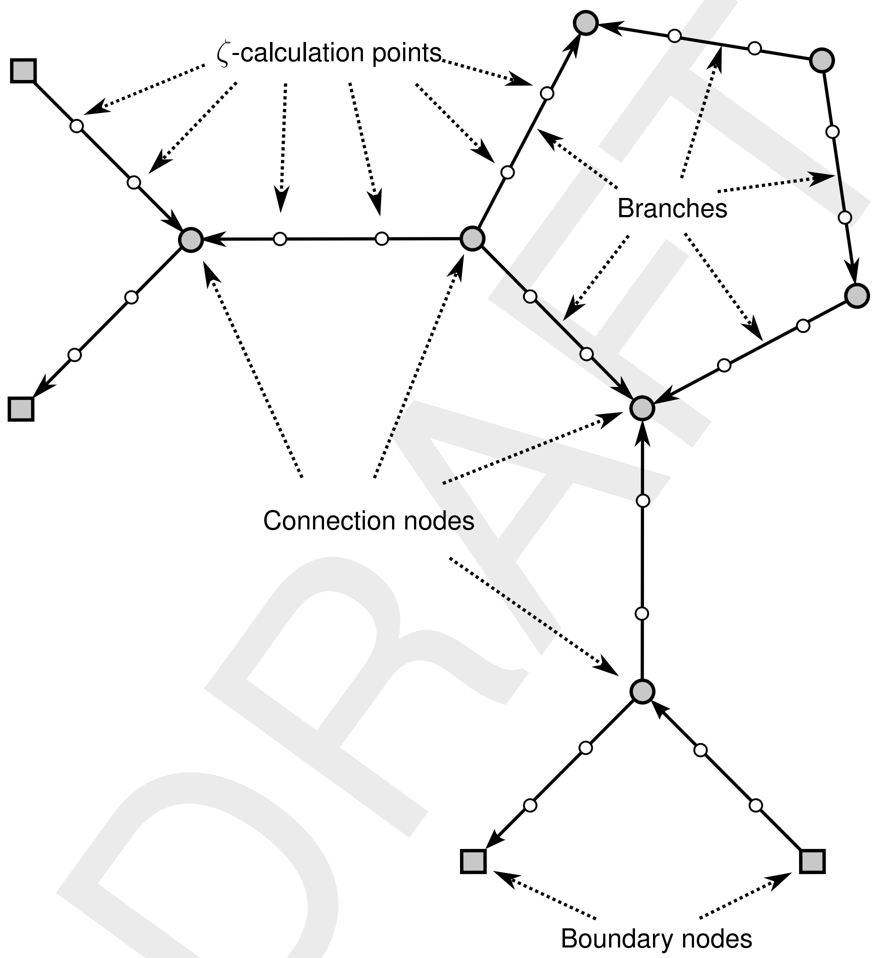


Figure 6.51: *Network*

In the Delft-scheme used by the D-Flow 1D module, the network is split up into a staggered grid. The water levels are defined at the connection nodes and the ζ-calculation points of this grid, while the discharges are defined at the branch segments.

6.1.26.1 Branch

A branch in a SOBEK-Flow-model is one of the basic elements in the schematisation. A branch connects two nodes, and has the following attributes:

- ◇ Begin node
- ◇ End node
- ◇ Branch length

◇ A geographical orientation (used only in case wind friction effect is included in your model).

The direction of the branch (positive x-direction) is from begin node to end node.

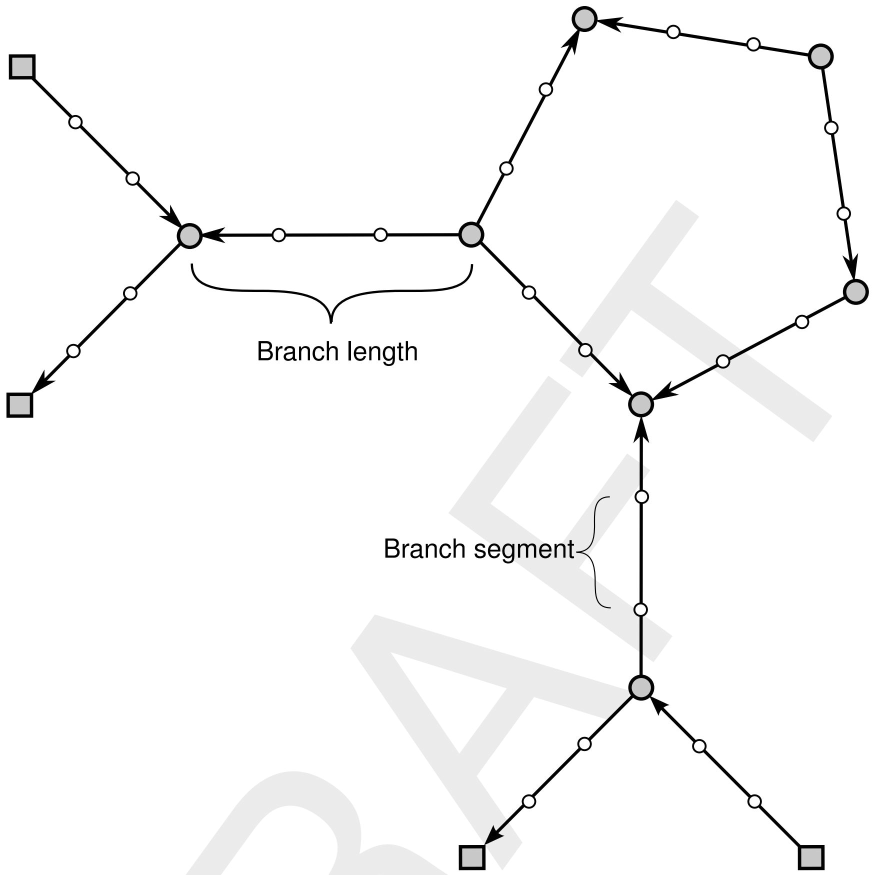


Figure 6.52: *Branch length in model network*

When using channel flow, ζ-calculation points can be specified on a branch. If this is done, the branch is split up into branch segments. The numerical simulation will be carried out based on these ζ-calculation points and branch segments in a staggered grid. The discharges are calculated at the branch or branch segments Water levels are calculated at the connection nodes and ζ-calculation points.

In case of sewer flow, each branch has a maximum of two ζ-calculation points at the beginning and end of that branch. The branch can be interpreted as one branch segment

6.1.26.2 Branch length

In sewer systems the length of a branch is the length from the begin connection node to the end connection node. This length is referred to as the co-ordinate length. In open channel systems, a branch can be made longer than this minimum co-ordinate length, with the use of vector points on the branch. This length is referred to as vector length. With this vector length the actual length of a meandering river or channel can be modelled. If the vector length is defined on a branch, this length is used in the computation of the water flow equations. If this length is not defined, the co-ordinate length is used.

Note: The vector length is always equal to or greater than the co-ordinate length

6.1.26.3 Branch segment

A SOBEK-Flow-network consists of branches that are connected in connection nodes. These branches can be divided by ζ-calculation points into segments; the so-called branch segments. The discharges that result from the water flow equations are defined in these branch segments

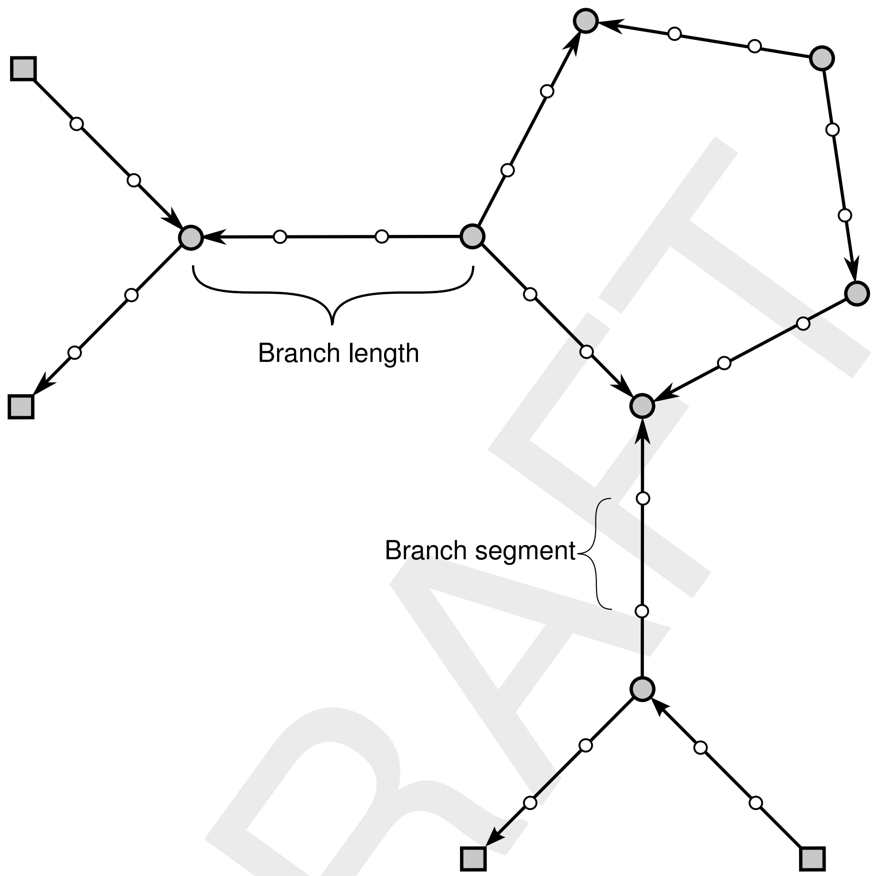


Figure 6.53: *Branch segment*

In sewer networks no ζ-calculation points can be used. So each branch consists of one branch segment.

6.1.26.4 Connection node

A connection node in a SOBEK-Flow-model is a location where branches can be joined to other branches. Water levels are defined at the connection nodes and the ζ-calculation points.

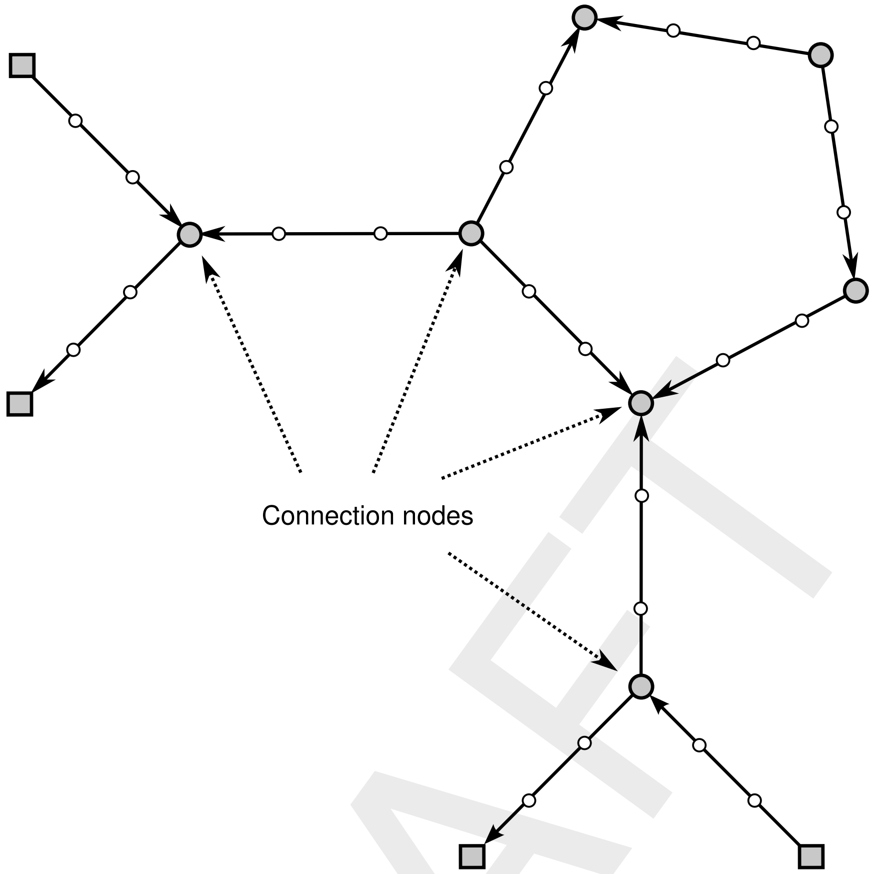


Figure 6.54: *Connection nodes*

Note: A node located at the bounds of the model can be a connection node (with storage) or a boundary node (no storage).

6.1.27 Robustness

The Delft-scheme is developed with robustness as its most important design aspect. This scheme can deal with phenomena such as drying/flooding, supercritical flow and flow transitions (hydraulic jumps) and guarantees a solution for every time step, though the actual computational time step can be reduced internally under certain flow conditions (large discharges in combination with little storage).

The computational time step reduction is done by a time step estimation procedure before a new time step or at the end of a time step when negative depths are computed. In the latter case the step is repeated with a time step reduced by a factor 2 until a correct solution is achieved.

6.1.28 Simulation output parameters at branch segments

For more background information on time step reductions, see section Time step reductions during the simulation. The simulation output parameters available at each branch segment (see option "Simulation Info at the Branch Segments" in the file <flowanal.his> or in Results in Charts and Results in Maps) are:

- **Time step estimation**: This refers to the number of times that the time step is reduced, since the allowable Courant time step of this branch segment is smaller than the user-defined time step.
- **No iteration up and No iteration down**: Parameters "No iteration up" and "No iteration down", respectively refer to the number of times that convergence in the Newton iteration process is not satisfied at the water level point with the lowest x-coordinate ("No iteration up") and the water level point with highest x-coordinate ("No iteration down") of the branch segment.
- **Negative depth up and Negative depth down**: Parameters "Negative depth up" and "Negative depth down", respectively refer to the number of times that a time step reduction is needed for avoiding negative depths at the upstream water level point (lowest x-coordinate) and the downstream water level point (highest x-coordinate) of the branch segment.
- **Total**: The number of times that the allowable Courant time step is smaller than the user-defined time step and the number of times that the time step is reduced for either satisfying the convergence condition or for avoiding negative depths.

6.1.29 Time step reductions during the simulation

The user can define a time step in the Settings Task block. During the computation this user-defined time step may be reduced for stability reasons (e.g. to prevent the simulation from crashing). D-Flow 1D performs the following checks:

- At the start ($t = t^k$) of each computational time-step, the allowable Courant time-step is determined. This is done as follows:
  - For each 1D branch segment (i) its 1D Courant time step is computed as:

$$\Delta t_i = \sigma \frac{Vol_{i,tot}}{Q_{i,t}} \qquad \text{1D Courant} \tag{6.157}$$

    where:

    σ the user-defined value for "maximum flow (1D) and velocity (2D) Courant number" in Settings
    $Vol_{i,tot}$ the total volume of water at the upwind (or upstream) water level point
    Q_{i,t^k} the discharge at $t = t^k$

    Then the 1D branch segment having the smallest 1D Courant time step (e.g. the allowable 1D Courant time step) is determined. The applicable 1D time step at $t = t^k$ is the minimum of the user-defined time step and the allowable 1D Courant time step. The branch segment with the smallest 1D Courant time step is logged in case its 1D Courant time step is smaller than the user-defined time step.
  - The 2D Courant time step for all 2D links in U-direction and for all 2D links in V-direction are respectively computed as:

$$\Delta t = \sigma \frac{\Delta x_m}{U_{m,t^k}} \qquad \text{2D Courant, U-direction} \tag{6.158}$$

    and

$$\Delta t = \sigma \frac{\Delta y_n}{V_{n,t^k}} \qquad \text{2D Courant, V-direction} \tag{6.159}$$

where:

σ	the user-defined value for "maximum flow (1D) and velocity (2D) Courant number" in Settings
Δx_m	the grid spacing in x-direction
Δy_n	the grid spacing in y-direction
U_{m,t^k}	the depth-averaged flow velocity in x-direction at $t = t^k$
V_{n,t^k}	the depth-averaged flow velocity in y-direction at $t = t^k$

The allowable 2D Courant time step is the minimum of the 2D Courant time steps of all 2D U-links and 2D V-links. The applicable 2D time step is the minimum of the user-defined time step and the allowable 2D Courant time step.

- In case of a combined 1D-2D hydraulic computation, the applied time step (Δt) is the minimum of the applicable 1D time step and the applicable 2D time step.

◇ In each computational time step (from $t = t^k$ to $t^{k+1} = t^k + \Delta t$), it is checked that for each water level point yields that the lateral inflow volume is not larger than its volume of water at $t = t^k$. If this condition is not satisfied, the user defined time step is reduced to such extend that this condition is met. The reduced time step, however, is not made smaller than the user-defined "minimum time step" in Settings.

◇ Due to various circumstances (discontinuities in model schematization, unrealistic boundary conditions, etc.), the above determined actual time step (Δt) may still be too large for finding a valid solution at $t^{k+1} = t^k + \Delta t$. Two conditions are to be fulfilled in finding a valid solution:

- The first condition is that the Newton iteration process converges in terms of water levels and volumes (both computed at water level points). In other words differences in water levels and volumes between the last and previous Newton solution are to be within the user-defined values for "epsilon for water depth" and "epsilon for volume". In Settings, the user can define the "maximum number of Newton iterations". If the convergence condition is not satisfied within the maximum number of Newton iterations, the time step is reduced by a factor 2. If this does not satisfy the convergence condition, the time step is again reduced by a factor 2. The simulation is terminated if a time step smaller than the user-defined "minimum time step" (see Settings) is needed for satisfying the convergence condition. All water level points, that after the maximum number of Newton iterations are conducted, still having a water level convergence error are logged during a computational time step.
- The second condition is that no negative depths (e.g. water level is below bed level) may occur. Negative depths at a particular water level point may occur if the outflow during a time step is larger than the volume stored at $t = t^k$. If negative depths occur, the time step is reduced up to a time step for which yields that the outflow during the reduced time step is smaller than the volume stored at $t = t^k$ and hence no negative depth occurs. The simulation is terminated if a reduced time step smaller than the user-defined "minimum time step" (see Settings) is needed to avoid negative depths. All water level points for which the time step needs to be reduced in order to avoid negative depths are logged during a computational time step.

Note: If a model run requires a lot of computational effort (e.g. an unexpected long simulation time is needed), often a large number of time step reductions during simulation is performed. These time step reductions are needed to find a valid solution and/or to prevent the model from crashing. Defining a smaller calculation time step in Settings may result in shorter simulation times. The reason for this, the smaller user-defined time step allows more moderated intermediate solutions to be found. More moderate solutions reduce the number of circumstances in which the model encounters large discontinuities, which require a considerable amount of time step reductions before finding a valid solution. The final actual time step can be much smaller than the user-defined time step. In other words: A smaller user-defined time step can result in significant less total number of time step reductions leading to a decrease of total computational time. This is because the total required simulation time is directly related to the total number of computational time steps.

6.1.30 Slope

In the D-Flow 1D module slope can refer to the slope of the water level or the bed slope. If the word "slope" is used without further indication, then it pertains to the slope of the water level in the model. The symbol used is i:

$$i = \frac{\partial \zeta}{\partial x} \tag{6.160}$$

where

ζ	Water level $[m]$
x	Distance $[m]$

For bed slope, the symbol used is i_b:

Bed slope:

$$i_b = \frac{\partial z_b}{\partial x} \tag{6.161}$$

with z_b as the bed level.

6.1.31 Stationary computation

A stationary computation is performed in the D-Flow 1D module by running a non-stationary simulation until a steady state is branched. During this simulation, all time depending parameters, such as lateral and boundary conditions, are set to their initial value and kept constant.

In some cases a steady state cannot be branched, for example, when a pump is switching on and off. In that case the computation stops after 1 000 simulation steps (not time steps, see Time step estimation) and the maximum difference of water level compared to the last time step in the network is given. If this value is low enough ($<$1 mm for example) the solution can be considered as steady.

The results of a stationary computation can be used as an initial condition for a non-stationary simulation.

6.1.32 Summer dike

A flood plain might be separated from a river by means of a small dike, having a crest level lower than the crest level of the main dike (see Figure 6.55). In the Netherlands large floods usually occur in winter. The main dike and the small dike are, therefore, respectively referred to as the “winter dike” and the “summer dike”.

The presence of a summer dike implies that the cross-sectional profile as measured in situ (see shadowed line in Figure 6.55) does not increase monotonously with rising water levels. When the water level becomes slightly higher than the crest level of the summer dike, first the area behind the summer dike is to be filled, before water levels can become any higher. This implies a local attenuation of the flood wave at water levels just above the crest level of

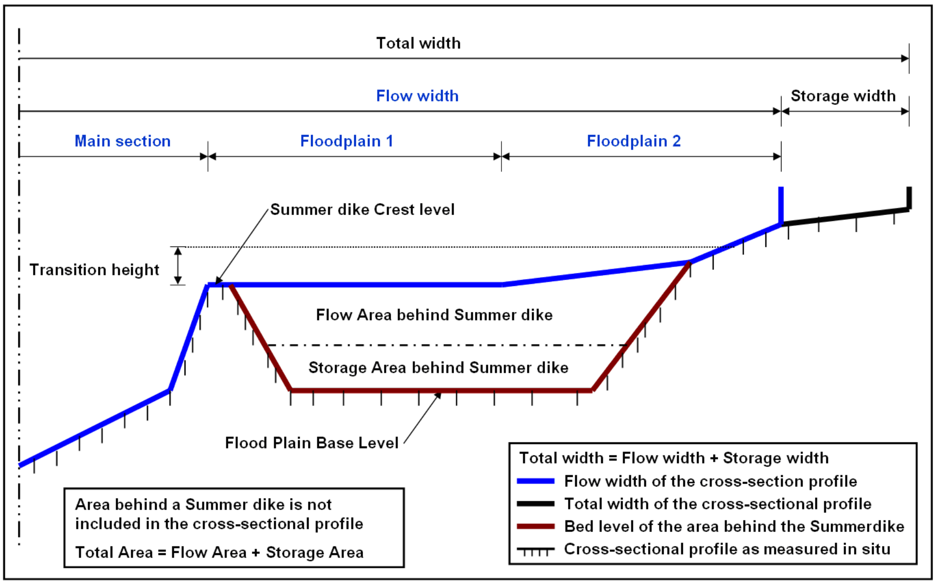


Figure 6.55: *Summer dike option, available in a river profile*

the summer dike. This hydraulic phenomenon can be modelled using the summer dike option available at a river profile.

In defining a summer dike in a river profile a distinction is to be made between:

◇ **The cross-sectional profile**
The cross-sectional profile comprises of a "flow area" and a "storage area", which in Figure 6.55 are respectively located above the blue lines and the black lines. The summation of flow area and storage area is called the "total area". Both flow area and total area are defined by its width as function of elevation. At each elevation in the cross-sectional profile yields that storage width equals total width minus flow width. Hence, no storage is defined if total width equals flow width. The flow area of the cross-sectional profile can be divided in three separate flow sections, respectively called the (i) main section, (ii) floodplain 1 and (iii) floodplain 2 . Each such flow section can have a different roughness value and a different roughness formulation.

◇ **The area behind the summer-dike**
In analogy with the cross-sectional profile, the "total area behind a summer dike" is divided in a "flow area" and a "storage area". The "Flood Plain Base Level" coincides with the lowest part of the area behind the summer dike (see Figure 6.55).

Remark:

◇ Both the cross-sectional profile and the area behind the summer dike should be mutually consistent with the cross-sectional profile as measured in situ (see shadowed line in Figure 6.55). Hence, both the flow area and the storage area behind the summer dike should not be included in the cross-sectional profile.

The so-called "transition height" is used for avoiding numerical oscillations. More precisely, the transition height ensures that the flow area and storage area behind the summer dike are gradually taken into computation. Transition height for summer dikes is a model-wide parameter, that is defined in the Settings Taskblock (*Edit* button next to 1DFLOW (River); Advanced settings Tab).

Lets consider the following summer dike properties:

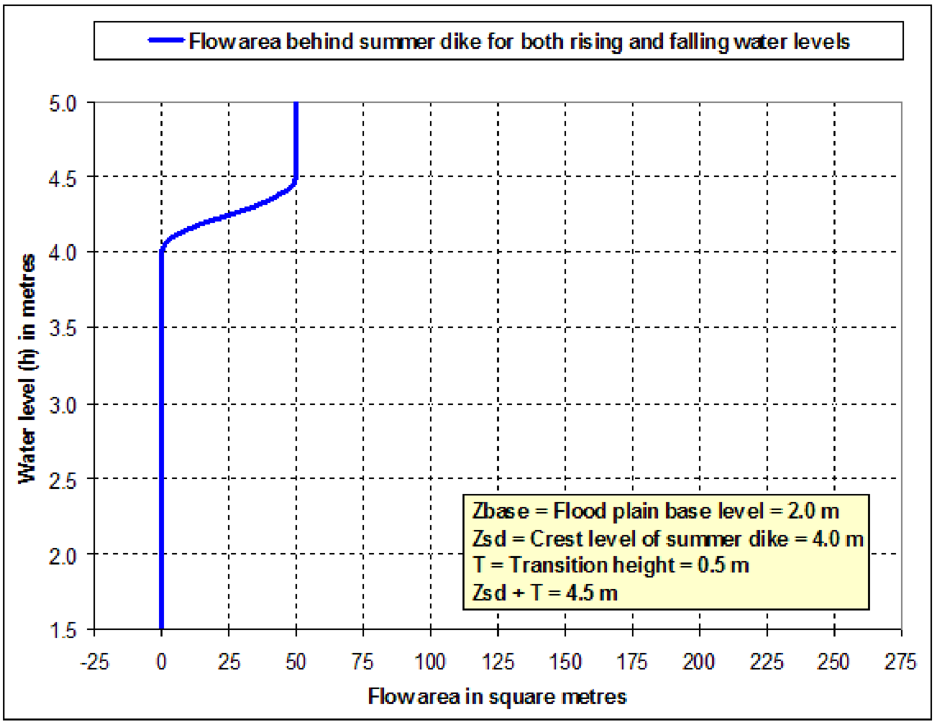


Figure 6.56: *Flow area behind the summer dike as function of local water levels.*

- flood plain base level (z_{base}) of 2 m,
- summer dike crest level (z_{sd}) of 4 m,
- transition height (T) of 0.5 m,
- storage area behind the summer dike of 200 m^2,
- flow area behind the summer dike (ExtraFlowArea) of 50 m^2, and
- total area behind the summer dike (ExtraTotalArea) of 250 m^2.

In explaining how a summer dike is taken into computation, we make a distinction between:

- The actual value of the (extra) flow area behind the summer dike,
- The flow velocity applied to the (extra) flow area behind the summer dike, and
- The actual value of the (extra) total area behind the summer dike.

The actual (extra) flow area is a unique function of the local water level (h) in the cross-sectional profile (see Figure 6.56). More precisely:

- If $\zeta < z_{\text{sd}}$; Flow area behind the summer dike is zero.
- If $z_{\text{sd}} \leq \zeta \leq z_{\text{sd}} + T$; Flow area behind the summer dike varies as a quadratic function of water level (ζ) between zero and ExtraFlowArea (= 50 m^2).
- If $\zeta > z_{\text{sd}} + T$; Flow area behind the summer dike is equal to ExtraFlowArea.

The flow velocity applied to the (extra) flow area behind the summer dike is the flow velocity in one of the three available flow sections. Hence, either the flow velocity in the main section, in floodplain 1 or in floodplain 2. The actual flow velocity applied to the (extra) flow area behind the summer dike is determined as follows:

1 Default the flow velocity in floodplain 2 is used.
2 However, if the wetted area of floodplain 2 becomes less than 0.4 m^2, the flow velocity in floodplain 1 is used,

3 Furthermore, if the wetted area of floodplain 1 becomes less than 0.4 m^2, the flow velocity of the main section is used.

Remarks:

- ◇ Please contact Deltares if you like to use a different threshold value (i.e. not 0.4 m^2) for discerning which flow velocity is to be applied to the (extra) flow area behind the summer dike.
- ◇ The flow velocity applied to the (extra) flow area behind the summer dike is computed as follows

$$U_i = C_i\sqrt{R_i S} \qquad i \in \{\text{main section}, \text{floodplain 1}, \text{floodplain 2}\} \tag{6.162}$$

where:

U_i	average flow velocity
C_i	Chézy coefficient of the flow section
R_i	$(A_i + A_{\text{sd}})/P_i$: hydraulic radius of the flow section
A_i	wetted area of the flow section
A_{sd}	extra flow area behind the summer dike
P_i	wetted perimeter of the flow section
S	water level slope

- ◇ The discharge including the (extra) flow area behind the summer dike is computed as follows

$$Q_i = U_i(A_i + A_{\text{sd}}) \tag{6.163}$$

Figure 6.57 depicts the actual (extra) total area behind the summer dike as function of the local water level (ζ) in the cross-sectional profile. The (extra) total area behind the summer dike is added to the total area of the cross-sectional profile. There is a hysteresis in the extra total area for water levels varying from flood plain base level ($z_{\text{base}} = 2.0\ m$) to crest level of summer dike plus transition height ($z_{\text{sd}} + T = 4.5\ m$). We call line ABC the rising limp and line CA the falling limp of the hysteresis. The so-called “hysteresis flag” determines which limp of the hysteris is to be followed. In case the hysteresis `flag=1` the rising limp of the hysteresis is followed, else the falling limp is followed. At the onstart of a computation yields that the hysteresis `flag=1`. The actual applied extra total area behind the summer dike is computed as follows:

1 If $\zeta < z_{\text{base}}$; Hysteresis flag is set to 1. Total area behind the summer dike is zero.

2 If $z_{\text{base}} \leq \zeta \leq z_{\text{sd}} + T$ and hysteresis `flag=1`;

- ◇ For $z_{\text{base}} \leq \zeta < z_{\text{sd}}$; Total area behind the summer dike is zero.
- ◇ For $z_{\text{sd}} \leq \zeta \leq z_{\text{sd}} + T$; Total area behind the summer dike varies as a quadratic function of water level (ζ) between zero and ExtraTotalArea (= 250 m^2)

3 If $z_{\text{base}} \leq \zeta \leq z_{\text{sd}} + T$ and hysteresis `flag=0`; Total area behind the summer dike varies as a quadratic function of water level (ζ) between zero and ExtraTotalArea.

4 If $\zeta > z_{\text{sd}} + T$; Hysteresis flag is set to 0. Total area behind the summer dike is equal to ExtraTotalArea.

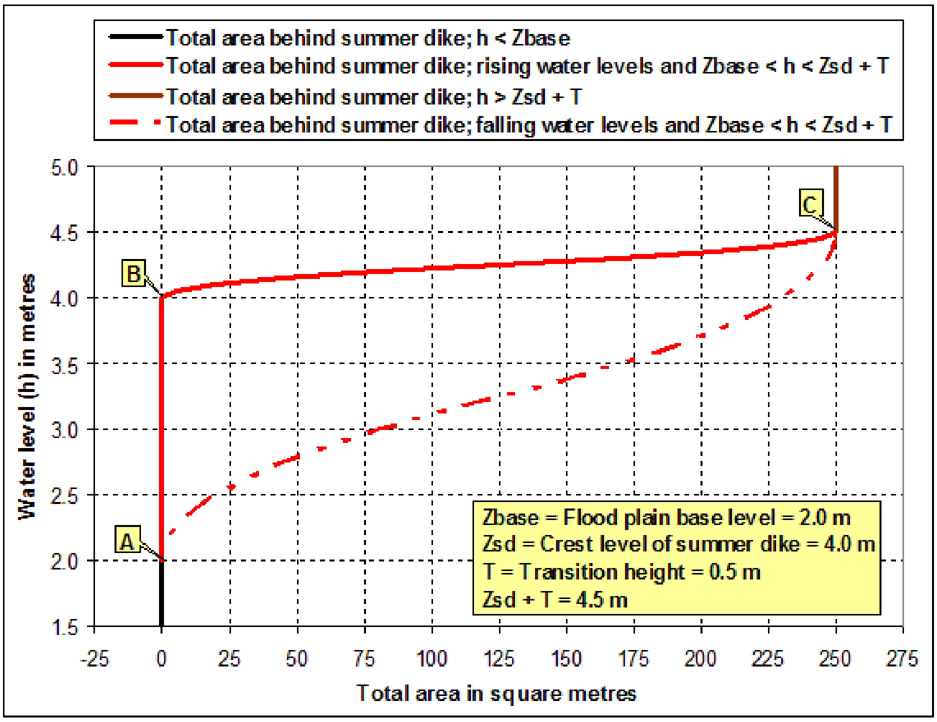


***Figure 6.57:** Total area behind the summer dike as function of local water levels.*

6.1.33 Super-critical flow

The D-Flow 1D module can deal with super-critical flow. The flow in a branch or branch segment is super-critical if the Froude number [$-$] is higher than 1:

$$\text{Froude} = \frac{u}{\sqrt{gA_f/W_f}} \tag{6.164}$$

u	Velocity [m/s]
g	Acceleration due to gravity [m/s^2] (≈ 9.81)
A_f	Wetted area [m^2]
W_f	Flow width [m]

Remarks:
- ◇ When the Froude number is less than 1, the flow is sub-critical.
- ◇ Transitions from super-critical to sub-critical flow causes an hydraulic jump.

6.1.34 Surface level

The surface level is the level of the soil surface i.e. the embankment level of the cross section. When the water level is higher than this level there is a flood. Normally water is stored above this level on a certain storage area. When the water level drops, this storage is emptying back into the branch or node.

6.2 Transport equation

The transport of salt, temperature and sediment in estuaries, tidal rivers and alluvial rivers can be considered as transport of conservative substance in water. These transports are described by an advection-diffusion equation, which is called the transport equation.

Next to the additional transport equation, density differences have to be accounted for in the momentum equation of the water flow module. The water flow module is therefore coupled with the salt intrusion module and temperature by the density and the flow field. The concentrations are denoted by C.

The transport equation is described by an advection-diffusion equation including source term and read:

$$\frac{\partial A_T C}{\partial t} + \frac{\partial}{\partial x}\left(Q\,C - A_F D \frac{\partial\,C}{\partial x}\right) = S_s \tag{6.165}$$

in which

C	concentration of salt or chloride, averaged over the total cross-sectional area [kg/m^3]
D	dispersion coefficient [m^2/s]
S_s	source term [$kg/(ms)$]
Q	discharge (water) [m^3/s]
A_T	total cross-sectional area [m^2]
A_F	flow area [m^2]

6.2.1 Temperature: heat flux model

6.2.1.1 General

In D-Flow 1D the heat exchange at the free surface is modelled by taking into account the separate effects of solar (short wave) and atmospheric (long wave) radiation, and heat loss due to back radiation, evaporation and convection. In literature there is a great variety of empirical formulations to calculate these heat fluxes across the sea surface. Most formulations differ in the dependency of the exchange on the meteorological parameters such as wind speed, cloudiness and humidity. Some formulations were calibrated for coastal seas others for lakes.

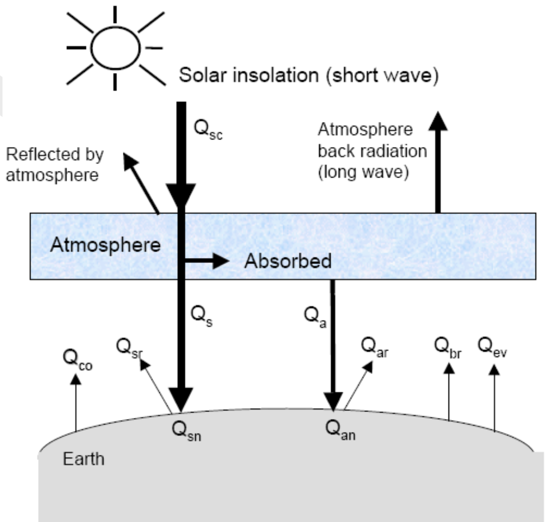


Figure 6.58: *Overview of the heat exchange mechanisms at the surface*

Legend for Figure 6.58:

Q_{sc}	radiation (flux) for clear sky condition in [J/m^{2}s]
Q_{co}	heat loss due to convection (sensible) in [J/m^{2}s]
Q_{sr}	reflected solar radiation in [J/m^{2}s]
Q_s	solar radiation (short wave radiation) in [J/m^{2}s]
Q_{sn}	net incident solar radiation (short wave), $= Q_s - Q_{sr}$
Q_a	atmospheric radiation (long wave radiation) in [J/m^{2}s]
Q_{an}	net incident atmospheric radiation (long wave)
Q_{ar}	reflected atmospheric radiation in [J/m^{2}s]
Q_{br}	back radiation (long wave radiation) in [J/m^{2}s]
Q_{ev}	heat loss due to evaporation (latent) in [J/m^{2}s]

In D-Flow 1D the heat exchange at the free surface is modelled by taking into account the separate effects of solar (short wave) and atmospheric (long wave) radiation, and heat loss due to background radiation, evaporation and convection. Two heat flux models have been implemented:

1 *Excess temperature model*
The heat exchange flux at the air-water interface is computed. Only the background temperature is required.
2 *Composite temperature model*
The fraction of the sky covered by clouds is prescribed (in %). The effective back radiation and the heat losses due to evaporation and convection are computed by the model. Additionally, when air and water densities and/or temperatures are such that free convection occurs, free convection of latent and sensible heat is computed by the model. This model formulation typically applies for large water bodies.

For the physical background of the heat exchange at the air-water interface and the definitions, we refer to Octavio *et al.* (1977) for the excess model and Lane (1989) for the composite model.

6.2.1.2 Heat balance

The change in temperature T_s (in °C) is given by:

$$\frac{\partial T_s}{\partial t} = \frac{Q_{tot}}{\rho_w c_p \Delta z_s} \tag{6.166}$$

with:

Q_{tot}	total heat flux through the air-water surface (in J/(m^{2}s))
ρ_w	specific density of water (in kg/m^3)
c_p	specific heat capacity of sea water (= 3930 J/(kg K))
Δz_s	water depth

In D-Flow 1D, the heat exchange at the bed is assumed to be zero. This may lead to overprediction of the water temperature in shallow areas.

Remarks:

- ◇ The temperature T is by default expressed in °C. However, in some formulas the absolute temperature $\bar{T}$ in K is used. They are related by:

$$\bar{T} = T + 273.15. \tag{6.167}$$

- ◇ In Equation (6.166) the total incoming heat flux is assumed to be absorbed in the water column. This may result in an unrealistically high surface temperature when the top

layer is thin. This can be prevented by absorbing the incoming radiation as a function of depth. Currently, this is only implemented in heat flux models 4 and 5.
◇ When using the composite heat flux model, the free convection of latent and sensible heat is also determined.

6.2.1.3 Excess temperature model

The excess temperature model computes the heat fluxes through the water surface in such a way that the temperature of the surface layer relaxates to the natural background temperature specified by you. The heat transfer coefficient mainly depends on the water temperature and the wind speed.

In the excess temperature model the solar radiation does not play an explicit role, but is part of the background temperature.

$$Q_{tot} = -\lambda \left(T_s - T_a \right), \tag{6.168}$$

with:

λ Heat exchange coefficient, is defined by:

$$\lambda = 4.48 + 0.049 T_s + f\left(U_{10}\right)\left(1.12 + 0.018 T_s + 0.00158 T_s^2\right) \tag{6.169}$$

T_a air temperature
T_s water surface temperature
$f\left(U_{10}\right)$ wind function:

$$f\left(U_{10}\right) = \left(3.5 + 2.0 U_{10}\right)\left(\frac{5.0 \times 10^6}{S_{area}}\right)^{0.05} \tag{6.170}$$

U_{10} wind speed
S_{area} exposed water surface in m^2

6.2.1.4 Composite temperature model

In the Composite temperature model, the heat fluxes through the water surface by incoming radiation, back radiation, evaporation and convection are computed separately. Evaporation and convection depend on the air temperature, the water temperature near the free surface, relative humidity, and wind speed.

The total heat flux through the free surface reads:

$$Q_{tot} = Q_{sn} + Q_{an} - Q_{br} - Q_{ev} - Q_{co}, \tag{6.171}$$

with:

Q_{sn} net incident solar radiation (short wave)
Q_{an} net incident atmospheric radiation (long wave)
Q_{br} back radiation (long wave)
Q_{ev} evaporative heat flux (latent heat)
Q_{co} convective heat flux (sensible heat).

$$Q_{tot} = Q_{sn} - Q_{eb} - Q_{ev} - Q_{co}, \tag{6.172}$$

with:

$$Q_{eb} \quad = Q_{br} - Q_{an}$$

The subscript n refers to a net contribution.

6.2.1.5 Solar radiation

The short-wave radiation emitted by the sun that reaches the earth surface under a clear sky condition is computed by means of an emperical formula

The solar radiation is computed by D-Flow 1D and is dependent on the geographical position at the earth and the local time.

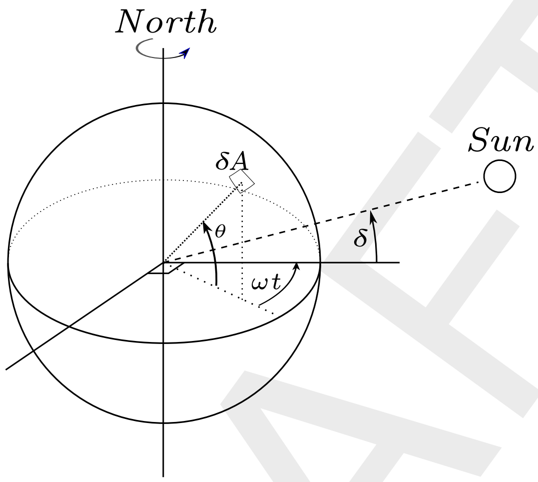


Figure 6.59: *Co-ordinate system position Sun*
δ: declination; θ: latitude; ωt: angular speed

Not all of the radiation is absorbed at the water surface. A part is transmitted to deeper water. Short waves can penetrate over a distance of 3 to 30 meters, depending on the clarity of the water, while the relatively longer waves are absorbed at the surface. Therefore, it is convenient to separate the incoming solar insolation into two portions:

1. βQ_{sn}, the longer wave portion, which is absorbed at the surface and
2. $(1-\beta)\,Q_{sn}$, the remainder part, which is absorbed in deeper water.

The absorption of heat in the water column is an exponential function of the distance H from the water surface:

$$(1-\beta)\,Q_{sn} = \int_0^H e^{-\gamma z}\,dz \Rightarrow \tag{6.173}$$

$$Q_{sn}\,(h) = \frac{\gamma e^{-\gamma h}}{1-e^{-\gamma H}}(1-\beta)Q_{sn}, \tag{6.174}$$

with:

β	part of Q_{sn} absorbed at the water surface which is a function of the wavelength. The default value of β in Delft3D-FLOW is 0.06.
γ	extinction coefficient (measured) in m^{-1}, also related to the so-called Secchi-depth $\gamma = \frac{1.7}{H_{Secchi}}$
h	distance to the water surface in meters.
H	total water depth.

Remark:

◇ The exponential decay function, Equation (6.174) has only been implemented in 3D-computations. Since D-Flow 1D is a depth averaged model, the distinction between long and short wave radiation is not relevant.

The incoming energy flux at the water surface depends on the angle (declination) between the incoming radiation and the Earth's surface. This declination depends on the geographical position on the Earth and the local time. The Earth axis is not perpendicular to the line connecting the Sun with Earth. This tilting (angle δ) varies with the time of the year and it leads to a seasonal variation of the radiation flux. At June 21, the declination is maximal, 23.5 degrees. Of course, by the rotation of the Earth the solar radiation also varies during the day. Near twelve o'clock local time, the sun elevation above the horizon is maximal. For an overview of the angles used to determine the solar elevation angle γ, see Figure 6.59.

The temporal and latitude-dependent solar elevation angle γ is estimated by:

$$\sin(\gamma) = \sin(\delta)\sin\left(\frac{\pi\phi}{180}\right) - \cos(\delta)\cos\left(\frac{\pi\phi}{180}\right)\cos(\omega_1 t) \qquad (6.175)$$

with:

δ	$= \frac{23.5\pi}{180}\cos(\omega_0 t - 2.95)$
ω_0	frequency of the annual variation ($= 2\pi/(365.24 \times 24)$)
ω_1	frequency of the diurnal variation ($= 2\pi/24$)
ϕ	latitude
t	number of hours since 1st of january

The incoming short-wave solar radiation through a clear sky at ground level Q_{sc} is about 0.76 of the flux incident at the top of the atmosphere (Gill, 1982):

$$Q_{sc} = \begin{cases} 0.76 S \sin(\gamma), & \sin(\gamma) \geq 0, \\ 0.0, & \sin(\gamma) < 0. \end{cases} \qquad (6.176)$$

The solar constant $S = 1\,368$ J/(m^{2}s) or W/m^2. This is the average energy flux at the mean radius of the Earth.

A part of the radiation that reaches the water surface is reflected. The fraction reflected or scattered (surface albedo) is dependent on latitude and season. Cloud cover will reduce the magnitude of the radiation flux that reaches the sea surface. The cloudiness is expressed by a cloud cover fraction F_c, the fraction of the sky covered by clouds. The correction factor for cloud cover is an empirical formula. The absorption of solar radiation is calculated (Gill, 1982) as the product of the net downward flux of short wave-radiation in cloudless conditions and factors correcting for reflection and cloud cover:

$$Q_{sn} = Q_s - Q_{sr} = (1 - \alpha)\, Q_{sc} f(F_c), \qquad (6.177)$$

with:

Q_s	solar radiation (short wave radiation) in [$J/(m^2 s)$]
Q_{sr}	reflected solar radiation in [$J/(m^2 s)$]
Q_{sc}	see Equation (6.176)
$\sin(\gamma)$	see Equation (6.175)
$f(F_c)$	$= 1.0 - 0.4 F_c - 0.38 F_c^2$

α	albedo (reflection) coefficient (= 0.06)
F_c	fraction of sky covered by clouds (user-defined input)
S	solar constant (=1368 J m^{-2} s^{-1})

6.2.1.6 Effective back radiation

The total net long wave radiation flux is called the effective back radiation:

$$Q_{eb} = Q_{br} - Q_{an}. \tag{6.178}$$

The atmospheric radiation depends on the vapour pressure e_a, the air temperature T_a and the cloud cover F_c. The back radiation depends on the surface temperature T_s.

$$Q_{eb} = \varepsilon\sigma\bar{T}_s^4\left(0.39 - 0.05\sqrt{e_a}\right)\left(1.0 - 0.6F_c^2\right), \tag{6.179}$$

with:

ε	emissivity factor (0.985)
σ	Stefan-Boltzmann's constant = 5.67×10^{-8} in [$J/(m^2sK^4$]
$\bar{T}_s$	the (absolute) water surface temperature in [K].
e_a	actural vapour pressure:

$$e_a = r_{hum}10^{\frac{0.7859+0.03477T_a}{1.0+0.00412T_a}} \tag{6.180}$$

F_c	fraction of sky covered by clouds (user-defined input)

6.2.1.7 Evaporative heat flux

Evaporation is an exchange process that takes place at the interface between water and air and depends on the conditions both in the water near the surface and the air above it. The evaporation depends on meteorological factors (wind-driven convection) and vapour pressures.

The evaporative heat flux Q_{ev} is defined by:

$$Q_{ev} = L_vE, \tag{6.181}$$

with L_v the latent heat of vaporisation in J/kg water:

$$L_v = 2.5 \times 10^6 - 2.3 \times 10^3T_s. \tag{6.182}$$

The evaporation rate E is defined as the mass of water evaporated per unit area per unit time [kg/(m^{2}s)]. It is computed by using a form of Dalton's law of mass transfer:

$$E = f\left(U_{10}\right)\left(e_s - e_a\right), \tag{6.183}$$

where the actual vapour pressure e_a and the saturated vapour pressure e_s, is defined as:

$$e_s = 10^{\frac{0.7859+0.03477T_s}{1.0+0.00412T_s}}, \tag{6.184}$$

$$e_a = r_{hum}10^{\frac{0.7859+0.03477T_a}{1.0+0.00412T_a}}. \tag{6.185}$$

Here r_{hum} is the relative humidity in [-].

Remarks:

◇ The relative humidity r_{hum} is specified as a function of time.
◇ The vapour pressures are calculated using air and water surface temperature in $^{\circ}$C.
◇ When the computed E is negative, it is replaced by zero, assuming that it is caused by modelling misfit and not by the actual physical process of water condensation out ot the air into the water.

The evaporation rate is computed from the difference in relative humidity, rather than from the difference in vapour pressure. The contribution to the evaporative heat flux is split in a contribution by forced convection and a contribution by free convection. These are discussed separately.

Forced convection of latent heat

The latent heat flux due to forced convection reads:

$$Q_{ev,\text{forced}} = L_V \rho_a f\left(U_{10}\right)\left\{q_s\left(T_s\right) - q_a\left(T_a\right)\right\}, \tag{6.186}$$

with q_s and q_a the specific humidity of respectively saturated air and remote air (10 m above water level):

$$q_s\left(T_s\right) = \frac{0.62 e_s}{P_{atm} - 0.38 e_s}, \tag{6.187}$$

$$q_a\left(T_a\right) = \frac{0.62 e_a}{P_{atm} - 0.38 e_a}. \tag{6.188}$$

The saturated and remote vapour pressures e_s and e_a are given by Eqs. (6.184) and (6.185). The latent heat L_v is given by equation (6.182)

The wind function in Equation (6.186) is defined as:

$$f\left(U_{10}\right) = c_e U_{10}, \tag{6.189}$$

Free convection of latent heat

Loss of heat due to evaporation occurs not only by forced convection, wind driven, but also by free convection. Free convection is driven by buoyant forces due to density differences (by temperature and/or water vapour content) creating unstable conditions in the atmospheric boundary layer (ABL). Evaporation due to free convection is important in circumstances where inverse temperature/density gradients are present and wind speeds are almost negligible so that the amount of forced convection is small. Neglecting free convection in this situation will lead to underestimating the heat loss. (Ryan *et al.*, 1974) developed a correction to the wind function, accounting for free convection. The derivation of evaporation by just free convection is based on the analogy of heat and mass transfer.

The latent heat flux due to free convection reads:

$$Q_{ev,\text{free}} = k_s L_V \overline{\rho}_a \left(q_s - q_a\right), \tag{6.190}$$

with the average air density:

$$\overline{\rho}_a = \frac{\rho_{a0} + \rho_{a10}}{2}, \tag{6.191}$$

and with the heat transfer coefficient defined as:

$$k_s = \begin{cases} 0 & \text{if} \quad \rho_{a10} - \rho_{a0} \leq 0 \\ c_{fr.\text{conv}} \left\{ \frac{g\alpha^2}{\nu_{air}\overline{\rho}_a} \left(\rho_{a10} - \rho_{a0}\right) \right\}^{1/3} & \text{if} \quad \rho_{a10} - \rho_{a0} > 0 \end{cases} \tag{6.192}$$

where the coefficient of free convection $c_{fr.\text{conv}}$ was calibrated to be 0.14, see (Ryan *et al.*, 1974). The viscosity of air ν_{air} is assumed to have the constant value 16.0×10^{-6} m^2/s. The molecular diffusivity of air α m^2/s is defined as

$$\alpha = \frac{\nu_{air}}{\sigma}, \tag{6.193}$$

with $\sigma = 0.7$ (for air) the Prandtl number. In Equation (6.190), the saturated air density is given by:

$$\rho_{a0} = \frac{\frac{100P_{atm} - 100e_s}{R_{dry}} + \frac{100e_s}{R_{vap}}}{T_s + 273.15}, \tag{6.194}$$

the remote air density (10 m above the water level):

$$\rho_{a10} = \frac{\frac{100P_{atm} - 100e_a}{R_{dry}} + \frac{100e_a}{R_{vap}}}{T_{air} + 273.15}, \tag{6.195}$$

where R_{dry} is the gas constant for dry air: 287.05 J/(kg K) and R_{vap} is the gas constant for water vapour: 461.495 J/(kg K). The specific humidity of respectively saturated air and remote air (10 m above the water level), q_s and q_a are given by Eqs. (6.187) and (6.198). The saturated and remote vapour pressure e_s and e_a are defined in Eqs. (6.184) and (6.185).

Q_{ev} *evaporative heat flux summarized*

The total heat flux due to evaporation then results from adding the forced convection of latent heat in Equation (6.186) and the free convection of latent heat in Equation (6.190):

$$Q_{ev} = Q_{ev,\text{forced}} + Q_{ev,\text{free}} \tag{6.196}$$

$$Q_{ev,\text{free}} = k_s L_V \overline{\rho}_a \left\{ q_s\left(T_s\right) - q_a\left(T_a\right) \right\} \tag{6.197}$$

$$Q_{ev,\text{forced}} = L_V \overline{\rho}_a f\left(U_{10}\right) \left\{ q_s\left(T_s\right) - q_a\left(T_a\right) \right\} \tag{6.198}$$

with:

k_s heat transfer coefficient:

$$k_s = \begin{cases} 0 & \text{if} \quad \rho_{a10} - \rho_{a0} \leq 0 \\ c_{fr.\text{conv}} \left\{ \frac{g\alpha^2}{\nu_{air}\overline{\rho}_a} \left(\rho_{a10} - \rho_{a0}\right) \right\}^{1/3} & \text{if} \quad \rho_{a10} - \rho_{a0} > 0 \end{cases} \tag{6.199}$$

L_V L_v the latent heat of vaporisation

$$L_V = 2.5 \times 10^6 - 2.3 \times 10^3 T_s \tag{6.200}$$

$q_s\left(T_s\right)$ The specific humidity of saturated air

$$q_s\left(T_s\right) = \frac{0.62e_s}{P_{atm} - 0.38e_s} \tag{6.201}$$

$q_a\left(T_a\right)$ The specific humidity of remote air

$$q_a\left(T_a\right) = \frac{0.62e_a}{P_{atm} - 0.38e_a} \tag{6.202}$$

$f(U_{10})$ wind function: $c_e U_{10}$
$\overline{\rho}_a$ average air density:

$$\overline{\rho}_a = \frac{\rho_{a0} + \rho_{a10}}{2} \tag{6.203}$$

ρ_{a0} The saturated air density is given by:

$$\rho_{a0} = \frac{\frac{100P_{atm} - 100e_s}{R_{dry}} + \frac{100e_s}{R_{vap}}}{T_s + 273.15} \tag{6.204}$$

ρ_{a10} remote air density (10 m above the water level):

$$\rho_{a10} = \frac{\frac{100P_{atm} - 100e_a}{R_{dry}} + \frac{100e_a}{R_{vap}}}{T_a + 273.15} \tag{6.205}$$

e_s saturated pressure:

$$e_s = 10^{\frac{0.7859 + 0.03477T_s}{1.0 + 0.00412T_s}} \tag{6.206}$$

e_a remote vapour pressure:

$$e_a = r_{hum} 10^{\frac{0.7859 + 0.03477T_a}{1.0 + 0.00412T_a}} \tag{6.207}$$

R_{dry} gas constant for dry air: 287.05 J/(kg K) is the
R_{vap} gas constant for water vapour: 461.495 J/(kg K)
P_{atm} Atmospheric pressure
U_{10} Wind velocity at 10 m
c_e Dalton number (=0.0015)
r_{hum} relative humidity
T_a air temperature
T_s water surface temperature
$c_{fr.\text{conv}}$ coefficient of free convection (= 0.14)
g gravity
α molecular diffusivity (= ν_{air}/σ)
ν_{air} viscosity of air (=16.0×10^{-6} m^2/s)
σ Prandtl number (= 0.7)

6.2.1.8 Convective heat flux

The convective heat flux is split into two parts, just as the evaporative heat flux. The convective heat flux is divided into a contribution by forced convection and a contribution by free convection.

$$Q_{co} = Q_{co,\text{forced}} + Q_{co,\text{free}} \tag{6.208}$$

Forced convection of sensible heat

The sensible heat flux due to forced convection is computed by:

$$Q_{co,\text{forced}} = \rho_a c_p g(U_{10})(T_s - T_a), \tag{6.209}$$

with:

ρ_a air density as specified in input
$g(U_{10})$ wind speed function is defined following Gill (1982): $c_H U_{10}$

c_H	Stanton number (= 0.0013)
T_a	air temperature
T_s	water surface temperature
c_{pa}	specific heat capacity of air; = 1004.0 J/(kg K)

Free convection of sensible heat

The sensible heat flux due to free convection is computed by:

$$Q_{co,\text{free}} = k_s \overline{\rho}_a c_{pa} \left(T_s - T_a\right) \tag{6.210}$$

with:

k_s heat transfer coefficient:

$$k_s = \begin{cases} 0 & \text{if} \quad \rho_{a10} - \rho_{a0} \leq 0 \\ c_{fr.\text{conv}} \left\{ \frac{g\alpha^2}{\nu_{air}\overline{\rho}_a} \left(\rho_{a10} - \rho_{a0}\right) \right\}^{1/3} & \text{if} \quad \rho_{a10} - \rho_{a0} > 0 \end{cases} \tag{6.211}$$

$\overline{\rho}_a$ average air density:

$$\overline{\rho}_a = \frac{\rho_{a0} + \rho_{a10}}{2} \tag{6.212}$$

ρ_{a0} The saturated air density is given by:

$$\rho_{a0} = \frac{\frac{100P_{atm} - 100e_s}{R_{dry}} + \frac{100e_s}{R_{vap}}}{T_s + 273.15} \tag{6.213}$$

ρ_{a10} remote air density (10 m above the water level):

$$\rho_{a10} = \frac{\frac{100P_{atm} - 100e_a}{R_{dry}} + \frac{100e_a}{R_{vap}}}{T_a + 273.15} \tag{6.214}$$

c_H	Stanton number (= 0.0013)
T_a	air temperature
T_s	water surface temperature
c_{pa}	specific heat capacity of air; = 1004.0 J/(kg K)

6.2.1.9 Input parameters for composite model

The input parameters for the composite model are:

F_c	percentage of sky covered by clouds (time dependent, Meteo data)
T_a	air temperature in °C (time dependent, Meteo data)
r_{hum}	percentage of relative humidity (time dependent, Meteo data)
c_e	Dalton number (default value: 0.0013)
c_H	Stanton number (default value: 0.0013)
c_p	specific heat capacity of water (default value: 3930 J/(kg K) for sea water)

6.2.2 Salinity dispersion

When one of the boundaries of a river or estuary is in contact with the sea, salt water may enter the system. Salt intrusion into a river or estuary is determined by the fresh water discharge on the river side and mixing of the water column by the tides. Mixing occurs due to tidal shear between streamlines of different velocities, tidal trapping of water in tidal flats or side channels, residual currents within the cross section (tidal pumping) and turbulent mixing at small scales. The dominant physical process is the gravitational circulation: it transports salt and thus more dense water near the bed in landward direction and fresh water near the surface in seaward direction. Mixing of the water column by the tide reduces the gravitational circulation.

For the one-dimensional D-Flow 1D the three-dimensional mixing processes are parametrized into a dispersion term. This dispersion term represents the processes that cannot be described in a cross section averaged model. The advection term in D-Flow 1D only represents the transport of salt with the cross sectional averaged flow. In D-Flow 1D a dispersion coefficient is used in the advection-diffusion equation SOBEK3_TRM (2013, §1.7). The advection-diffusion equation is coupled via a state equation, relating salinity and density, to the momentum equation.

For estuaries several formulations for the dispersion coefficient are available. Thatcher and Harleman (1972) defined a dispersion formula based on field observations in three estuaries in the USA and on a tidal flume experiment. Within their formula they made distinction between shear dispersion which is also present in the fresh water region of an estuary (first part) and tidal dispersion (second part):

$$D = 77\,u\,n\,R^{5/6} + kL_e^2 \frac{u_0}{s_{0,max}} N_r^{1/4} \left| \frac{\partial s}{\partial x} \right| \tag{6.215}$$

in which:

u	flow velocity [m s^{-1}],
n	Manning coefficient [s m$^{-\frac{1}{3}}$],
R	hydraulic radius [m],
k	parameter for tidal dispersion to be defined by the user [-],
L_e	the estuary length [m],
u_0	absolute maximum flood flow velocity [m s^{-1}],
$s_{0,max}$	the maximal salt concentration in the estuary mouth [kg m^{-3}],
N_r	estuarine Richardson number [-],
s	salt concentration [kg m^{-3}],
x	Coordinate along branch [m].

The estuarine Richardson number (N_r) expresses the balance between the stratifying factors (fresh water input) and the mixing factors due to the tide. Therewith it is a measure for the degree of stratification. It is used in all formulations. For low numbers of N_r the estuary is well mixed, while for high numbers it is highly stratified. According Fischer *et al.* (1979) the transition for N_r occurs between 0.08 and 0.8. The estuarine Richardson number is defined as:

$$N_r = \frac{\Delta\rho}{\rho_w} \frac{ghQ_fT}{u_0^2 P_e} \tag{6.216}$$

in which:

$\Delta\rho$	density of sea water minus density of inflowing fresh water [kg/m^3]
ρ_w	density of inflowing fresh water [kg/m^3]
Q_f	fresh water discharge [m^3/s]

T tidal period [s]
P_e tidal prism: the water entering the estuary from the sea side during flood [m^3]
g acceleration due to gravity [m s^{-2}],
h Total water depth [m]

Various newer dispersion formulations exist. Kuijper and Van Rijn (2011) modified a formulation for the tidal dispersion term described by Savenije (2012). Based on Ippen and Harleman (1961) they added a friction parameter to account for vertical mixing related to the ratio of dissipated energy by means of bed friction and gained potential energy of the fresh river water due to an increase in density. Furthermore, they added depth explicitly in the formulation. They distinguished between convergent (trumpet shaped) and prismatic estuaries (having straight banks). The formulation for prismatic channels is defined at high water slack (HWS):

$$D^{HWS} = 6\alpha_c u_0 d_0 N_r^{1/2} \frac{C}{\sqrt{g}} e^{\phi x} \frac{s^{HWS}}{s_0^{HWS}} \tag{6.217}$$

and for convergent estuaries:

$$D^{HWS} = 60\alpha_c u_0 \frac{h_0}{a} E_0 N_r^{1/2} \frac{C}{\sqrt{g}} e^{\phi x} \frac{s^{HWS}}{s_0^{HWS}} \tag{6.218}$$

in which:

$$\phi = \frac{\delta_u}{2} + \frac{3(a-b)}{2ab} + \frac{1}{2a} \tag{6.219}$$

with:

α_c calibration factor with default 1, and $0.7 \leq \alpha_c \leq 1.3$ [-],
s_0 salt concentration in the mouth [kg m^{-3}],
ϕ damping term for the damping of tidal amplitude [m^{-1}],
x Location increasing into the estuary [m],
δ_u damping coefficient [m^{-1}],
a cross-sectional convergence length [m],
b width convergence length [m].
h_0 average water depth in a branch [m], ,
E_0 tidal excursion length [m],
C Chézy coefficeint [m$^{0.5}$ s^{-1}],

6.2.3 Sediment transport capacity

The D-Flow 1D module computes the sediment transport capacity according to the following two formulations.

Frijlink

$$T_v = D_{50}\sqrt{g\mu R|S_e|}5\exp\left(\frac{-0.27D_{90}}{\mu R|S_e|}\right) \tag{6.220}$$

$$\mu = \left(\frac{C}{18\ {}^{10}\log\left(\frac{12R}{D_{90}+\frac{3\nu}{\sqrt{gR|S_e|}}}\right)}\right)^{3/2} \tag{6.221}$$

$$S_E = \frac{\Delta\left(h+\frac{u^2}{2g}\right)}{\Delta x} \tag{6.222}$$

T_v Sediment transport capacity per m bed width [m^2/s]
μ Bottom roughness factor (ripple factor)
D_{50} Grain diameter at which 50 % of the grains is smaller [m]
D_{90} Grain diameter at which 90 % of the grains is smaller [m]
R Hydraulic radius [m]
S_e Energy slope [$-$]
C Chézy coefficient [$m^{1/2}/s$]
ν Kinematic viscosity [m^2/s]. Normally 1.002×10^{-6}
h Water level [m]
u Velocity [m/s]
g Acceleration due to gravity [m/s^2] (≈ 9.81)
x Length of branch segment [m]

Van Rijn

$$T_v = 0.053 \frac{T_{gr}^{2.1}}{D_{gr}^{0.3}} \sqrt{(s-1)g} D_{50}^{1.5} \tag{6.223}$$

$$T_{gr} = \frac{u_*^2 - u_{*c}^2}{u_{*c}^2} \text{ if } u_*^2 > u_{*c}^2 \tag{6.224}$$

$$T_{gr} = 0 \text{ if } u_*^2 \leq u_{*c}^2 \tag{6.225}$$

$$u_* = \frac{u\sqrt{g}}{C} \tag{6.226}$$

$$C = 18 \; {}^{10}\log \left(\frac{12R}{3D_{90}} \right) \tag{6.227}$$

C Chézy coefficient related to bed material
u Average flow velocity [m/s]
u_{*c} Critical shear stress velocity according to Shields [m/s]

The grain size under water is defined dimensionless by $D_g r$:

$$D_{gr} = D_{50} \sqrt[3]{\frac{(s-1)g}{\nu^2}} \tag{6.228}$$

$(s-1)$ Relative density
s s = s/w
r_s Density of sediment [kg/m^3]. Normally 2650 kg/m^3
r_w Density of water [kg/m^3]. Normally 1000 kg/m^3
ν Kinematic viscosity [m^2/s]

The critical shear stress velocity according to Shields u_{*c} can be found with

$$u_{*c} = \sqrt{\theta_{cr}(s-1)gD_{50}} \tag{6.229}$$

$$\theta_{cr} = \begin{cases} 0.24 D_{gr}^{-1} & \text{if } D_{gr} \leq 4 \\ 0.14 D_{gr}^{-0.64} & \text{if } 4 < D_{gr} \leq 10 \\ 0.04 D_{gr}^{-0.1} & \text{if } 10 < D_{gr} \leq 20 \\ 0.013 D_{gr}^{-0.29} & \text{if } 20 < D_{gr} \leq 150 \\ 0.055 & \text{if } D_{gr} > 150 \end{cases} \tag{6.230}$$

Note: The sediment transport capacity computed with these formulas is always positive. To give an indication of the direction of the sediment transport in the network, the result is multiplied by the sign of the velocity.

The mentioned formulas are defined in sediment transport capacity per meter bed width. To arrive at a more comprehensible definition the results are multiplied with a certain characteristic width. For closed cross sections, the maximum width of the profile is taken. For open cross sections the wetted perimeter at a certain level (normally 0.30 m) above the bed level is taken.

Note: The parameters that are used in the formulations, like grain diameters, are constant over the whole network

6.3 Hydrodynamics Overland 2DFLOW

6.3.1 Continuity equation (2D)

The flow in two dimensions is described by three equations:

1. the continuity equation,
2. the momentum equation for the x-direction and
3. the momentum equation for the y-direction.

The continuity equation reads:

$$\frac{\partial \zeta}{\partial t} + \frac{\partial hu}{\partial x} + \frac{\partial hv}{\partial y} = 0 \tag{6.231}$$

where:

u	velocity in x-direction [m/s]
v	velocity in y-direction [m/s]
ζ	water level above plane of reference [m]
h	total water depth [m]
d	depth below plane of reference [m]

The continuity equation ensures the conservation of fluid.

6.3.2 Momentum equations (2D)

For two dimensional flow, two momentum equations are calculated, together with the continuity equation 2D. The momentum equations read:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + g\frac{\partial \zeta}{\partial x} + g\frac{u|\vec{u}|}{C^2 h} + au|u| = 0 \tag{6.232}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + g\frac{\partial \zeta}{\partial y} + g\frac{v|\vec{u}|}{C^2 h} + av|v| = 0 \tag{6.233}$$

where:

u	velocity in x-direction [m/s]
v	velocity in y-direction [m/s]
$\lvert\vec{u}\rvert$	velocity magnitude ($= \sqrt{u^2 + v^2}$) [m/s]
ζ	water level above plane of reference [m]
C	Chézy coefficient [$\sqrt{m}/s$]
h	total water depth ($= d + \zeta$) [m]
d	depth below plane of reference [m]
a	wall friction coefficient [$1/m$]

They consist of acceleration terms, the horizontal pressure gradient terms, convective terms, bottom friction terms and wall friction terms. These equations are non-linear and they are a subset of the well-known shallow water equations, that describe water motion for which vertical accelerations are small compared to horizontal accelerations (this applies to tidal flow, river flow, flood flow).

6.3.3 Branch growth formulae available at a "Flow 1D Dam Break Branch"

Two bbranch growth formulae are available at a "Flow 1D Dam Break Branch":

◇ The Verheij-vdKnaap(2002) formula (see section 6.3.3.1), and
◇ The vdKnaap(2000) formula section 6.3.3.2).

6.3.3.1 Verheij-vdKnaap(2002) bbranch growth formula

First the Verheij-vdKnaap(2002) bbranch growth formula is discussed. Thereafter, default values (as well as a range of values) for the parameters used in the Verheij-vdKnaap(2002) formula are given.

The Verheij-vdKnaap(2002) formula:

Two phase can be discerned:

◇ **Phase 1**: during which the bbranch, for a constant initial width(B_0), is lowered from its initial dike level or crest level ($z_{\text{crest level}}$) to its final bed level or crest level (Z_{min}).
◇ **Phase 2**: during which the bbranch only grows in width as long as the flow velocity through the bbranch is larger than the critical flow velocity for eroding sediment/soil particles (u_c).

Phase 1:

For $T_{start} \leq t \leq T_{start} + t_{\text{Phase 1}}$

$$B(t) = B_0 \tag{6.234}$$

$$z(t) = z_{\text{crest level}} - \left(\frac{t - T_{Start}}{t_{\text{Phase 1}}}\right)(z_{\text{crest level}} - z_{\min}) \tag{6.235}$$

Phase 2:

For $t > T_{start} + t_{\text{Phase 1}}$ (hence $B(t) \geq B_0$):

$$z(t) = z_{\min} \tag{6.236}$$

$$B(t_{i+1}) = B(t_i) + \Delta t \left(\frac{\partial B}{\partial t}\right)_{t_i} \tag{6.237}$$

$$\left(\frac{\partial B}{\partial t}\right)_{t_i} \begin{cases} \frac{f_1 f_2}{\ln(10)} \frac{(g(h_{up} - max(h_{down}, z_{min})))^{3/2}}{u_c^2} \frac{1}{1 + \frac{f_2 g}{u_c}(t_i - t_0)} & \text{if} \quad |u| > u_c \\ 0 & \text{if} \quad |u| \leq u_c \end{cases} \tag{6.238}$$

where:

f_1	factor1: constant factor (input parameter) $[-]$ (default value =1.3)
f_2	factor 2: constant factor (input parameter) $[-]$ (default value = 0.04)

$B(t)$	width of the dike-bbranch at point-in-time t [m]
B_0	Initial width of the dike-bbranch [m]
$\left(\frac{\partial B}{\partial t}\right)_{t_i}$	Rate of bbranch width growth in time-step Δt from t_i to t_{i+1} [m/h]
g	acceleration due to gravity [m/s^2]
h_{up}	upstream water level at point-in-time t [m]
h_{down}	downstream water level at point-in-time t [m]
t	actual computational point-in-time [h]
$T_{start} + t_{\text{Phase 1}}$	= the point-in-time when the maximum bbranch-depth (z_{min}) is branched,
$t_{\text{Phase 1}}$	duration of phase 1, the time period over which the bbranch for a constant initial width (B_0) is lowered from its initial dike level or crest level ($z_{\text{crest level}}$) to its final bed level or crest level ($z_{\min}$) [h]. **Note:** T_0 in hours relative to a reference time!
T_{start}	computational point in time at which the development of the initial bbranch starts [h]
u	average flow velocity in the bbranch [m/s]
u_c	constant critical flow velocity for eroding sediment/soil particles (input parameter). If u > u_c bbranch material is picked up by the flow [m/s]
$z(t)$	elevation of the dike-bbranch at point-in-time t [m]
$z_{crest\ level}$	elevation of the crest level of the dike at $t = t_{start}$ (input parameter) [m]
z_{min}	Elevation of the dike-bbranch at $t = t_0$ (input parameter) [m]

Defaults in the Verheij-vdKnaap(2002) formula

To submit proper values a list of default values as well as the range of values given in the table below can be used.

Parameter	Default	Range
f1	1.3	0.5 - 5
f2	0.04	0.01 - 1
B0	10	1 - 100 m
T0	0.1 hours	0.1 - 12 hours
U_c	0.2 m/s	0.1 - 10 m/s
Zcrest level	-	-
Zmin	-	-

Characterising soil strength:
The parameter u_c, applied in the Verheij-vdKnaap(2002) formula may be derived from the table given below.

Table 6.6

Soil type	Bodemtype (Dutch)	U_c [m/s]	τ_c [Pa]

grass, good	gras, goed	7	185
grass, moderate	gras, matig	5	92.5
grass, bad	gras, slecht	4	62
clay, very good (compacted)	klei, zeer goed	1.0	4
	(compact; $t_{\text{ongedraineerd}}$ = 80 - 100 kPa)		
clay, good (firm)	klei met 60% zand, (stevig; $t_{\text{ongedraineerd}}$ = 40 - 80 kPa)	0.8	2.5
clay, moderate (little structure)	goede klei met weinig structuur	0.7	2
clay moderate (considerable structured)	goede klei, sterk	0.6	1.5
clay, bad (weak)	slechte klei (slap; $t_{\text{ongedraineerd}}$ = 20 - 40 kPa)	0.4	0.65
sand with 17% silt	zand met 17% silt	0.225	0.20
sand with 10% silt	zand met 10% silt	0.20	0.15
sand with 0% silt7	zand met 0 % silt	0.16	0.10

In addition, it is possible to calculate u_c from τ_c by applying the following formula:

$$u_c = 0.5\sqrt{\tau_c} \tag{6.239}$$

or

$$u_c = u_{c,\text{sand}}(1 + 0.01\alpha P_{\text{clay}} + \beta(0.65 - v) \tag{6.240}$$

using the defaults, $\alpha = 15$, $\beta = 1$, $U_{c,\text{sand}} = 0.2$ m/s

Further on yields:

$$v = n/(1 - n) \tag{6.241}$$

where n = the soils pore fraction (default $n = 0.4$), and P_{clay} = percentage of clay.

Literature:

Formula Verheij-vdKnaap (2002):Verheij (2002)

6.3.3.2 vdKnaap(2000) bbranch growth formula

This formula consists of equations: one for sand dikes, and one for clay dikes.

The following information is used in calculating the bbranch area (or time-Area) table:

- ◇ maximum allowed bbranch width $B_{\max}$; 200 m for sand and 75 m for clay
- ◇ maximum number of steps for the bbranch area (or time-Area) curve when the dam break grows in width ($n_{\max} = 20$)
- ◇ minimum step size for generated a bbranch area (or time-Area) table (s_{min} = 5 s)
- ◇ minimum increase in bbranch area per time step for a generated bbranch area (or time-Area) table ($dA_{\min}$ =0.01 m^2)
- ◇ percentage areal increase for 1st step on bbranch area (or t-A) curve to smooth linear part with logarithmic curve (p = 10 %)

Info supplied by the user through the user interface:

- ◇ initial bbranch width (B_{ini})
- ◇ maximum bbranch depth ($D_{\max}$)
- ◇ start of dam break (t_0)
- ◇ time to branch maximum bbranch depth (d_{t1})
- ◇ type of dam material (i.e. sand or clay)
- ◇ either maximum bbranch width ($B_{\max}$) or time to branch max. bbranch width (d_{tmax})

The dam material (i.e. sand or clay) determines the equations that will be used to compute the time-dependent development of the branch, viz:

- ◇ for sand: $B(t) = 67\ {}^{10}\log(t/522)$ and $t(B) = 522 \times 10^{B/67}$
- ◇ for clay: $B(t) = 20\ {}^{10}\log(t/288)$ and $t(B) = 288 \times 10^{B/20}$

The info supplied by the user is checked on the following conditions:

- ◇ $B_{ini} > 0$
- ◇ $D_{max} > 0$
- ◇ $B_{ini} + B(s_{min} n_{max}) \leq B_{max}$
- ◇ $d_{t1} + s_{min} n_{max} \leq d_{tmax} < t(B_{max})$

If all conditions are met, two initial time steps (the linear part) are calculated:

1st time step: A_0 = 0 m^2
2nd time step: $t_1 = t_0 + d_{t1}$ and $A_1 = D_{max} B_{ini}$

Then, based on the end condition, the last time step is calculated according to the applicable equation. The starting point for the equation ($t = 0$) is supposed to coincide with t_1.

NB: The total number of time steps is $n_{\max}$ time steps + 2 initial time steps (i.e. 22).
22nd time step: $t_{21} = t_0 + d_{tmax}$ or $t_{21} = t_1 + t(B_{max})$ and $A_{21} = D_{max} B(t_{21} - t_1)$

Now all intermediate time steps (3rd through 21st) can be calculated. Because of the equations being logarithmic functions, these time steps are not linearly divided over the remaining interval, but as shown in the figure below, thus producing more values for B in the region where the curve is steep.

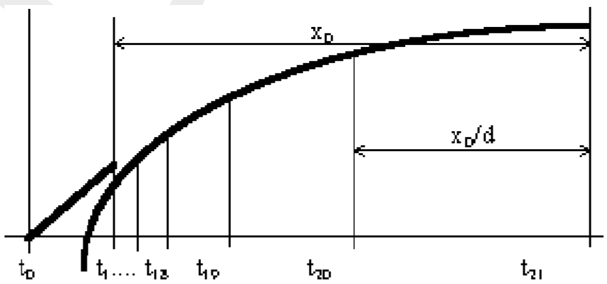


Figure 6.60: *Dam break Bbranch vdKnaap(200), timestep subdivision*

When $x_0 = t_{21} - t_1$, $x_1 = t_{20} - t_1$ and d is a divider such that $d = x_0/(t_{21} - t_{20})$ then:

$$x_n = x_0(1 - \frac{1}{d})\frac{1}{n_{max}} \text{ and } x_n \geq s_{min} \tag{6.242}$$

or

$$d \geq \frac{1}{(1 - s_{min}/x_0)} \frac{1}{n_{max} - 1} \tag{6.243}$$

Based on this divider d the time steps t_{20} through t_2 can be calculated:

$$t_i = t_{i+1} + (t_1 - t_{i+1})/d \tag{6.244}$$

$$A_i = D_{max} B(t_i - t_1) \tag{6.245}$$

Since the logarithmic functions will evaluate negative values when closing in to $t = 0$, there will often be some values (from t_2 upto t_k) calculated for the bbranch width smaller than B_{ini}.

Of course the bbranch width can not decrease, so these values are removed from the table, except for the last one (t_k with A_k such that $A_{k+1} > A_1$), which is used to smoothen the linear part with the logarithmic curve.

A percentage p is introduced to determine the bbranch width value for this point according to the following rule:

$$A_k = A_1 + p(A_{k+1} - A_1) \tag{6.246}$$

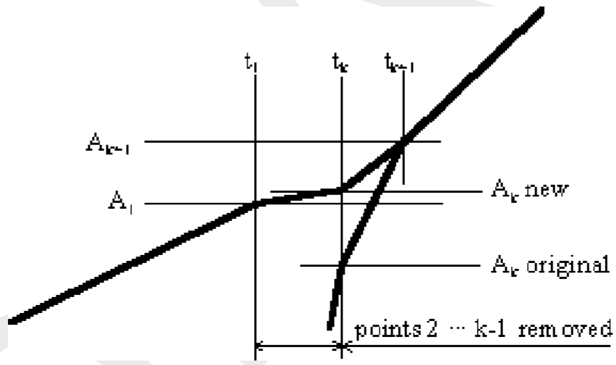


Figure 6.61: *Bbranch width computation according vdKnaap(2000)*

Literature:

Formulas vdKnaap (2000): Van der Knaap (2000),

6.3.4 1D-2D connection

The 1D network is linked with the 2D grid in the following ways:

- ◇ The connection between the 1D connection Node and 2D grid cell;
- ◇ The connection between the 1D ζ-calculation Points and 2D grid cell.

The following rules should be kept in mind, only one connection per grid cell is allowed. In other words, you cannot have both a connection node and calculation point in one grid cell, nor more that one connection node or ζ-calculation point per grid cell. It is simpler to assume that 1D and 2D networks are two independent map layers, with the 2D network map layer overlapping a 1D network. The computational code determines the connection points between 1D and 2D based on the map coordinates for the centre of 2D grid cell and the 1D connection node / ζ-calculation node. If they fall within certain criteria, then the connection is made between them, else not. The criteria, if expressed in mathematical terms, are as follows:

if ($|X1 - X2| \leq \Delta x/2$) and ($|Y1 - Y2| \leq \Delta y/2$), where:

$X1$	x map coordinate for 1D point
$X2$	x map coordinate for 2D grid cell
$Y1$	y map coordinate for 1D point
$Y2$	y map coordinate for 2D grid cell
Δx	width of grid cell in x-direction
Δy	width of grid cell in y-direction (Δx and Δy are equal)

then the 1D point is assumed to lie completely within the 2D grid cell.

The connection between the 2D cells and the 1D network is done in the following way (see figure below):

- The center of 1D node is internally moved to match with the center of 2D grid cell, without changing the length of the connecting 1D branches.
- The 2D Grid Cell is counted as part of 1D Node.
- The flow in 1D channel below the 2D grid level is treated as 1D flow, while the flow above the 2D Grid level is treated as 2D flow with the area of 2D grid cell.

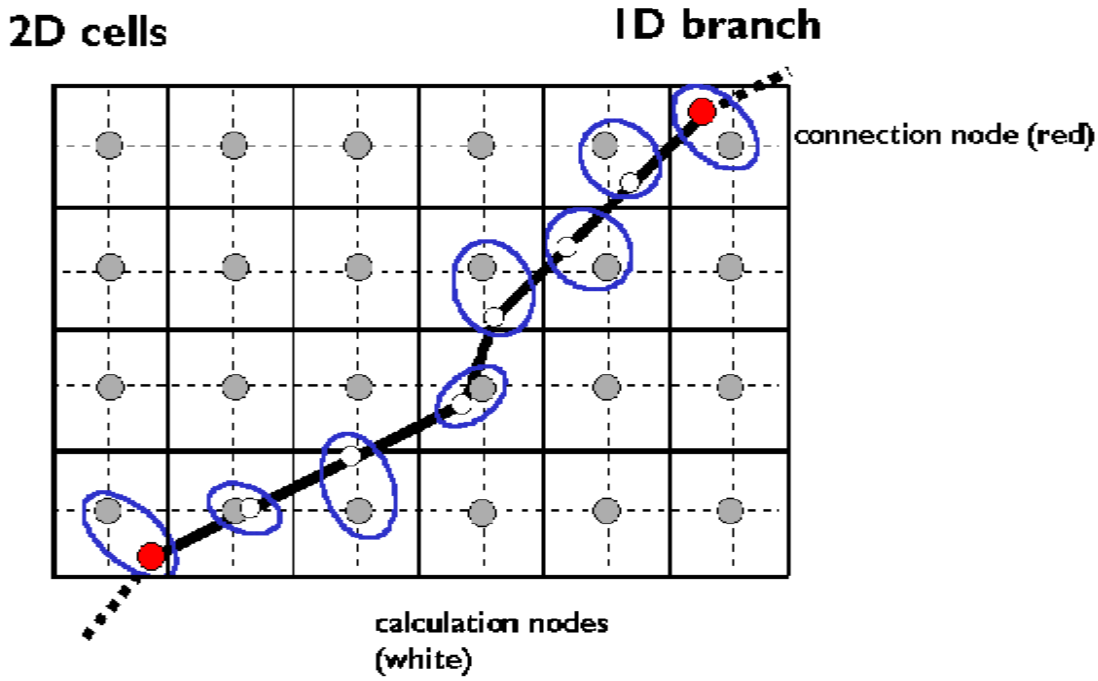


Figure 6.62: *Connections between 1D and 2D*

Modelling 1D2D

Principle is that the top width of the remaining (clarification below) 1D channel should be smaller than the 2D grid cell size. The way of 1D-2D linking depends on the option you choose in the Advanced Settings tab:

- **Assume No Embankments**: In this case the elevation of the underlying 2D grid cell remains the same. The 1D cross-sectional profile above the elevation of the underlying 2D grid cell is omitted (hence excluded from computation).
  - In case the water level is below 2D grid elevation, the flow is fully 1D only (note that the width of the 1D channel might be (and preferably: should be) smaller than the 2D grid cell size).
  - In case water level is above the 2D grid cell elevation; there is a 1D flow as well as 2D flow. In the 1D cross-sectional profile below the 2D grid cell elevation, 1D flow is computed in accordance with the underlying 1D profile and 1D hydraulic roughness. Above the 2D grid cell elevation, there is only 2D flow and this is of course over the full 2D grid cell size, both in x- and y- direction.
- **Assume Highest Level of Embankments**: In this case SOBEK raises the elevation of the underlying 2D grid cell to the highest embankment level. No part of the 1D cross-sectional profile is omitted. Thereafter the computation is identical as described under option 1 (Assume no embankments), with the notation that not the original 2D grid cell elevation is used but the raised grid cell elevation.
- **Assume Lowest Level of Embankments**: Same as option 2 above, but with notation that the part of the 1D cross-sectional profile above the lowest embankment level (i.e. minimum of left and right 1D embankments) is omitted and that the elevation of the underlying 2D gridcell is raised to the elevation of the lowest embankment level.

In general you can best model your river fully 2D when your 1D profile width $>$ 2D grid cell size.

6.3.5 2D-2D connection

The 2D grid cell connection to another 2D grid cell when one grid is partially or fully overlapping another grid, is done in such a way that the 2D Grid cell of the nested grid is connected to the 2D grid cell of underlaying grid by a branch. This branch carries the flow to and from the two connecting grid cells. This branch is given the property of the smallest grid cell and with a very low friction coefficient so that the hindrance of flow is negligible.

The computational core defines the connection between two 2D grid cells based on the map coordinates for the center of each grid cells. If they fall within certain criteria, then the connection is made between them else not. The criteria, expressed in mathematical terms, are:

$$(|X1 - X2| \leq \Delta x) \text{ and } (|Y1 - Y2| < YH \ldots 1) \tag{6.247}$$

$$(|Y1 - Y2| \leq \Delta y) \text{ and } (|X1 - X2| < XH \ldots 2) \tag{6.248}$$

where:

$X1$	x map coordinate for 2D grid cell in first grid
$X2$	x map coordinate for 2D grid cell in second grid
$Y1$	y map coordinate for 2D grid cell in first grid
$Y2$	y map coordinate for 2D grid cell in second grid
Δx	$(\Delta x_1/2 + \Delta x_2/2)$
Δx	$(\Delta y_1/2 + \Delta y_2/2)$
XH	maximum of $\Delta x_1/2$ or $\Delta x_2/2$

YH	maximum of $\Delta y_1/2$ or $\Delta y_2/2$
Δx_1	width of grid cell in x-direction in first grid
Δy_1	width of grid cell in y-direction in first grid
Δx_2	width of grid cell in x-direction in second grid
Δy_2	width of grid cell in y-direction in second grid

The connection is made, if any of the above two criteria matches. It is possible that there are more connections made to the same 2D grid cell from different 2D grid cells in another grid.

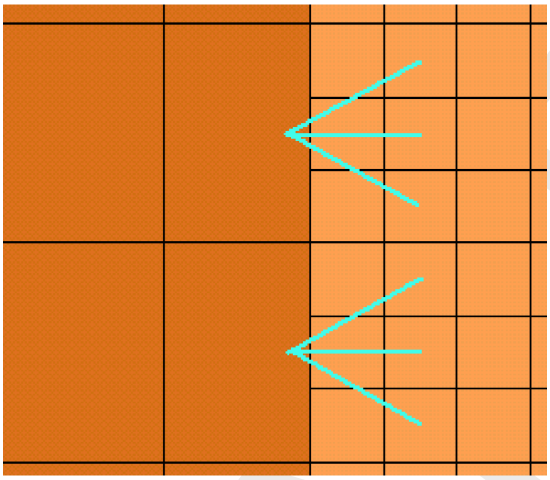


Figure 6.63: *Connections made between different grids*

Note: please bear in mind that the outside cells of any grid are removed from the calculation. So, in the example of Figure 6.63, the connections are made between the large (parent) grid cell and the second column of grid cells of the child grid!

6.4 Triggers and Controllers

6.4.1 Controller

The SOBEK-Flow-module offers the option to operate hydraulic structures.

The adjustable part of a hydraulic structure (e.g. crest level, crest width, Gate Lower Edge Level [m AD]) can be adjusted by a controller. In the River Flow and Real Time Control modules, more than one controller can operate on the same hydraulic structure, however, a specific control parameter (crest level, etc.) can be steered by only one controller at a time. Also, one controller can steer the same control parameter of more than one hydraulic structure, e.g., it is possible to steer the gate lower edge level of a series of structures by only one controller.

In case no trigger is assigned to a controller, this controller will be active during the entire computation.

Six controller types have been implemented:

1 Interval controller
2 Hydraulic controller
3 PID controller
4 Time controller
5 Relative time controller

6 Relative from value (time) controller

The parameters that can be controlled are:

- Crest level of a weir
- Crest level and crest width of River weir
- Crest level of Advanced weir
- Crest level, crest width and gate lower edge level of General structure
- Crest level of Database structure
- Opening height of a orifice
- Capacity of a pump (Rural Flow module only)
- Valve opening of a culvert
- Valve opening of a siphon
- Valve opening of an inverted siphon

For more details on controllers available for the Advanced weir, Database structure, General structure and River weir, see also Appendix B (River Flow controller options).

A control frequency should be given. This frequency defines when the controller function should be updated. This may be every time step (control frequency 1), but also after a number of time steps (control frequency n).

Procedure for the River Flow module (usage of triggers):
The controller function is activated once for each time step in the computational process (the controller is not updated during the iteration process of solving the flow equations). Just before the start of the iteration process the controller is updated in the following way.

For each controller:

```
if (a trigger is defined for this structure) then
    if 'trigger==ON' then
        activate controller function
    else
        do not activate controller function
    endif
else
    activate controller function
endif
```

As you see, a controller can be activated by a trigger, but it can also be activated by a combination of triggers. The computation of the actual control parameter (the carrying out of controller function) for a structure is as follows:

if controller should be evaluated (depending on the user defined control frequency) then

```
if 'time controller' then   evaluate time controller
if 'hydraulic controller' then   evaluate hydraulic controller
if 'Interval controller' then   evaluate interval controller
if 'pid controller' then   evaluate pid controller
if 'relative time controller' then   evaluate relative time controller
if 'relative from value controller' then   evaluate relative from value controller
```

A control frequency should also be given. This defines when the controller function should be updated. This may be every time step (control frequency 1), but also after a higher number of time steps.

The control frequency of an hydraulic controller can also be specified zero. This means that the control parameter will be updated only the first time after the end of an inactive period. A trigger should be used to activate the hydraulic controller.

If a time controller, a relative time controller or a relative from value controller is used the parameter 'dvalue/dt' must be given a value. This parameter defines the maximum speed of increase or decrease of the control parameter. It provides some physical damping, to prevent that instabilities arise due to sudden changes in the structure opening. With for instance a value of 'dvalue=0.1', the speed of opening or closing of a gate is 0.1 m/s (e.g., 1 m change of gate lower edge level in a computational time step of 10s).

6.4.1.1 Combinations of controllers

The crest level, crest width or gate lower edge level of a hydraulic structure can be operated by more than one controller. SOBEK allows up to four controllers for one structure. A specific crest level, crest width or gate lower edge level of a hydraulic structure can be operated by only one controller at a time. In case two or more controllers are given a value for a particular structure parameter (e.g. crest level), then SOBEK stops the computation. By using triggers the user should avoid the situation that two or more controllers are simultaneous active for one and the same structure parameter.

6.4.1.2 Hydraulic controller

The hydraulic controller is a relatively simple controller type which can be used to operate a structure as a (tabulated) function of a specified hydraulic parameter. In other words: the crest level, opening height, pump capacity or valve opening is a function of one of the following parameters:

- ◇ Water level at a specified measurement station
- ◇ Discharge at a specified measurement station

And, only for the **River Flow** module:

- ◇ head difference over a hydraulic structure(perhaps another structure than the controlled one!);
- ◇ flow velocity at a specified measurement station;
- ◇ flow direction at a specified measurement station;
- ◇ pressure difference over a structure or compound structure member (perhaps another structure than the controlled one!).

The computation procedure is straightforward. First, the value of the hydraulic argument at the previous time step is determined. Next, the control parameter is computed by interpolation in the controller table. In case of the discharge a user specified time lag with respect to the current time can be taken into account.

Figure 1 Example of hydraulic controller

Figure 6.64: *Example of hydraulic controller*

If the control frequency of an hydraulic controller is specified zero, the control parameter will be updated only at the start of an active period. A trigger should be used to activate the hydraulic controller

In case the head difference Δh over a structure is used as controller, the following formula will be used:

$$\Delta h = h_1 - h_2 \tag{6.249}$$

In case the pressure difference ΔP over a structure is used as controller, the following formula will be used:

$$\Delta P = \frac{1}{2} g(\rho_1 (h_1 - z_s)^2 - \rho_2 (h_2 - z_s)^2) \tag{6.250}$$

This is in fact the difference between the pressure triangles with respect to the crest level z_s on both sides.

where

h_1	water level at structure facing beginning of the branch;
h_2	water level at structure facing end of the branch;
z_s	crest level of structure;
ρ_1	water density at structure facing beginning of the branch;
ρ_2	water density at structure facing end of the branch.

6.4.1.3 Interval controller

The interval controller can be used to operate a structure in such a way that a specified hydraulic parameter is maintained. The controlled parameter can be the water level at a specified measurement station in the model, the discharge at a specified measurement station in the model, or the sum of up to five discharges at specified measurement stations in the model .

The user defines the following input for the interval controller:

◇ Control parameter (crest level, crest width, gate lower edge level)
◇ Control frequency (see controller).
◇ A selection of the controlled parameter (water level or discharge at the measurement station).

- ◇ Setpoints for the controlled parameter (constant or tabulated function of time).
- ◇ The so called ”when below deadband” and ”when above deadband” control values. These “$<$ *Deadband*” and “$>$ *Deadband*” values are the extreme values or upper and lower boundaries allowed for the control parameter (e.g. crest level) (see also River Flow Controller options). The desired direction is also determined from these values. (upward or downward of the control parameter (e.g. crest level).
- ◇ A selection of the deadband type:
  - □ fixed deadband: enter a value
  - □ deadband as a percentage of the discharge: you must enter percentages and maximum and minimum deadband values
- ◇ selection of the controller interval type (or maximum allowable change in value of control parameter):
  - □ fixed interval: you must enter fixed interval value of the controlled parameter
  - □ check velocity: enter the controller velocity or speed in m/s (positive values only)

The value of the control parameter ν_s is computed as follows:

$$\nu_s = \begin{cases} \nu_{s,old} + d\Delta\nu_s & \text{if } e < -\frac{1}{2}D \\ \nu_{s,old} & \text{if } -\frac{1}{2}D \leq e \leq \frac{1}{2}D \\ \nu_{s,old} - d\Delta\nu_s & \text{if } e > \frac{1}{2}D \end{cases} \tag{6.251}$$

where:

D Deadband (in $[m]$ in case of water level, and in $[m^3/s]$ in case of discharge)

e Deviation of the controlled variable, e = setpoint - actual measured (computed) value

d Control direction (1 or -1). We define control values a and b:

- ◇ Define a as the value of the lower boundary of the control parameter. (User interface: ”when below deadband”)
- ◇ Define b as the value of the upper boundary of the control parameter. (User interface: ”when above deadband”)

The control direction is positive if $a > b$ and negative if $b > a$.

ν_s Maximum change of control structure parameter, $\nu_s = v \times dt$

$v =$ maximum change of control structure parameter per second
$dt =$ timestep in seconds

So, the rough algorithm is as follows:

if ’interval controller’ present, then

- ◇ determine value at the previous time step of controlled parameter
- ◇ compute deviation of the value of the controlled parameter at the previous time step from the setpoint
- ◇ compute the value of the control parameter (according to the algorithm described)
- ◇ check if the control parameter is within its extreme values (if not: set the extreme value). The extreme values are the allowed value for the control parameter (e.g. crest level) and are equal to “$<$ Deadband (or when below deadband)” and “$>$ Deadband (or when above deadband)” values.

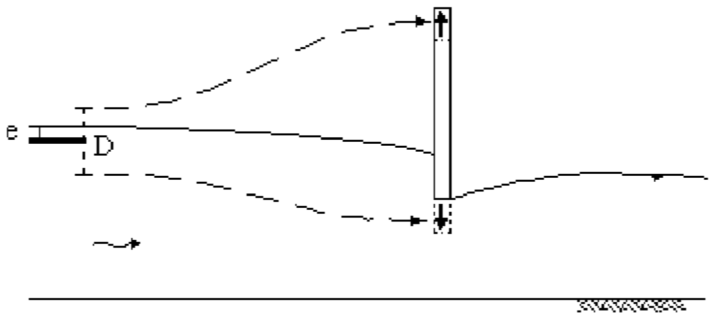


Figure 1 Example of interval controller

Figure 6.65: *Example of interval controller*

The interval controller is not a very advanced type of controller. It is, for instance, sensitive to instabilities, particularly if the controller velocity, control frequency or deadband are not selected properly. Also the control history (before the last time step) is not taken into account to determine the control parameter.

A more advanced controller (the PID controller) is available as well. The PID controller takes the control history into account, thus enabling a more appropriate behaviour of the control parameter.

6.4.1.4 PID controller

The PID-controller can be used to operate a structure in such way, that a specified hydraulic parameter (e.g. water level or discharge) is maintained. The PID-controller consist of a Proportional, Integrating and Differentiating part. The controlled parameter can be the water level or the discharge at a specified measurement station in the model. The user defines the following input for the PID controller:

- ◇ control parameter (crest height, width, gate lower edge level)
- ◇ control frequency (see controller)
- ◇ a selection of the controlled parameter (water level or discharge at the measurement station). For river flow, also the sum of discharges adding up to five measuring stations.
- ◇ setpoints for the controlled parameter (constant or tabulated function of time)
- ◇ initial value of the controlled parameter; note that the initial control value should preferably be the same as for the structure definition
- ◇ maximum and minimum value of the control parameter (e.g. control parameter can be between fixed limits)
- ◇ maximum change of control structure parameter per second ($v_{\max}$ in m/s)
- ◇ Proportional gain factor K_p (determines the reaction time of the controller)
- ◇ Integral gain factor K_i (reduces the standard deviation induced by K_p)
- ◇ Differential gain factor K_d (provides damping in the controller)
- ◇ A control (Update) frequency which may be used to filter out high frequency oscillations in the input signal, but which must be sufficiently small to reproduce the relevant waves in the controlled variable.

The new value of the control parameter $w(t)$ is computed as follows:

1 First compute the correction value

$$u(t) = K_p^{'}e(t) + K_i^{'}\sum_{t=0}^{t} e(t) + K_d^{'}(e(t) - e(t-1)) \tag{6.252}$$

$$K_p^{'} = k_1 K_p, \qquad K_i^{'} = k_2 K_i, \qquad K_d^{'} = k_3 K_d \tag{6.253}$$

2 Perform this correction value to the control parameter $w(t)$

$$w(t) = w(t-1) + u(t) \tag{6.254}$$

In which:

K_p, K_i, K_d	Coefficient of the PID-controller.
k_j	$j = 1, 2, 3$. Conversion factor between the controll parameter (e.g. discharges [m^3/s]) and the controlled parameter (e.g. water levels [m])
$e(t)$	deviation of the controlled parameter $y(t)$, $e(t) = y_{sp}(t) - y(t)$
$u(t)$	correction factor on the control parameter, e.g. $e(t) \equiv 0 \Rightarrow u(t) = 0 \Rightarrow w(t)$ is unchanged.

Remark:

- Equation (6.254) is slightly different from a the PID-controller as reported in Åström and Hägglund (1995) and as report in RTC-Tools TRM (2012).

The algorithm to compute the control parameter is as follows:

- determine value of controlled parameter at previous time level
- compute deviation of this value from the setpoint
- compute change of deviation from the setpoint ($e = e(t) - e(t-1)$)
- compute summation of deviation from the setpoint
- compute the value of the control parameter $w(t)$
- check if the computed change of the control parameter $\Delta w = w(t) - w(t-1)$ is within the user-defined limitations

The gain factors K_p, K_i, K_d must be calibrated for optimal performance of the PID controller. See section B.6

6.4.1.5 Relative time controller

The relative time controller can be used to specify the controlled parameter as a function of time, where the time (in seconds) is given relative to the moment that the controller is activated for the first time by a trigger. When the controller is activated for the first time, the relative controller table is made absolute (= computational time + relative time), thereafter the controller starts at the top of the table and continues downward until it is turned off by the trigger. The controller table will remain absolute during the user-defined so called Start period. In case the controller is activated after this start period has passed, the controller table will be made absolute again. Start period $= 0$, means that the controller table is made absolute each and every time that the controller is triggered. In case the user defined value for $d(\text{value})/dt$ is too small to allow for the in the controller Table defined changes in control parameter, SOBEK will divert from these defined controller parameter values in such way as to best fit the overall controller table. $d(\text{value})/dt = 0$, means that there is no restriction in change in controller parameter over one time step. When it branches the end of the table, the value of the controlled parameter is kept constant at the last value.

6.4.1.6 Relative from value (time) controller

The relative time controller can be used to specify the controlled parameter (e.g. crest level) as a function of time, where the time (in seconds) is given relative. The difference with the relative time controller is the fact that when activated for the first time (or after start period has passed, see here after), the controller does not start at the top of the controller table but at the row that coincides with the actual value of the structure controlled parameter (e.g. crest level) and continues from there on downwards in the controller table. The advantage of the relative from value (time) controller is that if used cleverly discontinuities in structure controlled parameter can be avoided.

The first time that the relative-from-value (time) controller is activated, the controller table will be made absolute (= computational time + relative time), the controller table will remain absolute during the so called start period. Hence in case the controller is de-activated and activated again within the start period, the controller table remains absolute. In case the controller is activated after the start period has passed, the controller table will be made absolute for the then yielding computational time again. Start period = 0, means that the controller table is made absolute each and every time that the controller is triggered. In case the user defined value for $d(\text{value})/dt$ is too small to allow for the in the controller table defined changes in controlled parameter (e.g. crest level), SOBEK will divert from these defined controller parameter values in such way as to best fit the overall controller table. A value of $d(\text{value})/dt = 0$, means that there is no restriction in change in controlled parameter over one computational time step. When it branches the end of the Table, the value of the controlled parameter is kept constant at the last value.

6.4.1.7 Time controller

The time controller is the most simple and straight forward controller. The control parameter (e.g. crest level, opening height, pump capacity, valve opening) is given explicitly as a (tabulated) function of time.

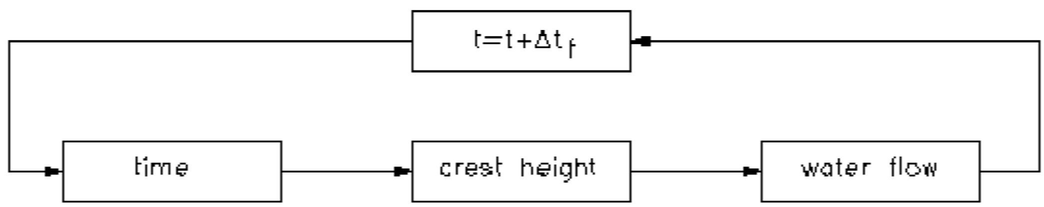


Figure 1 Example of time controller

Figure 6.66: *Example of time controller*

6.4.2 Triggers

A hydraulic structure can be adjusted by a controller. Each controller in SOBEK can be activated or de-activated by a trigger, which can have the value ON or OFF. Also a combination of triggers can activate a controller, while more than one controller can be activated by one trigger or trigger combination. In case no trigger is assigned to a controller, this controller will be active during the entire computation.

Three different trigger types are available:

1. time trigger;
2. hydraulic trigger;
3. Time & Hydraulic trigger (combined trigger).

Moreover, it is possible to perform a logical "and" or logical "or" function on the combination of time trigger and hydraulic trigger.

Whether a trigger is ON or OFF is determined as follows:

```
if AND/OR period defined then
    determine AND/OR period (AND/OR)
    evaluate time trigger (ON/OFF)
    evaluate hydraulic trigger (ON/OFF)
    trigger = OFF
if (AND and (time trigger = ON and hydraulic trigger = ON)) then
    trigger = ON
if (OR and (time trigger = ON or hydraulic trigger = ON)) then
    trigger = ON
else if time trigger then
    evaluate time trigger (ON/OFF)
else if hydraulic trigger then
    evaluate hydraulic trigger (ON/OFF)
```

6.4.2.1 Combinations of triggers

Up to four triggers can be combined to activate or de-activate a controller. When the user specifies more than one trigger, he must specify how the triggers work together by specifying the relationship between the triggers with logical and-or statements. Evaluation of AND-OR statements are according to the usual logical rules. This means that AND statements are evaluated before OR statements.

This results in the following:

◇ the combination of triggers 'trig1 and trig2', 'trig12', is evaluated as follows:
`if` $(trig1 == ON$ `and` $trig2 == ON)$ `then`
    $trig12 = ON$
`else`
    $trig12 = OFF$
`endif`

◇ the combination of triggers 'trig1 or trig2', '$trig12$', is evaluated as follows:
`if` $(trig1 == ON$ `or` $trig2 == ON)$ `then`
    $trig12 = ON$
`else`
    $trig12 = OFF$

endif

◇ the combination of triggers 'trig1 and trig2 and trig3', '$trig123$', is evaluated as follows:

if $(trig1 == ON$ and $trig2 == ON$ and $trig3 == ON)$ then
$trig123 = ON$
else
$trig123 = OFF$
endif

◇ the combination of triggers 'trig1 and trig2 or trig3', '$trig123$', is evaluated as follows:

if $((trig1 == ON$ and $trig2 == ON)$ or $trig3 == ON)$ then
$trig123 = ON$
else
$trig123 = OFF$
endif

◇ the combination of triggers 'trig1 or trig2 and trig3', '$trig123$', is evaluated as follows:

if $((trig1 == ON$ or $trig2 == ON)$ and $trig3 == ON)$ then
$trig123 = ON$
else
$trig123 = OFF$
endif

6.4.2.2 Time trigger

To determine if a time trigger is ON or OFF, a user-defined time table, is used. In the time table periods are defined during which the trigger is either ON or OFF.

6.4.2.3 Hydraulic trigger

To determine if a hydraulic trigger is ON or OFF, first the hydraulic parameter will be calculated or the gate lower edge level will be retrieved. After this
SOBEK checks if the computed value exceeds a user defined threshold value.

Eight hydraulic trigger types are possible:

1 water level at a specified location
2 discharge at a specified location
3 gate lower edge level at a hydraulic structure
4 crest level at a hydraulic structure
5 crest width at a hydraulic structure
6 the water level in retention area
7 head difference at a hydraulic structure
8 force difference per unit width at a hydraulic structure

The head difference at a structure used by a trigger is defined as:

$$\Delta h = h_1 - h_2 \tag{6.255}$$

The force difference (ΔF) per unit width over a structure is defined as:

$$\Delta F = \frac{1}{2} g(\rho_1 (h_1 - z_s)^2 - \rho_2 (h_2 - z_s)^2) \tag{6.256}$$

where

g	acceleration due to gravity;
h_1	water level at structure, facing beginning of branch;
h_2	water level at structure, facing end of branch;
z_s	crest level of structure;
ρ_1	water density at structure, facing beginning of branch;
ρ_2	water density at structure, facing end of branch

If *gate lower edge level*, *crest level* or *crest width* is chosen two options are available:

1 the value is checked against the threshold
2 the direction of variation (increase or decrease of the value) can be checked

The following procedure is used:

◇ determine hydraulic trigger type
◇ determine location
◇ determine threshold value and whether it should be triggered on a "higher" value or a "lower" value

```
trigger = OFF
if 'hydraulic trigger type = water level' then
    if 'trigger on higher value'
        if 'h(location) > value'
            trigger = ON
    if 'trigger on lower value'
        if 'h(location) < value'
            trigger = ON
elseif 'trigger type = head difference' then
    head difference = h_left − h_right of structure
    if 'trigger on higher value'
        if 'head difference > value'
            trigger = ON
    if 'trigger on lower value'
        if head difference < value
            trigger = ON
endif
```

6.4.2.4 Time & Hydraulic triggers

The Time & Hydraulic trigger is actually a combination of a Hydraulic trigger with a Time trigger. This can be applied in “AND” or in “OR” mode.

◇ AND mode means: Time & Hydraulic trigger value is ON if both the Hydraulic trigger and the Time trigger are ON.
◇ OR mode means: Time & Hydraulic trigger value is ON if either the Time trigger or the Hydraulic trigger is ON, but also if both Time and Hydraulic triggers are ON.

6.5 Hydrology (Rainfall Runoff modules)

6.5.1 SOBEK-Rural RR (Rainfall Runoff) concept

6.5.1.1 Alpha reaction factor

The reactionfactor α is an important parameter in the De Zeeuw-Hellinga equation which is used in SOBEK to calculate the different components of the groundwater/subsurface flow:

$$q_t = q_{t-1}e^{-\alpha\Delta t} + (I + S)(1 - e^{-\alpha\Delta t}) \quad (6.257)$$

q_t	specific discharge at time t [m/d]
q_{t-1}	specific discharge at time $t-1$ [m/d]
Δt	time step [d]
α	reaction factor [$1/d$]
I	infiltration [m/d]
S	seepage (percolation) [m/d]

In SOBEK, the De Zeeuw-Hellinga equation is used to calculate the following components of flow:

- ◇ groundwater drainage (towards drainpipes or channels);
- ◇ surface run-off;
- ◇ infiltration from open water.

For each process, the user must define specific reaction factors.

Reaction factor groundwater drainage
In SOBEK, the ground can be divided into different ground layers, each with it's own a. The total specific discharge is calculated by applying the De Zeeuw-Hellinga equation to all layers, and summing the result (see Figure 6.67)

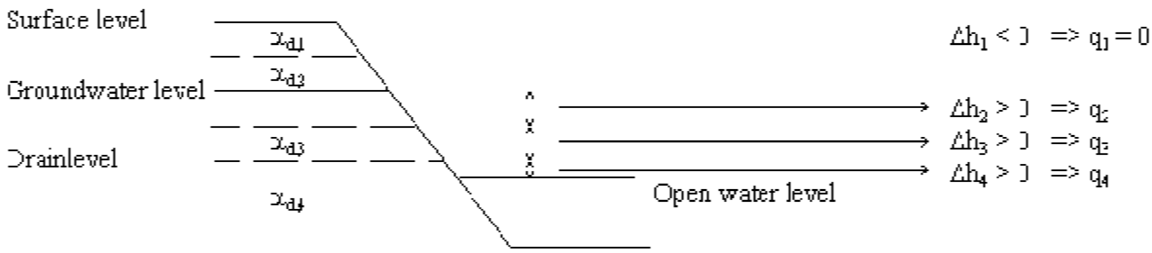


Figure 6.67: *Drainage levels*

Usually, the division between the ground layers is placed at the level of the drains, which are then simulated by giving the above drain level ($\alpha_{d,1}$ to $\alpha_{d,3}$ in the figure) a much higher value than the a below drain level ($\alpha_{d,4}$ in Figure 6.67).

To obtain indications for the values of the reaction factor one can:

- ◇ use the values in the Table below (Vademecum, 1988):

α	Discharge type
100–200	Surface runoff from steep slopes
1–10	Surface runoff from soils with impervious subsoil
0.3–0.7	Drainage discharge from well-drained agricultural soil
0.03–0.07	Discharge from grassland without drainage system

- ◇ Measure the decrease of discharge in time. This only can be done after a period of (heavy)rain, and no additional precipitation, and no (or very little) evaporation.

◇ Derive the α value from

$$q = \alpha \mu \Delta h \tag{6.258}$$

and

$$q = \frac{\Delta h}{W} \tag{6.259}$$

and equals

$$\alpha = \frac{1}{\mu W} \tag{6.260}$$

q	specific discharge $[m/d]$
Dh	groundwater level above drainage depth or open water level $[m]$
α	reaction coefficient $[1/d]$
m	storage coefficient $[-]$
W	drainage resistance $[d]$

Reaction factor surface run-off

When the groundwater level branches the surface level or the precipitation excess exceeds the infiltration capacity, water is stored on land. When the storage on land is filled in a SOBEK-RR unpaved node the surface runoff process starts.

Therefore, the user must define a reaction factor α_s. The above table gives indications for the values of this reaction factor.

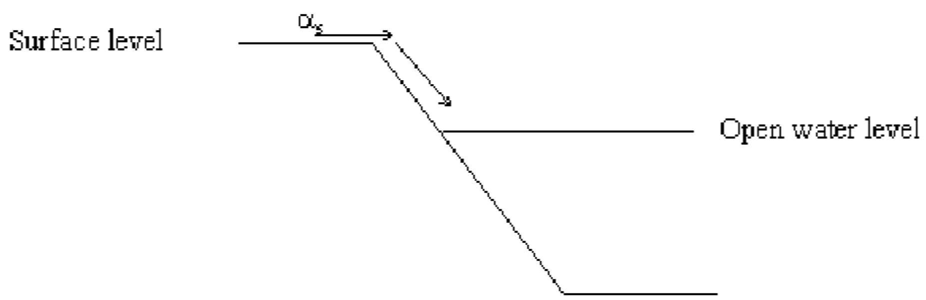


Figure 6.68: *Surface run-off*

Reaction factor infiltration from open water

In dry periods the groundwater level can decrease. When the groundwater level is lower than the open water level, infiltration from the open water occurs. In SOBEK-RR, this process is described by the modified De Zeeuw-Hellinga equation. Again, the user must define a specific reaction factor, α_i

6.5.1.2 Capillary rise

Capillary rise describes the unsaturated flow in the subsoil from the groundwater to the root zone. Capillary rise occurs in case of a water deficit in the root zone. Water deficit is defined as a water content less than equilibrium moisture storage. If the water content in the root zone exceeds equilibrium the excess water percolates to the groundwater. The excess water is the potential root zone volume minus the equilibrium moisture content. To assess whether capillary rise or percolation will occur over a time step, the *potential root zone volume* is determined from the net precipitation and the evapotranspiration, based on the soil moisture storage in the root zone at the previous time step:

$$V'(t) = V(t - \Delta t) + (P_n - E)\Delta t \tag{6.261}$$

where:

$V'(t)$	potential root zone volume at time t [m]
$V(t - \Delta t)$	root zone volume after the previous time step [m]
P_n	net precipitation [m/d]
E	evapotranspiration [m/d]
Δt	time step [d]

In case of a water deficit in CAPSIM the capillary rise flux is calculated in 2 steps. First the potential capillary rise flux is calculated. The potential capillary rise flux depends on the groundwater level, the soil physical unit and the root zone thickness:

$$q_{pot} = f(s, d_g, d_r) \tag{6.262}$$

with:

q_{pot}	Potential capillary rise [mm/d]
s	Soil physical unit [-]
d_g	Depth groundwater level [m]
d_r	Root zone thickness [m]

The potential capillary rise flux is calculated with the 1-D, steady state simulation model CAPSEV (Wesseling, 1991), assuming $pF = 3$ in the root zone. The potential capillary rise fluxes are tabulated.

Secondly SOBEK-CAPSIM uses the potential capillary rise flux, the actual moisture storage in the root zone and the equilibrium moisture storage to calculate the actual capillary rise flux, according to:

$$q_{act} = \begin{cases} q_{pot}\frac{v_{eq} - v_{act}}{v_{eq} - v_{pF3}} & v_{act} > v_{pF3} \\ q_{pot} & v_{act} < v_{pF3} \end{cases} \tag{6.263}$$

where:

q_{act}	Actual capillary rise flux [mm/d]
v_{eq}	Equilibrium soil moisture content [m]
v_{act}	Actual soil moisture content [m]

Equilibrium soil moisture content v_{eq}

The moisture storage of the root zone at equilibrium condition is calculated with the function:

$$V_{eq} = f(s, d_g, d_r) \tag{6.264}$$

where:

V_{eq}	the moisture storage of the root zone at equilibrium condition [m]
s	soil physical unit [-]
d_r	thickness of root zone [m]
d_g	depth of the groundwater level [m]

Soil moisture content at $pF = 3$***,*** v_{pF3}

The moisture storage of the root zone at $pF = 3$ is calculated assuming steady conditions and a groundwater level of 10 m minus soil surface.

6.5.1.3 Crop factors agricultural crops

The fixed input file with crop names and crop factors as a function of time. 1 year of data is enough, since it is assumed that the crop factors are constant over the years; the variation is taken into account in the reference evaporation data and not in the crop factors.

The header of the file contains the number of crops and crop names.

```
*Number of crops
16
*Names
’1 grass ’
’2 corn ’
’3 potatoes ’
’4 sugarbeet ’
’5 grain ’
’6 miscellaneous ’
’7 non-arable land ’
’8 greenhouse area ’
’9 orchard ’
’10 bulbous plants ’
’11 foliage forest ’
’12 pine forest ’
’13 nature ’
’14 fallow ’
’15 vegetables ’
’16 flowers ’
*Year/Month/Day/Cropfact 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16
 0000 1 1 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 2 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 3 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 4 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 5 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 6 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 7 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 8 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 9 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 10 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 11 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 12 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
 0000 1 13 0.95 1.00 1.00 1.00 1.00 0.95 1.00 0.00 1.00 0.00 0.90 1.20 0.95 1.00 1.00 0.00
etc.
```

6.5.1.4 Crop factors open water

The fixed input file with open water evaporation factors is a function of time. 1 year of data is enough; it is assumed that the evaporation factors are the same for all years.

```
*Surface water
1
*Names
’0.0 Surface water ’
*Year/Month/Day/Cropfact 1
 0000 1 1 0.50
 0000 1 2 0.50
 0000 1 3 0.50
 0000 1 4 0.50
```

```
 0000 1 5 0.50
 0000 1 6 0.50
 0000 1 7 0.50
 0000 1 8 0.50
 0000 1 9 0.50
 0000 1 10 0.50
 0000 1 11 0.50
 0000 1 12 0.50
 0000 1 13 0.50
etc.
```

6.5.1.5 De Zeeuw-Hellinga drainage formula

To simulate the flow of groundwater towards the drainage system (i.e. drain pipes and/or channels) the modified equation of De Zeeuw-Hellinga is used in the Rainfall-Runoff module. The theory behind this equation is treated here.

The head loss of groundwater flowing to a drain can be subdivided in head loss caused by:

- ◇ vertical flow;
- ◇ horizontal flow;
- ◇ radial flow nearby the drain/channel;
- ◇ entrance in the drain/channel.

In the equation of De Zeeuw-Hellinga it is assumed that the head loss is mainly caused by radial flow and entrance loss. The head loss by horizontal flow and vertical flow is neglected.

This equation is based on a reservoir containing a volume of ground. The reservoir discharges through a capillary tube. The discharge depends on the difference in pressure head. Thus:

$$Q = c\Delta h \tag{6.265}$$

Q	discharge $[m^3/d]$
c	proportional coefficient $[m^2/d]$
Δh	difference in pressure head $[m]$

and

$$q = \frac{Q}{A} = c_2 h = \alpha\mu h \tag{6.266}$$

q	specific discharge $[m/d]$
Q	discharge $[m^3/d]$
A	area $[m^2]$
c_2	proportional coefficient $[1/d]$
h	groundwater level above drainage depth $[m]$
α	reaction coefficient $[1/d]$
μ	storage coefficient

The water balance for time dt is:

$$(I + S - q)\Delta t = \mu\Delta h \tag{6.267}$$

I	infiltration $[m/d]$
S	seepage (percolation) $[m/d]$
t	time $[d]$

This equals:

$$\Delta t = \frac{\mu}{I + S - \alpha\mu h}\Delta h \tag{6.268}$$

and eliminating the integration constant ($t = 0$, $q = q_0$) they lead to the equation of De Zeeuw-Hellinga:

$$q_t = q_0 e^{-\alpha t} + (I + S)(1 - e^{-\alpha t}) \tag{6.269}$$

q_t specific discharge at time t [m/d]
q_0 specific discharge at time 0 [m/d]

With this equation the groundwater flow to a drain can be calculated for a constant $I + S$. When $I + S$ is not constant, the $I + S$ series can be seen as a series of constant $I + S$, positive and negative. For each constant $I + S$ in the series the q_t can be calculated, with t depending on the start of the component. Because the De Zeeuw-Hellinga equation is linear, superposition may be applied. It is easier though to calculate the q_t for several short successive periods wherein $I + S$ is constant. This results in the following modified equation:

$$q_t = q_{t-1} e^{-\alpha\Delta t} + (I + S)(1 - e^{-\alpha\Delta t}) \tag{6.270}$$

q_{t-1} specific discharge at time $t - 1$ [m/d]
Δt time step [d]

For each time step infiltration and seepage must be constant.

The total volume of water flowing to the drain can be calculated by integrating the equation of De Zeeuw-Hellinga and multiplying it with the area:

$$V_t = A \int_0^{\Delta t} q_t \, dt \tag{6.271}$$

V_t Volume of groundwater flow to drain at time t [m^3]

V_t becomes:

$$V_t = A \int_0^{\Delta t} q_t \, dt \tag{6.272}$$

$$= A \left[-\frac{1}{\alpha} q_0 e^{\alpha t} + (I + S)(t + \frac{1}{\alpha} e^{-\alpha t}) \right]_0^{\Delta t} \tag{6.273}$$

$$= A \left[-\frac{1}{\alpha} q_0 e^{\alpha\Delta t} + (I + S)(\Delta t + \frac{1}{\alpha} e^{-\alpha\Delta t}) + \frac{1}{\alpha} q_0 - \frac{1}{\alpha}(I + S) \right] \tag{6.274}$$

$$= A \left[\frac{q_0 - (I + S)}{\alpha}(1 - e^{-\alpha\Delta t}) + (I + S)\Delta t \right] \tag{6.275}$$

and

$$q_0 = \alpha\mu\Delta h_0 \tag{6.276}$$

Substituting gives:

$$V_t = \frac{A\alpha\mu\Delta h - A(I + S)}{\alpha}(1 - e^{-\alpha\Delta t}) + A(I + S)\Delta t \tag{6.277}$$

The average discharge $Q_{t,average}$ $[m^3/d]$ is computed by

$$Q_{t,average} = \frac{V_t}{\Delta t} \tag{6.278}$$

Thus the average discharge in a time step to the drain as is used by the Rainfall-Runoff module is:

$$Q_{t,average} = \frac{A\alpha\mu\Delta h - A(I+S)}{\alpha\Delta t}(1 - e^{-\alpha\Delta t}) + A(I+S) \tag{6.279}$$

This is the so-called modified equation of De Zeeuw-Hellinga. Two important differences from the 'normal' equation of De Zeeuw-Hellinga:

- the storage coefficient μ is explicit, and
- the pressure head Δh is in the Rainfall-Runoff module calculated per time step as the difference between the groundwater level and the open water level.

6.5.1.6 `DrainageDeltaH` option

```
DrainageDeltaH=-1 ! -1=parallell systems (default), 0=stacked system;
```

Given the definition of the drainage levels, and the Hellinga-de Zeeuw alfa reaction factors α, or the Ernst drainage resistances γ, there are two options for computing the relevant heads on which the α factors or drainage resistances are applied to.

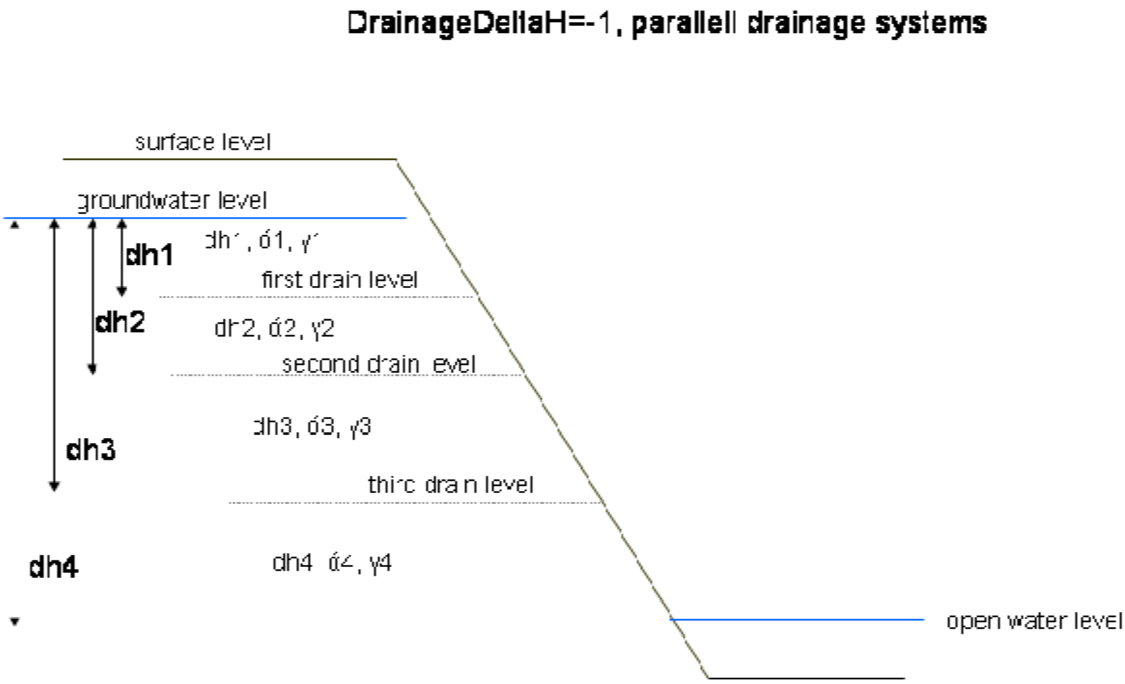


Figure 6.69: *Parallell drainage systems (default:* `DrainageDeltaH`*=-1)*

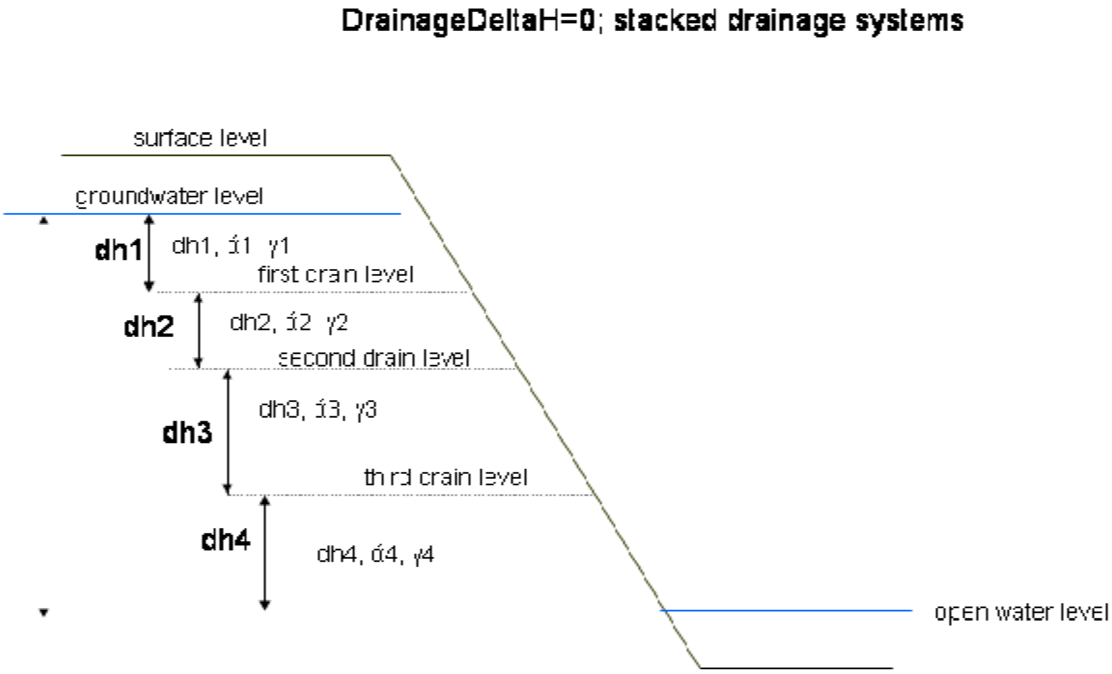


Figure 6.70: *Stacked drainage systems (*`DrainageDeltaH`*=0)*

See also: <Delft_3B.Ini> as described in SOBEK input file formats.

6.5.1.7 Dry Weather Flow (DWF)

The DWF or dry weather flow can be taken into account at the RR-paved node. Also the DWF is part of the formulation in the RR-NWRW node, Nationale Werkgroep Riolering en Waterkwaliteit (NLingenieurs, 1978).

The dry weather flow represents return flow from domestic use, like showers, washing, flushing the toilets, etc.

The dry weather flow can be specified in one of the following ways:

- as a constant flow $[m^3/s]$ during the whole day
- as a variable flow $[m^3/s]$ for 24 hours; 1 value for each hour.
- as a constant flow $[m^3/s]$ per day per person, multiplied with the number of persons
- as a variable flow $[m^3/s]$ per hour per person, multiplied with the number of persons

For paved area, the dry weather flow is discharged into the DWF sewer in case of a separated or improved separated system. In case of a mixed system it is discharged into the mixed sewer system, where it is mixed with sewer inflow due to rainfall.

For a NWRW node the dry weather flow is added to the computed sewer inflow with the NWRW rainfall-runoff model.

6.5.1.8 Equal filling controller

Given initial, target and maximum allowable upstream and downstream levels, the controller tries to operate the weir such that the upstream and downstream open water nodes will experience the same filling percentage.

The filling percentage is defined in terms of water levels as:

$$(\text{actual level} - \text{target level})/(\text{maximum allowable level} - \text{target level}) \times 100\,\%. \tag{6.280}$$

This means that the controller will try to achieve similar exceedance patterns of the target levels, taking into account the maximum allowable levels. In calculating the flow which results in equal filling percentages, a linear relation between water level and volume is assumed. For open waters with a constant area this is a valid assumption. However, if the open water area increases with increasing level, the procedure will not result in exactly equal filling percentages.

The calculated flow to branch equal filling percentages is limited by a maximum allowed flow Q_{max}. This maximum flow Q_{max} is determined by:

$$Q_{max} = \text{QMaxAtTargetLevel} + \text{FillingUpstream} \times (\text{QMaxAtMaxLevel} - \text{QMaxAtTargetLevel}). \tag{6.281}$$

The maximum flows *QMaxAtTargetLevel* and *QMaxAtMaxLevel* are user defined input data for the equal filling controller.

Furthermore, the flow Q_{max} is limited (taking into account other calculated flows from previous iterations) such that the upstream water level does not fall below target level.

6.5.1.9 Ernst drainage formula

One of the options of modelling the drainage from unpaved area towards open water is the Ernst formulation (Ernst, 1978). Other options available are the De Zeeuw-Hellinga formulation and the Krayenhoff van de Leur formulation.

The following figure illustrates the principles of the Ernst method.

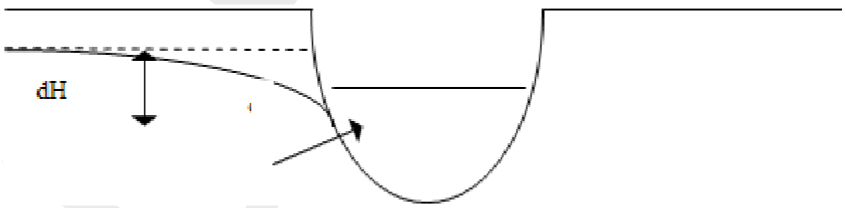


Figure 6.71: *Principles of Ernst method*

The equation reads:

$$q = \frac{dH}{\gamma f} \tag{6.282}$$

where:

q	drainage $[m/d]$
dH	difference between groundwater level and drainage basis $[m]$
γ	drainage resistance $[d]$
f	factor (0.65 - 0.85 (Ernst, 1978)), depending on the shape of the groundwatertable $[-]$

The f factor can not be entered via the User Interface of SOBEK. Instead, the user has to take this factor into account in the drainage resistance.

6.5.1.10 Evaporation (when using capsim)

The actual evapotranspiration ET_{act} is calculated as:

$$ET_{act} = \alpha_E E_r \tag{6.283}$$

with

$$\alpha_E = f\left(\frac{V}{V_{eq}}\right) \tag{6.284}$$

where:

α_E	relative evapotranspiration factor $[-]$
V	actual soil moisture storage in the root zone $[m]$
V_{eq}	equilibrium soil moisture storage in the root zone $[m]$
E_r	potential evapotranspiration $[m]$

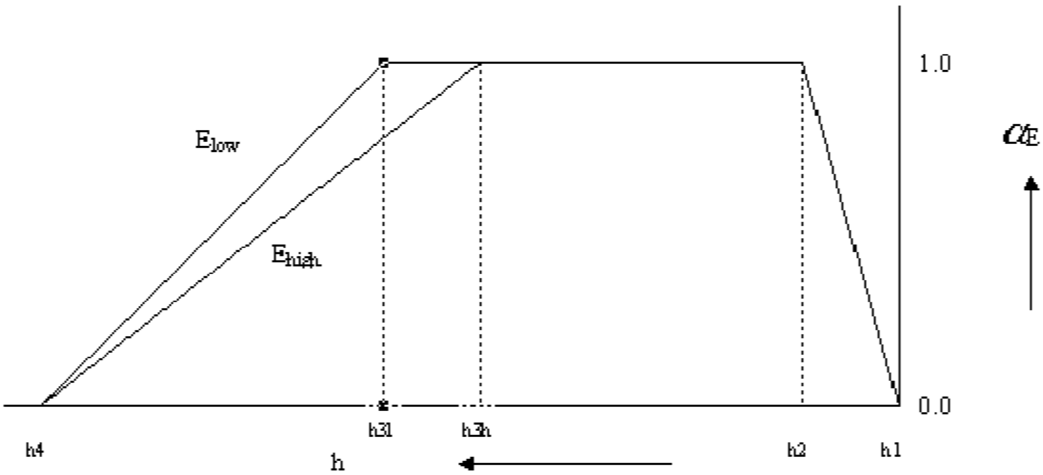


Figure 6.72: *Reduction coefficient for root water uptake, α_E, as a function of soil water pressure head h (Feddes et al., 1978).*

Water uptake by roots is zero at soil water pressure h_4 which is assumed to be the wilting point, see Figure 6.72. Soil water pressure h_3 is called the reduction point. In between the wilting point and the reduction point the potential evapotranspiration rate is linearly reduced. The location of the reduction point depends on the potential evapotranspiration. If the potential evapotranspiration is less than or equal to 1 mm/d then the curve E_{low} is used. If the potential evapotranspiration is equal to or greater than 5 mm/d then the curve E_{high} is used. In between SOBEK-CAPSIM linearly interpolates between the reduction curves E_{low} and E_{high}.

Between soil water pressures h_2 and h_3 the evapotranspiration is at maximum ('potential'). As a result of oxygen deficiency in the root zone, water uptake is hampered for some crops between soil water pressures h_2 and h_1. SOBEK-CAPSIM does not take this reduction into account ($\alpha_E = 1$ between soil water pressures h_2 and h_1).

SOBEK-CAPSIM doesn't compute the soil waterpressure head directly, but uses the relative root zone storage V/V_{eq}. The relative root zone storages for h_4, h_{3l}, h_{3h}, h_2 and h_1 are calculated by CAPSEV and tabulated in SOBEK-CAPSIM. Then Figure 6.72 can be better explained by:

$$\alpha_E = 0 \quad \text{when} \quad 0 \leq V/V_{eq} < V/V_{eq}(h_4) \tag{6.285}$$

$$0 \leq \alpha_E \leq 1 \quad \text{when} \quad V/V_{eq}(h_4) \leq V/V_{eq} < V/V_{eq}(h_{3l} \text{ or } h_{3h}) \tag{6.286}$$

$$\alpha_E = 1 \quad \text{when} \quad V/V_{eq} \geq V/V_{eq}(h_{3l} \text{ or } h_{3h}) \tag{6.287}$$

6.5.1.11 Evapo(transpi)ration

Open water

For open water, related to Makkink evapotranspiration, the following 'crop' factors are used. These values are based on Hooghart and Lablans (1988).

Decade[1)]	**Value**	**Decade**	**Value**	**Decade**	**Value**
jan-01	0.50	may-01	1.30	sep-01	1.17
jan-02	0.50	may-02	1.30	sep-02	1.17
jan-03	0.70	may-03	1.30	sep-03	1.17
feb-01	0.80	jun-01	1.31	oct-01	1.00
feb-02	1.00	jun-02	1.31	oct-02	0.90
feb-03	1.00	jun-03	1.31	oct-03	0.80
mar-01	1.20	jul-01	1.29	nov-01	0.80
mar-02	1.30	jul-02	1.27	nov-02	0.70
mar-03	1.30	jul-03	1.24	nov-03	0.60
apr-01	1.30	aug-01	1.21	dec-01	0.50
apr-02	1.30	aug-02	1.19	dec-02	0.50
apr-03	1.30	aug-03	1.18	dec-03	0.50

[1)] Decades are parts of a month, defined as: first 10 days, second 10 days and rest of the month.

Related input file:
crop factor open water

Unpaved areas

The potential evapotranspiration in unpaved areas depends on the vegetation. The standard way to determine the potential evapotranspiration is to multiply the reference evaporation (p.e. Makkink) by a so called crop factor. The crop factor can differ per vegetation, in time and per location on earth. The default crop factors are orientated towards the Dutch situation for a limited number of crops, listed in the table below.

crop nr.	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**	**15**	**16**
decade	**grass**	**corn**	**potatoes**	**sugarbeet**	**grain**	**miscellaneous**	**non-arable land**	**greenhouse area**	**orchard**	**bulbous plants**	**foliage forest**	**pine forest**	**nature**	**fallow**	**vegetables**	**flowers**
jan-01	0.95	1.00	1.00	1.00	1.00	0.95	1.00	0.00	1.00	1.00	0.90	1.20	0.95	1.00	1.00	0.00
jan-02	0.95	1.00	1.00	1.00	1.00	0.95	1.00	0.00	1.00	1.00	0.90	1.20	0.95	1.00	1.00	0.00

crop nr.	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
decade	grass	corn	potatoes	sugarbeet	grain	miscellaneous	non-arable land	greenhouse area	orchard	bulbous plants	foliage forest	pine forest	nature	fallow	vegetables	flowers
jan-03	0.95	0.71	0.71	0.71	0.71	0.95	0.71	0.00	0.71	0.71	0.90	1.20	0.95	0.71	0.71	0.00
feb-01	0.95	0.63	0.63	0.63	0.63	0.95	0.63	0.00	0.63	0.63	0.90	1.20	0.95	0.63	0.63	0.00
feb-02	0.95	0.50	0.50	0.50	0.50	0.95	0.50	0.00	0.50	0.50	0.90	1.20	0.95	0.50	0.50	0.00
feb-03	0.95	0.40	0.40	0.40	0.40	0.95	0.40	0.00	0.40	0.40	0.90	1.20	0.95	0.40	0.40	0.00
mar-01	0.95	0.33	0.33	0.33	0.33	0.95	0.33	0.00	0.33	0.33	1.00	1.20	0.95	0.33	0.33	0.00
mar-02	0.95	0.23	0.23	0.23	0.23	0.95	0.23	0.00	0.23	0.23	1.00	1.20	0.95	0.23	0.23	0.00
mar-03	0.95	0.23	0.23	0.23	0.23	0.95	0.23	0.00	0.23	0.78	1.00	1.20	0.95	0.23	0.23	0.00
apr-01	1.00	0.23	0.23	0.23	0.65	1.00	0.23	0.00	1.04	0.91	1.05	1.20	1.00	0.23	0.23	0.00
apr-02	1.00	0.23	0.23	0.23	0.78	1.00	0.23	0.00	1.04	0.91	1.05	1.20	1.00	0.23	0.52	0.00
apr-03	1.00	0.23	0.23	0.23	0.91	1.00	0.23	0.00	1.04	0.91	1.05	1.20	1.00	0.23	0.65	0.00
may-01	1.00	0.52	0.15	0.52	1.04	1.00	0.15	0.00	1.43	1.04	1.15	1.20	1.00	0.15	0.78	0.00
may-02	1.00	0.52	0.65	0.52	1.04	1.00	0.15	0.00	1.43	1.04	1.15	1.20	1.00	0.15	0.91	0.00
may-03	1.00	0.52	0.91	0.52	1.04	1.00	0.15	0.00	1.43	1.04	1.15	1.20	1.00	0.15	1.04	0.00
jun-01	1.00	0.79	1.05	0.79	1.18	1.00	0.15	0.00	1.57	1.05	1.20	1.20	1.00	0.15	1.18	0.00
jun-02	1.00	1.05	1.05	1.05	1.18	1.00	0.15	0.00	1.57	1.05	1.20	1.20	1.00	0.15	1.18	0.00
jun-03	1.00	1.18	1.18	1.18	1.18	1.00	0.15	0.00	1.57	0.92	1.20	1.20	1.00	0.15	1.18	0.00
jul-01	1.00	1.29	1.16	1.16	1.03	1.00	0.16	0.00	1.68	0.77	1.25	1.20	1.00	0.16	1.03	0.00
jul-02	1.00	1.27	1.14	1.14	0.89	1.00	0.16	0.00	1.65	0.64	1.25	1.20	1.00	0.16	0.76	0.00
jul-03	1.00	1.24	1.12	1.12	0.74	1.00	0.16	0.00	1.61	0.50	1.25	1.20	1.00	0.16	0.16	0.00
aug-01	1.00	1.21	1.09	1.09	0.61	1.00	0.17	0.00	1.33	0.17	1.10	1.20	1.00	0.17	0.17	0.00
aug-02	1.00	1.19	0.83	1.07	0.17	1.00	0.17	0.00	1.31	0.17	1.10	1.20	1.00	0.17	0.17	0.00
aug-03	0.90	1.18	0.83	1.06	0.25	0.90	0.25	0.00	1.18	0.25	1.10	1.20	0.90	0.25	0.25	0.00
sep-01	0.90	1.17	0.70	1.05	0.26	0.90	0.26	0.00	1.17	0.26	1.05	1.20	0.90	0.26	0.26	0.00
sep-02	0.90	1.17	0.26	1.05	0.26	0.90	0.26	0.00	1.17	0.26	1.05	1.20	0.90	0.26	0.26	0.00
sep-03	0.90	1.17	0.26	1.05	0.26	0.90	0.26	0.00	1.17	0.26	1.05	1.20	0.90	0.26	0.26	0.00
oct-01	0.90	0.40	0.30	0.40	0.30	0.90	0.30	0.00	0.30	0.30	1.00	1.20	0.90	0.30	0.30	0.00
oct-02	0.95	0.45	0.44	0.44	0.44	0.95	0.44	0.00	0.44	0.44	1.00	1.20	0.95	0.44	0.44	0.00
oct-03	0.95	0.50	0.50	0.50	0.50	0.95	0.50	0.00	0.50	0.50	1.00	1.20	0.95	0.50	0.50	0.00
nov-01	0.95	0.50	0.50	0.50	0.50	0.95	0.50	0.00	0.50	0.50	0.95	1.20	0.95	0.50	0.50	0.00
nov-02	0.95	0.71	0.71	0.71	0.71	0.95	0.71	0.00	0.71	0.71	0.95	1.20	0.95	0.71	0.71	0.00
nov-03	0.95	0.83	0.83	0.83	0.83	0.95	0.83	0.00	0.83	0.83	0.95	1.20	0.95	0.83	0.83	0.00
dec-01	0.95	1.00	1.00	1.00	1.00	0.95	1.00	0.00	1.00	1.00	0.90	1.20	0.95	1.00	1.00	0.00
dec-02	0.95	1.00	1.00	1.00	1.00	0.95	1.00	0.00	1.00	1.00	0.90	1.20	0.95	1.00	1.00	0.00
dec-03	0.95	1.00	1.00	1.00	1.00	0.95	1.00	0.00	1.00	1.00	0.90	1.20	0.95	1.00	1.00	0.00

Related input file:
crop factor agricultural crops

Remarks:

- ◇ The crop factor file is specified for a limited number of crops and suitable for the Dutch situation or for simular conditions only. In other situations (or other crops) please define your own crop factors. This can be done by changing the input file <../../fixed/cropfact>. We recommend saving the default crop factor file first. We also recommend to save the changed crop factor file at a different location as well, because when installing a new release of SOBEK the crop factor file will be replaced by a new default crop factor file.
- ◇ The values in the table above are based on the Makkink evapotranspiration. The used information comes from the Dutch Cultuurtechnisch Vademecum (3rd edition), Hooghart, Spieksma et. al, Lysimeter results and PAWN (DEMGEN) documentation.
- ◇ The growing season of the specified crops is marked yellow. Outside the growing season the values are the same as the values for fallow (bare soil).
- ◇ Non-arable land is assumed to be the same as fallow.
- ◇ Miscellaneous is assumed to be the same as grass.
- ◇ Nature depends entirely on the kind of nature. Therefore it is assumed to be the same as grass.
- ◇ For greenhouse crops (greenhouse and greenhouse-flowers), the values are set to zero.

For proper use, please use the greenhouse node. Other flowers are assumed to be bulbous plants.
- ◇ For vegetables peas and beans are assumed.
- ◇ For both open water as well as unpaved areas a fatal error (the programm stops) will occur when a crop factor is used with a value lower than 0 or higher than 2.5.
- ◇ When a crop factor with value 0 is used, a warning will be given in the log file (<SOBEK_3B.log>). The programm continues.

The actual evapotranspiration can be determined by using the potential evapotranspiration (default) or by taking into account the reduction for root water uptake as a function of soil water pressure head (using Capsim).

6.5.1.12 Fixed level difference controller

With the fixed level difference controller one can keep the difference between the upstream target level and downstream target level. By increasing downstream water level, SOBEK-RR will increase the crest level. In this way, more storage capacity is created in the upstream area.

The algorithm is given as follows:

User input:

- ◇ Upstream target level TL_{up}
- ◇ Downstream target level TL_{down}
- ◇ Model input, i.e. parameters computed by SOBEK-RR:
- ◇ Upstream level L_{up}
- ◇ Downstream level L_{down}
- ◇ Crest level = $L_{down} + (TL_{up} - TL_{down})$

Model input, i.e. parameters computed by SOBEK-RR:

- ◇ Upstream level L_{up}
- ◇ Downstream level L_{down}
- ◇ Crest level = $L_{down} + (TL_{up} - TL_{down})$

This formulation makes sure the upstream open water will only discharge to the downstream open water if the level difference exceeds the specified fixed level. The controller does not guarantee that the level difference is always equal to the specified fixed level. In fact, the level difference can be lower (if the upstream water level is below the crest level), or higher. In case the level difference is higher, the crest level is put at the desired fixed level difference, so the water levels will move towards the desired fixed level difference.

6.5.1.13 Fixed upstream level controller

This controller tries to maintain the upstream target level as long as the flow over the structure does not exceed the maximum flow. If the flow equals to the maximum flow, the level may increase up to the maximum level; at that time the maximum flow may also be exceeded. If the downstream level exceeds the specified maximum downstream level, a more strict flow limitation will be used.

The calculated flow by the controller is limited in two ways. Firstly, it is verified that this flow can be physically realised by lowering the weir to the specified lowest value z_{min} (either bottom level, or a higher user defined level). Secondly, there may be a maximum crest level z_{max} which limits the operation of the weir. If not specified, the controller operates such that negative weir flows are not allowed (this is equivalent with specifying z_{max} very high). If a z_{max} is specified, negative flows may occur (if the downstream level is higher than the upstream level); and also either the downstream level is above the z_{max} level or the upstream level is below the target level.

The algorithm is given as follows:

User input:

- ◇ Maximum flow MF
- ◇ Maximum level above which the maximum flow may be exceeded ML
- ◇ Maximum downstream level MD
- ◇ Minimum crest level related to the reference level z_{min}. If it is not specified, the lowest crest level is taken equal to the bottom level of the boundary or the RR-open water; this keyword provides a way to specify a higher lowest level.
- ◇ Maximum crest level related to the reference level z_{max}. If it is not specified, +9999.99 is used, thus preventing negative flow; this is backward compatible. This keyword provides a way to specify a lower maximum crest level.

6.5.1.14 Hydrologic Cycle

Water circulates through the hydrosphere through a maze of paths, in a cycle without a beginning or an end. Water evaporates from the oceans, is transported to other parts of the atmosphere, and precipitates on the land or the oceans. From there, it will eventually branch the seas via different processes known as infiltration, percolation, and groundwater flow. This cycle of processes is known as the *hydrologic cycle*.

Basically, SOBEK Rainfall Runoff focuses on the following transport processes:

- ◇ precipitation
- ◇ evapo(transpi)ration
- ◇ surface runoff
- ◇ infiltration
- ◇ (Drainage) outflow
- ◇ Seepage
- ◇ Percolation

In SOBEK RR, the user can choose from many different ways to model the flows of water over and under the ground surface, depending on the area type. The following area types are available:

- ◇ paved area
- ◇ unpaved area

- ◇ greenhouse area
- ◇ open water
- ◇ industrial area

Please refer to the corresponding sections for more information about the way the water transport and storage processes are modelled.

6.5.1.15 Improved separated sewer

The sewer type of a paved area node can be one of the following types:

- ◇ mixed sewer
- ◇ separated sewer
- ◇ improved separated sewer

A separated sewer system has separate sewer systems for rainfall and dry weather flow. The improved separated system is designed to reduce the sewer overflows of a separated system. Although overflows from the rainfall sewer system do not contain waste loads originating from DWF, they do contain waste loads from street surfaces, and thus can have negative impacts on water quality of the receiving water (the so-called 'first flush')

The improved separated sewer system has a connection between the rainfall sewer and the DWF sewer. This connection is used to store overflows from the rainfall sewer in the DWF sewer, if storage in the DWF sewer is available and no overflow of the DWF sewer is caused. If storage in the DWF sewer is not enough to store the overflow of the rainfall sewer system, the rainfall sewer will still spill into open water (but less than in case of a normal separated system).

6.5.1.16 Infiltration

Infiltration is the process by which water infiltrates from the surface of the ground into the *root zone* (subsoil), and is thus a part of the so-called *Hydrologic Cycle*. The infiltration capacity is influenced by many factors, like the condition of the soil surface and its vegetative cover, the soil properties and the current moisture content of the soil.

In SOBEK-RR, the infiltration capacity of the unpaved areas is considered to have a constant value in time, and can be entered in either mm/hour or mm/day.

6.5.1.17 Infiltration from open water

In dry periods the groundwater level can decrease. When the groundwater level is lower than the open water level, infiltration from the open water occurs. The infiltration rate depends on 1 the difference between open water level and groundwater level and 2 the resistance of the channel

In SOBEK-RR, this process is described by the modified De Zeeuw-Hellinga equation.

Therefore, the user must define a specific reaction factor, α. The difference between the open water level and groundwater level is computed by SOBEK-RR.

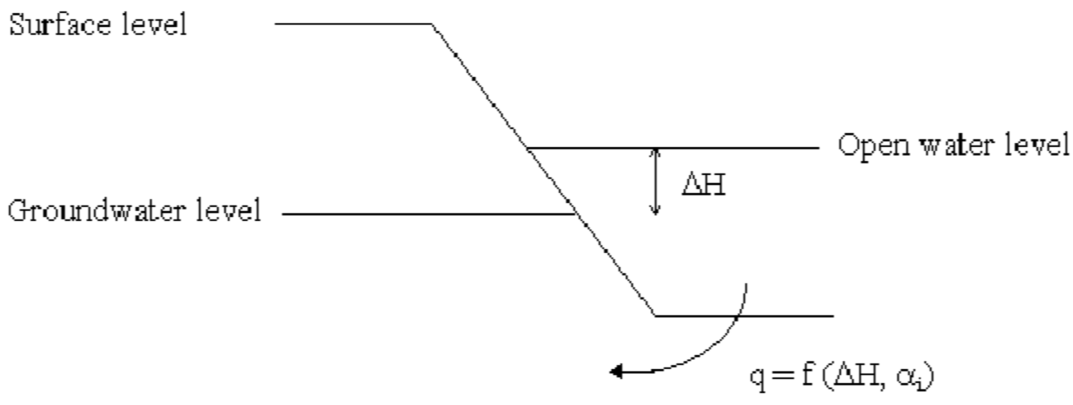


Figure 6.73: *Infiltration from open water*

6.5.1.18 Krayenhoff van de Leur drainage formula

One of the options of modelling the runoff for unpaved area is the Krayenhoff van de Leur formulation. Another available option is the de Hellinga-de Zeeuw formulation.

The following figure illustrates the principles of the Krayenhoff van de Leur method.

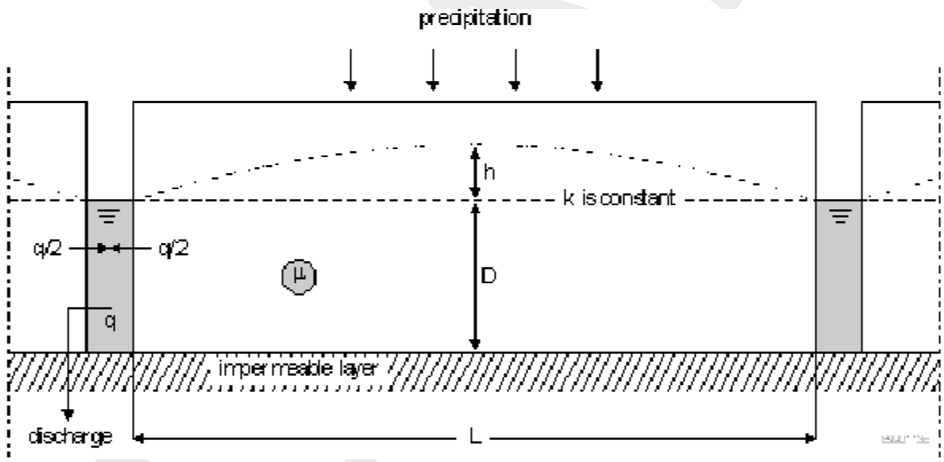


Figure 6.74: *Drainage according to Krayenhoff van de Leur*

The reservoir coefficient j is either specified directly, or calculated as:

$$j = \frac{\mu L^2}{\pi^2 k_D} \tag{6.288}$$

The (cumulative) drainage flow q is computed as:

$$q = \frac{8P}{\pi^2} \sum_{n=1,3,5,...}^{n \to \infty} \frac{1}{n^2}(1 - e^{-n^2 \frac{t}{j}}) \tag{6.289}$$

and the (cumulative) highest groundwater level between the ditches as computed as:

$$h = \frac{\pi^2 P}{8\mu} j \left(1 - \frac{32}{\pi^3} \sum_{n=1,3,5,\ldots}^{n \to \infty} \frac{1}{n^3} e^{-n^2 \frac{t}{j}} \right) \tag{6.290}$$

where:

j	reservoir coefficient [d]
m	storage coefficient (specific yield) [$-$]
L	distance between drainage ditches [m]
k_D	soil transmissibility (transmissivity) [m^2/d]

k_D can be derived from:

k	soil permeability [m/d]
D	thickness of permeable layer [m]
P	precipitation [m]
q	(cumulative) drainage flow [m/d]
h	difference between highest groundwater level and ditch level [m]
t	time [d]

Given the area A [m^2], precipitation and drainage flow can be converted to [m^3/s]

6.5.1.19 Minimum filling percentage for greenhouse storage basin

When the water level in the greenhouse storage basins becomes equal/lower than this minimum filling percentage, the withdrawal of water out of the basins will be stopped.

Default: 10 %

6.5.1.20 Minimum level difference controller

With the minimum level difference controller one can keep a minimum difference between the upstream water level and the downstream water level. This controller is closely related to the fixed level controller. There are two differences:

◇ the fixed level controller determines the level difference from the specified target levels which can be variable within the year, whereas the minimum level difference controller uses one constant minimum level difference;
◇ the fixed level controller allows a crest level below the values specified in the crest level table, whereas the minimum level difference controller does not allow this.

The algorithm of the minimum level difference controller is given as follows:

User input:

◇ Minimum level difference Min_dif
◇ initial crest level

Model input, i.e. parameters computed by SOBEK-RR:

◇ Downstream level L_{down}

The minimum level difference controller adjusts the crest level of the weir in the following way:

Crest level = Max (Crest level according to input table, (L_{down} + Min_dif))

6.5.1.21 Mixed sewer

The sewer type of a paved area node can be one of the following types:

- ◇ mixed sewer
- ◇ separated sewer
- ◇ improved separated sewer

A mixed sewer system means that there is only one system for the discharge of both rainfall water and dry weather flow (return flow from domestic water use, e.g. showers, toilets, etc.)

Spilling from the sewer system after heavy rainfall thus contains a mix of rainfall water and domestic return flow. Spilling from a mixed sewer system therefore in general has a negative effect on the water quality of the receiving open water.

6.5.1.22 Open water node

In SOBEK-RR the local surface water is represented by open water nodes. An open water node can also be considered a storage basin in which the variation of water level follows from the volume balance equation. In this equation an increase of the storage with higher water levels (effect from banks) can be taken into account. An open water node is always connected with a structure to another open water node or to a boundary node representing the boundary of the water system.

Storage at open water nodes increases through precipitation, seepage from deep groundwater, flows from adjacent paved area, unpaved area and greenhouse area, and inflow through structures from other open water nodes or form boundaries. Open water storage decreases through evaporation, percolation to deep groundwater, flows to unpaved area (if the open water level is higher than the groundwater level), and flows through structures to other open water nodes or boundaries.

In order to obtain an accurate computation of the open water level, the level changes and flow-through-structures, which depend on water levels, are computed iteratively.

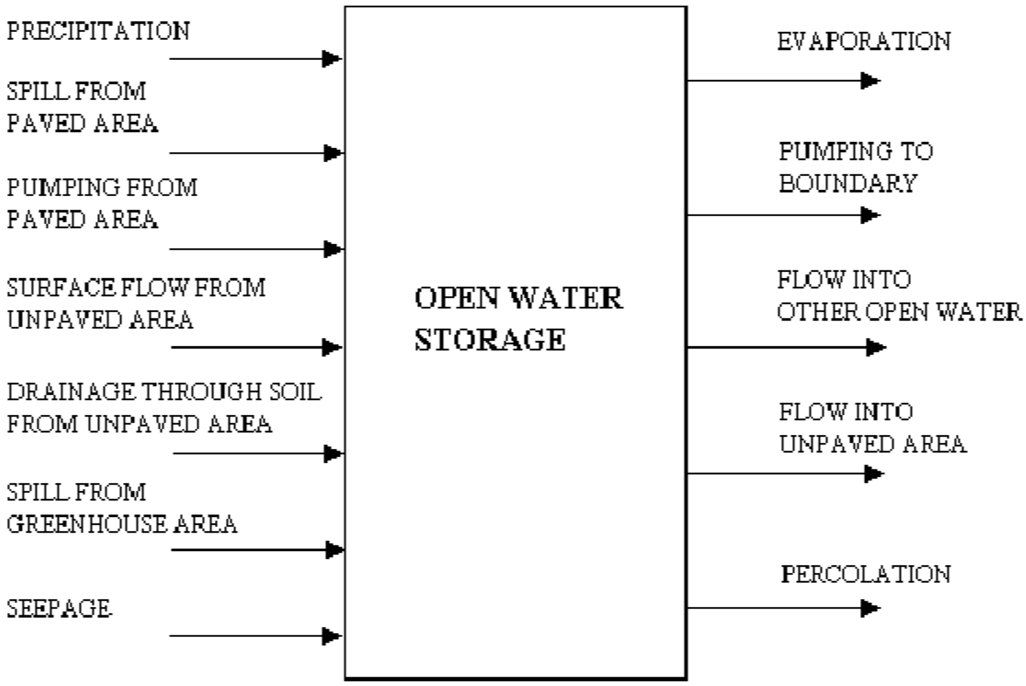


Figure 6.75: *Representation of the rainfall-runoff process of an open-water area*

Amount of rainfall on open water node

The amount of rainfall on a open water node depends on the surface area of open water, which can increase with an increasing water level. SOBEK does take this effect into account. But the increase of open water surface means that in SOBEK the total area also increases. In reality, the surface area of the unpaved area decreases. SOBEK does not take this effect into account.

6.5.1.23 Paved area node

General

The paved area node is used to simulate the rainfall-runoff process on paved or impervious areas.

A paved area is characterized by:

- ◇ area [ha]
- ◇ maximum storage on street [mm]
- ◇ maximum storage in sewer [mm]
- ◇ sewer pump capacity [m^3/s]
- ◇ sewer type (mixed, separated, or improved separated system)
- ◇ DWF (dry weather flow)

Important processes are surface runoff or sewer overflow, occurring when the storage on land or storage in the sewer exceeds its maximum and rapid runoff occurs.

6.5.1.24 Paved node surface runoff

When the amount of water on the RR-paved surface exceeds the maximum storage, surface runoff will occur. The surface runoff can be implemented using either

◇ no delay,
◇ using a runoff delay coefficient, or
◇ using a QH-relation.

The first option, no delay, means all surface runoff branches the outflow point in the same timestep. Surface runoff can be blocked however if the open water level at the connecting node exceeds the surface level of the paved node. But when using the 'no delay' option the surface storage will immediately runoff as soon as the water level drops below the paved surface level. This may lead to very spiky discharges from the paved node to the RR-open water or RR-Boundary node.

The second option, using a runoff delay coefficient, is similar to the RR-Urban runoff model.

(See also the description of delay of runoff in the RR-Urban documentation, section 6.5.4. This is formulating the surface runoff using the rational method ($q = ch$). The runoff delay coefficient c is specified as a number between zero and one (1/min). A coefficient of one means all runoff occurs in 1 minute, while a coefficient of 0.1 means only 10 % of the excess volume will branch the open water in 1 minute.

The impact of the runoff delay coefficient on the runoff pattern is shown in the following graph. The graph shows the rainfall time-series used, and the runoff using runoff delay coefficients of 0.1, 0.2 and 0.5. It is assumed that there is no infiltration or sewer, and that the maximum storage on the street is zero.

Note: The example Figure 6.76 is actually taken from a simulation of the RR-Urban inflow model, but the behaviour for the RR-Paved node is the same.

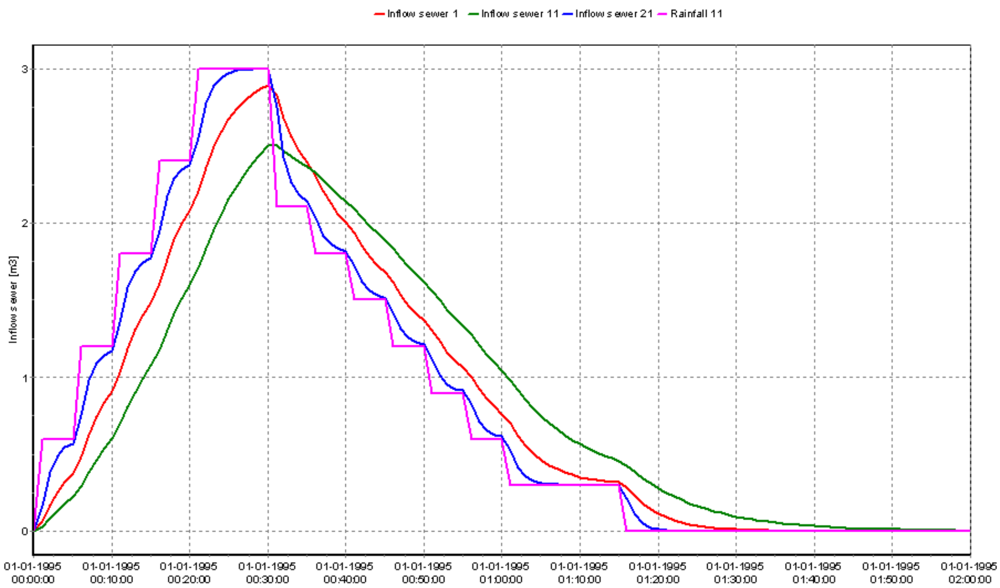


Figure 6.76: *Impact of delay coefficient on runoff pattern*

The third option, using a QH-relation, allows limiting of the Q based on the water level h. Based on the interpolation table defined by the user a maximum Q is determined with which the overflowing discharge is limited.

The used h is the water level of the open water node or RR-CF-connection node downstream of the paved node, where the overflowing discharge of the RR-paved node is discharged to. See the following two examples:

1 Figure 6.77 shows the result of water overflowing from two paved nodes to open water nodes without delay of discharge from the paved nodes.
2 Figure 6.78 shows the discharges after using a QH-relation.

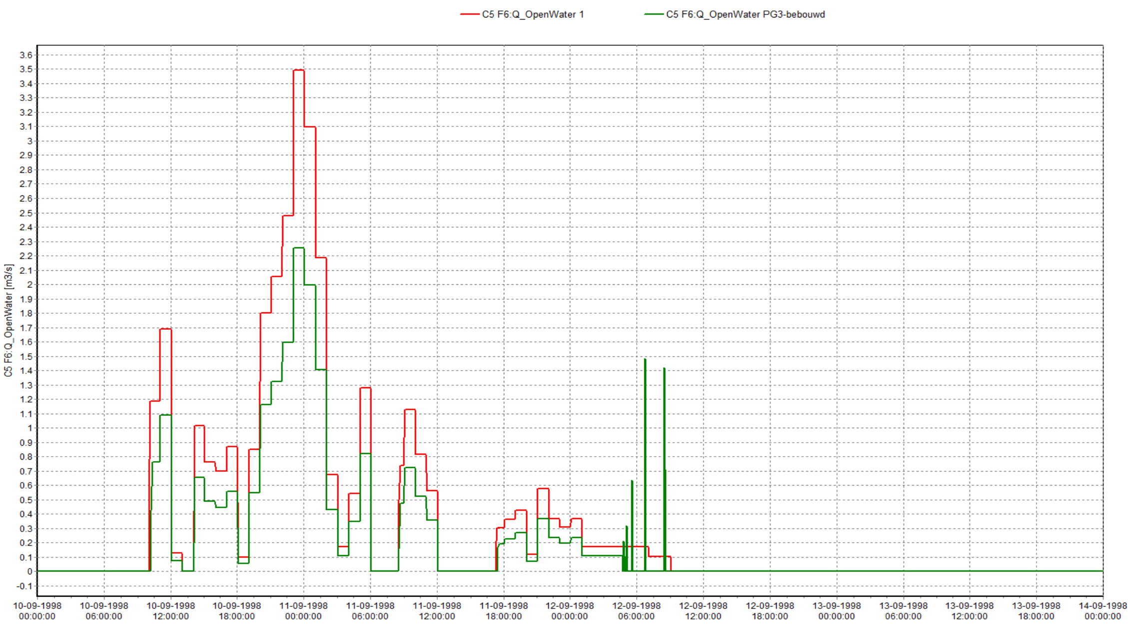


Figure 6.77: *Discharge from Paved to Open Water without delay*

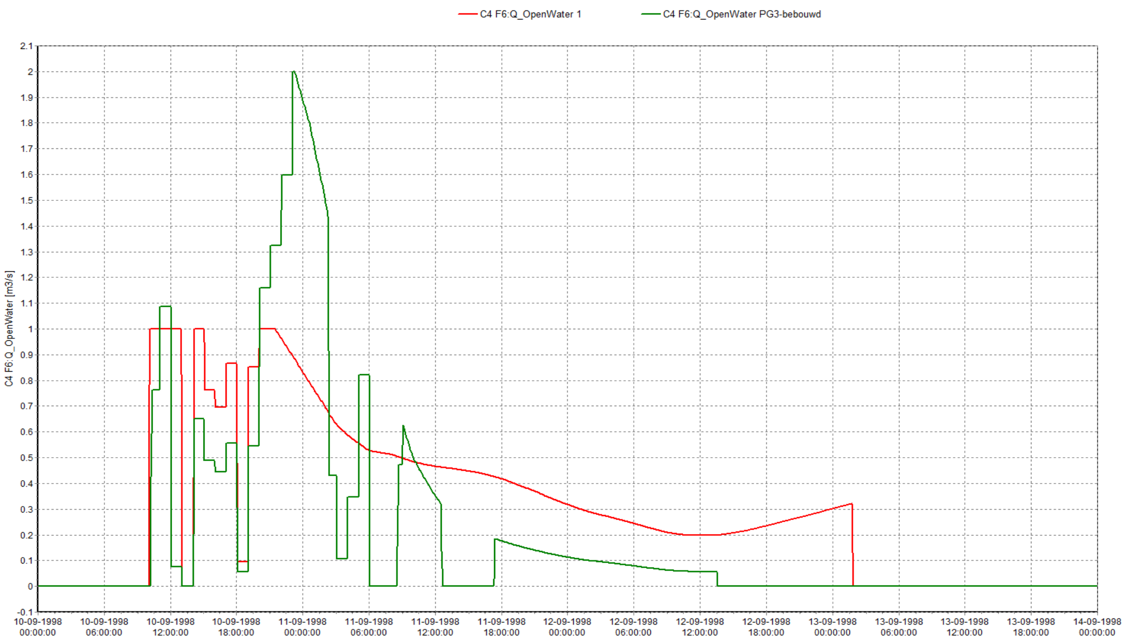


Figure 6.78: *Discharge from Paved to Open Water using QH-relation*

6.5.1.25 Percolation

If the moisture storage in the root zone exceeds the equilibrium content, percolation occurs from root zone to groundwater:

$$q_p = \frac{V_{eq} - (V(t - \Delta t) + \Delta V)}{\Delta t} \tag{6.291}$$

q_p	Percolation $[m/d]$
V	Soil moisture storage in the root zone $[m]$
V_{eq}	Equilibrium moisture storage in the root zone $[m]$
Δt	Time step $[d]$

If percolation occurs, it is assumed that the percolating water branches the groundwater table within the current time step of the groundwater.

The equilibrium moisture storage in the root zone is defined as the amount of moisture corresponding with a steady-state simulation with no-flow conditions to or from the root zone.

Q	h
0	0
0.01	0.1
0.02	0.2
0.03	0.3
0.04	0.4
0.05	0.5

6.5.1.26 QH-relation

A QH-relation can be used to represent any type of structure. When the user defines the QH-relation as follows:

then SOBEK-RR computes the discharge Q as a function of the water level h:

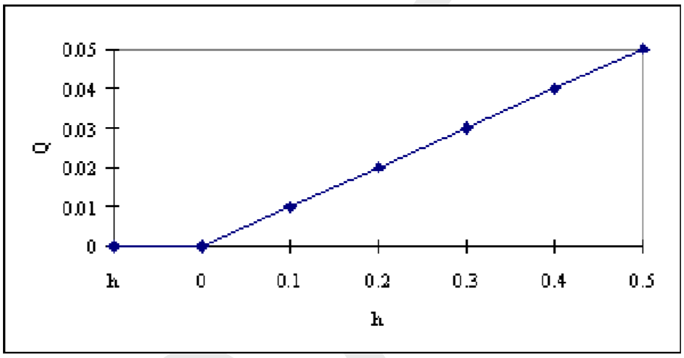


Figure 6.79: *QH-relation*

6.5.1.27 Root zone

The root zone for which the water balance is considered has a thickness which is a function of the land use type and the soil physical unit:

$$d_r = f(j, s) \tag{6.292}$$

where:

d_r	root zone thickness
j	land use type (1,. . . ,16)
s	soil physical unit (1,. . . ,21)

The thickness of the root zone is assumed to be constant, regardless of season and year. The moisture storage of the root zone at equilibrium condition is calculated with the function:

$$V_{eq} = f(s, d_r, dg) \tag{6.293}$$

where:

V_{eq}	the moisture storage of the root zone at equilibrium condition $[m]$
s	soil physical unit $[-]$
d_r	thickness of root zone $[m]$
d_g	depth of the groundwater level $[m]$

Initial storage in the root zone

The initial storage in the root zone can be defined by one of the three following options:

- ◇ the equilibrium root zone storage for the given crop-soil combination and specified initial groundwater level;
- ◇ the root zone storage at pF= 2;
- ◇ the root zone storage at pF= 3.

Root zone thickness defining more than one crop

When the user defines more than one crop per unpaved node SOBEK calculated the overall root zone by weighted averaging. So, the area of the individual crops is also taken into account.

For calculating the evaporation SOBEK uses the crop with the largest sub-areal.

6.5.1.28 RR controllers

The SOBEK Rainfall-Runoff module offers the option to operate hydraulic structures such as weirs and orifices to a certain extent.

The available controllers are:

- ◇ Time controller
- ◇ Target level controller
- ◇ Fixed Level difference controller
- ◇ Minimum level difference controller
- ◇ Equal filling controller

See their sections in the Technical Reference Manual for more information on each type of controller.

6.5.1.29 RR routing link

The standard RR-links are links indicating connections between e.g. an unpaved node and an open water node. The link flow in that case represents the flow from the unpaved area to the open water. There is no difference between link inflow and link outflow.

Within SOBEK, the hydrodynamic flow aspects involving time delay between link inflow and outflows are typically handled using the SOBEK-Channel Flow module. However, in some cases a simple approach may be necessary. For instance, if only flow measurements at an upstream location and a downstream location (without additional inflows in between) are known, but no cross-section information is available. In such situations a simple routing approach can be used. An example of such a simple approach is the well-known Muskingum method. This method is available in SOBEK-RR by using the so-called RR-Routing link.

Standard Muskingum method

The standard Muskingum method is using a layered routing approach. The inflow hydrograph is divided into a number of layers, where each layer has its own routing coefficient.

According to the standard Muskingum method (McCarthy, 1938) the outflow Q at time step $i+1$ is computed using outflow Q at time step i and inflows I at time step i and $i+1$:

$$Q_{i+1} = c_1 I_i + c_2 I_{i+1} + c_3 Q_i \tag{6.294}$$

where:

$$c_1 = \frac{\Delta T + 2Kx}{2K(1-x) + \Delta T} \tag{6.295}$$

$$c_2 = \frac{\Delta T - 2Kx}{2K(1-x) + \Delta T} \tag{6.296}$$

$$c_3 = \frac{2K(1-x) - \Delta T}{2K(1-x) + \Delta T} \tag{6.297}$$

The parameter K is a proportionality factor with the dimension of time, it is dependent on the time step you select. K is the travel time of a flood wave through the branch. The parameter x is a dimensionless weighting factor with $0 \leq x \leq 0.5$. In natural streams values of x ranging from 0 to 0.3 are often found. Great accuracy in determining x may not be necessary as the final result is relatively insensitive to the value of x.

It is easily verified that $c_1 + c_2 + c_3 = 1$ and further that ΔT and K should have the same time units. The use of the method requires choices on ΔT, K and x. The routing interval ΔT is the calculation time step size in SOBEK-RR. The chosen time step size should be less than K, as otherwise peaks will be missed at the downstream side. It is advised the SOBEK-RR computation time step size ΔT to be taken as $1/2$ to $1/4$ of K. If, however, a ΔT is selected which is much smaller than K, then c_2 becomes negative. This will lead to a dip in outflows, or even to negative outflows when the inflow starts rising. To avoid negative coefficients the routing interval should be within the range: $2Kx \leq \Delta T \leq K$. For values of x close to 0.5 this requirement leaves little freedom in the selection of ΔT. If one is mostly interested in the higher part of the flood wave the above condition can be relaxed somewhat by accepting slightly negative values for c_2.

The Muskingum routing method allows the routing to be split in multiple layers. This can be of great value when the river contains a minor bed and a floodplain. In the floodplain the Muskingum parameters will be completely different from the parameters of the minor bed.

An example of this computation is given below. Suppose that you use 2-layers for routing the

flood wave:
layer 1: $x = 0.4 \quad K = 3.5 \quad Q_{max} = 100\, m^3\, s^{-1}$
layer 2: $x = 0.1 \quad K = 9$

SOBEK-RR will then route all discharges up to $Q = 100\, m^3\, s^{-1}$ with the parameters of the first layer, all discharges above $Q = 100\, m^3\, s^{-1}$ will be routed with the parameters of the second layer

(and using $Q_{layer2} = Q_i - 100[m^3/s]$).

RR connection node

Similar to a Flow Connection node, now also a RR-connection node is available. The RR-Connection node does not require any input data. The basic function of the RR-Connection node is that of a confluence: adding up all flows from incoming links and supplying that to the downstream link.

Network validation rules for RR-routing link and RR-connection nodes

The following rules apply to the use of the RR-routing link and RR-connection nodes:

- The RR-routing link can be used to connect:
  - a Sacramento node to an RR-connection node;
  - an RR-connection node to an RR-connection node;
  - an RR-connection node to a RR-boundary node;
- At RR-connection nodes, multiple upstream links are possible, but only 1 downstream (routing) link is allowed.

Examples of network layout and ModelEdit user interface:

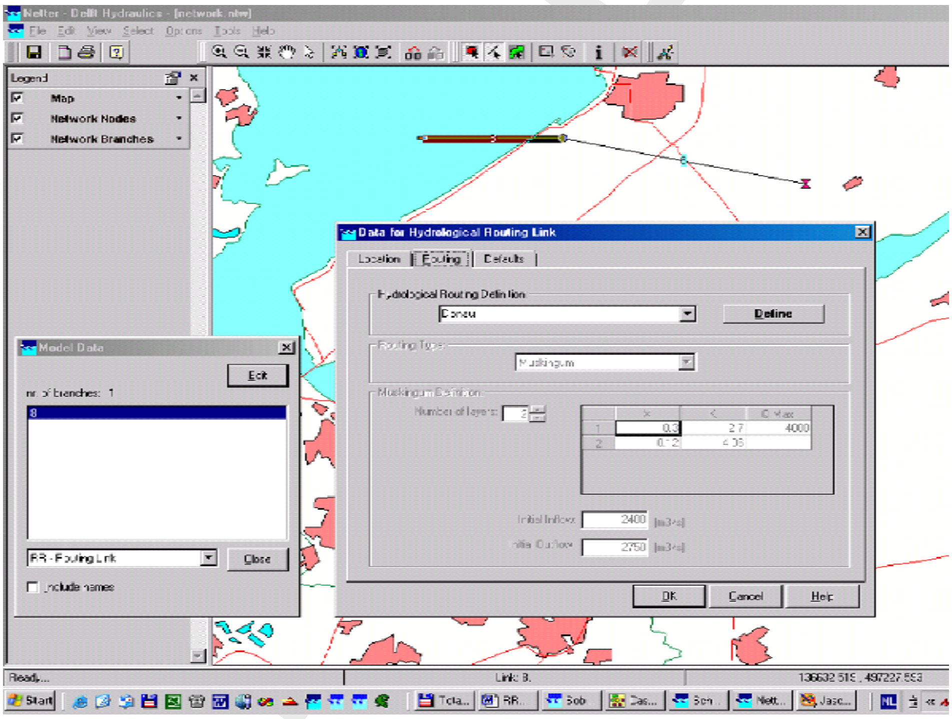


Figure 6.80: *Example input data*

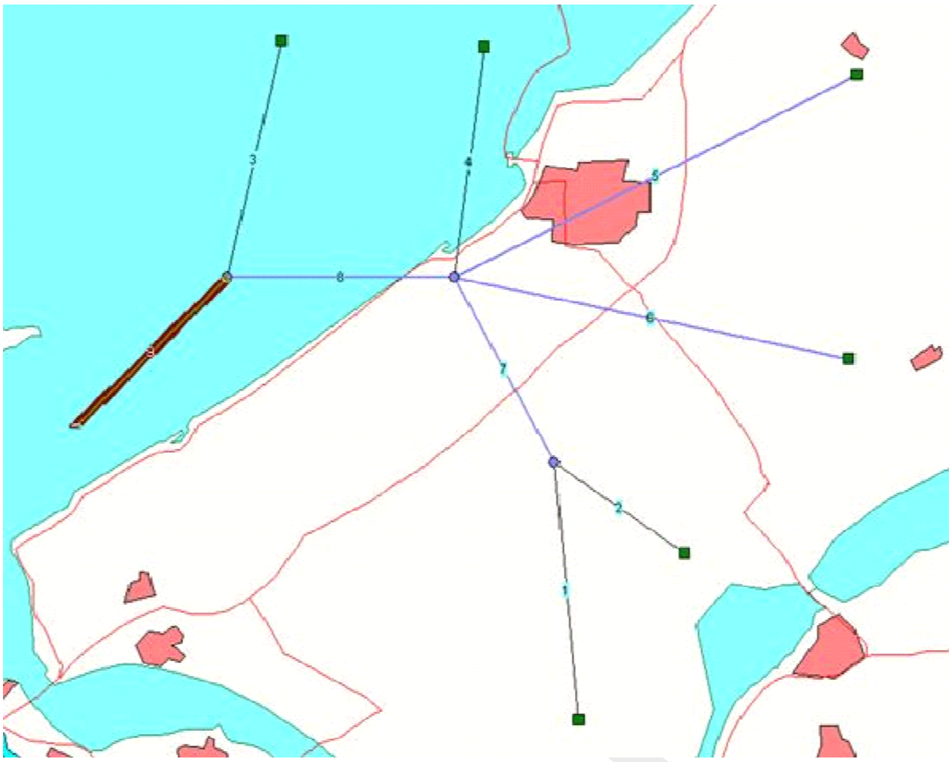


Figure 6.81: *Possible usage in a network*

6.5.1.30 RR - Orifice

The orifice is one of the structures available in the SOBEK-RR-module. The geometrical shape is given in Figure 6.82.

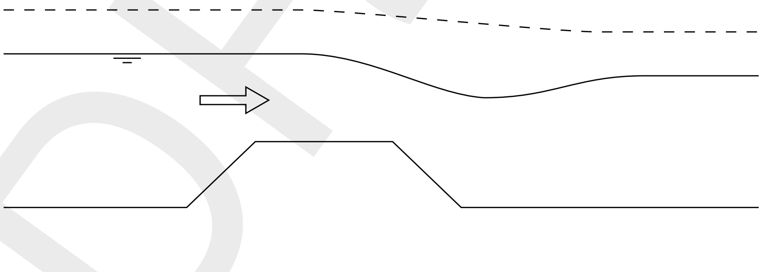


Figure 6.82: *Orifice*

Flow across the orifice can be of the following types: drowned weir flow, free weir flow, drowned orifice flow, free orifice flow or no flow (water levels below crest level or orifice closed) depending on the dimensions of the structure and the flow conditions.

The following discharge equations are applied during the computations:

Free orifice flow:

$$Q = \mu c B d \sqrt{2g(h_1 - (z + \mu d_g))} \tag{6.298}$$

Drowned orifice flow:

$$Q = \mu c B d \sqrt{2g(h_1 - h_2)} \tag{6.299}$$

Free weir flow:

$$Q = cB\frac{2}{3}\sqrt{\frac{2}{3}g}(h_1 - z_s)^{3/2} \tag{6.300}$$

Drowned weir flow:

$$Q = cB(h_1 - z)\sqrt{2g(h_1 - h_2)} \tag{6.301}$$

Q	Discharge across orifice $[m^3/s]$
μ	Contraction coefficient $[-]$
c	Discharge coefficient $[-]$
B	Crest width $[m]$
d_g	Openings height $[m]$ (openings level - crest level)
g	Gravity acceleration $[m/s^2]$
h_1	Upstream water level $[m]$
h_2	Downstream water level $[m]$
z_s	Crest level $[m]$

The different formulas are applied when the following conditions occur:

Free orifice flow:

$$h_1 - z \geq \frac{3}{2}d_g \text{ and } h_2 \leq z_s + d_g \tag{6.302}$$

Drowned orifice flow:

$$h_1 - z \geq \frac{3}{2}d_g \text{ and } h_2 > z_s + d_g \tag{6.303}$$

Free weir flow:

$$h_1 - z < \frac{3}{2}d_g \text{ and } h_1 - z_s > 3/2(h_2 - z_s) \tag{6.304}$$

Drowned weir flow:

$$h_1 - z < \frac{3}{2}d_g \text{ and } h_1 - z_s \leq \frac{3}{2}(h_2 - z_s) \tag{6.305}$$

6.5.1.31 RR - Weir

Three types of flow conditions can occur in the case of weir flow. These are free (modular) flow, drowned (submerged) flow and no flow (water levels below crest level). If high tail water conditions do affect the flow, the weir is said to be drowned.

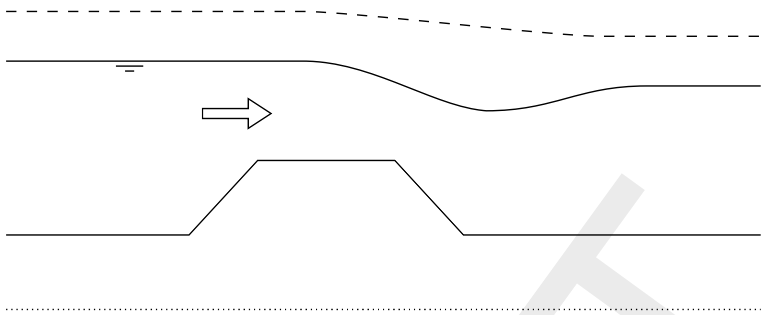


Figure 6.83: *Weir*

The discharge through the weir is computed with the following formulas:

Free weir flow:

$$Q = cB\frac{2}{3}\sqrt{\frac{2}{3}g}(h_1 - z_s)^{3/2} \tag{6.306}$$

Drowned weir flow:

$$Q = cB(h_1 - z_s)\sqrt{2g(h_1 - h_2)} \tag{6.307}$$

Q	Discharge across weir $[m^3/s]$
c	Discharge coefficient $[-]$
B	Crest width $[m]$
g	Acceleration due to gravity $[m/s^2]$
h_1	Upstream water level $[m]$
h_2	Downstream water level $[m]$
z_s	Crest level $[m]$

The different formulas are applied when the following conditions occur:

Free weir flow:

$$h_2 - z_s < \frac{2}{3}(h_1 - z_s) \tag{6.308}$$

Drowned weir flow:

$$h_2 - z_s \geq \frac{2}{3}(h_1 - z_s) \tag{6.309}$$

Two-stage weir:
The flow over a 2-stage weir is determined as the sum of the two weirs:

◇ A rectangular weir with crest level Crestlevel1 and width Width1; and

◇ A rectangular weir with crest level Crestlevel2 and width Width2.

The Rainfall Runoff module separately determines the flow conditions for both levels (free or drowned flow, or no flow). The normal Weir formulae are used.

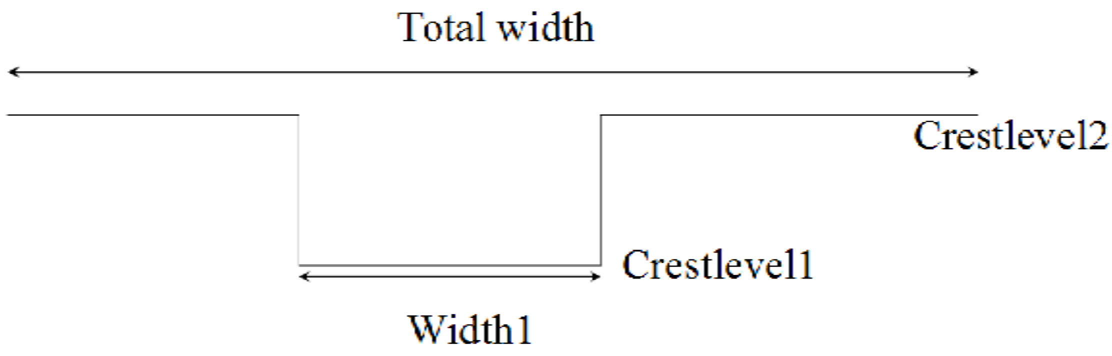


Figure 6.84: *Two-stage weir*

The crest level of the lower stage of the weir can optionally be adjusted according to the upstream open water target level (depending on the switch specified in the Settings module). The crest level of the second stage is not adjusted.

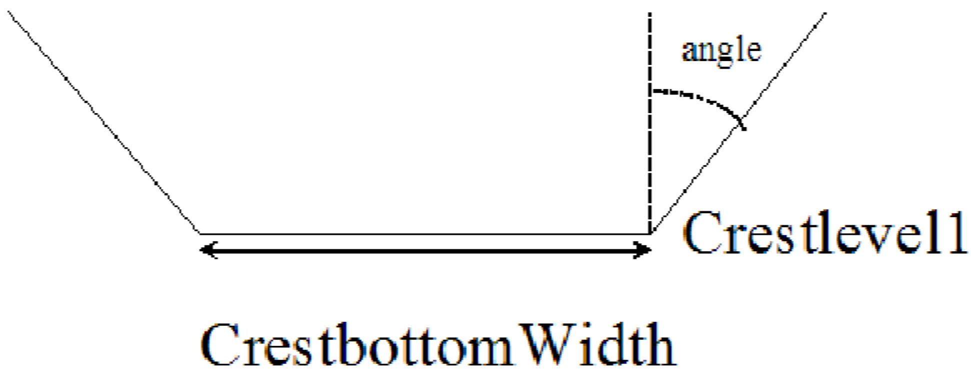


Figure 6.85: *Two-stage weir, crest bottom width at crestlevel 1*

Output for 2-stage weirs: Both the total flow over the weir and the seperate flows of the two levels are shown.

V-notch broad-crested weir

Free weir flow:

$$Q = d_c \frac{16}{25} \sqrt{\frac{2}{5} g} \tan(\alpha) H^{2.5} \tag{6.310}$$

where d_c is typically between 0.85 and 1.05.

Drowned weir flow

The drowned flow reduction factor is given as a function of the ratio of the upstream and downstream waterlevels above the crest level (Bos, 1989, Par. 4.3.3).

H2/H1	Reduction factor	Remark
1	0	H2=H1, no flow
0.995	0.1	
0.99	0.2	
0.985	0.3	
0.978	0.4	
0.97	0.5	
0.96	0.6	
0.95	0.67	
0.945	0.7	
0.925	0.8	
0.9	0.87	
0.885	0.9	
0.85	0.96	
0.8	1.0	free weir flow

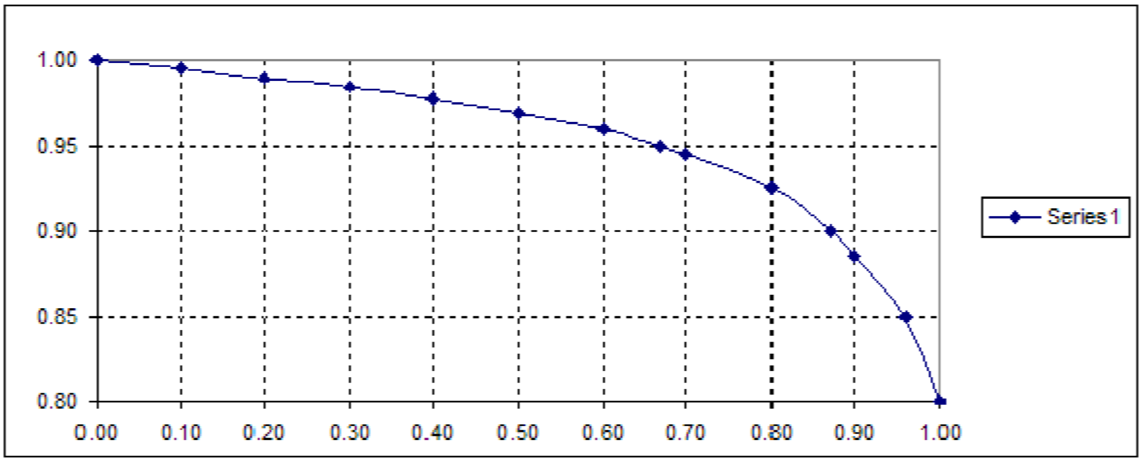


Figure 6.86: *V-notch broad-crested weir*

6.5.1.32 Separated sewer

The sewer type of a paved area node can be one of the following types:

◇ mixed sewer
◇ separated sewer
◇ improved separated sewer

A separated sewer has separate sewer systems for rainfall and dry weather flow. This system is designed to reduce water quality impacts of sewer overflows on the receiving open water. In a mixed sewer system spilling contains DWF water (domestic return flows). In a separated sewer system however, sewer overflows usually only originate from the rainfall sewer system and not from the DWF sewer system. Therefore, the sewer overflows do not contain the waste loads from the domestic return flows.

6.5.1.33 Silo capacity/Pump capacity

Apart from storage in rainfall basins in some greenhouse areas there is also storage of water in silo's. These silo's, usual capacity approximately 200 $m^3\,ha^{-1}$, function as a temporary storage before pumping the water into the soil (subsoil storage). The pump capacity for subsoil storage usually is about 15 $m^3\,h^{-1}$.

6.5.1.34 Soil surface level

There are two options to define the soil surface level:

Surface level = constant

The user can define any constant surface level he wants, usually the (almost) lowest surface level in the unpaved area.

Storage on land

- groundwater level rise: when the groundwater table branches the lowest surface level, surface runoff occurs only when the storage on land is filled.
- high precipitation rate: when the net precipitation rate exceeds the infiltration rate, water will be stored on land. Surface runoff occurs only when the storage on land is filled.

Surface level = not constant

When this option was chosen, the user can specify the soil surface level in a more detailed way.

SOBEK does take into account up to a maximum of 100 sub-areas (default). The user can define these areas in a table, see Figure 6.87.

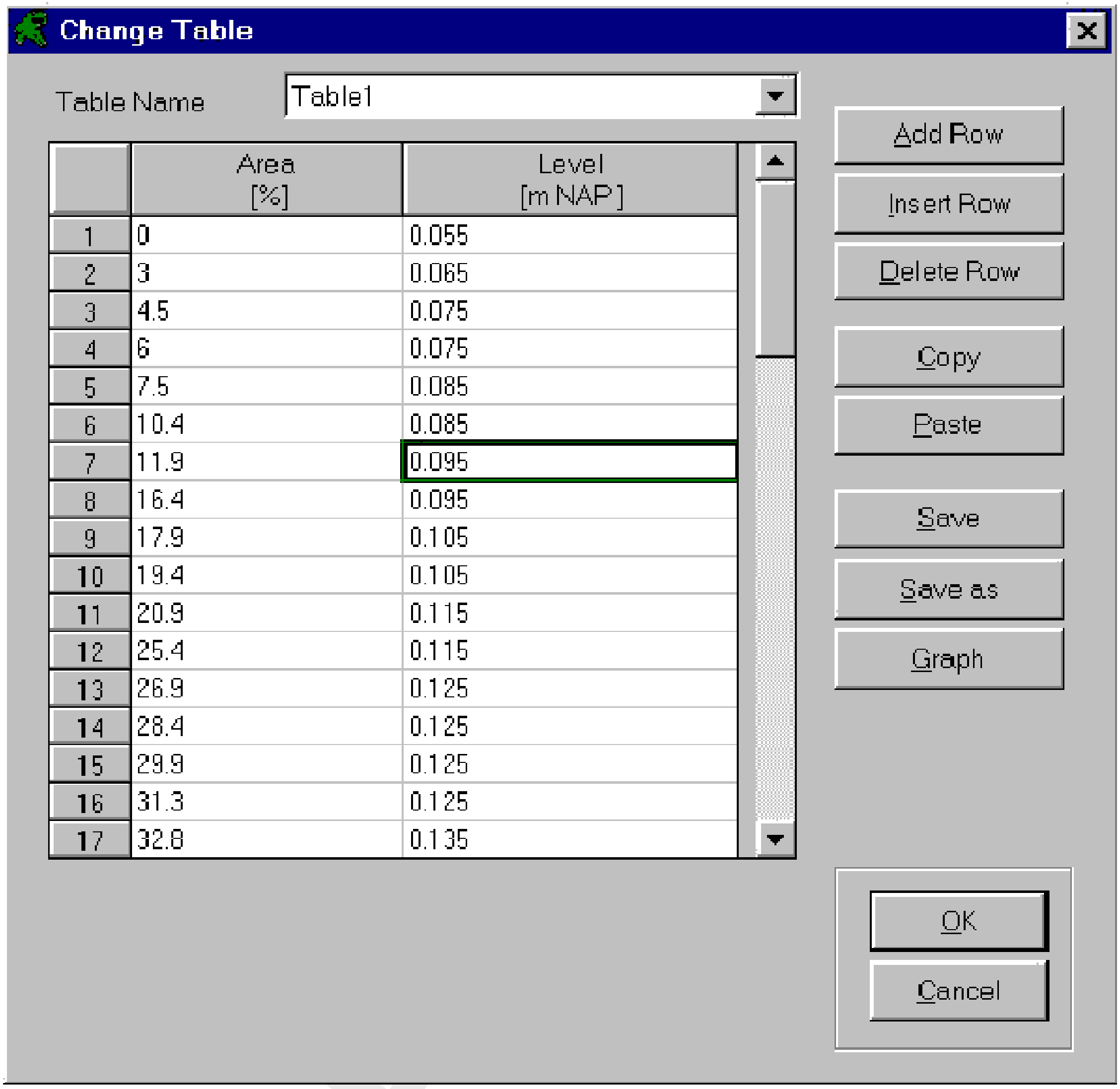


Figure 6.87: *Change table*

The user should make sure that 0 and 100 % are present in this table. In between the user can define as many rows as he wants. Above the 100 % level it is assumed that the area does not increase.

Note: By showing the graph of the S-curve you can check whether you entered the s-curve data correctly.

Inundation

When the open water level exceeds the lowest surface level, inundation occurs. See Figure 6.88. SOBEK does take into account the extra volume to store water. SOBEK does not take into account the precipitation and evaporation on the area of this extra storage volume. In fact, the precipitation and evaporation on this area was already taken into account with the surface of the unpaved area.

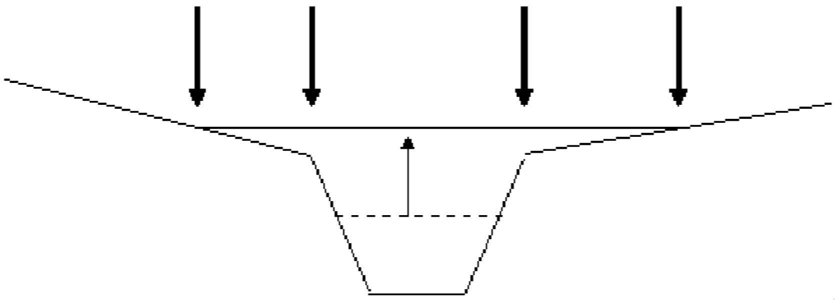


Figure 6.88: *Inundation*

Vertical infiltration of water in this extra storage area is not considered as well.

Finally, when the open water level exceeds the lowest surface level it is assumed that the potential storage on land is equal to zero.

Initial groundwater level

When the surface level is not constant, it is assumed that the initial groundwater level is defined w.r.t. the lowest surface level. For example, in Figure 6.87 the initial groundwater level is defined w.r.t. the level of 0.055 m NAP.

Reaction factor α

In SOBEK it is possible to schematise the drainage process by defining four different layers. For each layer a different reaction factor α is defined. When the surface level is not constant, the reaction factor layers are defined relative to the surface level according to the table, see Figure 6.89.

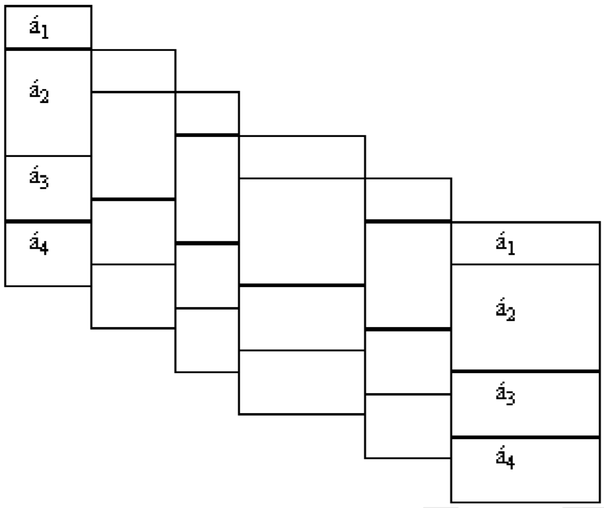


Figure 6.89: *Example of definition of reaction factor layers when the surface level is not constant (in this case the number of sub-areas defined in the table is equal to 6)*

Actually, the above figure is a bit more complex, because default a 100 sub-areas are considered. The surface level of each sub-area is determined by interpolation of the surface level table defined by the user.

Storage on land

Surface level is not constant:

- ◇ groundwater level rise: when the groundwater table branches the lowest surface level, immediately surface runoff occurs. Storage on land is not considered.
- ◇ high precipitation rate: when the net precipitation rate exceeds the infiltration rate, water will be stored on land. Surface runoff occurs only when the storage on land is filled.

Storage coefficient i

Surface level = not constant AND NOT using CAPSIM for unsaturated zone

The storage coefficient used to calculate the groundwater level fluctuation, is determined as the average of the initial storage coefficients for each sub-area. The initial storage coefficient is determined by the soil type and the distance between the soil surface and initial groundwater level.

When the initial groundwater level exceeds part of the surface level (storage on land) then the storage coefficients of this part of the surface are excluded from the averaging procedure.

Surface level = not constant AND using CAPSIM for unsaturated zone

CAPSIM needs , among others, the root zone depth and initial groundwater level. If the surface level is not constant, the root zone depth and initial groundwater level passed to CAPSIM are defined relative to the surface level (Figure 6.90). The root zone depth is equal for each sub-area, because the root zone depth only depends on the soil type and the crop.

For each sub-area, separate actual evaporation, percolation and capillary rise, and volume of water in the unsaturated zone are computed.

The groundwater level is determined using the total percolation/capillary rise flow for all sub-areas. The storage coefficient used to calculate the groundwater level fluctuation, is determined as the average of the actual storage coefficients for each sub-area. When the groundwater level exceeds part of the surface level (storage on land) then the storage coefficients of this part of the surface are excluded from the averaging procedure.

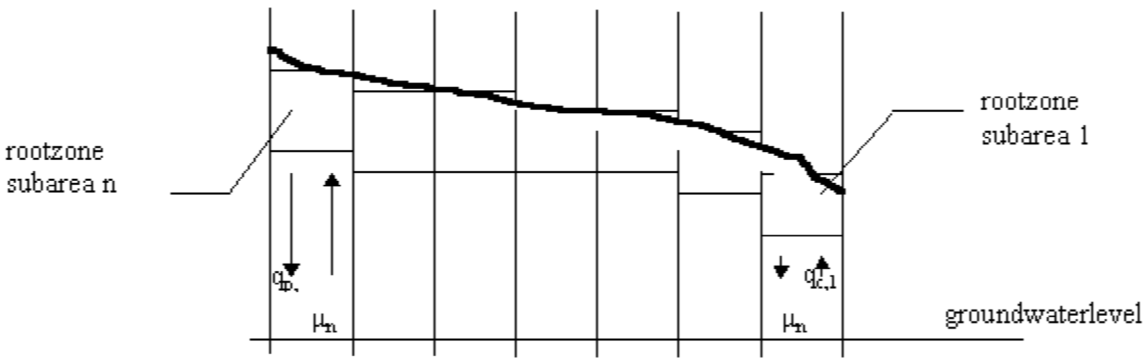


Figure 6.90: *Rootzone sub areas*

6.5.1.35 Storage coefficient

The storage coefficient m represents the percentage of soil-volume which is available for storage of water. Once the storage coefficient is known, the total storage capacity can be calculated:

$$V = \mu d \tag{6.311}$$

V	storage capacity $[mm]$
μ	storage coefficient $[m/m]$
d	depth to groundwater table $[mm]$

With the storage coefficient, the change of the groundwater level can be calculated.

Option 'Unsaturated zone = none'

Calculating groundwater level

For calculating the groundwater level the average storage coefficient is used during the simulation period (Table 6.10). This average storage coefficient depends on the drainage basis, i.e. the distance between the surface level and the initial groundwater level. These coefficients are provided by the Winand Staring Centre -DLO.

The soil types are:

1 loamy, humous fine sand,
2 peat,
3 heavy clay,
4 humous clay and peat,
5 light loamy sand, medium coarse sand,
6 loamy silt,
7 humous clay and peat with silty top layer,
8 clay and light clay,
9 loamless, medium coarse and coarse sand,

10 silt,
11 very light clay,
12 sand with a silty top layer.

Table 6.10: *Average storage coefficients depending on soil type and dranaige basis.*

Drainage basis [m]	Soil type											
	1	2	3	4	5	6	7	8	9	10	11	12
0.1	0.0074	0.0077	0.0094	0.0063	0.0046	0.0049	0.0049	0.0048	0.0027	0.0029	0.0025	0.0013
0.2	0.0183	0.0153	0.0166	0.0134	0.0124	0.0103	0.0118	0.0092	0.0098	0.0066	0.0052	0.0032
0.3	0.0302	0.0226	0.0226	0.0206	0.0220	0.0158	0.0181	0.0132	0.0162	0.0105	0.0079	0.0052
0.4	0.0419	0.0305	0.0278	0.0279	0.0323	0.0211	0.0237	0.0171	0.0228	0.0145	0.0107	0.0074
0.5	0.0532	0.0387	0.0323	0.0350	0.0427	0.0262	0.0289	0.0206	0.0294	0.0186	0.0134	0.0096
0.6	0.0664	0.0467	0.0363	0.0420	0.0528	0.0310	0.0339	0.0240	0.0359	0.0226	0.0160	0.0120
0.7	0.0805	0.0545	0.0399	0.0486	0.0625	0.0364	0.0386	0.0271	0.0422	0.0265	0.0185	0.0143
0.8	0.0938	0.0621	0.0431	0.0551	0.0715	0.0416	0.0430	0.0300	0.0484	0.0303	0.0210	0.0167
0.9	0.1061	0.0702	0.0461	0.0613	0.0801	0.0466	0.0472	0.0328	0.0542	0.0341	0.0235	0.0191
1.0	0.1173	0.0784	0.0489	0.0673	0.0880	0.0514	0.0512	0.0355	0.0598	0.0377	0.0258	0.0214
1.2	0.1372	0.0939	0.0538	0.0786	0.1024	0.0605	0.0586	0.0404	0.0704	0.0446	0.0303	0.0261
1.5	0.1614	0.1158	0.0600	0.0941	0.1208	0.0729	0.0685	0.0470	0.0845	0.0541	0.0366	0.0329

Calculating drainage flux

For calculating the drainage flux according to De Zeeuw- Hellinga a constant storage coefficient is used. This storage coefficient, which is the same as for calculation the changes in groundwater level, is the storage coefficient corresponding with the initial groundwater level.

Option 'Unsaturated zone = CAPSIM'

Groundwater level

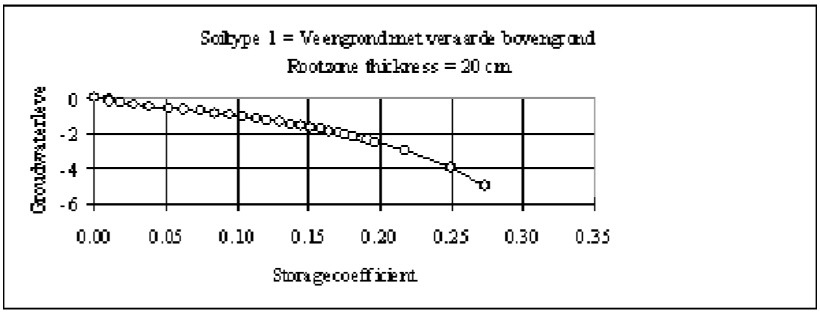
For calculating the groundwater level a storage coefficient is used depending on the actual groundwater level during the simulation period.

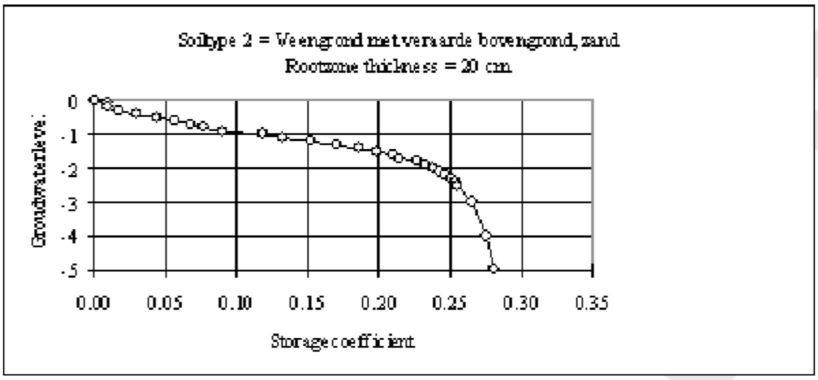
Drainage flux

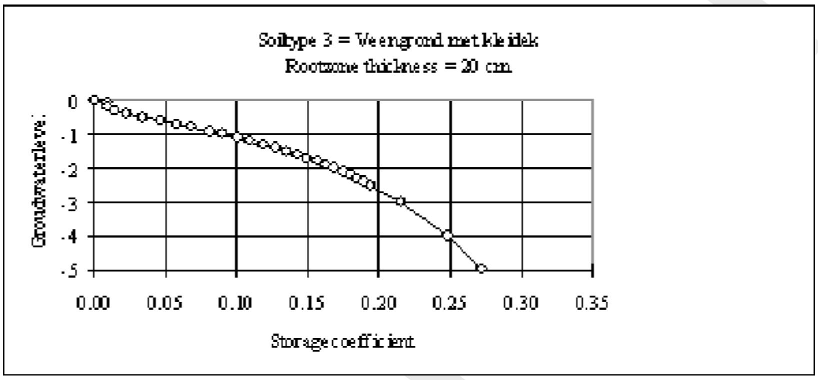
The storage coefficient is calculated by averaging all coefficients within in the domain of the groundwater level and the open water level. Since the groundwater level and the open water level may change during the simulation period, the average storage coefficient may change.

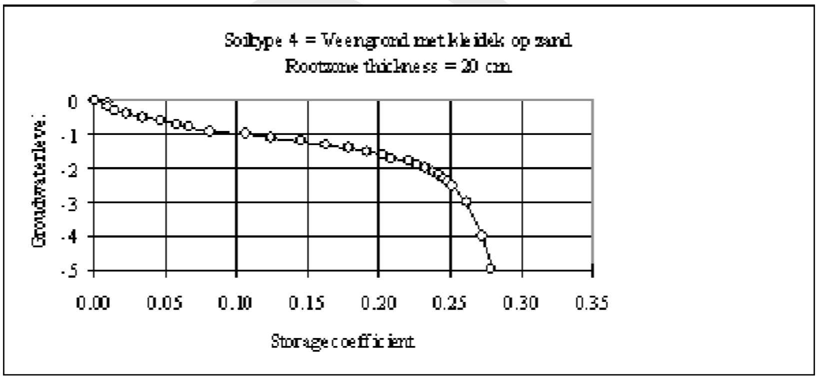
The coefficients for calculating the groundwater level and the drainage flux are tabulated in in SOBEK. These storage coefficients depend on the soil type, the root zone thickness end the groundwater level. Below the storage coefficients are depicted for a root zone thickness of 20 cm. SOBEK also takes into account storage coefficient for root zone thicknesses of 10 cm, 50 cm, 100 cm and 200 cm.

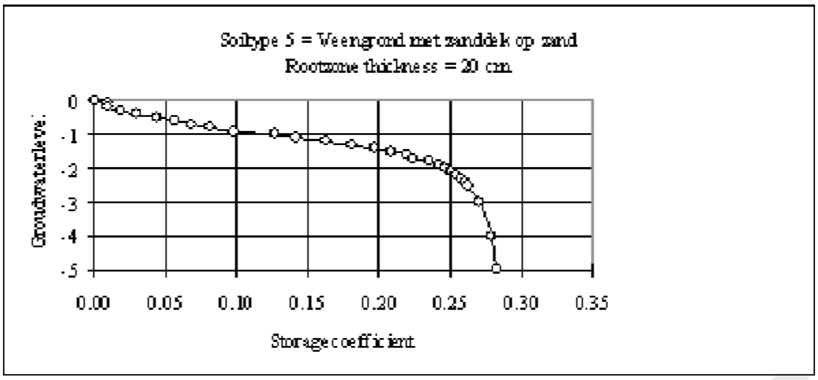
All coefficients are provided by Alterra.

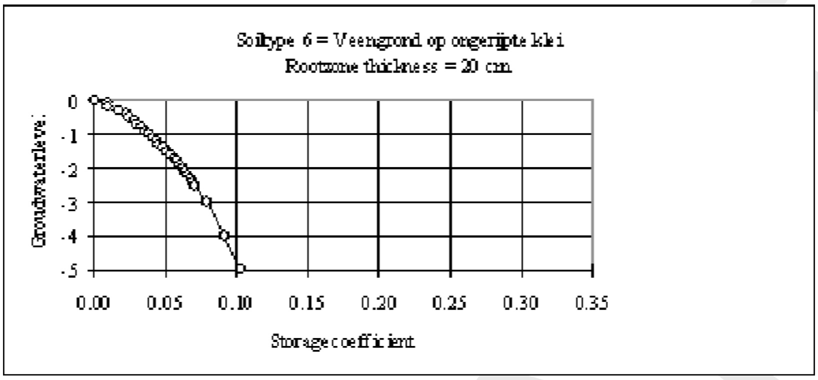

Soiltype 1 = Veengrond met veraarde bovengrond
Rootzone thickness = 20 cm.
Groundwaterleve
0
-2
-4
-6
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient

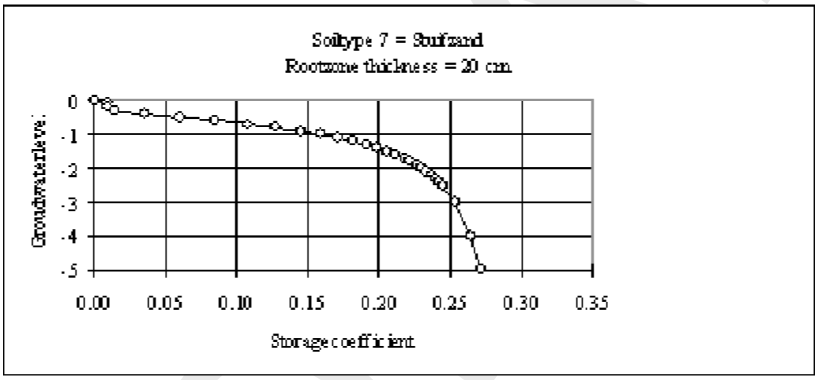

Soiltype 2 = Veengrond met veraarde bovengrond, zand
Rootzone thickness = 20 cm.
Groundwaterleve
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient

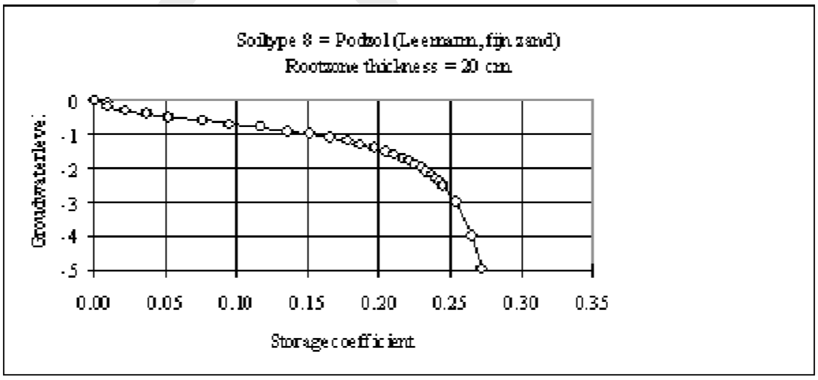

Soiltype 3 = Veengrond met kleidek
Rootzone thickness = 20 cm.
Groundwaterleve
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient

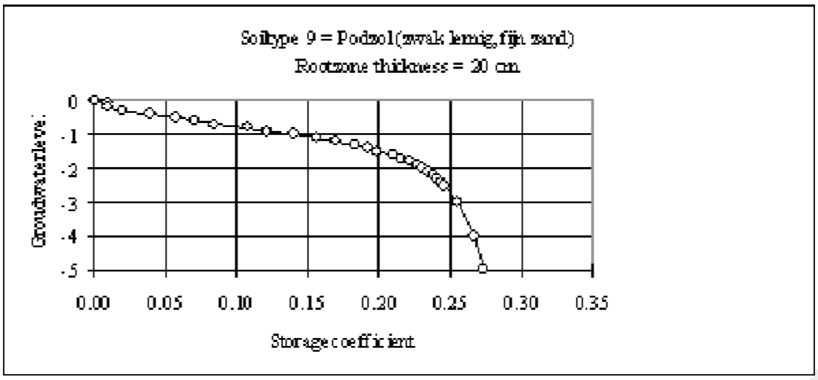

Soiltype 4 = Veengrond met kleidek op zand
Rootzone thickness = 20 cm.
Groundwaterleve
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient

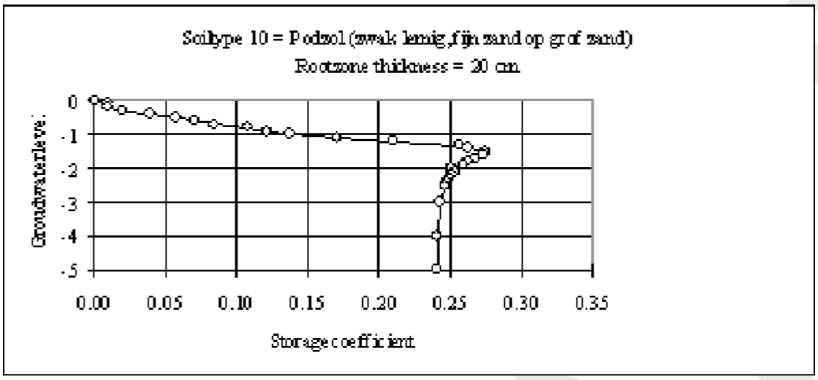

Soiltype 5 = Veengrond met zanddek op zand
Rootzone thickness = 20 cm
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00 0.05 0.10 0.15 0.20 0.25 0.30 0.35
Storage coefficient

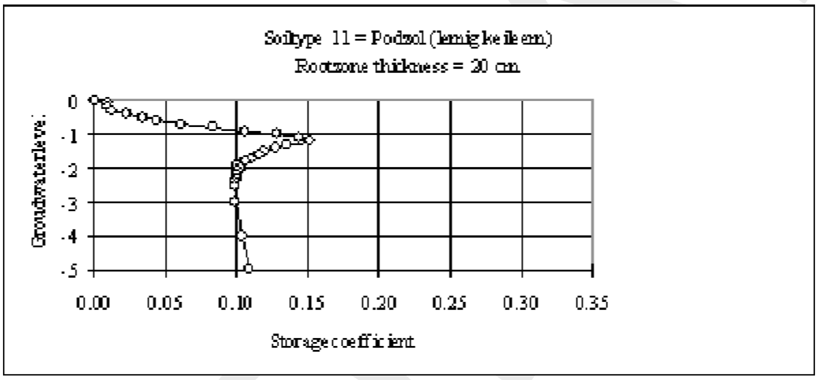

Soiltype 6 = Veengrond op ongerijpte klei
Rootzone thickness = 20 cm
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00 0.05 0.10 0.15 0.20 0.25 0.30 0.35
Storage coefficient

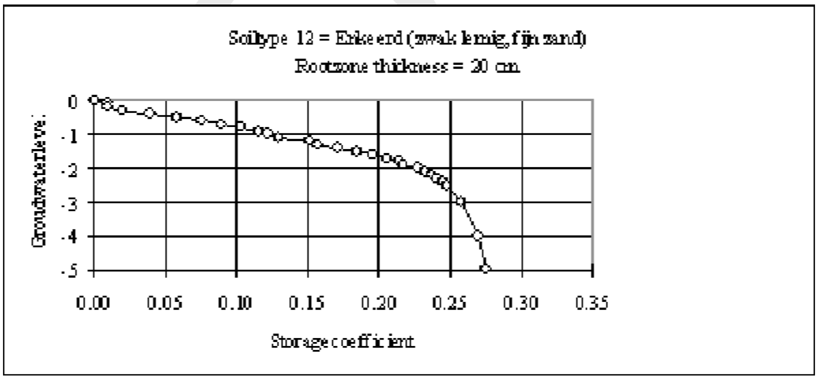

Soiltype 7 = Stuifzand
Rootzone thickness = 20 cm
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00 0.05 0.10 0.15 0.20 0.25 0.30 0.35
Storage coefficient

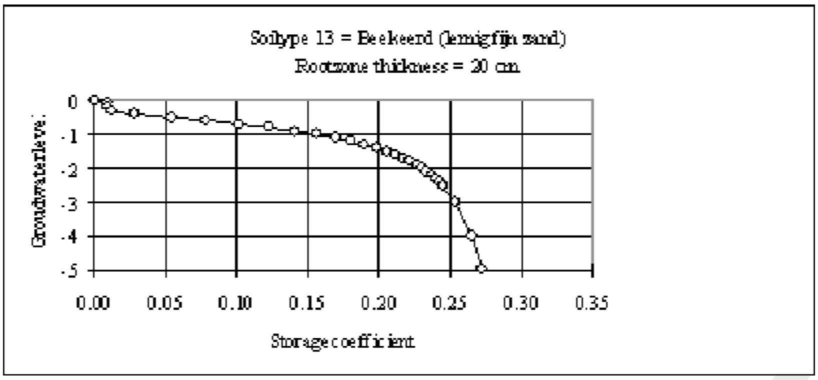

Soiltype 8 = Podzol (Leemarm, fijn zand)
Rootzone thickness = 20 cm
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00 0.05 0.10 0.15 0.20 0.25 0.30 0.35
Storage coefficient

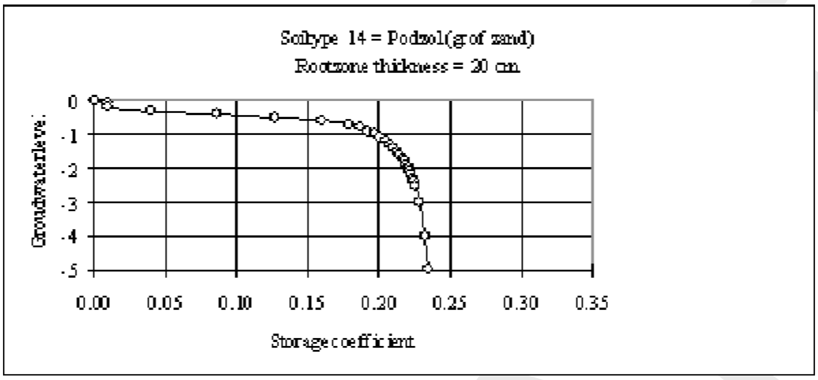

Soiltype 9 = Podzol (zwak lemig, fijn zand)
Rootzone thickness = 20 cm
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient

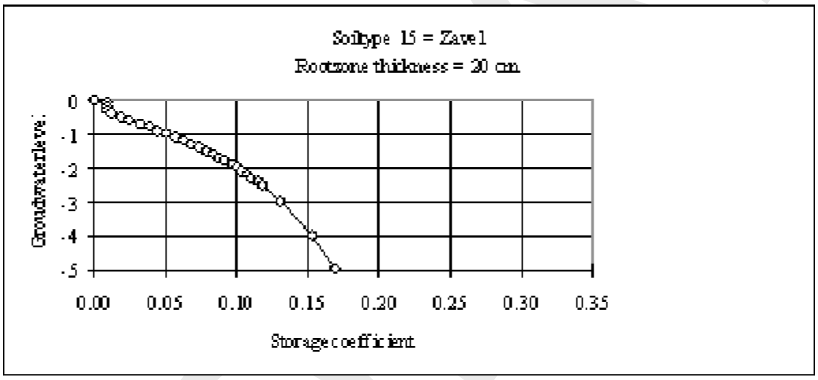

Soiltype 10 = Podzol (zwak lemig, fijn zand op grof zand)
Rootzone thickness = 20 cm
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient

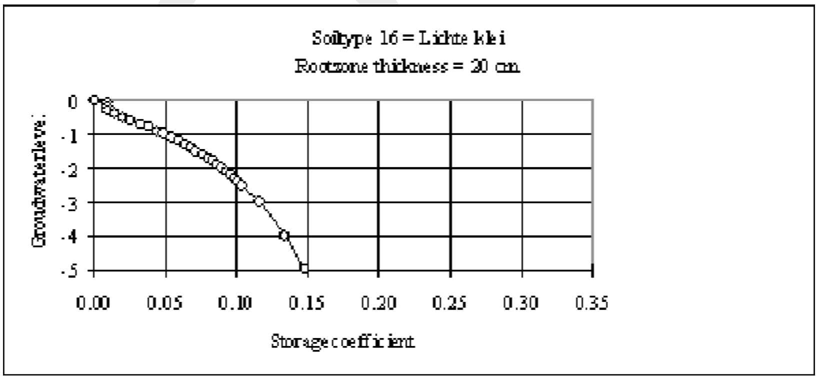

Soiltype 11 = Podzol (lemig keileem)
Rootzone thickness = 20 cm
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient

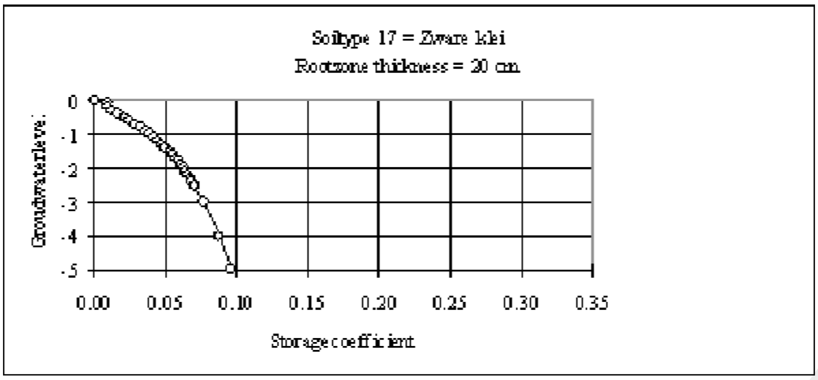

Soiltype 12 = Enkeerd (zwak lemig, fijn zand)
Rootzone thickness = 20 cm
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient

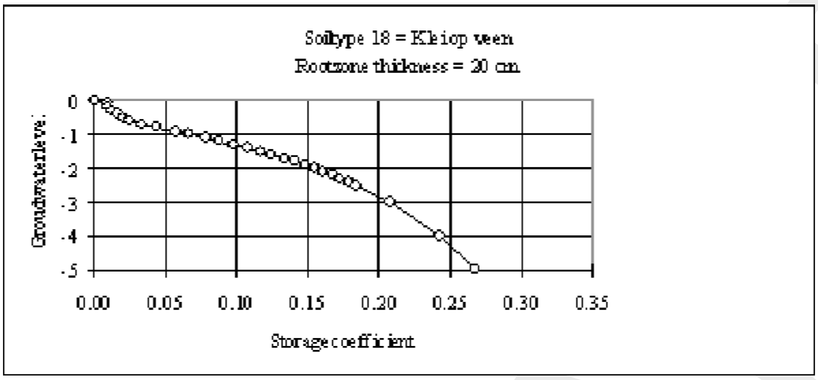

Soiltype 13 = Beekeerd (lemig fijn zand)
Rootzone thickness = 20 cm
Groundwaterlevel
Storage coefficient

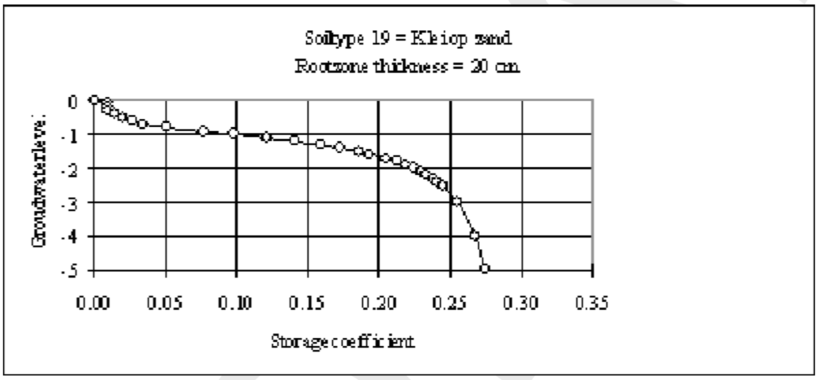

Soiltype 14 = Podzol (grof zand)
Rootzone thickness = 20 cm
Groundwaterlevel
Storage coefficient

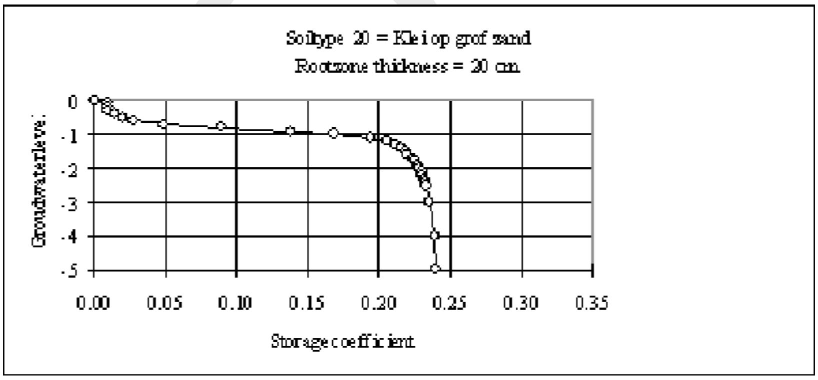

Soiltype 15 = Zavel
Rootzone thickness = 20 cm
Groundwaterlevel
Storage coefficient

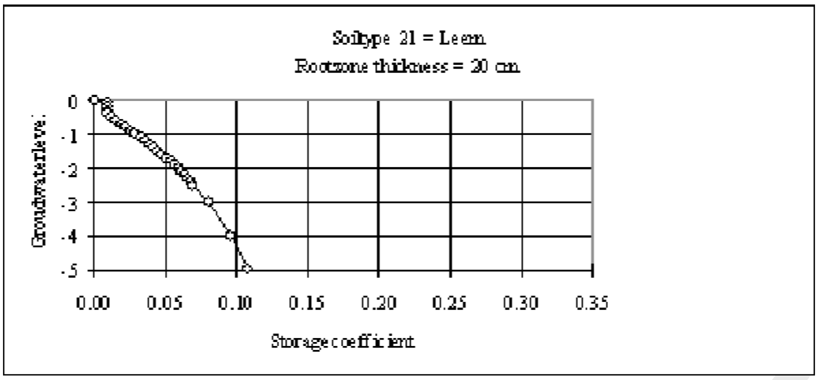

Soiltype 16 = Lichte klei
Rootzone thickness = 20 cm
Groundwaterlevel
Storage coefficient


Soiltype 17 = Zware klei
Rootzone thickness = 20 cm.
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient


Soiltype 18 = Klei op veen
Rootzone thickness = 20 cm.
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient


Soiltype 19 = Klei op zand
Rootzone thickness = 20 cm.
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient


Soiltype 20 = Klei op grof zand
Rootzone thickness = 20 cm.
Groundwaterlevel
0
-1
-2
-3
-4
-5
0.00
0.05
0.10
0.15
0.20
0.25
0.30
0.35
Storage coefficient



6.5.1.36 Surface runoff

Surface runoff in paved areas

In paved areas surface runoff occurs when the 'storage on street' reservoir is completely filled.

Surface runoff in unpaved areas

In unpaved nodes surface runoff to open water can occur in two cases:

1 when the 'storage on land' reservoir is filled by the precipitation (minus evaporation and infiltration into the soil).
2 when the groundwater level has branched the soil surface level. In this case, the storage on land has not been taken into account, so surface runoff will immediately take place when the groundwater level has branched the surface level.

In both cases the runoff process is simulated by means of the de Zeeuw-Hellinga equation. Therefore, the user must define a specific surface runoff reaction factor.

Notice that when the soil surface is defined as a constant level, the total area defined in the unpaved node is part of the surface runoff process. Often this causes very large discharges to the open water.

When the soil surface is defined as a variable level, only the inundated part of the soil surface is part of the surface runoff process.

6.5.1.37 Target level controller

With the target level controller one can control the upstream target level. The algorithm is given as follows:

User input:

- ◇ Maximum flow Q_{max}
- ◇ Maximum allowed upstream level MAXLup
- ◇ Maximum allowed downstream level MAXLdown
- ◇ Upstream target level TLup
- ◇ Downstream target level TLdown
- ◇ Parameters computed by SOBEK-RR:
- ◇ Upstream level L_{up}
- ◇ Downstream level L_{down}
- ◇ Weir flow Q

A weir with a target level controller tries to maintain the upstream water level at the user specified target level TLup. Upstream here means upstream with respect to the defined link direction. When the upstream water level gets above the target level TLup, the crest of the weir will be lowered in order to let the excess water flow away to the downstream node. The weir crest can be lowered until the maximum defined discharge Q_{max} is branched (or the crest branches is specified minimum level, which is by default the bed level).

The discharge is kept at or below the maximum defined discharge Q_{max} until the upstream water level branches the specified maximum allowed upstream level MAXLup. In that case, the specified maximum discharge Q_{max} may be exceeded, but only if the downstream water level is below the specified maximum allowed downstream water level MAXLdown.

In extreme conditions, the downstream level is allowed to exceed the specified maximum allowed downstream water level MAXLdown, but the upstream water level will not be allowed to rise above the maximum upstream water level MAXLup.

Note: that the controller only operates in case of upstream excess water; it is not active in case the upstream water level is below the target level.

The computation is done using the symbols as defined above, and additionally:

TimestepSize	computation time step size in seconds
UpLvlEst	estimated upstream level, without flow over the weir
DoLvlInit	downstream level at beginning of the computation timestep
VolNow	upstream volume in m^3 at level UpLvlEst
VolTarget	upstream volume in m^3 at upstream target level TLup
VolMax	upstream volume in m^3 at maximum allowed level MAXLup

A pseudo code of the computation is: (lines starting with ! are comments)

! Set upstream target end volume
`if` (DoLvlInit $>$ MAXLdown) `then`
    ! downstream water level above maximum allowed downstream level;
    ! upstream level may rise until maximum allowed upstream level
    $EndVolume = VolMax$
`else`
    ! normal situation: target end volume corresponding with upstream target level
    $EndVolume = VolTarget$
`endif`
! Set provisional flow over the weir to get to the target EndVolume
`if` (VolNow $>$ EndVolume) `then`
    $QOut = \min(Qmax, (VolNow - EndVolume)/TimestepSize)$
`else`
    $QOut = 0$
`endif`
! Set provisional upstream End volume
$EndVolume = VolNow - Qout * TimestepSize$
! Check provisional end volume with maximum allowed upstream level (volume):
! The upstream level may never exceed the maximum allowed upstream water level
`if` $EndVolume > VolMax$ `then`
    $QOut = \max(Qout, (VolNow - VolMax)/TimestepSize)$
`endif`
! Final computed weir flow
$Qout = \min(Qout, \mathit{flow\ with\ lowest\ possible\ crest\ level})$

6.5.1.38 Time step output

By default the time step of the results is equal to the time step of the computation. Especially when a small time step is chosen, this results in a very large amount of output values. Then the user can choose an other option in the Settings task in order to reduce the number of output values.

Output reduction options in Settings:

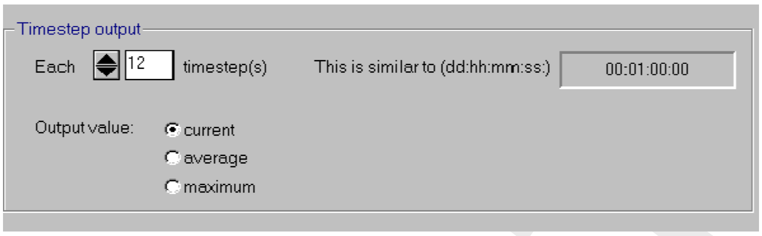


***Figure 6.91:** Output reductions in Settings*

In this case each 12th computed value is shown in the results graphs. The user can define any interval he wants.

Other options;

- ◇ Current: each nth computed value is written to the output file;
- ◇ Average: the average value of n computed values is written to the output file;
- ◇ Maximum: the maximum value of n computed values is written to the output file.

Note: the user should realise that the chosen options are of great importance to the shape of the output graph, see this example:

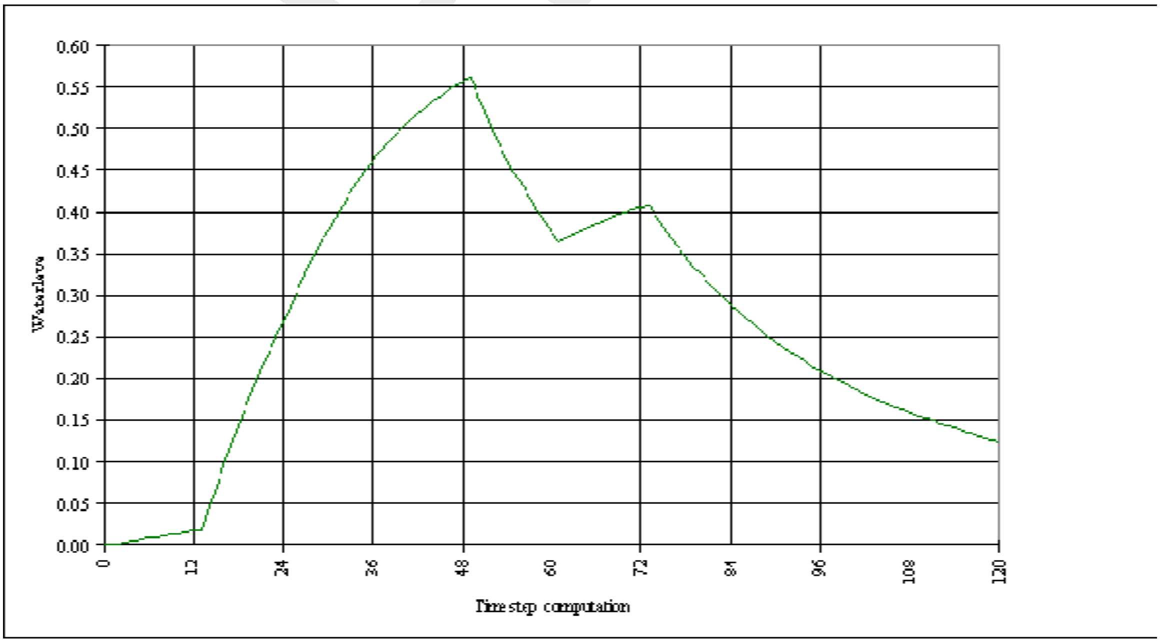


***Figure 6.92:** Time step computation*

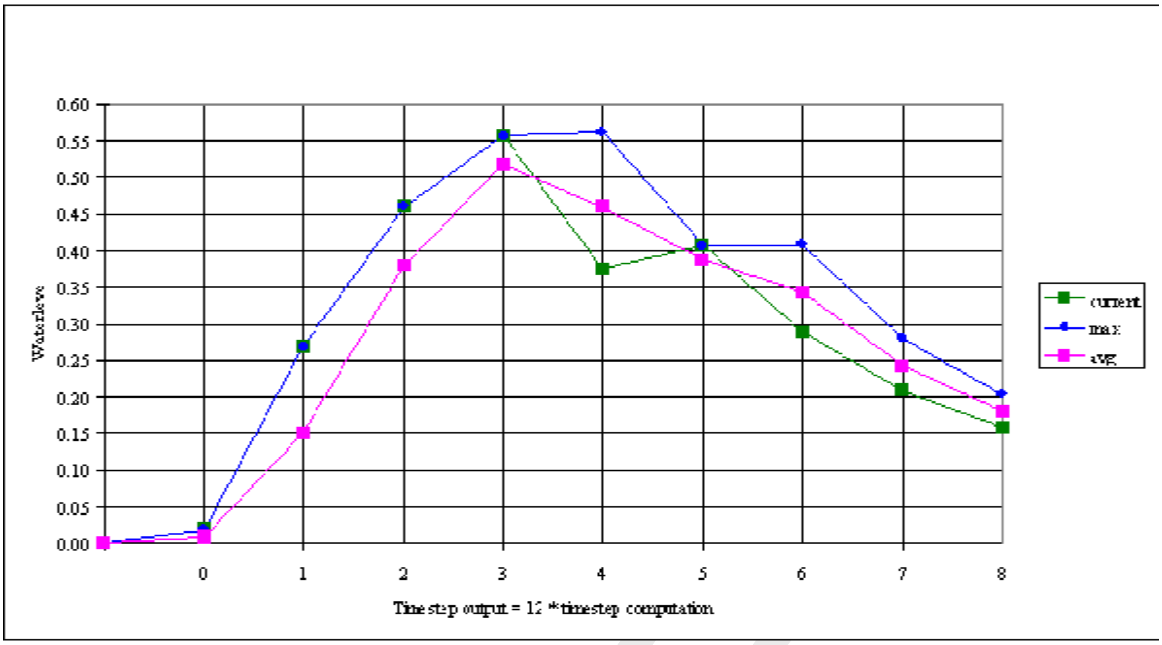


Figure 6.93: *Time step output = 12 ∗ time step computation*

When SOBEK is used to compute series of precipitation events, then the maximum realised level is written to the output file, based on all computed values. The options current, average and maximum are not relevant anymore.

6.5.1.39 Unpaved area node

General

The unpaved area node is used to simulate the rainfall-runoff process on unpaved areas.

An unpaved area is characterized by:

- ◇ total unpaved area (sum of crop areas)
- ◇ groundwater area
- ◇ area per crop
- ◇ soil surface level
- ◇ soil type
- ◇ storage coefficient
- ◇ root zone
- ◇ saturated zone
- ◇ evaporation
- ◇ capillary rise/percolation from root zone to groundwater
- ◇ storage on land
- ◇ infiltration capacity
- ◇ drainage resistance value / reaction factor (dependent on the chosen method)
- ◇ seepage/percolation (constant, variable in time or calculated from a defined groundwater head in the lower confined aquifer and the resistance value of the confining layer)
- ◇ surface runoff

The unpaved area is modeled using boxes representing storage on land, storage in the unsaturated zone, and the saturated zone. The unsaturated zone is optional. The surface area is divided into area for different crops. The sum of the crop area is the total unpaved area. This area is used for the surface storage and unsaturated zone computations. The saturated zone computations use the groundwater area. This area is by default equal to the total unpaved area, but can be defined separately by the user.

6.5.1.40 Unpaved surface flow link

The unpaved surface flow link is a branch type in RR which can be used to split the runoff from a RR-unpaved node into two components: groundwater outflow (drainage) and surface runoff. If an unpaved node has both a normal RR-link and an unpaved surface flow link connecting it to the 1DFlow schematisation, the RR link will carry the groundwater drainage, and the unpaved surface flow link will carry the surface runoff to the 1DFlow schematisation. In case no unpaved surface flow link is used, the RR link will transport the total flow (groundwater drainage plus surface runoff) to the 1DFlow schematisation.

This allows the user to specify different outflow locations for groundwater drainage and surface runoff, and also allows to specify different waste loads or background concentrations for the groundwater drainage and surface runoff in water quality computations.

The groundwater drainage computations use the open water level taken from the node downstream of the RR-link (remember that the driving force of the groundwater drainage is the head difference between this open water level and the groundwater level) , while the surface runoff computation uses the open water level taken from the node downstream of the RR-unpaved surface flow link. The surface runoff is stopped when the open water level exceeds the RR-unpaved surface level.

6.5.2 Sacramento Rainfall-Runoff model

6.5.2.1 Sacramento, the Segment module: implemented in SOBEK

The segment module simulates the rainfall-runoff process in part of the catchment, where the attention is on the land-phase of the rainfall-runoff process. It is assumed that the open water system in the segments contributes little to the shaping of the hydrograph. The conceptualisation of the processes as described in the segment module is presented in Figure 6.94.

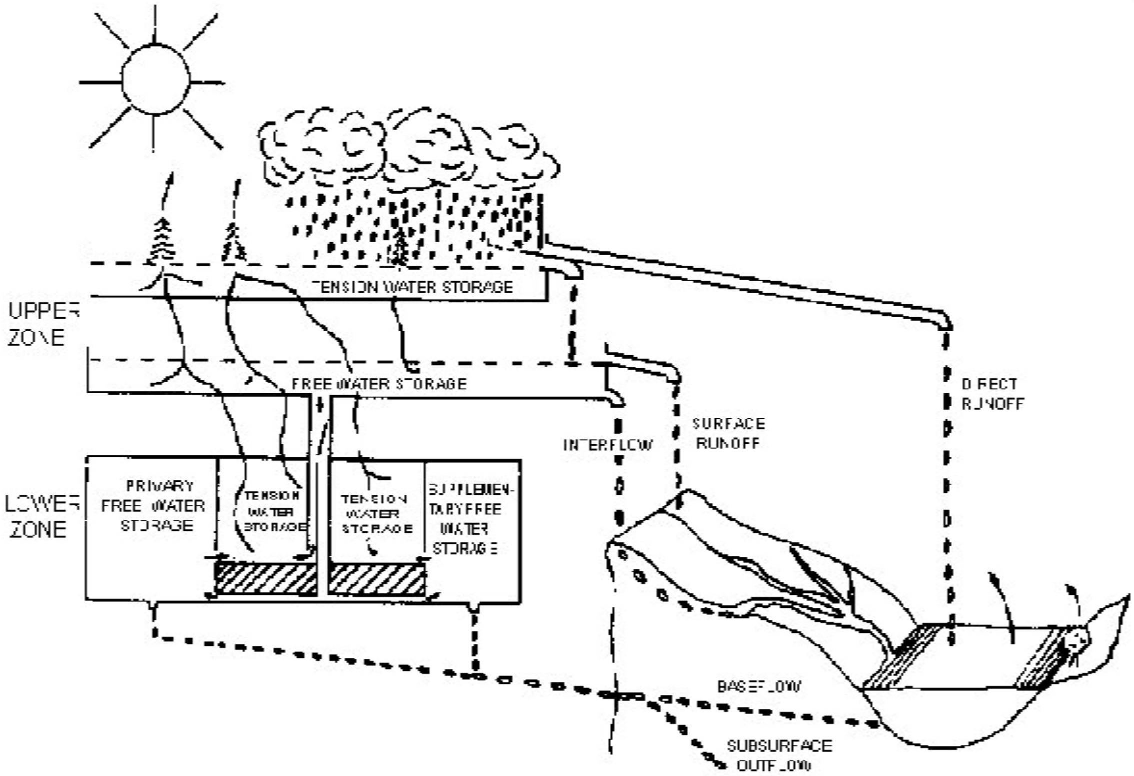


Figure 6.94: *Conceptualisation of the rainfall-runoff process in a segment.*

The segment module is divided into the following components, (see also Figure 6.95):

- ◇ Impervious area with transfers to direct runoff
- ◇ Pervious area
  - Upper zone
    - Tension storage with transfers to evaporation, free water storage
    - Free water storage with transfers to evaporation, percolation,
    - surface runoff and interflow
  - Lower zone
    - Tension storage with transfers to evaporation. free water storage
    - Free water storage with transfers to base flow

From the *impervious* areas, precipitation immediately discharges to the channel. However, impervious areas, which drain to a pervious part before branching the channel, are not considered impervious. Both zones have a tension and a free water storage element. Tension water is considered as the water closely bound to soil particles. Generally first the tension water requirements are fulfilled before water enters the free water storage.

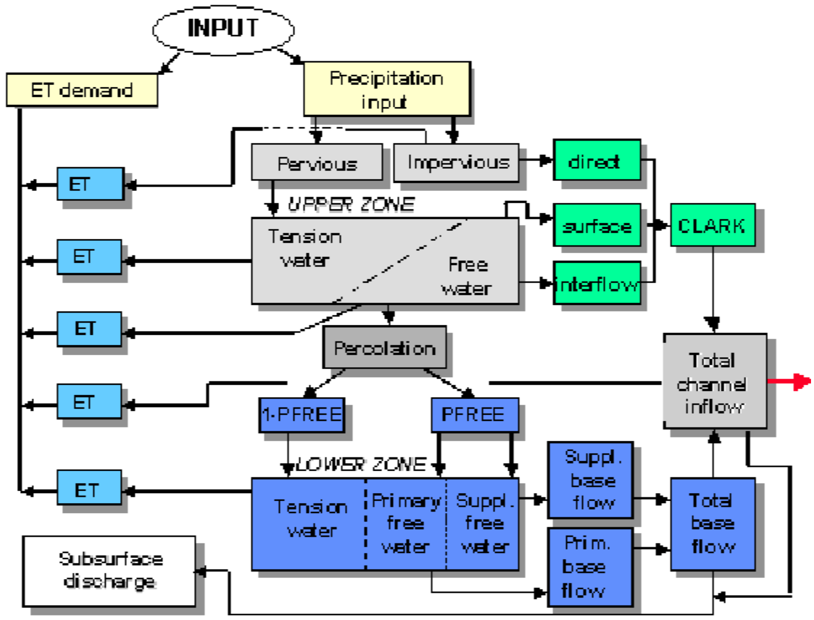


Figure 6.95: Schematisation of the rainfall-runoff process in a segment

In the following sub-sections the various components will be described in detail

6.5.2.2 Upper zone storage

The upper zone tension storage represents that precipitation volume required under dry conditions:

- to meet all interception requirements, and
- to provide sufficient moisture to the upper soil so that percolation can begin.

If the maximum storage capacity of the upper-zone tension storage is exceeded, water becomes available for the upper zone free water storage, a temporary storage from which water percolates to the lower zone system and from which water discharges to the channel via the interflow component. The preferred flow direction from the upper zone is the vertical direction, i.e. percolation to the lower zone system.

Interflow occurs only when the precipitation rate exceeds the percolation rate. The upper zone is treated as a linear storage element which is emptied exponentially: discharge = storage $*$ storage depletion coefficient. The upper zone free water storage depletion coefficient is denoted by UZK and the upper zone free water content by *UZFWC* then the interflow takes place at a rate:

$$Q_{\text{interflow}} = UZFWC * UZK \tag{6.312}$$

When the precipitation intensity exceeds the percolation intensity and the maximum interflow drainage capacity, then the upper zone free water capacity ($UZFWM$) is completely filled and the excess precipitation causes surface runoff.

6.5.2.3 Lower zone storage

The lower zone consists of the

- ◇ tension water storage, i.e. the depth of water held by the lower zone soil after wetting and drainage (storage up to field capacity) and
- ◇ two free water storages: the primary and supplemental storage elements representing the storages leading to a slow and a fast groundwater flow component, respectively. The introduction of two free lower zone storages is made for greater flexibility in reproducing observed recession curves caused by groundwater flow.

6.5.2.4 Percolation from upper to lower zones

The percolation rate from the upper zone to the lower zone depends on the one hand on the lower zone demand, i.e. requirements determined by the lower zone water content relative to its capacity and on the other hand on the upper zone free water content relative to its capacity.

The lower zone percolation demand is denoted by $PERC_{act.dem}$. The upper zone free water content relative to its capacity is $UZFWC/UZFWM$. Hence, the actual percolation intensity then reads:

$$PERC = PERC_{act.dem} \times UZFWC/UZFWM \tag{6.313}$$

The lower zone percolation demand has a lower and an upper limit:

- ◇ the minimum lower zone percolation demand, and
- ◇ the maximum lower zone percolation demand.

The minimum lower zone percolation demand occurs when all three lower zone storages are completely filled. Then by continuity the percolation rate equals the groundwater flow rate from full primary and supplemental reservoirs. Denoting the minimum demand by *PBASE* then it follows:

$$PERC_{min.dem} = PBASE = LZFPM \times LZPK + LZFSM \times LZSK \tag{6.314}$$

where:

$LZFPM$	lower zone primary free water storage capacity
$LZFSM$	lower zone supplemental free water storage capacity
$LZPK$	drainage factor of primary storage
$LZSK$	drainage factor of supplemental storage

The maximum lower zone percolation demand takes place if the lower zone is completely dried out i.e. if its content = 0. Then the maximum percolation rate is expressed as a function of *PBASE*:

$$PERC_{max.dem} = PBASE(1 + ZPERC) \tag{6.315}$$

with: $ZPERC \gg 1$ usually.

The actual lower zone percolation demand depends on the lower zone content relative to its capacity. Computationally it means that *ZPERC* has to be multiplied by a function G of the relative lower zone water content such that this function:

- ◇ equals 1 in case of a completely dry lower zone
- ◇ equals 0 in case of a completely saturated lower zone

◇ represents an approximate exponential decay of the percolation rate in case of a continuous recharge.

In the Sacramento model this function has the following form:

$$G = \left(\frac{\sum(\text{lower zone capacities} - \text{lower zone content})}{\sum(\text{lower zone capacities})} \right)^{REXP} \tag{6.316}$$

and the *actual percolation demand* is given by (see Figure 6.96):

$$PERC_{act.dem} = PBASE(1 + ZPERC * G) \tag{6.317}$$

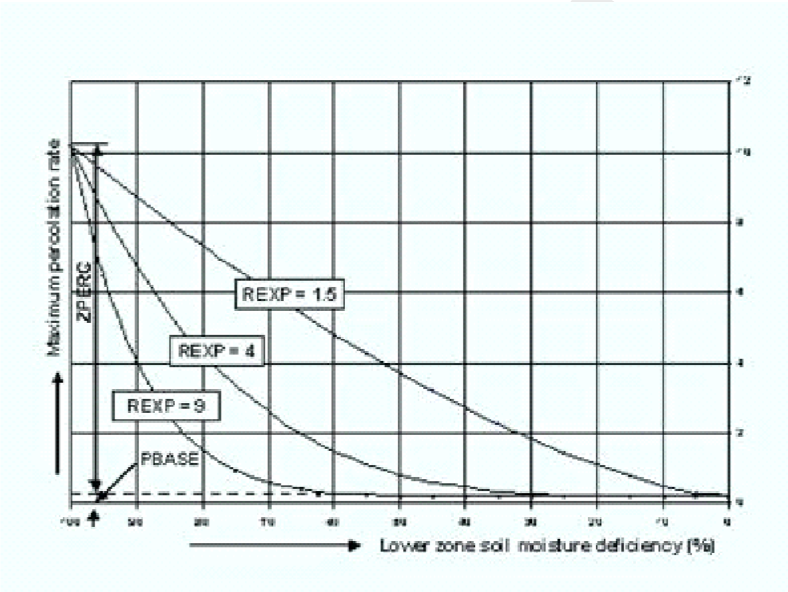


Figure 6.96: *Actual percolation demand representation*

6.5.2.5 Distribution of percolated water from upper zone

The percolated water drains to three reservoirs, one tension and two free water reservoirs. Based on the preceding comments one would expect that the lower zone tension storage is filled first before percolation to the lower zone free water storages takes place. However, variations in soil conditions and in precipitation amounts over the catchment cause deviations from the average conditions. This implies that percolation to the free water reservoirs and hence groundwater flow takes place before the tension water reservoir is completely filled. The model allows for this to let a fraction of the infiltrated water percolate to the two free water storages. When the tension water reservoir is full, all percolated water drains to the primary and supplemental free water storage in a ratio corresponding to their relative deficiencies.

6.5.2.6 Groundwater flow

Base flow to the river from groundwater depends on the contents of the two lower zone free water storages and two drainage constants expressed in fractions of the content per day. If the actual contents of the primary and supplemental free water zones are denoted by *LZFPC* and *LZFSC* respectively then the total base flow QBASE becomes, in accordance with the linear reservoir theory:

$$QBASE = LZFPC \times LZPK + LZFSC \times LZSK \tag{6.318}$$

The drainage factors $LZPK$ and $LZSK$ can be determined from the recession part of the hydrograph by plotting that part of the hydrograph on semi-logarithmic paper (Fig. 6).

In the lowest part of the recession curve only the slow base flow component is acting while in the higher stages both base flow components contribute.

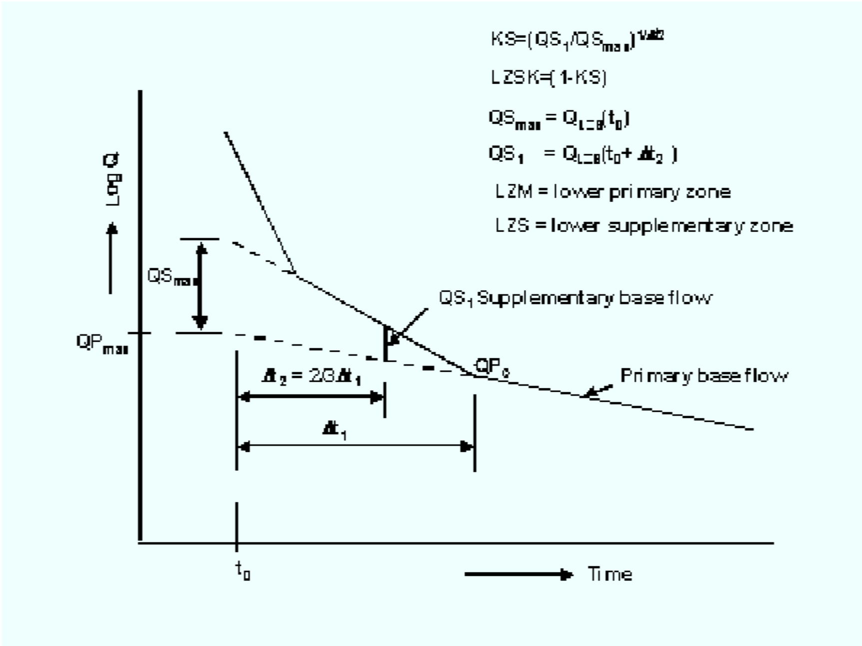


Figure 6.97: *Principle of computation of lower zone recession coefficient*

The drainage factor $LZPK$ follows from:

$$K = \left(\frac{QP_{t0+\Delta t}}{QP_{t0}}\right)^{1/\Delta t} \tag{6.319}$$

and

$$LZPK = 1 - K \tag{6.320}$$

where:

K	recession coefficient of primary base flow for the time unit used
Δt	number of time units, generally days
$QP_{t0+\Delta t}$	a discharge when recession is occurring at the primary base flow rate
QP_{t0}	the discharge t time units later

If QP_{max} represents the maximum value of the primary base flow, then the maximum water content of the lower zone becomes:

$$LZFPM = \frac{QP_{max}}{LZPK} \tag{6.321}$$

and similarly the supplemental lower zone free water capacity is determined; at least this procedure provides first estimates of the lower zone free water capacities (Figure 6.97).

The total base flow contributes completely or in part to the channel flow. A complete contribution occurs if subsurface discharge (i.e. discharge from the segment, which is not measured at the outlet) is absent. Otherwise a fraction of the total base flow represents the subsurface flow.

6.5.2.7 Actual evapotranspiration

Evaporation at a potential rate occurs from that fraction of the basin covered by streams, lakes and riparian vegetation. Evapotranspiration from the remaining part of the catchment is determined by the relative water contents of the tension water zones. If *ED* is the potential evapotranspiration, then the actual evapotranspiration from the upper zone reads:

$$E_1 = ED \times \frac{UZTWC}{UZTWM} \tag{6.322}$$

i.e. the actual rate is a linear function of the relative upper zone water content. Where $E_1 < ED$ water is subtracted from the lower zone as a function of the lower zone tension water content relative to the tension water capacity:

$$E_2 = (ED - E_1)\frac{LZTWC}{UZTWM + LZTWM} \tag{6.323}$$

If the evapotranspiration should occur at such a rate that the ratio of content to capacity of the free water reservoirs exceeds the relative tension reservoir content then water is transferred from free water to tension water such that the relative loadings balance. This correction is made for the upper and lower zone separately. However, a fraction $RSERV$ of the lower zone free water storage is unavailable for transpiration purposes.

6.5.2.8 Impervious and temporary impervious areas

Besides runoff from the pervious area, the channel may be filled by rainwater from the impervious area. With respect to the size of the impervious area it is noted that in the Sacramento model a distinction is made between **permanent and temporary impervious areas** where temporary impervious areas are created when all tension water requirements are met, i.e. an increasing fraction of the catchment assumes impervious characteristics.

6.5.2.9 Routing of surface runoff

Before the runoff from the impervious areas, the overland- and interflow branch the channel, they may be transformed according to a unit hydrograph leading to an adapted time distribution of these flow rates.

Use can be made here of the Clark method, which is a combined time-area and storage routing method. The model requires the construction of a time-area diagram. For this isochrones are constructed representing points of equal travel time to the segment outlet, see Figure 5. The areas between successive isochrones is determined and subsequently properly scaled by the time of concentration T_c. The latter is defined as the time required to have the effect of rainfall fallen in the most remote part felt at the segment outlet. The time-area diagram can be thought of as the outflow from the segment if only translation and no deformation takes place of an instantaneous unit supply of rain over the entire segment. Subsequently, the time area diagram flow is routed through a linear reservoir, which characterises the effect of storage in the open drainage system of the segment. This reservoir is represented by the second parameter: the recession coefficient k. It is noted that the output from the reservoir represents the instantaneous unit hydrograph (IUH). This has to be transformed into say a 1-hour unit hydrograph, dependent on the chosen routing interval

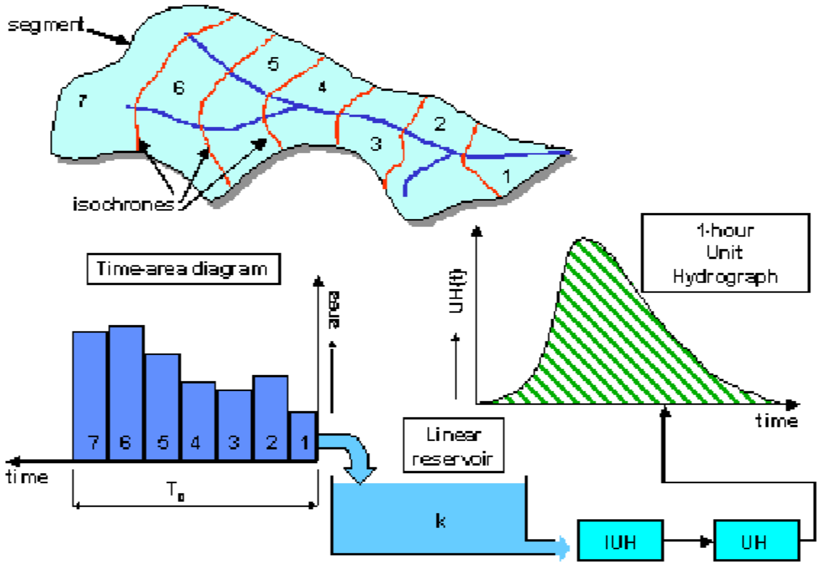


Figure 6.98: *Principles of the Clark method for simulating surface runoff and interflow.*

The two parameters T_c and k can be obtained from observed rainfall and discharge hydrographs. The time of concentration is equal to the time interval between cessation of rainfall and the time the hydrograph has receded to its inflection point. Alternatively it is determined from physical features of the segment as length and slope. A large number of empirical formulas are available which relate the time of concentration to topographical features of the basin. It is noted, though, that these formulas have generally only local validity. The best is to estimate the celerity from the flow velocities in the drainage system taking account of the following characteristics of celerity:

- ◇ If the rivulet remains inbank the celerity is about 1.5 to 1.7 times the cross-sectional flow velocity
- ◇ If the flow becomes overbank the above celerity has to be multiplied with the ratio of the drain width and the total width of the flow at the water surface (i.e. inclusive of the floodplain)

To the time required to travel through the drainage system one has to add the overland flow time.

The recession coefficient k is determined from the slope of recession part of the surface runoff hydrograph, similar to the procedure for groundwater.

6.5.2.10 Sacramento - Estimation of segment parameters

Overview of parameters

The following groups of parameters can be distinguished for a particular segment:

◇ Segment

[-] Segment area [km^2]

◇ Direct runoff

PCTIM	Permanently impervious fraction of segment contiguous with stream channels
ADIMP	Additional impervious fraction when all tension water requirements are met
SARVA	Fraction of segment covered by streams, lakes and riparian vegetation

◇ Upper soil moisture zone

UZTWM	Capacity of upper tension water zone [mm]
UZFWM	Capacity of upper free water zone [mm]
UZK	Upper zone lateral drainage rate (fraction of contents per day)

◇ Percolation

ZPERC	Proportional increase in percolation from saturated to dry conditions in lower zone
REXP	Exponent in percolation equation, determining the rate at which percolation demand changes from dry to wet conditions

◇ Lower zone

LZTWM	Capacity of lower zone tension water storage [mm]
LZFPM	Capacity of lower zone primary free water storage [mm]
LZFSM	Capacity of lower zone supplemental free water storage [mm]
LZPK	Drainage rate of lower zone primary free water storage (fraction of contents per day)
LZSK	Drainage rate of lower zone supplemental free water storage (fraction of contents per day)
PFREE	Fraction of percolated water, which drains directly to lower zone free water storages
RSERV	Fraction of lower zone free water storages which is unavailable for transpiration purposes
SIDE	Ratio of unobserved to observed base flow
SSOUT	Fixed rate of discharge lost from the total channel flow [$mm/\Delta t$]

◇ Surface runoff

[-] Unit hydrograph ordinates

◇ Internal routing interval

PM	Time interval increment parameter
PT1	Lower rainfall threshold
PT2	Upper rainfall threshold

Basically two procedures are available to get first estimates for the majority of the segment parameters:

◇ from observed rainfall and runoff records: this method is usually applied and works well provided that the model concepts are applicable and that reliable records are available for some time covering the majority of the range of flows
◇ from soil characteristics: this method is particularly suitable if no runoff records are available, i.e. for ungauged catchments.

With respect to gauged catchments the following grouping of parameters according to the method of estimation can be made:

- ◇ Parameters computed and estimated from basin map solely:
  segment area and SARVA
- ◇ Parameters estimated from observed rainfall and runoff records:
  readily: LZFPM, LZPK, LZFSM, LZSK, PCTIM
  approximately: UZTWM, UZFWM, UZK, LZTWM, SSOUT and PFREE
- ◇ Parameters estimated from topographic maps and rainfall and runoff records:
  unit hydrograph ordinates obtained from Clark method
- ◇ Parameters to be obtained through trial runs:
  ZPERC, REXP, SIDE, ADIMP, RSERV
- ◇ Internal routing parameters, as per requirement:
  PM, PT1, PT2
  In the next sub-sections guidelines are given for the determination and estimation of the segment parameters for gauged catchments.

6.5.2.11 Segment parameter estimation for gauged catchments.

The estimation of the segment parameters is presented according to their order of appearance in the previous sub-section. The sequence in which the estimation is done in practice is different from this order, for which reference is made to the end of the sub-section.

Segment

Segment area

To allow a good comparison between the observed and simulated runoff from the basin, the segment area (km^2) should refer to the total segment area draining upstream of the gauging station. Any difference between total segment area up to the main stream and the area upstream of the gauging station can be accommodated for in the channel routing part.

Direct runoff

PCTIM

Permanently impervious fraction of the basin contiguous with stream channels. It can be determined from small storms after a significant period of dry weather. Then the volume of direct runoff (= observed runoff - base flow) divided by the volume of rain gives the percentage impervious fraction of the basin. *PCTIM* should not be close to 1!

An example is given below.

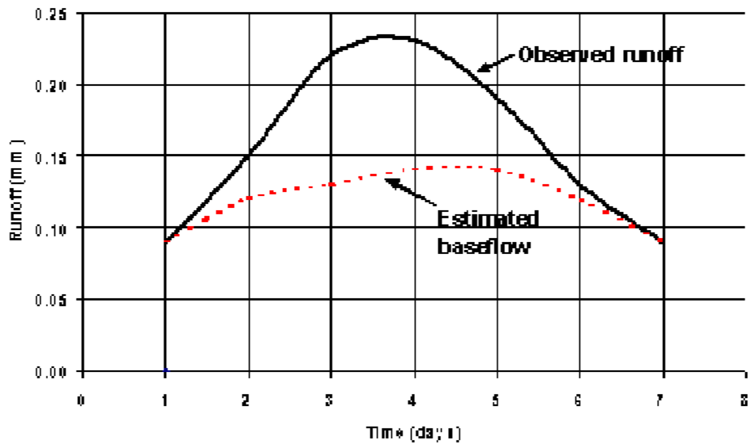


Figure 6.99: *Calculation of PCTIM*

ADIMP

Fraction of the basin, which becomes impervious as all tension water requirements are met. It can be estimated from small storms after a very wet period. As before, the volume of direct runoff divided by the volume of rain gives the total percentage of impervious area. The estimate for $ADIMP$ follows from:

$$ADIMP = \textbf{Total Percentage Impervious} - PCTIM \tag{6.324}$$

SARVA

Fraction of the basin covered by streams, lakes, and riparian vegetation, under normal circumstances. The *SARVA* area is considered to be the same as or less than *PCTIM* (see below). Detailed maps may be referred to in order to estimate the extent of paved areas, which drain directly to the streams so that differences between *PCTIM* and *SARVA* can be approximated. Generally, *SARVA* appears to range between 40 % and 100 % of the *PCTIM* value.

Upper soil moisture zone

UZTWM - the upper tension storage capacity

The depth of water, which must be filled over non-impervious areas before any water becomes available for free water storage. Since upper zone tension water must be filled before any stream flow in excess of the impervious response can occur, its capacity can be approximated from hydrograph analysis. Following a dry period when evapotranspiration has depleted the upper soil moisture, the capacity of upper zone tension water can be estimated. That volume of rainfall, which is retained before runoff from the pervious fraction is visible, is identified as *UZTWM.* To that rainfall volume the losses to evaporation during the considered period should be added. All periods of rain following a dry period should be checked for estimation of this parameter. Generally the capacity of the upper zone tension will vary between 25 and 175 mm, depending on the soil type.

Following the logic of the Curve Number method, where the initial abstraction before rainfall becomes effective is estimated as 20 % of the potential maximum retention, the *UZTWM* becomes:

$$UZTWM = 50.8\left(\frac{100}{CN} - 1\right)\ [mm] \tag{6.325}$$

CN-values range from 30 to about 90 for rural areas and are a function of:

- ◇ soil type (soil texture and infiltration rate); hydrological soil groups A–D are distinguished
- ◇ land use, type of land cover, treatment and hydrologic or drainage condition

It is also a function of antecedent moisture condition, for which the condition "dry" should be taken in view of the meaning of $UZTWM$. Based on this assumption $UZTWM$ would vary between 120 and 6 mm, values which are in the range of those given above, particularly if one realises that the 20 % of the potential maximum as initial abstraction is an average value. Reference is made to standard textbooks on hydrology for CN-values

UZFWM - the upper free water storage capacity

Upper zone free water represents that depth of water, which must be filled over the non-impervious portion of the basin in excess of $UZTWM$ in order to maintain a wetting front at maximum potential. This volume provides the head function in the percolation equation and also establishes that volume of water, which is subject to interflow drainage. Generally its magnitude ranges from 10–100 mm. It is not generally feasible to derive the magnitude of the upper zone free water from direct observations, and successive computer runs are required in order to establish a valid depth.

However, if a rough estimate of UZK is available (see below), then a rough value of $UZFWM$ can be obtained from the hydrograph at the time of the highest interflow, by reducing the flow value with primary and supplemental base flow.

UZK - the upper zone lateral drainage rate

The upper zone lateral drainage rate is expressed as the ratio of the daily withdrawal to the available contents. Its range is roughly 0.18 to 1.0, with 0.40 generally serving as an effective initial estimate. Though basically, this factor is not capable of direct observation and must be determined by successive computer runs, Peck (1978) suggests the following approximate procedure. UZK is roughly related to the amount of time that interflow occurs following a period with major direct and surface runoff. A long period of interflow results in a small value for UZK. Assuming that interflow is observed during N consecutive days and that interflow becomes insignificant when it is reduced to less than 10 % of its maximum value it follows:

$$(1 - UZK)^N = 0.10 \quad \text{or} \quad UZK = 1 - 0.1^{1/N} \tag{6.326}$$

Values for UZK as a function of N can be read from Figure 6.100.

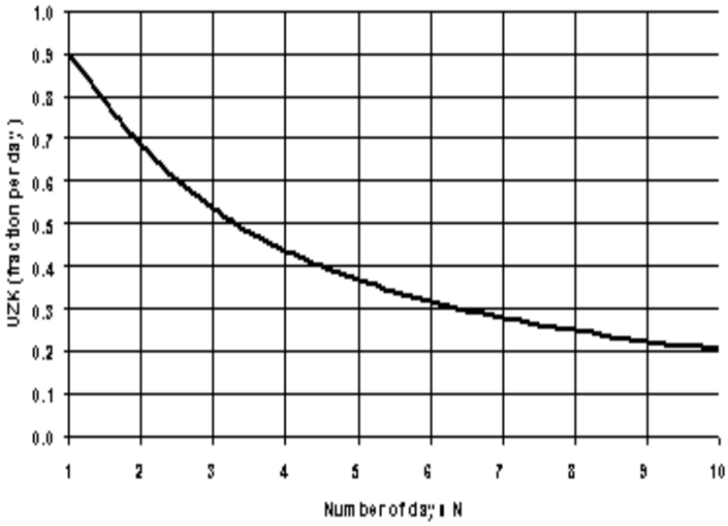


Figure 6.100: *UZK as function of number of days with significant interflow*

Percolation

ZPERC

The proportional increase in percolation from saturated to dry condition is expressed by the term *ZPERC*. The value of *ZPERC* is best determined through computer trials. The initial estimate can be derived by sequentially running one or two months containing significant hydrograph response following a dry period. The value of *ZPERC* should be initially established so that a reasonable determination of the initial run-off conditions is possible.

Amstrong (1978) provides a procedure to derive *ZPERC* from the lower zone tension and free water reservoir capacities and drainage rates, using Equation (6.314) and Equation (6.315). The maximum percolation takes place when the upper zones are full and the lower zones are empty. Assuming that the maximum daily percolation will be the maximum contents of the lower zones, from equation Equation (6.315) it follows for *ZPERC*:

$$ZPERC = \frac{LZTWM + LZFPM + LZFSM - PBASE}{PBASE} \tag{6.327}$$

If data would be available on maximum percolation rates *ZPERC* can be estimated using equation Equation (6.315). Values for *ZPERC* ranging from 5 to 80 have been used.

REXP

The exponent in the percolation equation which determines the rate at which percolation demand changes from the dry condition, $(ZPERC + 1)PBASE$, to the wet condition, *PBASE*. Figure 6.98 illustrates how different values of the exponent affect the infiltration rate. It is recommended that an initial estimate of this exponent is made from the same record which is used in determining an initial estimate of *ZPERC*. The interaction between *PBASE*, *ZPERC* and *REXP* may require a shift of all three terms whenever it becomes clear that a single term should be changed. Visualising the percolation curve generated by these three terms helps to ascertain the necessary changes. The observed range of *REXP* is usually between 1.0 and 3.0. Generally a value of about 1.8 is an effective starting condition. Values for *REXP* for different soils are given by Amstrong (1978) and are presented in Table 6.11.

Table 6.11: Percolation exponent REXP for different soil types

Soil classification	REXP
Sand	1.0
Sandy loam	1.5
Loam	2.0
Silty loam	3.0
Clay, silt	4.0

Lower zone

LZTWM - lower zone tension water capacity [mm]

This volume is one of the most difficult values to determine effectively. In as much as carryover moisture in this storage may exist for a period of many years, its total capacity may not be readily discernible from available records. If a drought condition during the period of record in the basin or in the area being studied has been sufficient to seriously affect the transpiration process of deep rooted plants, then the period of record is usually sufficient to determine the maximum storage value of lower zone tension water. Often, however, field data is not adequate for this purpose. As a result, unless great care is taken, the depth of lower zone tension water storage may inadvertently be set near the maximum deficit experienced during the period of record rather than the actual capacity of the zone. It has been noted that the plant growth of an area is a relatively effective indicator of the capacity of the lower zone tension water zone. In heavily forested regions of deep-rooted conifers, this zone may be approximately 600 mm in magnitude. In areas of deep-rooted perennial grasses this depth is more likely to be close to 150 mm. Where vegetation is composed primarily of relatively shallow-rooted trees and grasses, this depth may be as little as 75 mm. It should be realised that this zone represents that volume of water, which will be tapped by existing plants during dry periods.

An approximate procedure to estimate *LZTWM* from a water balance analysis is presented by Peck (1978). For this a period is selected with direct and/or surface runoff following an extended dry spell. The selected period is bounded by the times t_1 and t_2. At both times t_1 and t_2 only base flow occurs. The start t_1 is selected immediately prior to the occurrence of direct/surface runoff and t_2 immediately following a period of interflow. The times t_1 and t_2 can best be selected from a semi-log plot of the runoff, see Figure 6.101.

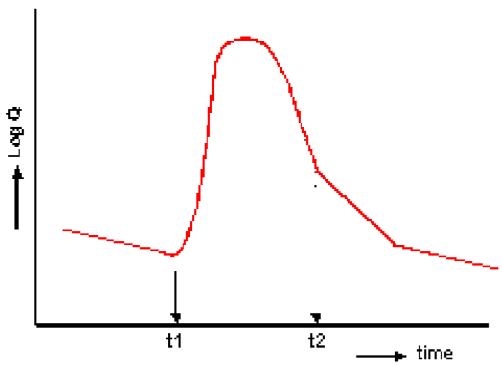


Figure 6.101: *Selection of period for LZTWM estimation*

Assuming that *UZTW* is full and *UZFWC* is empty at times t_1 and t_2 the water balance for the period $t_1 - t_2$ then reads:

$$P - R - E - DLZFPC - DLZFSC = DLZTWC \tag{6.328}$$

where:

P	precipitation from t_1 to t_2 $[mm]$
R	total runoff from t_1 to t_2 $[mm]$
E	segment evaporation $[mm]$; this amount would small for most wet period and may be neglected
$DLZFPC$	change in storage in LZ primary free water reservoir from t_1 to t_2 $[mm]$
$DLZFSC$	change in storage in LZ supplemental free water reservoir from t_1 to t_2 $[mm]$
$DLZTWC$	change in the lower zone tension water $[mm]$

DLZTWC is a lower limit of *LZTWM* since:

1 The lower zone tension water reservoir may not have been fully empty at t_1
2 The lower zone tension water reservoir may not have been completely filled at t_2

Hence some 10 to 20 % (or more) may be added to the value obtained through Equation (6.328). If such ideal cases as assumed above cannot be found, water balances for periods of 3 to 4 months may be considered.

In Equation (6.328) *LZFPC* and *DLZFSC* are computed as follows:

$$DLZFPC = LZFPC(t_2) - LZFPC(t_1) \quad \text{where} \quad LZFPC(t) = QP(t)/LZPK \tag{6.329}$$

$$DLZFSC = LZFSC(t_2) - LZFSC(t_1) \quad \text{where} \quad LZFSC(t) = QS(t)/LZSK \tag{6.330}$$

The primary baseflows QP at times t_1 and t_2 are estimated by extrapolation from other periods. Let the discharges at t_1 and t_2 be denoted by $Q1$ and $Q2$, then the supplemental baseflows follow from:

$$QS(t_1) = Q1 - QP(t_1) \quad \text{and} \quad QS(t_2) = Q2 - QP(t_2) \tag{6.331}$$

LZFPM - lower zone primary free water storage

The maximum capacity of the primary lower zone free water, which is subject to drainage at the rate expressed by $LZPK$. The value of the lower zone primary free water maximum can be approximated from hydrograph analysis. For this the primary base flow, obtained from a semi-log plot of the lower end of the recession curve, is extended backward to the occurrence of a peak flow. Assuming that the primary free water reservoir is completely filled then, so that it outflow is at maximum ($QPmax$), its value is determined from Equation (6.321). The effectiveness of this computation in determining the maximum capacity is dependent upon the degree to which the observed hydrograph provides a representation of the maximum primary base flow. If only a portion of the groundwater discharge is observable in the stream channel, the estimated capacity based upon surface flows must be increased to include the non-channel components by applying the term *SIDE* (See below).

LZFSM - lower zone supplemental free water storage

The maximum capacity of the lower zone supplemental free water reservoir, which is subject to drainage at the rate expressed by *LZSK*. A lower limit of the lower zone free water supplemental maximum can be approximated from hydrograph analysis. Figure 6.97 illustrates the computation of the lower zone free water supplemental maximum. Note that first the primary base flow has to be identified and corrected for, see also Equation (6.331). The effectiveness of this computation in determining the maximum capacity is dependent upon the degree to which the observed hydrograph provides a representation of the maximum base flow capability of the basin. If only a portion of the groundwater discharge is observable in the stream channel, the estimated capacity based upon surface flows must be increased to include the non-channel components by applying the term *SIDE* (See below).

LZPK - lateral drainage of the lower zone primary free water reservoir.

Lateral drainage rate of the lower zone primary free water reservoir expressed as a fraction of the contents per day. The coefficient is determined from the primary base flow recession curve. Selecting flow values from this curve at some time interval Dt apart provides with the help of Equation (6.319) and Equation (6.320) the required estimate, see also Figure 6.97.

LZSK - lateral drainage of the lower zone supplemental free water reservoir

Lateral drainage rate of the lower zone supplemental free water reservoir, expressed as a fraction of the contents per day. Its computation is outlined in Figure 6.97. The procedure is similar to that of *LZPK*, with the exception that the flow values have to be corrected for the primary base flow.

PFREE

The fraction of the percolated water, which is transmitted directly to the lower zone free water aquifers. Its magnitude cannot generally be determined from hydrograph analysis. An initial value of 0.20 is suggested. Values will range between 0 and 0.50. The analysis of early season base flow allows an effective determination of *PFREE*. The relative importance of *PFREE* can be determined from storms following long dry spells that produce runoff (*UZTW* completely filled). If the hydrograph returns to approximately the same base flow as before then little filling of the lower zone free water reservoirs did take place and hence the *PFREE*-value can be rated small, 0 to 0.2. If, on the contrary, the base flow has increased significantly a *PFREE*-value as high as 0.5 may be applicable.

RSERV

Fraction of the lower zone free water, which is unavailable for transpiration purposes. Generally this value is between zero and 0.40 with 0.30 being the most common value. This factor has very low sensitivity.

SIDE

Represents that portion of base flow, which is not observed in the stream channel. When the soil is saturated, if percolation takes place at a rate, which is greater than the observable base flow, the need for additional soil moisture drainage becomes manifest. *SIDE* is the ratio of the unobserved to the observed portion of base flow. When the saturated soils do not drain to the surface channel, *SIDE* allows the correct definition of *PBASE*, in order that the saturated percolation rate may be achieved. In an area where all drainage from base flow aquifers branches surface channels, *SIDE* will be zero. Zero or near zero values occur in a large proportion of basins. However, in areas subject to extreme subsurface drainage losses, *SIDE* may be as high as 5.0. It is conceivable that in some areas the value of *SIDE* may be even higher.

SSOUT

The sub-surface outflow along the stream channel, which must be provided by the stream before water is available for surface discharge. This volume expressed in mm/time interval is generally near zero. It is recommended that the value of zero be utilised, and *SSOUT* is applied only if the ${}^{10}\log Q$ vs. time plot requires a constant addition in order to achieve a valid recession characteristic. If constant volumes of flow are added to observed stream flow, the slope of the discharge plot will be altered. That value, which is required to linearize the primary recession, is the appropriate value of *SSOUT*. It should be realised that where *SSOUT* is required, an effective determination of lower zone free water storages and discharge rates will require inclusion of the *SSOUT* value (mm/Δt)

Surface runoff

Unit hydrograph ordinates for the routing of flow from the impervious and pervious surfaces as well as interflow towards the segment outlet can be obtained through standard unit hydrograph procedures. It requires the selection of rainfall events (corrected for losses) with their resulting flood hydrographs (corrected for base flow). Note that for each event the net rainfall amount should match with the surface runoff and interflow amount. Various procedures are available to arrive at a unit hydrograph. If the rainfall intensity during the storm varies, multiple linear regression and discrete convolution techniques may be applied. The regression technique is readily available in spreadsheet software. The resulting unit hydrographs generally will show some irregularities and hence requires some smoothing afterwards. Unit hydrographs from various storms may appropriately be averaged to arrive at a representative unit hydrograph for the segment.

Another option is to use the Clark method. The principle of the Clark method was dealt with in Sub-section 2.2.8. First requirement is the derivation of a time-area diagram. If a Digital Elevation Model (DEM) is available from a catchment with appropriate software automatic calculation of the time-area diagram is possible. In the absence from a DEM the time-area diagram is derived from a basin map. By estimating travel times to the basin outlet (from river and terrain slopes, assumed roughness and flow depth) isochrones can be determined. The areas between successive isochrones is determined leading to a first estimate of the time-area diagram. The total time base of the time-area diagram should be the concentration time T_c, but due to inaccurate assessment of celerities in the basin it may differ from that. Therefore, the time base of the time-area diagram is scaled by a more appropriate estimate of T_c. An estimate for T_c may be obtained as the time lapse between the cessation of rainfall and the occurrence of recession on the falling limb of the hydrograph of surface runoff. The time base of the time-area diagram is scaled by this time lapse. Alternatively, the concentration time is estimated from an empirical formula applicable to the region. E.g. for a number of small catchments in the Indus basin the following equation applies:

$$T_c = \frac{1}{119}\frac{L}{\sqrt{S}} \tag{6.332}$$

where:

T_c	concentration time [h]
L	length of river [km]
S	slope of main river

The units of the time-area diagram (km^2) are converted into m^3/s by multiplication with $0.278/\Delta t$, with Δt in hours. Subsequently, the time-area diagram is routed through a linear reservoir, with reservoir coefficient k, estimated from the slope of the recession curve of the surface water hydrograph. The routing is carried out by the following equation:

$$O_{i+1} = c_1 I_{av} + c_2 O_i \tag{6.333}$$

$$I_{av} = \frac{1}{2}(I_{i+1} + I_i) \tag{6.334}$$

$$c_1 = \frac{\Delta t}{k + \Delta t/2} \tag{6.335}$$

$$c_2 = \frac{k - \Delta t/2}{k + \Delta t/2} \tag{6.336}$$

$$c_1 + c_2 = 1 \tag{6.337}$$

where:

I_{av}	average inflow during Δt (input is in form of histogram)
O	outflow from the linear reservoir

The outflow from the reservoir is the Instantaneous Unit Hydrograph (IUH) for the basin, which has to be transformed by averaging or S-curve technique into the Unit Hydrograph resulting from a rainfall of duration equal to the routing interval.

Internal routing interval

PM	Time interval increment parameter
Pt_1	Lower rainfall threshold
Pt_2	Upper rainfall threshold

In case the time step used in the model is larger than 1 hour, the model simulates the redistribution of water between the various reservoirs with a time step, which is smaller than the time interval of the basic data. Particularly for the infiltration process this effect could be important. Also the rainfall will be lumped to that smaller interval. The number of increments in the time interval is derived from:

$$N_{\Delta t} = 1 + PM * (UZFWC * F + P_{eff}) \tag{6.338}$$

where:

$$F = 1 \quad \text{for} \quad P_{eff} < Pt_1 \tag{6.339}$$

$$F = 1/2 P_{eff}/Pt_2 \quad \text{for} \quad Pt_1 \leq P_{eff} \leq Pt_2 \tag{6.340}$$

$$F = 1 - 1/2 Pt_2/P_{eff} \quad \text{for} \quad P_{eff} > Pt_2 \tag{6.341}$$

The most important parameter is seen to be PM. Taking a very small value for PM (say $PM = 0.01$), then $N_{\Delta t}$ remains approximately1. If e.g. $PM = 0.1$ then $N_{\Delta t}$ becomes substantially larger than 1. To limit the increase of $N_{\Delta t}$ a low value for Pt_1 is to be chosen in combination with a large value of Pt_2, which will reduce the value of F.

Sequence of parameter estimation

From the presentation above it will be clear that certain parameters should be estimated before other can be assessed. The following sequence is recommended of which the first three steps are mandatory:

1 Segment area
2 Lower zone primary free water parameters LZPK and LZFPM
3 Lower zone supplemental free water parameters LZSK and LZFSM
4 Impervious fraction PCTIM
5 Upper zone parameters UZTWM, UZK and UZFWM
6 Lower zone tension capacity LZTWM
7 Percolation parameters ZPERC and REXP
8 Remaining parameters

Linear reservoirs

An essential feature of the Sacramento model is that the free water reservoirs are considered as linear reservoirs, i.e. there is a linear relation between the reservoir storage *S* and the outflow Q:

$$S = kQ \tag{6.342}$$

If the recharge is indicated by I, the continuity equation for the linear reservoir reads:

$$\frac{dS}{dt} = I - Q \tag{6.343}$$

Eliminating S from above equations results in a linear first order differential equation in Q:

$$\frac{dQ}{dt} + \frac{1}{k}Q - \frac{1}{k}I = 0 \tag{6.344}$$

With I constant and at $t = t_0$, $Q_t = Q_{t_0}$ the solution to Equation (6.344) reads:

$$Q_t = I\left(1 - \exp\left(-\frac{t - t_0}{k}\right)\right) + Q_{t_0}\exp\left(-\frac{t - t_0}{k}\right) \quad \text{for} \quad t \geq t_0 \tag{6.345}$$

When there is no recharge to the reservoir ($I = 0$) Equation (6.345) reduces to:

$$Q_t = Q_{t_0}\exp\left(-\frac{t - t_0}{k}\right) \quad \text{for} \quad t \geq t_0 \tag{6.346}$$

This equation can be compared with Equation (6.319), using the same notation:

$$Q_t = Q_{t_0}K^{(t - t_0)} \tag{6.347}$$

Hence:

$$K = \exp\left(-\frac{1}{k}\right) \quad \text{or} \quad k = -\frac{1}{\ln K} \tag{6.348}$$

Expressing time in days, then the amount of water released from the reservoir in 1 day amounts according to equation Equation (6.342):

$$S_0 - S_1 = kQ_0 - kQ_1 = kQ_0\left(1 - \exp\left(-\frac{1}{k}\right)\right) = S_0(1-K) \tag{6.349}$$

This is seen to match with e.g. the equations for the lower zone primary free water reservoir, where:

$$S_0 = LZFPC \quad \text{and} \quad 1 - K = LZPK \tag{6.350}$$

Equation (6.348) provides a means to express the lower zone free water parameters in terms of dimensions and physical properties of aquifers. Consider the phreatic aquifer shown in Figure 6.102, which has the following dimensions and properties:

1 The width of the aquifer perpendicular to the channel is L
2 The water table at the divides is h_0 above the drainage base
3 The specific aquifer yield is m
4 The aquifer transmissivity is T.

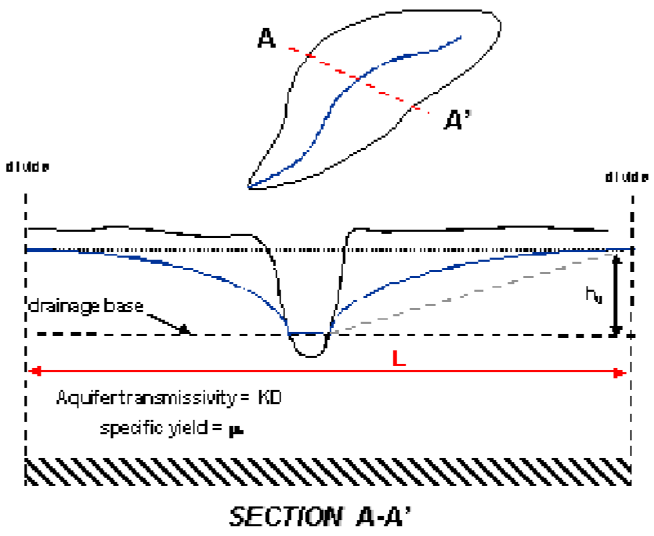


***Figure 6.102:** Schematic cross-section through basin aquifer*

The amount of water stored above the drainage base per unit length of channel available for drainage is:

$$S = mc_1Lh_0 \quad \text{with} \quad \frac{1}{2} < c_1 < 1 \tag{6.351}$$

The discharge to the channel per unit length of channel according to Darcy with the Dupuit assumption

$$Q = -2T\frac{dh}{dx} = \frac{2Tc_2h_0}{(L/2)} \quad \text{with} \quad c_2 > 1 \tag{6.352}$$

Combining the above two equations by eliminating h_0 and bringing it in the form of the linear storage discharge relation Equation (6.342) :

$$S = \frac{\mu L^2}{4cT}Q \quad \text{with} \quad c = \frac{c_2}{c_1} > 1 \tag{6.353}$$

Hence for the reservoir coefficient k in Equation (6.342) it follows:

$$k = \frac{\mu L^2}{4cT} \tag{6.354}$$

The reservoir coefficient k is seen to be proportional to the square of the aquifer width and inversely proportional to T, which is logical as k is a measure for the reside-time of the percolated water in the groundwater zone. The value of c varies between 2 and 2.5 dependent on the shape of the water table. For the parameters K and $LZPK$ for the lower zone primary free water storage it then follows:

$$K = \exp\left(-\frac{4cT}{\mu L^2}\right) \quad \text{and} \quad LZPK = 1 - \exp\left(-\frac{4cT}{\mu L^2}\right) \tag{6.355}$$

A similar story applies for the lower zone free supplemental reservoir, which can be viewed as representing the drainage from the shallower based denser network of the smaller channels, see Figure 6.103. Since its main difference is with the aquifer width L, which is much smaller than for the deeper based primary channel network, its reservoir coefficient will be smaller than of the primary free water storage and consequently $LZSK \gg LZPK$.

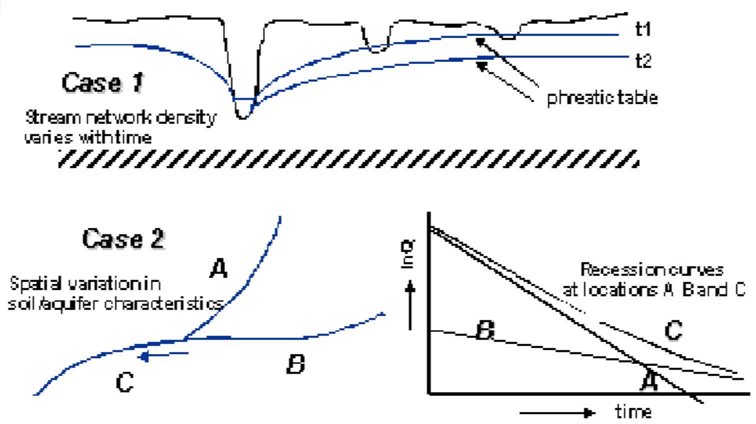


__Figure 6.103:__ Cases of multiple exponential decay of recession curve

Note that similar differences in a basin between fast and slow draining aquifers if different soils are present leading to different transmissivities.

6.5.3 Description of the D-NAM rainfall-runoff model

The "**D-NAM**" rainfall-runoff model is developed at Deltares. It concerns an improvement and expansion of the NAM rainfall-runoff model, developed at the Institute of Hydrodynamics and Hydraulic Engineering of the Technical University of Denmark (see Nielsen and Hansen (1973)). NAM is the abbreviaton of the Danish "Nedbør-Afstrømmings-Model" (in Dutch "Neerslag-Afvoer Model"), meaning rainfall-runoff model.

The D-NAM rainfall-runoff model is a deterministic, lumped and conceptual model, that treats each catchment as a single unit. Model parameters and variables, therefore, represent average values for the entire catchment area.

A comparison of the D-NAM model and the NAM model is made in section 6.5.3.18.

6.5.3.1 External forces acting on a D-NAM model

External forces (or boundary conditions) acting on a D-NAM model comprise of:

1. Rainfall (P), defined in the "Meteo" Task block.
2. Potential evapotranspiration (EP), defined in the "Meteo" Task block.
3. Groundwater pump discharge ($GWPump$), defined on the "GWPump" tab of the D-NAM Node in the "Schematisation" Task block.
4. External water level (h). Either defined at a RR boundary Node (rainfall-runoff modelling only) or computed by a 1D flow model (combined rainfall-runoff and 1D flow modelling).

6.5.3.2 D-NAM storages and their water-storage capacity

The D-NAM model comprises of three different reservoirs:

1. The surface storage, that is situated on the surface of the catchment area and has an **unlimited** water-storage capacity.
2. The lower zone storage, having a limited water-storage capacity only and representing the field capacity of the root zone layer (s_{fc} [-]). In other words the part of the soil moisture content that can be retained against gravity.
3. The groundwater storage, having a limited water-storage capacity only and representing the soil moisture content that can not be retained against gravity.

The lower zone storage and the groundwater storage jointly represent the water-storage-capacity of the modelled soil layer thickness (see Figure 6.104). The soil beneath the surface level (SL [m AD]) is divided into a **root zone layer** with bed level $RZBL$ [m AD] and specific yield $s_{y,rz}$ [-] and a **subsoil layer** with bed level $GWSBL$ [m AD] and specific yield $s_{y,ss}$ [-]. For water-storage-capacities yield (see Figure 6.104):

- ◇ The maximum water depth in the lower zone storage (L_{max} [mm]) amounts to $s_{fc}(SL - RZBL)$.
- ◇ The maximum water depth in the root zone part of the groundwater storage ($GWSD_{rz,max}$ [mm]) amounts to $(s_{y,rz} - s_{fc})(SL - RZBL)$.
- ◇ The maximum water depth in the subsoil part of the groundwater storage ($GWSD_{ss,max}$ [mm]) amounts to $s_{y,ss}(RZBL - GWSBL)$.
- ◇ The maximum water depth in the groundwater storage ($GWSD_{max}$ [mm]) amounts to $GWSD_{rz,max} + GWSD_{rz,max}$.

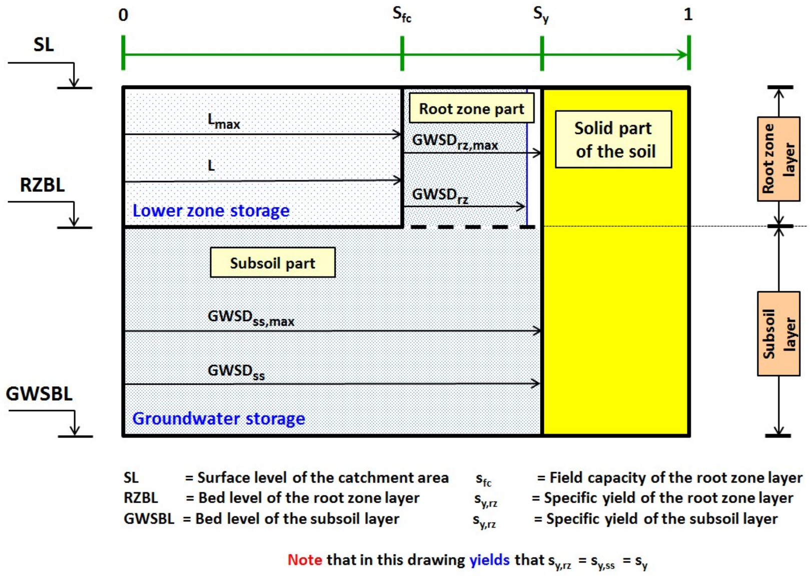


Figure 6.104: *Maximum water depths in lower zone storage and groundwater storage. The solid part of the soil (coloured yellow) can not contain any groundwater.*

6.5.3.3 D-NAM external and internal fluxes

The D-NAM model discerns nine external fluxes (see Figure 6.105):

1 Rainfall into the surface storage (P).
2 Evaporation from the surface storage ($E1$).
3 Interflow out of the surface storage (IF).
4 Overland flow out of the surface storage (OF).
5 Transpiration from the root zone layer ($E2$); being the summation of the transpiration from the lower zone storage ($E2_{LZS}$) and the transpiration from the root zone part of the groundwater storage ($E2_{GWS,rz}$).
6 Groundwater Pump. Either the abstraction of water contained in the groundwater storage ($GW_{ABS,Act}$) or the supply of external water ($GW_{SUP,Act}$); being the summation of the supply of external water into the lower zone storage (DL_{GWPump}) and the supply of external water into the groundwater storage (GW_{GWPump}).
7 Fast base flow component out of the groundwater storage ($FastBF$).
8 Slow base flow component out of the groundwater storage ($SlowBF$).
9 Inflow of external (ground)water ($GWInflow$); being the summation of the inflow of external (ground)water into the lower zone storage (DL_{Ext}) and the inflow of external (ground)water into the groundwater storage (GW_{Ext}).

The D-NAM model discerns four internal fluxes (see Figure 6.105):

1 Infiltrated water (INF) or water in the surface storage that infiltrates into the soil.
2 Infiltration into the lower zone storage (DL).
3 Percolation into the groundwater storage (G).
4 Capillary rise of water (CR), contained in the subsoil part of the groundwater storage.

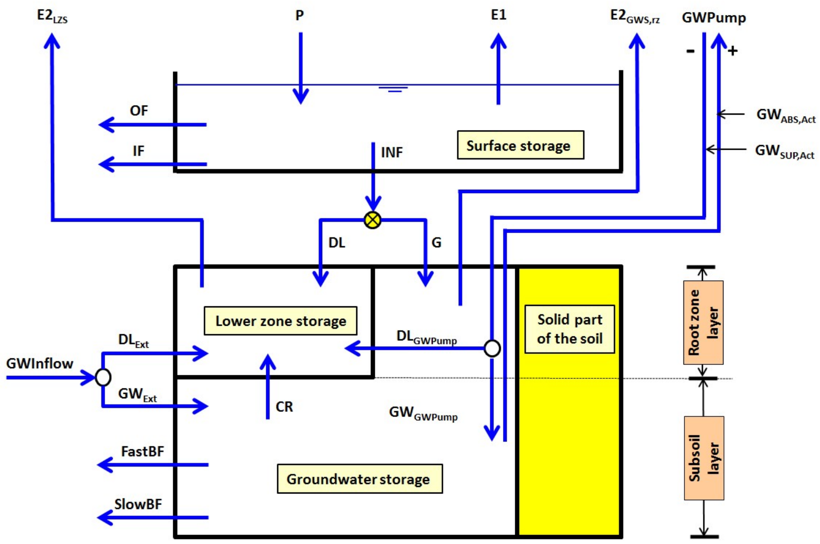


Figure 6.105: *Schematic representation of fluxes in the D-NAM rainfall runoff model.*

6.5.3.4 Computing water depths in the surface flow storage

Water depths in the surface storage (U [mm]) are computed accounting for fluxes in the following sequence (see Figure 6.106):

1. Rainfall P [mm] is added.
2. Evaporation $E1$ [mm] (see section 6.5.3.6) is subtracted.
3. Interflow IF [mm] (see section 6.5.3.7) is subtracted.
4. Infiltrated water INF [mm] (see section 6.5.3.8) is subtracted.
5. Overland flow OF [mm] (see section 6.5.3.9) is subtracted.

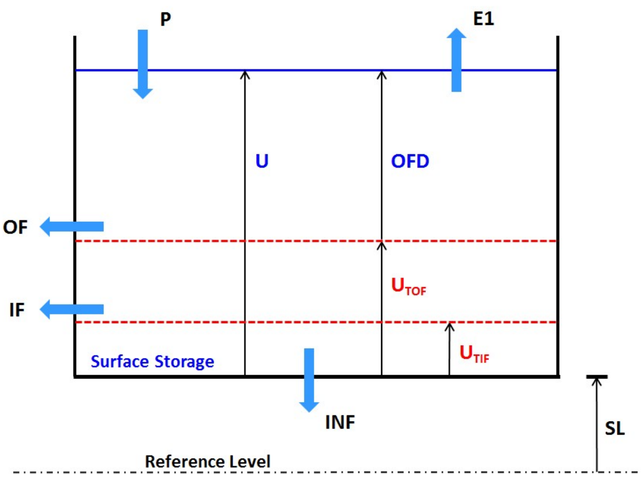


Figure 6.106: *Water depths, thresholds and fluxes related to the surface storage*

6.5.3.5 Computing water depths in the lower zone storage and overland flow storage

Water depths in the lower zone storage (L [mm]) and the groundwater storage ($GWSD$ [mm]) are computed accounting for fluxes (see Figure 6.107) in the following sequence:

1. Infiltration DL [mm] (see section 6.5.3.10) is added to the lower zone storage.
2. Percolation G [mm] (see section 6.5.3.10) is added to the groundwater storage.

3 Transpiration $E2_{LZS}$ [mm] (see section 6.5.3.11) is subtracted from the lower zone storage.
4 Transpiration $E2_{GWS_{rz}}$ [mm] (see section 6.5.3.11) is subtracted from the groundwater storage.
5 Capillary rise CR [mm] (see section 6.5.3.12) is subtracted from the groundwater storage and added to the lower zone storage.
6 Fast base flow $FastBF$ [mm] (see section 6.5.3.13) is subtracted from the groundwater storage.
7 Slow base flow $SlowBF$ [mm] (see section 6.5.3.13) is subtracted from the groundwater storage.
8 External (ground)water DL_{Ext} [mm] (see section 6.5.3.14) is added to the lower zone storage.
9 External (ground)water GW_{Ext} [mm] (see section 6.5.3.14) is added to the groundwater storage.
10 Abstraction by the groundwater pump $GW_{ABS,Act}$ [mm] (see section 6.5.3.15) is subtracted from the groundwater storage.
11 Supply by the groundwater pump DL_{GWPump} [mm] (see section 6.5.3.16) is added to the lower zone storage.
12 Supply by the groundwater pump GW_{GWPump} [mm] (see section 6.5.3.16) is added to the groundwater storage.

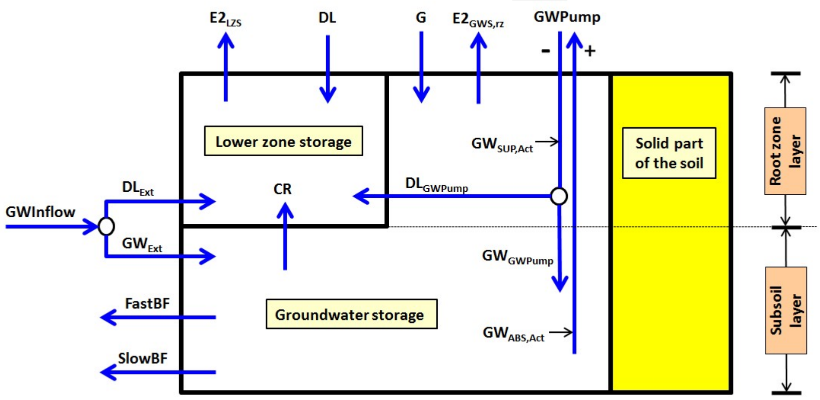


Figure 6.107: *Fluxes related to the lower zone storage and groundwater storage. The solid part of the soil (coloured yellow) can not contain any groundwater.*

6.5.3.6 Evaporation from the surface storage

For the evaporation ($E1$ [mm]) from the surface storage yields:

$$E1 = min(EP, U)$$

where:

EP	User-defined potential evapotranspiration (see section 6.5.3.1) [mm].
U	Water depth in the surface storage [mm].

6.5.3.7 Interflow out of the surface storage

For interflow (IF [mm]) out of the surface storage yields:

$$L_\alpha = (L - L_{TIF})/(L_{max} - L_{TIF}), \text{ where } L_\alpha = 0 \text{ if } L \leq L_{TIF}$$

$$CKIF_{\Delta t} = 1/(1-(1-1/CKIF)^{\Delta t/86400})$$

$$IF = (L_\alpha/CKIF_{\Delta t})\ max(0, U - U_{TIF})$$

Note: No interflow occurs if $U \leq U_{TIF}$ or $L \leq L_{TIF}$

where:

$CKIF$	User-defined reservoir coefficient applied in routing interflow [day].
$CKIF_{\Delta t}$	Reservoir coefficient used in each time-step Δt [s] for routing interflow [Δt].
L	Water depth in lower zone storage [mm].
L_{max}	Maximum water depth in lower zone storage (see section 6.5.3.2 and Figure 6.104) [mm].
L_{TIF}	User-defined lower-zone-storage threshold for interflow (Figure 6.106) [mm].
U	Water depth in the surface storage [mm].
U_{TIF}	User-defined surface-storage threshold for interflow (Figure 6.106) [mm].
Δt	User-defined time-step [s].

6.5.3.8 Infiltrated water into the soil

For infiltrated water (INF [mm]) from the surface storage into the soil yields:

$$INF_{cap,\Delta t} = INFCap\ (\Delta t/3600)$$

$$DL_{max} = L_{max} - L.$$

$$G_{max} = min(GWSD_{max} - GWSD\ ,\ G_{pot,max}\ (\Delta t/86400)\ )$$

$$INF = min(U\ ,\ INF_{cap,\Delta t}\ ,\ DL_{max} + G_{max})$$

Note: Infiltrated water (INF) equals zero if $INFCap = 0$

where:

DL_{max}	Available water-storage-depth in the lower zone storage [mm].
G_{max}	Water depth that can potentially be stored in the groundwater storage [mm].
$G_{pot,max}$	User-defined maximum rate of percolation into the groundwater storage [mm/day].
$GWSD$	Water depth in the groundwater storage [mm].
$GWSD_{max}$	Maximum water depth in the groundwater storage (see section 6.5.3.2 and Figure 6.104) [mm].
$INFCap$	User-defined maximum infiltration rate for water contained in the surface storage [mm/hr].
$INFCap_{\Delta t}$	Amount of water contained in the surface storage that infiltrates into the soil in time-step Δt [mm/hr].
L	Water depth in lower zone storage [mm].
L_{max}	Maximum water depth in lower zone storage (see section 6.5.3.2 and Figure 6.104) [mm].

U	Water depth in the surface storage [mm].
Δt	User-defined time-step [s].

6.5.3.9 Overland flow out of the surface storage

Overland flow (OF [mm]) is determined using an approximate analytical formula that:

1 Describes overland flow with the Manning roughness formula for open channel flow.
2 Assumes that the catchment area is a rectangular container with drainage length CL [m] and width B [m], having a slope S [-] towards its overland outflow point.
3 Assumes that overland flow occurs in such way, that at each point-in-time yields that water depths in the surface storage are constant in space.

This approximate analytical formula is given below:

◇ $OFD = max(0\ ,\ U - U_{TOF})$

◇ $OF = OFD - 1000\left((OFD/1000)^{-2/3} + (2\,\Delta t\,\sqrt{S})/(3\,n\,CL)\right)^{-3/2}$

Note: No overland flow occurs if $U \leq U_{TOF}$.

where:

OFD	Overland flow depth (see Figure 6.106), defined as the water depth in the surface storage above U_{TOF} [mm].
U	Water depth in the surface storage [mm].
U_{TOF}	Surface-storage threshold for overland flow [mm].
n	Manning value for the surface roughness of the catchment area [$s/m^{1/3}$].
Δt	User-defined time-step [s].

6.5.3.10 Infiltration into the lower zone storage and percolation into the groundwater storage

Infiltrated water (INF [mm], see section 6.5.3.8) is distributed as infiltration in the lower zone storage (DL) [mm] and percolation in the groundwater storage (G) [mm] (see Figure 6.108). This distribution is done as follows:

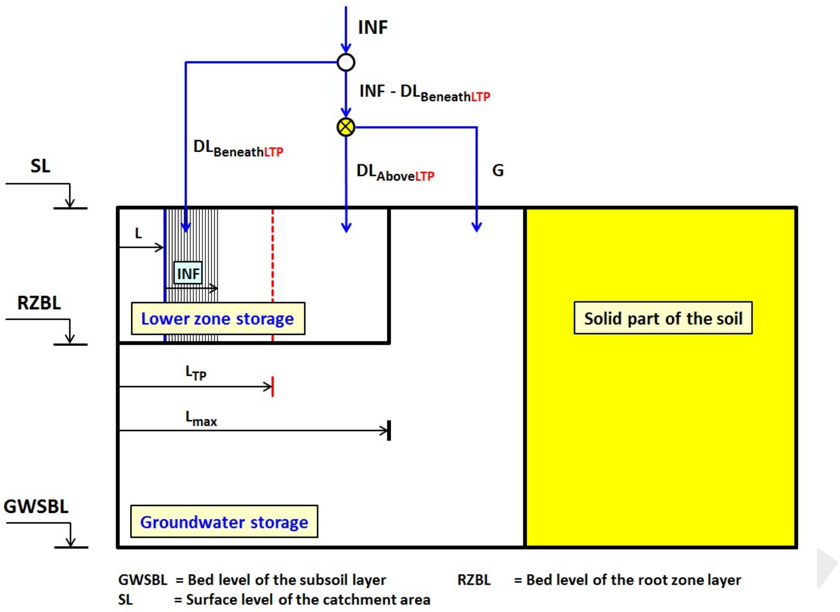


Figure 6.108: *Distributing infiltrated water (INF) over the lower zone storage and the groundwater storage. The solid part of the soil (coloured yellow) can not contain any groundwater.*

1 $DL_{BeneathL_{TP}}$ [mm] is the part of the infiltrated water (INF) that is stored in the lower zone storage beneath the lower-zone-storage threshold for percolation (L_{TP} [mm], see Figure 6.108).

$$DL_{BeneathL_{TP}} = max(0\ ,\ min(L_{TP} - L\ ,\ INF)).$$

where:

L	Water depth in the lower zone storage [mm].
L_{TP}	Lower-zone-storage threshold for percolation [mm].

The value for L_{TP} determines the degree in which infiltrated water (INF) is distributed over the lower zone storage and the groundwater storage as proposed by Nielsen and Hansen (1973). More precisely:

- If $L_{TP} = L_{max}$ the distribution proposed by Nielsen and Hansen (1973) is overruled. Since, first the lower zone storage is completely filled up before infiltrated water can percolate into the groundwater storage.
- If $0 \leq L_{TP} < L_{max}$ the distribution proposed by Nielsen and Hansen (1973) becomes applicable as soon as the lower zone storage is filled up to L_{TP}, meaning that part of the infiltrated water (INF) can percolate into the groundwater storage (or $G > 0$) before the lower zone storage is at field capacity (or $L < L_{max}$).

2 $DL_{NH,AboveL_{TP}}$ [mm] (see Figure 6.108) is the part of the infiltrated water (INF) that, in line with the Nielsen and Hansen (1973) distribution, infiltrates into the lower zone storage above L_{TP}.

$$DL_{NH,AboveL_{TP}} = min(AD_{AboveL_{TP}}\ , (1 - (L/L_{max}))(INF - DL_{BeneathL_{TP}}))$$

where:

$AD_{AboveL_{TP}}$	Available water-storage-depth in the lower zone storage above L_{TP} [mm].

L_{max}	Maximum water depth in the lower zone storage (see section 6.5.3.2 and Figure 6.104) [mm].

3 G_{NH} [mm] (see Figure 6.108) is the part of the infiltrated water (INF) that, in line with the Nielsen and Hansen (1973) distribution, percolates into the groundwater storage.

$$G_{NH} = INF - DL_{BeneathL_{TP}} - DL_{NH,AboveL_{TP}}$$

4 The actual percolation (G [mm]) follows from limiting G_{NH} to the user-defined maximum percolation rate $G_{pot,max}$ [mm/day].

$$G = max(G_{NH}\ ,\ G_{pot,max}\ (\Delta t/86400))$$

where:

$G_{pot,max}$	User-defined maximum rate of percolation into the groundwater storage [mm/day].
Δt	User-defined time-step [s].

5 For the actual infiltration into the lower zone storage (DL [mm], see Figure 6.108) yields.

$$DL = INF - G$$

6.5.3.11 Transpiration from the root zone layer

Transpiration from the root zone layer ($E2$ [mm], see Figure 6.107) respectively comprises of transpiration from the root zone part of the groundwater storage ($E2_{GWS,rz}$ [mm]) and transpiration from the lower zone storage ($E2_{LZS}$ [mm]). $E2_{GWS,rz}$ and $E2_{LZS}$ are computed as follows:

$$GWSD_{rz} = max(0,\ GWSD - GWSD_{ss,max})$$

$$E2 = min(L + GWSD_{rz}\ ,\ (EP - E1)(L + GWSD_{rz})/(L_{max} + GWSD_{rz}))$$

$$E2_{GWS,rz} = min(E2\ ,\ GWSD_{rz}).$$

$$E2_{LZS} = E2 - E2_{GWS,rz}$$

where:

$E1$	Evaporation from the surface storage (see section 6.5.3.6) [mm].
EP	User-defined potential evapotranspiration (see section 6.5.3.1) [mm].
$GWSD$	Water depth in the groundwater storage [mm].
$GWSD_{rz}$	Water depth in the root zone part of the groundwater storage [mm].
$GWSD_{ss,max}$	Maximum water depth in the root zone part of the groundwater storage (see section 6.5.3.2 and Figure 6.104) [mm].
L	Water depth in the lower zone storage [mm].
L_{max}	Maximum water depth in the lower zone storage (see section 6.5.3.2 and Figure 6.104) [mm].

6.5.3.12 Capillary rise

For capillary rise (CR [mm], see Figure 6.107) yields:

$$CR = \min(L_{max} - L\ ,\ GWSD\ ,\ (\Delta t/86400)\ CR_{pot,max}\ )$$

where:

$CR_{pot,max}$	User-defined maximum capillary rise [mm/day].
$GWSD$	Water depth in the groundwater storage [mm].
L	Water depth in the lower zone storage [mm].
L_{max}	Maximum water depth in the lower zone storage (see section 6.5.3.2 and Figure 6.104) [mm].
Δt	User-defined time-step [s].

Note: The surface level (SL [m AD]) and the bed level of the groundwater storage ($GWSBL$ [m AD]) determine the thickness of the soil layer, that is considered in a D-NAM model. So the bed level of the groundwater storage is to be considered as watertight. Therefore, $CR_{pot,max}$ is set equal to zero if $GWSD = 0$, this irrespective of the user-defined value for $CR_{pot,max}$.

6.5.3.13 Fast and slow base flow component

Base flow occurs only if the external water level (h [m AD]) is below the groundwater level (GWL [m AD]). Base flow equals zero if $h \geq GWL$ (see Figure 6.109). Water contained in the lower zone storage can not flow out as base flow, since this water can be retained against gravity.

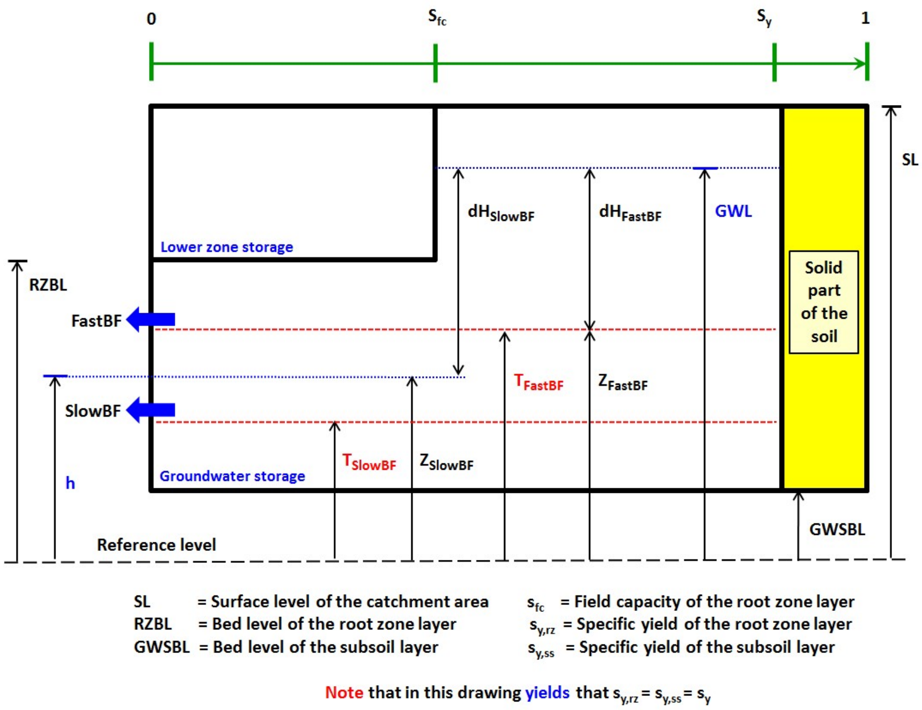


Figure 6.109: *Computing the Fast base flow component and the Slow base flow component. The solid part of the soil (coloured yellow) can not contain any groundwater.*

The fast base flow component ($FastBF$) and the slow base flow component ($SlowBF$) are computed as follows (see Figure 6.109):

$$GWSD_{ss} = min(GWSD\ ,\ GWSD_{ss,max})$$

$$GWSD_{rz} = max(0\ ,\ GWSD - GWSD_{ss,max})$$

$$GWL = GWSBL + (GWSD_{ss}/1000\ s_{y,ss}) + ((GWSD_{rz}\ +\ L)/1000\ s_{y,rz})$$

$$Z_{FastBF} = max(h, T_{FastBF})$$

$$Z_{SlowBF} = max(h, T_{SlowBF})$$

$$dH_{FastBF} = max(0,\ GWL - Z_{FastBF})$$

$$dH_{SlowBF} = max(0,\ GWL - Z_{SlowBF})$$

$$GWSD_{Z_{FastBF}} = f(Z_{FastBF}\ ,\ SL\ ,\ RZBL\ ,\ GWSBL\ ,\ s_{y,rz}\ ,\ s_{fc}\ ,\ s_{y,ss})$$

$$GWSD_{Z_{SlowBF}} = f(Z_{SlowBF}\ ,\ SL\ ,\ RZBL\ ,\ GWSBL\ ,\ s_{y,rz}\ ,\ s_{fc}\ ,\ s_{y,ss})$$

$$V_{pot,FastBF} = max(0,\ GWSD - GWSD_{Z_{FastBF}})$$

$$V_{pot,SlowBF} = max(0,\ GWSD - GWSD_{Z_{SlowBF}})$$

$$CKFastBF_{\Delta t} = 1/(1 - (1 - 1/CKFastBF)^{\Delta t/86400})$$

$$CKSlowBF_{\Delta t} = 1/(1 - (1 - 1/CKSlowBF)^{\Delta t/86400})$$

$$FastBF_{pot} = min(V_{pot,FastBF}\ ,\ 1000(dH_{FastBF}/CKFastBF_{\Delta t})\ )$$

$$SlowBF_{pot} = min(V_{pot,SlowBF}\ ,\ 1000(dH_{SlowBF}/CKSlowBF_{\Delta t})\ )$$

$$GWOutflow_{pot} = FastBF_{pot} + SlowBF_{pot}$$

$$FastBF = FastBF_{pot} \quad \text{if} \quad GWOutflow_{pot} \leq V_{pot,FastBF}$$

$$FastBF = V_{pot,FastBF}\left(\frac{FastBF_{pot}}{GWOutflow_{pot}}\right) \quad \text{if} \quad GWOutflow_{pot} > V_{pot,FastBF}$$

$$SlowBF = SlowBF_{pot} \quad \text{if} \quad FastBF + SlowBF_{pot} \leq V_{pot,SlowBF}$$

$$SlowBF = V_{pot,SlowBF}\ -\ FastBF \quad \text{if} \quad FastBF + SlowBF_{pot} > V_{pot,SlowBF}$$

where:

$CKFastBF$	User-defined reservoir coefficient applied in routing the fast base component ($CKFastBF > 1$) [day].
$CKFastBF_{\Delta t}$	Reservoir coefficient used in each time-step Δt [s] for routing the fast base component [Δt].
$CKSlowBF$	User-defined reservoir coefficient applied in routing the slow base component ($CKSlowBF > 1$) [day].
$CKSlowBF_{\Delta t}$	Reservoir coefficient used in each time-step Δt [s] for routing the slow base component [Δt] .
dH_{FastBF}	Hydraulic head (or driving force) of the fast base flow component [m].
dH_{SlowBF}	Hydraulic head (or driving force) of the slow base flow component

	[m].
$FastBF_{pot}$	Potential fast base flow component [mm].
$SlowBF_{pot}$	Potential slow base flow component [mm].
GWL	Groundwater level [m AD].
$GWOutflow_{pot}$	Potential groundwater outflow (=$FastBF_{pot}$ + $SlowBF_{pot}$) [mm].
$GWSBL$	Bed level of the groundwater storage [m AD].
$GWSD$	Water depth in the groundwater storage [mm].
$GWSD_{rz}$	Water depth in the root zone part of the groundwater storage [mm].
$GWSD_{ss}$	Water depth in the subsoil part of the groundwater storage [mm].
$GWSD_{ss,max}$	Maximum water depth in the subsoil part of the groundwater storage (see section 6.5.3.2 and Figure 6.104) [mm].
$GWSD_{Z_{FastBF}}$	Water depth in the groundwater storage if GWL equals Z_{FastBF} [mm].
$GWSD_{Z_{SlowBF}}$	Water depth in the groundwater storage if GWL equals Z_{SlowBF} [mm].
h	External water level [m AD].
L	Water depth in the lower zone storage [mm].
$RZBL$	Bed level of the root zone layer [m AD].
s_{fc}	Field capacity of the soil in the root zone layer [-].
SL	Surface level of the catchment area [m AD]
$s_{y,rz}$	Specific yield of the root zone layer (ratio of the volume of water in a fully saturated root zone layer to its total soil volume) [-].
$s_{y,ss}$	Specific yield of the subsoil layer (ratio of the volume of water in a fully saturated subsoil layer to its total soil volume) [-].
T_{FastBF}	Threshold for the fast base flow component. Only water in the groundwater storage above T_{FastBF} may flow out as fast base flow [m AD].
T_{SlowBF}	Threshold for the slow base flow component. Only water in the groundwater storage above T_{SlowBF} may flow out as slow base flow [m AD].
$V_{pot,FastBF}$	Volume of water in the groundwater storage, that is potentially available for drainage by the fast base flow component [mm].
$V_{pot,SlowBF}$	Volume of water in the groundwater storage, that is potentially available for drainage by the slow base flow component [mm].
Z_{FastBF}	Drainage level of the fast base flow component [m AD].
Z_{SlowBF}	Drainage level of the slow base flow component [m AD].
Δt	User-defined time-step [s].

6.5.3.14 External (ground)water flowing into the lower zone storage and groundwater storage

Inflow of external (ground)water into the soil ($GWInflow$ [mm]) occurs only if the external water level (h [m AD]) is above the groundwater level (GWL [m AD]). $GWInflow$ equals zero if $h \leq GWL$ (see Figure 6.110). The sequence of filling the lower zone storage and the groundwater storage by inflowing external (ground)water is as follows:

- ◇ Firstly, the subsoil part of the groundwater storage is completely filled-up
- ◇ Secondly, the lower zone storage is completely filled-up.
- ◇ Thirdly and finally, the root zone part of the groundwater storage is filed-up.

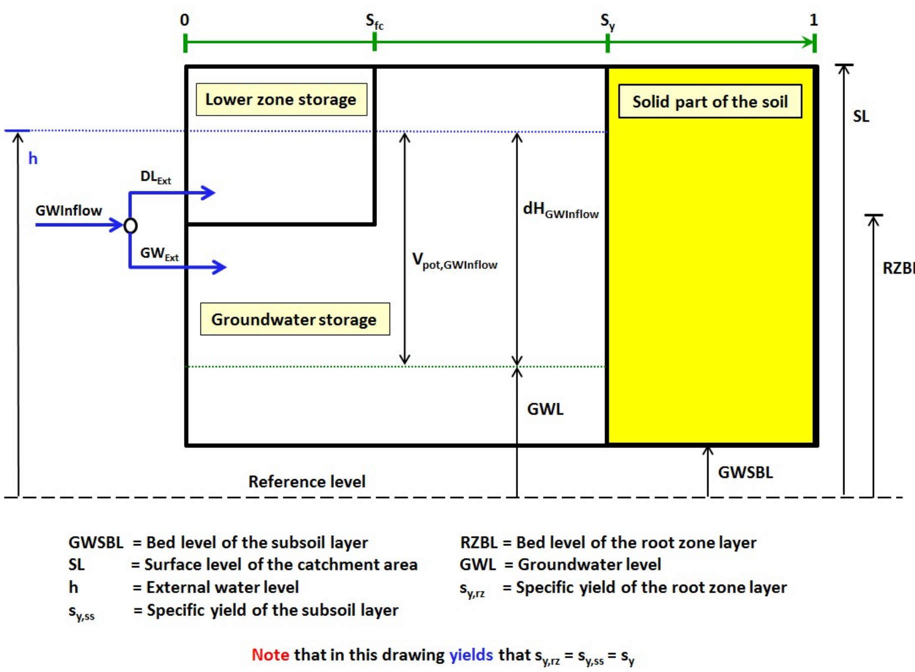


Figure 6.110: *Determining the inflow of external (ground)water ($GWInflow$). The solid part of the soil (coloured yellow) can not contain groundwater.*

The inflow of external (ground)water ($GWInflow$ [mm]), the part following into the lower zone storage (DL_{Ext} [mm]), and the part flowing into the groundwater storage (GW_{Ext} [mm]) are computed as follows (see Figure 6.110):

$$GWSD_{ss} = min(GWSD\ ,\ GWSD_{ss,max})$$

$$GWSD_{rz} = max(0\ ,\ GWSD - GWSD_{ss,max})$$

$$GWL = GWSBL + (GWSD_{ss}/1000\ s_{y,ss}) + ((GWSD_{rz}\ +\ L)/1000\ s_{y,rz})$$

$$dH_{GWInflow} = max(0, h - GWL)$$

$$V_{pot,GWInflow} = f(h\ ,\ GWL\ ,\ SL\ ,\ RZBL\ ,\ GWSBL\ ,\ s_{y,rz}\ ,\ s_{y,ss})$$

$$CKGWInflow_{\Delta t} = 1/(1 - (1 - 1/CKGWInflow)^{\Delta t/86400})$$

$$GWInflow = min(1000\ (dH_{GWInflow}/CKGWInflow_{\Delta t})\ ,\ V_{pot,GWInflow})$$

$$DL_{Ext} = max(0\ ,\ min(L_{max} - L\ ,\ GWSD + GWInflow - GWSD_{ss,max})$$

$$GW_{Ext} = GWInflow - DL_{Ext}$$

where:

$CKGWInflow$	User-defined reservoir coefficient, applied for the inflow of external (ground)water ($CKGWInflow > 1$) [day].
$CKGWInflow_{\Delta t}$	Reservoir coefficient used in each time-step Δt [s] for computing the inflow of external (ground)water [Δt].
$dH_{GWInflow}$	Hydraulic head (or driving force) for the inflow of external (ground)water [m].

GWL	Groundwater level [m AD].
$GWSBL$	Bed level of the groundwater storage [m AD].
$GWSD$	Water depth in the groundwater storage [mm].
$GWSD_{rz}$	Water depth in the root zone part of the groundwater storage [mm].
$GWSD_{ss}$	Water depth in the subsoil part of the groundwater storage [mm].
$GWSD_{ss,max}$	Maximum water depth in the subsoil part of the groundwater storage (see section 6.5.3.2 and Figure 6.104) [mm].
h	External water level [m AD].
L	Water depth in the lower zone storage [mm].
L_{max}	Maximum water depth in the lower zone storage (see section 6.5.3.2 and Figure 6.104) [mm].
$RZBL$	Bed level of the root zone layer [m AD].
s_{fc}	Field capacity of the soil in the root zone layer [-].
SL	Surface level of the catchment area [m AD].
$s_{y,rz}$	Specific yield of the root zone layer (ratio of the volume of water in a fully saturated root zone layer to its total soil volume) [-].
$s_{y,ss}$	Specific yield of the subsoil layer (ratio of the volume of water in a fully saturated subsoil layer to its total soil volume) [-].
$V_{pot,GWInflow}$	Volume of water in both the lower zone storage and the groundwater storage, that potentially can be filled-up by the inflow of external (ground)water [mm].
Δt	User-defined time-step [s].

6.5.3.15 Abstraction by the groundwater pump

Abstraction by the groundwater pump ($GW_{ABS,Act}$ [mm], see Figure 6.107) occurs only if the groundwater pump discharge ($GWPump\ [m^3/s]$) is greater than zero. Water contained in the lower zone storage can not be abstracted by the groundwater pump. $GW_{ABS,Act}$ is computed as follows:

$$GW_{ABS,max} = GWSD$$

$$GW_{ABS,pot} = max(0\ ,(1000\Delta t\ /CatchmentArea)\ GWPump)$$

$$GW_{ABS,Act} = min(GW_{ABS,max}\ ,\ GW_{ABS,pot})$$

where:

$GW_{ABS,max}$	Maximum water depth in the groundwater storage that can be abstracted by the groundwater pump [mm].
$GW_{ABS,pot}$	Potential water depth that can be abstracted according to the user-defined groundwater pump discharge [mm].
$GWSD$	Water depth in the groundwater storage [mm].
Δt	User-defined time-step [s].

6.5.3.16 Supply by the groundwater pump

Supply of water by the groundwater pump ($GW_{SUP,Act}$ [mm]), see Figure 6.107) occurs only if the groundwater pump discharge ($GWPump$ [m^3/s]) is less than zero. The sequence of filling the lower zone storage and the groundwater storage by the groundwater pump is as follows:

- Firstly, the subsoil part of the groundwater storage is completely filled-up
- Secondly, the lower zone storage is completely filled-up.
- Thirdly and finally, the root zone part of the groundwater storage is filed-up.

Supply of water by the groundwater pump ($GW_{SUP,Act}$) comprises of supply to the lower zone storage (DL_{GWPump} [mm]) and supply to the groundwater storage (GW_{GWPump} [mm]). DL_{GWPump} and GW_{GWPump} are computed as follows:

$$GW_{SUP,max} = (GWSD_{max} - GWSD) + (L_{max} - L)$$

$$GW_{SUP,pot} = max(0\ ,\ -(1000\Delta t\ /\ CatchmentArea)\ GWPump)$$

$$GW_{SUP,Act} = min(GW_{SUP,max}\ ,\ GW_{SUP,pot})$$

$$AD_{GWS,ss,GWPump} = max(0, GWSD_{ss,max} - GWSD))$$

$$DL_{GWPump} = min(L_{max} - L, GW_{SUP,Act} - min(AD_{GWS,ss,GWPump}\ , GW_{SUP,Act}))$$

$$GW_{GWPump} = GW_{SUP,Act} - DL_{GWPump}$$

where:

$AD_{GWS,ss,GWPump}$	Water-storage-depth available in the subsoil part of the groundwater storage for the supply of water by the groundwater pump [mm].
$GWSD$	Water depth in the groundwater storage [mm].
$GWSD_{max}$	Maximum water depth in the groundwater storage (see section 6.5.3.2 and Figure 6.104) [mm].
$GWSD_{ss,max}$	Maximum water depth in the subsoil part of the groundwater storage (see section 6.5.3.2 and Figure 6.104) [mm].
$GW_{SUP,max}$	Maximum water-storage-depth available for the supply of water by the groundwater pump. Sum of available water-storage-depth in the lower zone storage and groundwater storage. [mm].
$GW_{SUP,pot}$	Potential water depth that can be supplied according to the user-defined groundwater pump discharge [mm].
L	Water depth in the lower zone storage [mm].
L_{max}	Maximum water depth in the lower zone storage (see section 6.5.3.2 and Figure 6.104) [mm].
Δt	User-defined time-step [s].

6.5.3.17 D-NAM output time-series

Table 6.12 provides an overview of the output time-series available for each D-NAM rainfall-runoff model. The second column contains for each output time-series the symbols that are used in equations.

Please note that the "Runoff node" is to be checked on the "Output options" tab in RR Settings in order to obtain the output time-series (see Table 6.12) available under "Runoff Node" in "Results in Charts" Task block.

Table 6.12: *Overview of D-NAM output time-series*

Name of Output Time Series	Symbols	Unit	Description
Rainfall	P	mm	Rainfall as defined in the "Meteo Task" block
PotEvap	EP	mm	Evapotranspiration as defined in the "Meteo Task" block.
ActEvap		mm	Sum of evaporation from surface storage ($E1$) and transpiration from the lower zone storage ($E2$).
Runoff		mm	Equals overland flow (OF) + interflow (IF) + fast base flow component ($FastBF$) + slow base flow component ($SlowBF$) - inflow of external (ground)water into the soil ($GWInflow$).
Outflow		m^3/s	Equals overland flow (OF) + interflow (IF) + fast base flow component ($FastBF$) + slow base flow component ($SlowBF$) - inflow of external (ground)water into the soil ($GWInflow$).
D-NAM ExternalWaterLevel	h	$m\,AD$	External water level.
D-NAM GWPumpDefinedDischarge	$GWPump$	m^3/s	Groundwater pump discharge.
D-NAM EvapSurfaceStorage	$E1$	mm	Evaporation from the surface storage.
D-NAM EvapRootZoneLayer	$E2$	mm	Transpiration from the root zone layer.
D-NAM EvapRootZonePartGWStorage	$E2_{GWS,rz}$	mm	Transpiration from the root zone part of the groundwater storage.
D-NAM EvapLowerZoneStorage	$E2_{LZS}$	mm	Transpiration from the lower zone storage.
D-NAM OverlandFlow	OF	mm	Overland flow (outflow from surface storage).
D-NAM Interflow	IF	mm	Interflow (outflow from surface storage).
D-NAM FastBaseFlow	$FastBF$	mm	Fast base flow component (outflow from groundwater storage).
D-NAM SlowBaseFlow	$SlowBF$	mm	Slow base flow component (outflow from groundwater storage).
D-NAM ExternalGWInflow	$GWInflow$	mm	Inflow of external (ground)water into the soil.
D-NAM ExternalGWInflowLowerZoneStorage	DL_{Ext}	mm	Inflow of external (ground)water into the lower zone storage.
D-NAM ExternalGWInflowGroundwaterStorage	GW_{Ext}	mm	Inflow of external (ground)water into the groundwater storage.
D-NAM GWPumpAbsGroundwaterStorage	$GW_{ABS,Act}$	mm	Abstraction of water from the groundwater storage by the groundwater pump.
D-NAM GWPumpSupLowerZoneStorage	DL_{GWPump}	mm	Supply of water into the lower zone storage by the groundwater pump.
D-NAM GWPumpSupGroundwaterStorage	GW_{GWPump}	mm	Supply of water into the groundwater storage by the groundwater pump.
D-NAM Infiltration	INF	mm	Water contained in the surface storage that infiltrates into the soil.
D-NAM InfiltrationLowerZoneStorage	DL	mm	Water contained in the surface storage that infiltrates into the lower zone storage.
D-NAM Percolation	G	mm	Water contained in the surface storage that percolates into the groundwater storage.
D-NAM CapillaryRise	CR	mm	Capillary rise of water from the subsoil part of the groundwater storage into the lower zone storage.
D-NAM DepthSurfaceStorage	U	mm	Water depth in the surface storage.

Table 6.12: Overview of D-NAM output time-series

Name of Output Time Series	Symbols	Unit	Description
D-NAM DepthLowerZoneStorage	L	mm	Water depth in the lower zone storage.
D-NAM DepthGroundwaterStorage	$GWSD$	mm	Water depth in the groundwater storage.
D-NAM VolumeSurfaceStorage	V_U	m^3	Volume of water contained in the surface storage.
D-NAM VolumeLowerZoneStorage	V_L	m^3	Volume of water contained in the lower zone storage.
D-NAM VolumeGroundwaterStorage	V_{GWS}	m^3	Volume of water contained in the groundwater storage.
D-NAM GroundWaterLevel	GWL	mAD	Groundwater level (measured in metres above datum).
D-NAM GroundWaterTableDepth	$GWTD$	mBS	Groundwater table depth (measured in metres below surface level).
D-NAM AvailableSoilStorage	AV_{soil}	m^3	Volume in the soil that is still available for the storage of water.
D-NAM BaseFlow	BF	mm	Equals fast base flow component ($FastBF$) + slow base flow component ($SlowBF$) - inflow of external (ground)water into the soil ($GWInflow$).
D-NAM GWPumpActualDischarge	$GWPump_{Act}$	m^3/s	Actual groundwater pump discharge. Positive if water is abstracted. Negative if water is supplied.
D-NAM GWPumpActual-DefinedDischarge	$GWPump$ $Act-Defined$	m^3/s	Actual groundwater pump discharge **minus** the user-defined groundwater pump discharge. If positive, the amount of discharge that could not be supplied. If negative, the amount of discharge that could not be abstracted.

6.5.3.18 Comparing the D-NAM model and the NAM model

Concepts of the Nielsen and Hansen (1973) NAM model implemented into the D-NAM model are:

1 Interflow occurs only if the water depth in the lower zone storage (L) is above the lower-zone-threshold for interflow (L_{TIF}). Interflow is a function of the relative moisture content in the lower zone storage above L_{TIF}. (see factor L_α in section 6.5.3.7).
2 Percolation may occur if the water depth in the lower zone storage is below field capacity (see section 6.5.3.10).
3 Transpiration from the root zone layer ($E2$) is a function of the relative moisture content in the root zone layer (see section 6.5.3.11).

Reasons for developing the D-NAM rainfall-runoff model are:

1 The NAM model stores water on the surface of a catchment area in three different stacked on each other storages, being the NAM surface storage, the NAM interflow storage and the NAM overland flow storage. As a result hereof, water contained in the latter two NAM storages is not available for evaporation and for infiltration into the soil.
In the D-NAM model all water on the surface of a catchment area is contained in one surface storage only and hence all water on the surface of a D-NAM model is at any point-in-time available for evaporation, interflow, infiltration into the soil and overland flow (see Figure 6.106).
2 The NAM surface storage, receiving precipitation has a watertight bed and a maximum water-storage-capacity. Hence, water contained in the NAM surface storage can not directly infiltrate into the soil. Only if the water depth in the NAM surface storage, after accounting for evaporation and inflow into the interflow storage, exceeds its maximum water-storage-capacity, the surplus of water in the NAM surface storage is divided into infiltration into the soil and inflow into the overland flow storage. This division of surplus water in the NAM surface storage is, however, made irrespective of the actual water depth

in the NAM overland flow storage.
In the D-NAM model the surface storage has an unlimited water-storage-capacity and its bed is not watertight, allowing water to infiltrate into the soil (INF). Infiltrated water is limited by the actual water depth in the surface storage, the available water-storage-depth in the soil, and the user defined maximum infiltration and percolation capacity (see section 6.5.3.8).

Additional functionalities in the D-NAM model with respect to the NAM model are:

1 The D-NAM model makes a distinction between the specific yield of root zone layer and the specific yield of the subsoil layer (see section 6.5.3.2)
2 The NAM model is suited for free discharging catchment areas only. The D-NAM model allows for the inflow of external (ground)water (see section 6.5.3.14), that is required in modelling deltaic areas.
3 The D-NAM model determines overland flow using the Manning formula for open channel flow (see section 6.5.3.9).
4 The D-NAM model makes a distinction between a fast and a slow base flow component (see Figure 6.109).
5 The D-NAM model includes a groundwater pump, that can abstract water (see section 6.5.3.15 or supply water (see section 6.5.3.16).

6.5.4 SOBEK-Urban RR (Rainfall Runoff) concept

Delay of Runoff

The delay of runoff (in SOBEK-Urban Rainfall-Runoff) depends on the average distance to the inflow location in sewer system, the slope and the geometry of the catchment. The formula which describes the runoff to the sewer system is the formula of the Rational Method:

$$q = ch \tag{6.356}$$

where:

q	inflow into sewer [mm/min]
c	runoff factor [1/min]
h	rainfall, dynamic storage on catchment [mm]

The runoff factor, c, is a function of length, roughness and slope. Twelve different area types are described in the 'Dutch Guidelines for sewer systems computations, hydraulic functioning'. The types and default values are presented in Table 6.13.

Table 6.13: *Default parameters of Delay of Runoff (Rational Method)*

Number	Area type	Runoff type	Runoff factor, c	Surface storage, h
1	closed paved	with a slope	0.5	0.0
2	closed paved	flat	0.2	0.5
3	closed paved	stretched flat	0.1	1.0
4	open paved	with a slope	0.5	0.0
5	open paved	flat	0.2	0.5
6	open paved	stretched flat	0.1	1.0
7	roof	with a slope	0.5	0.0
8	roof	flat	0.2	2.0
9	roof	stretched flat	0.1	4.0

Table 6.13: Default parameters of Delay of Runoff (Rational Method)

Number	Area type	Runoff type	Runoff factor, c	Surface storage, h
10	unpaved	with a slope	0.5	2.0
11	unpaved	flat	0.2	4.0
12	unpaved	stretched flat	0.1	6.0

An area 'with a slope' is an area with a slope more than 4 %. A stretched flat area is an area with the distance to the nearest inflow point in the sewer larger than 100 meters.

Horton equation

The description of the infiltration in the Urban version of the Rainfall Runoff module (NWRW model; Nationale Werkgroep Riolering en Waterkwaliteit (NLingenieurs, 1978)) is based on the formula of Horton:

Decreasing infiltration capacity:

$$f_t = f_e + (f_b - f_e)e^{-k_a t}$$

Recovering infiltration capacity:

$$f_t = f_e - (f_b - f_e)e^{-k_h t}$$

where:

$_ft$	infiltration capacity at moment of time t $[mm/h]$
f_b	maximum infiltration capacity at t=0 $[mm/h]$
f_e	minimum infiltration capacity $[mm/h]$
k_a	time factor decreasing infiltration capacity $[1/h]$
k_h	time factor recovering infiltration capacity $[1/h]$
t	time $[h]$

The rate of decreasing and recovering infiltration capacity between the maximum value fb and the minimum value fe depends on the time factors ka and kh. See also the table below. It is assumed that at the beginning of each rainfall event, the infiltration capacity is at its maximum. Infiltration capacity is decreasing as long as there is water stored on the surface. If infiltration capacity branches the minimum value, and there is still water on the surface, it will remain at the minimum value. Infiltration capacity will increase as soon as the surface is dry and it is not raining. If infiltration capacity branches the maximum value and there is no water on the surface, it will remain at the maximum value.

Table 6.14: Default parameters Horton equation

nr	Area type	Runoff type	Infiltration capacity maximum, fb	Infiltration capacity minimum, fe	Time factor decreasing, ka	Time factor recovering, kh
1	closed paved	with a slope	0.0	0.0	0.0	0.0
2	closed paved	flat	0.0	0.0	0.0	0.0
3	closed paved	stretched flat	0.0	0.0	0.0	0.0
4	open paved	with a slope	2.0	0.5	3.0	0.1

Table 6.14: *Default parameters Horton equation*

nr	Area type	Runoff type	Infiltration capacity maximum, fb	Infiltration capacity minimum, fe	Time factor decreasing, ka	Time factor recovering, kh
5	open paved	flat	2.0	0.5	3.0	0.1
6	open paved	stretched flat	2.0	0.5	3.0	0.1
7	roof	with a slope	0.0	0.0	0.0	0.0
8	roof	flat	0.0	0.0	0.0	0.0
9	roof	stretched flat	0.0	0.0	0.0	0.0
10	unpaved	with a slope	5.0	1.0	3.0	0.1
11	unpaved	flat	5.0	1.0	3.0	0.1
12	unpaved	stretched flat	5.0	1.0	3.0	0.1

Choosing the appropriate Rainfall Runoff concept

SOBEK provides three different Rainfall Runoff concepts. Basically all three have the task to transform rainfall intensities into an outflow discharge towards open water. However, they all have different environments in which they fit best. In this chapter, you are explained which Rainfall Runoff concept to use.

◇ The original SOBEK-Rural Rainfall Runoff concept.
  This concept has originally been developed by WL | Delft Hydraulics for use in low lying areas, such as polders. It provides a wide range of modelling objects, such as unpaved areas, weirs, greenhouses and waste water treatment plants.
◇ The SOBEK-Urban Rainfall Runoff concept (NWRW; NLingenieurs (1978))
  This concept is only active for modelling objects of the Sewer Flow module, thus for sewer pipes and manholes. The rainfall runoff options that are attached to those object types are always of this concept.
◇ The Sacramento Concept
  Throughout the world, Sacramento has become a hugely popular Rainfall Runoff concept for use in river basins and catchment areas. SOBEK now provides modelling objects that support this concept. Objects of this type can easily be combined with objects from the original SOBEK-Rural Rainfall Runoff concept!
◇ The HBV Concept
  The Hydrologiska Byråns Vattenbalansavdelning (HBV) model was introduced back in 1972 by the Swedisch Meteological and Hydrological Institute (SMHI). It has started as a simple lumped model for river basins, but currently also distributed model versions (Lindström *et al.*, 1997) are available. These HBV model concept are also available in SOBEK.
◇ The SCS Curve Number Concept
  The SCS Curve Number Concept is a simple, widely used and efficient method for determining the amount of runoff from a rainfall event in a river basin. Although the method is designed for a single storm event, it can be scaled to find average annual runoff values. This concept is available in SOBEK too.

The HBV, SCS and External Runoff Concepts

A runoff timeseries file, necessary for the RR-external runoff node, and the temperature time-series file, necessary when the RR-HBV node is used, can not be created in the SOBEK interface yet. It is assumed these are created outside of SOBEK and put in the <{Sobek}\Fixed> directory, where {Sobek} indicates the directory where SOBEK is installed, e.g. <d:\SOBEK215>.

The format of the runoff time-series and temperature time-series file is exactly the same as the format of the SOBEK rainfall (<*bui>) file. The files must be located in the <{Sobek}\Fixed> directory, the same directory where the SOBEK rainfall files are located. SOBEK assumes the extension <*.RNF> for the runoff file, and <*.TMP> for the temperature file.

So when the rainfall file <Default.Bui> is selected, the corresponding runoff file should be called <Default.RNF>.

The runoff file can contain data series for a different number of stations (and other stations) as the rainfall file.

The runoff station mentioned in the RR-runoff node input data screen refers to a station in the runoff file, just like the meteo station specified at RRnodes refers to the meteo station (rainfall station) in the rainfall file.

Note that the timestep of the runoff file can be different from the timestep in the rainfall file. Just as for the rainfall timestep and the RR computation timestep, the runoff timestep and the RR computation timestep should 'fit' in the sense that the timestep in the runoff file should be a multiple of the RR computation timestep.

The runoff timeseries are specified in $[m^3/s]$. The runoff timeseries is directly put on the downstream RRlink of the RR-runoff node. This link typically connects the RRrunoff node with an RR-connection node, an RR- boundary node or an RR-CF connection.

Definitions:
Area adjustment factor
The area adjustment factor allows you to specify an (optional) area adjustment factor on the rainfall data, to reflect differences between point station rainfall and areal basin rainfall. The factor is usually between 0 and 1, although for flexibility a value larger than 1 could be allowed. Default value is 1.

Initial AMC
The SCS model derives maximum retention and initial losses based on the Curve Number. The present implementation assumes average antecedent moisture conditions, the curve number is referred to as CNII. The curve number CNII for a watershed can be estimated as a function of land use, soil type, and antecedent watershed moisture, using tables published by the SCS.

6.5.4.1 Real Time Control (RTC module)

Reservoir in SOBEK

Reservoirs are important structures in practical water resources management. A reservoir is considered as advanced control, so it is implemented in the SOBEK real time control module (SOBEK-RTC).

The following figure illustrates some of the reservoir concepts and terminology, like:

- ◇ dead storage;
- ◇ bottom gate (main gate);
- ◇ turbine intake; and
- ◇ spillway level.

Backwater outlets are not yet shown in the figure. The backwater gates represent abstractions or spills from the reservoir which are not located at the main dam site.

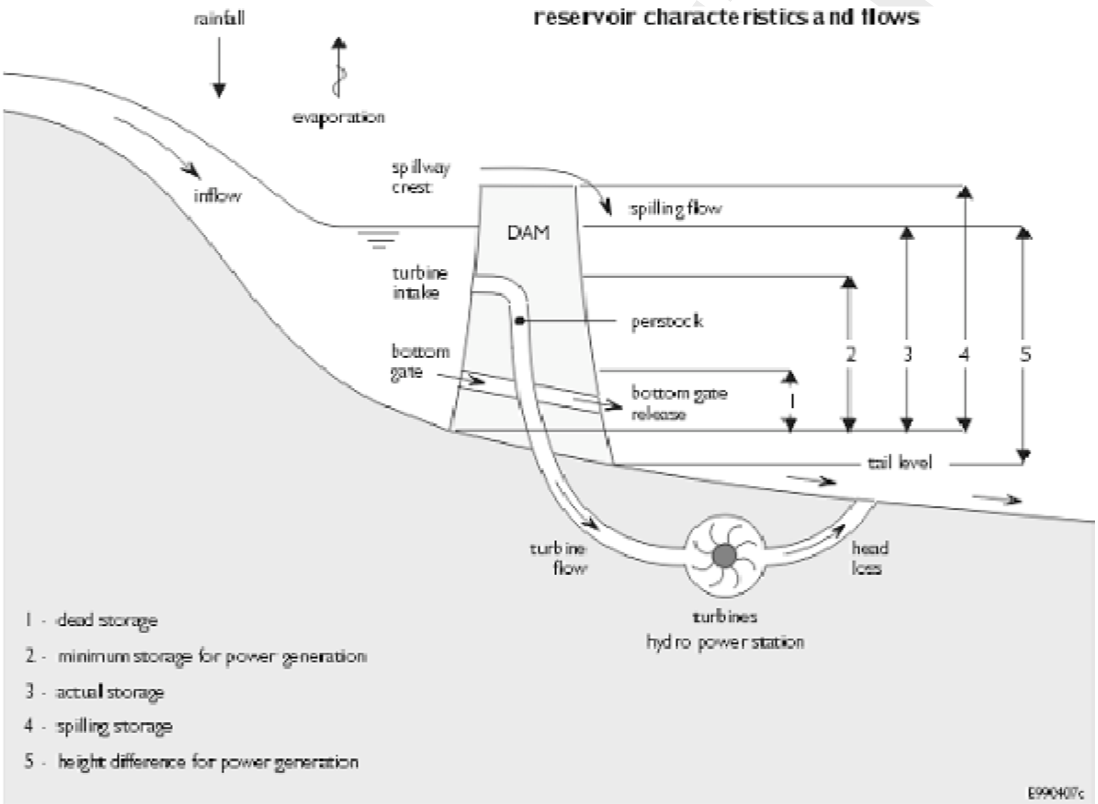


Figure 6.111: *Reservoir*

Reservoir concept

The SOBEK reservoir includes the following aspects:

- ◇ Reservoir operation rules (flood control curve, target curve, firm storage curve, hedging rules below firm storage);
- ◇ Multiple outlets (bottom gates, turbines, spillways, including backwater outlets)

Important here are the following 2 remarks:

- ◇ not all outlets are obligatory,
- ◇ outlets may be located on the same link, or on different links.

- ◇ Q-H relations describing the relation between maximum flow and net-head for all outlets (reflecting physical limitations);
- ◇ Time series of maximum flows per outlet (reflecting imposed management);
- ◇ Time series of release targets for downstream demands for all outlet links;
- ◇ Time series of release targets for energy generation;
- ◇ The operation of the SOBEK reservoir is based on the initial reservoir level and expected inflow (defined by other decision parameters in SOBEK-RTC)

Application possibilities

The SOBEK-RTC reservoir module uses the initial reservoir level and an expected reservoir inflow, together with the reservoir input data, to compute the desired release through the bottom gate, turbine and spillway gate. The initial reservoir level and the expected reservoir inflow are to be defined using decision parameters in SOBEK-RTC. This also allows for applications in flood-early warning systems (FEWS), where the determination of expected inflow is an important aspect.

Furthermore, the reservoir module computes decision parameters for all outlet gates, which can be used to define other decision parameters in RTC. The user thus has the possibility to modify or overrule the decision parameters computed by the reservoir module. The setting of the actual structure flows in SOBEK is determined by the decision parameter used to set the structure flow, as specified by the user in the measure input file.

An important extra possibility of the SOBEK reservoir is that the outlets may be located on different links. In practice, it quite often occurs that the outflows of main gate (bottom gate), turbine and spillway do not come together on the same downstream link.

In the SOBEK reservoir design, there can be multiple bottom gates, turbines, and spillways. Each of them may be on different outlet links, but also there may be several bottom gates, on the same link or each on a different link. Backwater gate can thus be modelled in SOBEK using a bottom gate on a separate link, with an appropriate downstream demand time series, and appropriate Q-H relation describing the (physical) maximum abstraction capacity as a function of the net head (=the head above intake level).

If these abstractions are to be modelled separate from the reservoir, SOBEK has the possibility to use a lateral discharge node. However, abstractions at these nodes are not depending on the actual water level.

Rainfall and evaporation

In SOBEK, rainfall and evaporation can be modelled by specifying a lateral discharge using a time series of rainfall or evaporation values in mm/timestep, and specifying a constant area on which this rainfall/evaporation series applies. When applying this for a reservoir, one can define the area at average target level as the area to be used in this computation.

Release targets

For the SOBEK reservoir, the user should specify time series of downstream release targets (demands) to be supplied from the reservoir. Since there may be different outlet links in SOBEK, these demands need to be specified for all different outlet links. This is done using input time series. It is anticipated that future versions allow to define the reservoir release targets by using decision parameters, summing the demands of various lateral discharges from SOBEK-CF and/or SOBEK-RR. This gives a huge flexibility. However, the current approach of directly specifying the release targets as input variables is very appropriate for testing and comparing with any distribution model. In SOBEK, the user specifies a time series of desired turbine flows as well.

Maximum flows

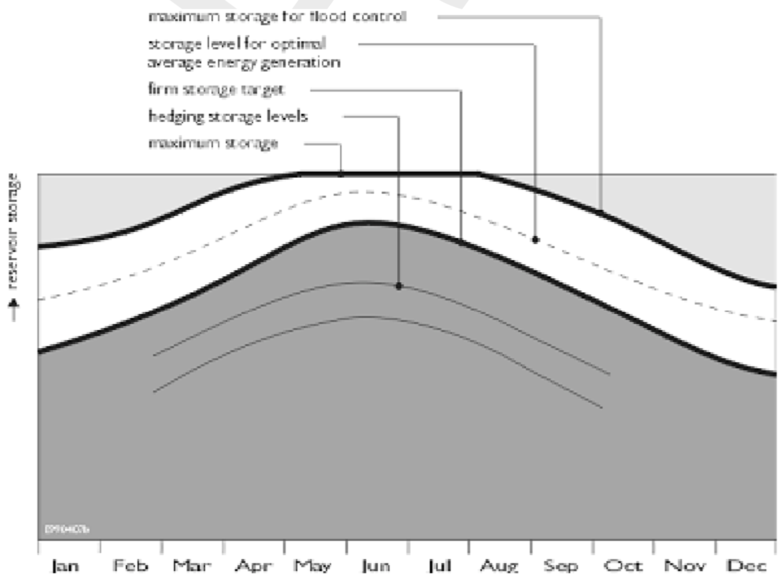
The maximum capacities of all outlet gates can be specified using a Q-h interpolation relation.

For the turbine gate, a distribution model determined the maximum possible flow through the turbines based on the turbine characteristics (installed capacity, generation efficiency, head loss relation etc.). In SOBEK, these hydropower data is not included, and a Q-h interpolation relation for the turbines is specified directly.

SOBEK offers the extra possibility to impose other maximum flows as well. These can be maximum flows reflecting e.g. water rights, or other limits imposed by the water authorities. These maximum flows can be specified as time series for all outlet gates separately.

Reservoir operation

The rule curves of the SOBEK reservoir are using the same concepts as Deltares RIBASIM distribution model, i.e. distinction is made between a flood control curve, a target curve and a firm storage curve. Furthermore, the hedging rules on storage are implemented. These hedging rules specify how releases are reduced when the reservoir is below firm storage.



The operation of the reservoir is as follows:

◇ If the reservoir is above flood control level, try to come down to the flood control level by any means. Just like in a distribution model, SOBEK uses first the turbines, then the bottom gate, and finally the spillway. In case of multiple bottom gates, SOBEK simply uses first the first bottom gate, then the second, etc.

- ◇ If the reservoir is between flood control level and target level, the reservoir is allowed to release more water, but only through the turbines (i.e. if extra energy can be generated). In case of multiple turbine outlets, SOBEK simply uses the order in which the turbine outlets are specified in the input data.
- ◇ If the reservoir is between target level and firm storage level, no special action is taken.
- ◇ If the reservoir is below firm storage, hedging rules are applied. They specify how much the discharge from the reservoir is reduced, depending on the actual reservoir water level. The releases on all outlet different links are reduced with the same percentage.

References

Ackers, P., W. White, J. Perkins and A. Harrison, 1978. *Weirs and Flumes for Flow Measurement*. John Wiley & Sons Ltd.

Amstrong, B. L., 1978. *Derivation of initial soil moisture accounting parameters from soil properties for the national weather service river forecast system.* Tech. rep., NOAA Technical Memorandum NWS HYDRO 37.

Åström, K. J. and T. Hägglund, 1995. *PID controllers: theory, design and tuning*. Instrument Society of America, Research Triangle Park, NC, USA, 2nd ed.

Becker, B., 2013. *Inzet RTC-Tools voor het boezemmodel "Wetterskip Fryslân"*. Report 1205773-000, Deltares, Delft. In Dutch.

Becker, B., R. Dahm, K.-J. van Heeringen, N. Goorden, N. Kramer, K. Kooij, J. Gooijer and J. Jansen, 2012a. "Op zoek naar een optimaal ontwerp voor een groot uitwateringsgemaal in het Lauwersmeer." *H2O* 44 (13): 11–13. In Dutch.

Becker, B. and Q. Gao, 2012. *Koppelen Sobek-modellen "Wetterskip Fryslân" en "Waterschap Noorderzijlvest" via OpenMI*. Report 1204514-000-ZWS-0007, Deltares, Delft. In Dutch.

Becker, B. and J. Talsma, 2014. "On the external and iterative coupling of multiple open channel flow models with OpenMI." *Revista de Ingeniería Innova* 6: 55–66.

Becker, B. P. J., D. Schwanenberg, T. Schruff and M. Hatz, 2012b. "Conjunctive real-time control and hydrodynamic modelling in application to Rhine River." In *Proceedings of 10th International Conference on Hydroinformatics*. TuTech Verlag TuTech Innovation GmbH, Hamburg, Germany.

Becker, B. P. J., J. Talsma, Q. Gao and E. Ruijgh, 2012c. "Coupling of multiple channel flow models with OpenMI." In *Proceedings of 10th International Conference on Hydroinformatics*. Hamburg, Germany.

Bos, 1989. *Discharge Measuring Structures*. International Institute for Land Reclamation and Improvement/ILRI, Wageningen, The Netherlands.

Cunge, J. A., F. M. Holly and A. Verwey, 1980. *Practical aspects of computational river hydraulics*. Boston; London; Melbourne; Boston; London; Melbourne: Pitman. Includes index.

D-WAQ PLT, 2013. *D-Water Quality Processes Library Tables, Technical Reference Manual*. Deltares, 4.00 ed.

D-WAQ UM, 2013. *D-Water Quality User Manual*. Deltares, 4.02 ed.

Deltares, 2013. "RTC-Tools a toolbox for real-time control of hydraulic structures." Published: Deltares.

Dutch Hydrological Society, 2002. "Hydrologische woordenlijst." Nederlandse Hydrologische Vereniging.

Ellis, J. B. and J.-L. Bertrand-Krajewski, eds., 2010. *Assessing infiltration and exfiltration on the Performance of Urban Sewer Systems (APUSS)*. IWA Publishing, London, UK. ISBN: 978-1-84-339149-4.

Engelund, F. and E. Hansen, 1967. *A monograph on Sediment Transport in Alluvial Streams*. Teknisk Forlag, Copenhagen.

Ernst, L., 1978. "Drainage of undulating sandy soils with high groundwatertables." *Journal of Hydrology* 39: 1–50.

Feddes, R. A., P. J. Kowalik, K. K. Malinka and H. Zaradny, 1978. *Simulation of Field Water Use and Crop Yield*. John Wiley and Sons. PUDOC Wageningen.

Fischer, H., E. List, R. Koh, J. Imberger and N. Brooks, 1979. *Mixing in inland and Coastal Waters*. Academic Press.

Gill, A. E., 1982. *Atmosphere-Ocean dynamics*, vol. 30 of *International Geophysics Series*. Academic Press.

Gregersen, J., P. Gijsbers and S. Westen, 2007. "OpenMI: Open modelling Interface." *Journal of Hydroinformatics* 9 (3): 175–191. DOI: 10.2166/hydro.2007.023.

Hooghart, J. and W. Lablans, 1988. *Van Penman naar Makkink: een nieuwe berekeningswijze voor de klimatologische verdampingsgetallen*. Rapporten en nota's. Ministerie van Verkeer en Waterstaat, Koninklijk Nederlands Meteorologisch Instituut. (In Dutch), Project- en Begeleidingsgroep Verdampingsberekeningen.

Ippen, A. and D. Harleman, 1961. "One dimensional analysis of salinity intrusion in estuaries." *Civ. Eng. J. Hydraul.* 96 (13-15): 1691-1709. Tech. Bull.

Karpf, C., J. Traenckner and P. Krebs, 2008. "Hydraulic modelling of sewage exfiltration." In *11th International Conference on Urban Drainage*. Edinburgh, Scotland, UK.

Knaap, F. Van der, 2000. *Breach growth as a function of time; Deltares, memo's 2 and 3 of May 21 and September 5 respectively*. Tech. Rep. Q2655, WL | Delft Hydraulics, Delft, The Netherlands.

Kuijper, C. and L. van Rijn, 2011. "Analytical and Numerical Analysis of Tides and Salinities in Estuaries; Part II: Salinity Distributions in Prismatic and Convergent Tidal Channels." *Ocean Dynamics* 61 (11): 1743:1765.

Lane, A., 1989. *The heat balance of the North Sea*. Tech. Rep. 8, Proudman Oceanographic Laboratory.

Lindström, G., B. Johansson, M. Persson, M. Gardelin and S. Bergström, 1997. *Development and test of the distributed HBV-96 hydrological model*. Tech. rep., Swedisch Meteological and Hydrological Institute.

McCarthy, G., 1938. "The unit hydrograph and flood routing." In *Conference of North Atlantic Division*. US Engineering Office, Providence RI, New London, CT. Unpublished Paper.

Moore, R., P. Gijsbers, D. Fortune, J. Gregersen and M. Blind, 2005. *OpenMI Document Series: Part A – Scope for the OpenMI*. Version 1.0 ed.

Moore, R. V. and C. I. Tindall, 2005. "An overview of the open modelling interface and environment (the OpenMI)." *Environmental Science & Policy* 8 (3): 279–286. DOI: 10.1016/j.envsci.2005.03.009.

Nielsen, S. A. and E. Hansen, 1973. "Numerical simulation of the rainrain runoff process on a daily basis." *Nordic Hydrology* 4: 171–190.

NLingenieurs, 1978. *Position paper Riolering*. Tech. rep., NLingenieurs.

Octavio, K. A. H., G. H. Jirka and D. R. F. Harleman, 1977. *Vertical Heat Transport Mechanisms in Lakes and Reservoirs*. Tech. Rep. 22, Massachusetts Institute of Technology.

Peck, E. L., 1978. *Catchment modeling and initial parameter estimation for the National Weather Service River Forecast System*. Tech. rep., NOAA Tech. Memo. NWS HYDRO-31, US Dept of Commerce, Silver Spring, Maryland, USA.

RTC-Tools TRM, 2012. *RTC-Tools a software package for modelling real-time control / Technical Reference Manual and Configuration Guidelines*. Deltares, Delft. Version: 0.1.0.22313.

Rutsch, J., M., J. Rieckermann, C. J.B., J. Ellis, Vollertsen and P. Krebs, 2008. “Towards a better understanding of sewer exflitration,.” In *Water Research*, vol. 42, pages 2385–2394.

Ryan, P. J., D. R. F. Harleman and K. D. Stolzenbach, 1974. “Surface Heat Loss From Cooling Ponds.” *Water Resources Research* 10 (5): 930–938.

Savenije, H., 2012. “Salinity and Tides in Alluvial Estuaries.” 2nd completely revised edition.

Schellekens, J., B. P. J. Becker, G. Donchyts, N. Goorden, J. C. Hoogewoud, S. Patzke and D. Schwanenberg, 2012. “OpenStreams: Open Source Components as Building Blocks for Integrated Hydrological Models.” In *Geophysical Research Abstracts*, vol. 14 EGU2012, page 3953. Vienna, Austria.

Schmidt, 1955. “Unknown.” .

Schwanenberg, D., B. Becker and T. Schruff, 2011. *SOBEK-Grobmodell des staugeregelten Oberrheins*. Report no. 1201242-000-ZWS-0014, Deltares. In German.

SOBEK3_TRM, 2013. *SOBEK 3 / Hydrodynamics Technical Reference Manual / SOBEK in Delta Shell*. Deltares, Delft. Version: 3.0.1.27817.

Stelling, G. S. and S. P. A. Duinmeijer, 2003. “A staggered conservative scheme for every Froude number in rapidly varied shallow water flows.” *International Journal Numerical Methods In Fluids* 43: 1329–1354.

Thatcher, M. L. and D. R. F. Harleman, 1972. *A mathematical model for the prediction of unsteady salinity intrusion in estuaries*. Report no. 144, MIT School of Engineering Massachusetts Institute of Technologie, Department of Civil Engineering.

Vademecum, C., 1988. *Werkgroep Herziening Cultuurtechnisch vademecum*. Tech. rep., Cultuurtechnische vereniging.

Verheij, H., 2002. *Modification breach growth model in HIS-OM*. Tech. Rep. Q3299, WL | Delft Hydraulics, Delft, The Netherlands. (in Dutch).

Wesseling, J., 1991. *CAPSEV; steady state moisture flow theory. Program description user manual.* Tech. rep., Staring Centrum, Wageningen.

A Dimension of Steel Cunnete Cross-sections

Table A.1: *Steel cunette cross-section*

Type 1: Three circle segments, a >= 120 degrees, a1 = 0 degrees; r3 =0

Type 2 & 3: Three circle segments only, a < 120 degrees, a1 = 0 degrees; r3 =0

Type 4: Four circle segments

W = Width, including wave height (WH)
H = Height, including wave height (WH)
r = Radius r, including half the wave height (WH)
r1 = Radius r1, including half the wave height (WH)
r2 = Radius r2, including half the wave height (WH)
r3 = Radius r3, including half the wave height (WH)
a = Angle a
a1 = Angle a1
WH = Wave height of corrugated construction material

W	**H**	**r**	**r1**	**r2**	**r3**	**a**	**a1**	**WH**	**Remarks**
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
100	78	50	80	20	0	28	0	2	SGME formerly: EM 1
120	98	60	90	30	0	38	0	2	SGME formerly: EM 2
140	116	70	110	30	0	41	0	2	SGME formerly: EM 3
160	131	80	182	30	0	38	0	2	SGME formerly: EM 4
180	136	90	150	30	0	46	0	2	SGME formerly: EM 5
200	154	100	170	30	0	49	0	2	SGME formerly: EM 6
210	161	105	202	30	0	47	0	2	SGME formerly: EM 7
220	168	110	242	30	0	45	0	2	SGME formerly: EM 8
230	179	115	222	30	0	50	0	2	SGME formerly: EM 9
240	174	120	210	30	0	54	0	2	SGME formerly: EM 10
250	185	126	200	30	0	59	0	2	SGME formerly: EM 11
260	192	130	230	30	0	57	0	2	SGME formerly: EM 12
270	189	133	224	30	0	59	0	2	SGME formerly: EM 13
280	210	140	250	30	0	59	0	2	SGME formerly: EM 14
290	206	144	242	30	0	63	0	2	SGME formerly: EM 15
300	212	150	270	30	0	62	0	2	SGME formerly: EM 16
310	224	154	261	30	0	66	0	2	SGME formerly: EM 17
320	230	160	290	30	0	65	0	2	SGME formerly: EM 18
190	147	94,5	220,5	48,5	0	55,5	0	5	SGM-1-1
209	141	109,5	252,5	48,5	0	56,5	0	5	SGM-1-2
209	155	104,5	277,5	48,5	0	55,5	0	5	SGM-1-3
220	160	109,5	277,5	48,5	0	56,5	0	5	SGM-1-4
226	166	113,5	348,5	48,5	0	56,5	0	5	SGM-1-5
234	171	116,5	291,5	48,5	0	58	0	5	SGM-1-6
246	175	122,5	302,5	48,5	0	59,5	0	5	SGM-1-7
265	173	136,5	312,5	48,5	0	62,5	0	5	SGM-1-8

continued on next page

Table A.1 – continued from previous page

W	H	r	r1	r2	r3	a	a1	WH	Remarks
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
265	180	134,5	312,5	48,5	0	63,5	0	5	SGM-1-9
274	190	136,5	397,5	48,5	0	60,5	0	5	SGM-1-10
283	192	142,5	437,5	48,5	0	60,5	0	5	SGM-1-11
290	196	144,5	442,5	48,5	0	61,5	0	5	SGM-1-12
296	201	147,5	442,5	48,5	0	61,5	0	5	SGM-1-13
305	207	152,5	452,5	48,5	0	62,5	0	5	SGM-1-14
317	211	158,5	462,5	48,5	0	63,5	0	5	SGM-1-15
327	220	163,5	462,5	48,5	0	65,5	0	5	SGM-1-16
337	221	169,5	462,5	48,5	0	66,5	0	5	SGM-1-17
353	226	178,5	462,5	48,5	0	68,5	0	5	SGM-1-18
366	229	186,5	464,5	48,5	0	71,5	0	5	SGM-1-19
365	237	183,5	552,5	48,5	0	66,5	0	5	SGM-1-20
385	237	196,5	492,5	48,5	0	73,5	0	5	SGM-1-21
386	246	195,5	492,5	48,5	0	73,5	0	5	SGM-1-22
392	252	196,5	656,5	48,5	0	66,5	0	5	SGM-1-23
396	259	198,5	552,5	48,5	0	70,5	0	5	SGM-1-24
414	262	207,5	652,5	48,5	0	69,5	0	5	SGM-1-25
419	269	209,5	672,5	48,5	0	69,5	0	5	SGM-1-26
434	272	217,5	702,5	48,5	0	70,5	0	5	SGM-1-27
440	277	220,5	712,5	48,5	0	71,5	0	5	SGM-1-28
445	285	222,5	712,5	48,5	0	72,5	0	5	SGM-1-29
472	286	239,5	578,5	48,5	0	83,5	0	5	SGM-1-30
478	292	241,5	652,5	48,5	0	79,5	0	5	SGM-1-31
483	300	243,5	722,5	48,5	0	77,5	0	5	SGM-1-32
489	304	244,5	862,5	48,5	0	71,5	0	5	SGM-1-33
505	308	254,5	742,5	48,5	0	78,5	0	5	SGM-1-34
510	315	256,5	742,5	48,5	0	80,5	0	5	SGM-1-35
410	290	206,5	492,5	81,5	0	101,5	0	5	SGM-1-36
417	296	209,5	601,5	81,5	0	99,5	0	5	SGM-1-37
431	300	216,5	505,5	81,5	0	104,5	0	5	SGM-1-38
438	306	219,5	567,5	81,5	0	101,5	0	5	SGM-1-39
444	311	222,5	654,5	81,5	0	99,5	0	5	SGM-1-40
459	316	230,5	580,5	81,5	0	104,5	0	5	SGM-1-41
473	320	238,5	532,5	81,5	0	108,5	0	5	SGM-1-42
480	326	241,5	592,5	81,5	0	107,5	0	5	SGM-1-43
487	331	244,5	664,5	81,5	0	104,5	0	5	SGM-1-44
500	336	252,5	601,5	81,5	0	110,5	0	5	SGM-1-45
508	341	254,5	670,5	81,5	0	107,5	0	5	SGM-1-46
522	346	263,5	612,5	81,5	0	113,5	0	5	SGM-1-47
527	351	265,5	679,5	81,5	0	111,5	0	5	SGM-1-48
534	357	268,5	752,5	81,5	0	108,5	0	5	SGM-1-49
551	361	277,5	692,5	81,5	0	114,5	0	5	SGM-1-50
557	367	280,5	762,5	81,5	0	111,5	0	5	SGM-1-51
571	371	288,5	702,5	81,5	0	116,5	0	5	SGM-1-52
578	377	290,5	772,5	81,5	0	113,5	0	5	SGM-1-53

continued on next page

Table A.1 – continued from previous page

W	H	r	r1	r2	r3	a	a1	WH	Remarks
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
593	381	299,5	702,5	81,5	0	119,5	0	5	SGM-1-54
599	387	302,5	777,5	81,5	0	116,5	0	5	SGM-1-55
605	392	304,5	852,5	81,5	0	114,5	0	5	SGM-1-56
611	398	306,5	952,5	81,5	0	111,5	0	5	SGM-1-57
627	402	315,5	862,5	81,5	0	118,5	0	5	SGM-1-58
632	408	317,5	902,5	81,5	0	116,5	0	5	SGM-1-59
185	155	93	172	63	0	159	0	5,5	SGMA formerly: MA 1
194	160	97	226	63	0	166	0	5,5	SGMA formerly: MA 2
228	173	118	177	63	0	137	0	5,5	SGMA formerly: MA 3
254	188	128	291	63	0	158	0	5,5	SGMA formerly: MA 4
289	207	145	476	63	0	167	0	5,5	SGMA formerly: MA 5
328	220	169	316	63	0	143	0	5,5	SGMA formerly: MA 6
343	230	174	419	63	0	155	0	5,5	SGMA formerly: MA 7
370	244	187	506	63	0	158	0	5,5	SGMA formerly: MA 8
377	249	190	602	63	0	163	0	5,5	SGMA formerly: MA 9
410	257	216	383	63	0	137	0	5,5	SGMA formerly: MA 10
418	262	217	425	63	0	143	0	5,5	SGMA formerly: MA 11
439	277	222	617	63	0	157	0	5,5	SGMA formerly: MA 12
446	367	223	394	131	0	175	0	5,5	SGMA formerly: MA 13
454	372	227	426	131	0	178	0	5,5	SGMA formerly: MA 14
489	387	245	392	131	0	165	0	5,5	SGMA formerly: MA 15
497	392	249	417	131	0	168	0	5,5	SGMA formerly: MA 16
519	409	259	511	131	0	176	0	5,5	SGMA formerly: MA 17
526	414	263	550	131	0	179	0	5,5	SGMA formerly: MA 18
548	418	276	441	131	0	161	0	5,5	SGMA formerly: MA 19
563	429	282	493	131	0	167	0	5,5	SGMA formerly: MA 20
584	445	292	592	131	0	175	0	5,5	SGMA formerly: MA 21
611	461	305	645	131	0	176	0	5,5	SGMA formerly: MA 22
630	472	315	658	131	0	175	0	5,5	SGMA formerly: MA 23
649	475	326	581	131	0	165	0	5,5	SGMA formerly: MA 24
676	498	338	724	131	0	175	0	5,5	SGMA formerly: MA 25
683	503	341	768	131	0	177	0	5,5	SGMA formerly: MA 26
702	514	351	779	131	0	176	0	5,5	SGMA formerly: MA 27
716	512	360	654	131	0	165	0	5,5	SGMA formerly: MA 28
729	523	366	721	131	0	169	0	5,5	SGMA formerly: MA 29
748	540	374	846	131	0	176	0	5,5	SGMA formerly: MA 30
768	550	384	856	131	0	175	0	5,5	SGMA formerly: MA 31
794	566	397	913	131	0	176	0	5,5	SGMA formerly: MA 32
814	576	407	922	131	0	175	0	5,5	SGMA formerly: MA 33
840	592	420	980	131	0	176	0	5,5	SGMA formerly: MA 34
860	603	430	989	131	0	175	0	5,5	SGMA formerly: MA 35
886	618	443	1047	131	0	176	0	5,5	SGMA formerly: MA 36
906	629	453	1055	131	0	175	0	5,5	SGMA formerly: MA 37
932	645	466	1114	131	0	176	0	5,5	SGMA formerly: MA 38

continued on next page

Table A.1 – continued from previous page

W	H	r	r1	r2	r3	a	a1	WH	Remarks
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
953	655	477	1121	131	0	175	0	5,5	SGMA formerly: MA 39
978	671	489	1180	131	0	176	0	5,5	SGMA formerly: MA 40
1001	737	501	923	166	0	177	0	5,5	SGMA formerly: MA 41
1027	754	513	963	166	0	178	0	5,5	SGMA formerly: MA 42
1047	765	523	974	166	0	177	0	5,5	SGMA formerly: MA 43
1072	781	536	1015	166	0	178	0	5,5	SGMA formerly: MA 44
1092	792	546	1026	166	0	177	0	5,5	SGMA formerly: MA 45
1118	809	559	1067	166	0	178	0	5,5	SGMA formerly: MA 46
1138	820	569	1077	166	0	178	0	5,5	SGMA formerly: MA 47
1163	836	582	1118	166	0	178	0	5,5	SGMA formerly: MA 48
1183	847	592	1129	166	0	178	0	5,5	SGMA formerly: MA 49
1208	864	604	1170	166	0	178	0	5,5	SGMA formerly: MA 50
219	169	115	155	63	0	129	0	5,5	SGMB formerly: MB 1
229	173	118	177	63	0	137	0	5,5	SGMB formerly: MB 2
240	178	127	173	63	0	128	0	5,5	SGMB formerly: MB 3
302	205	163	225	63	0	124	0	5,5	SGMB formerly: MB 4
340	224	179	297	63	0	136	0	5,5	SGMB formerly: MB 5
372	266	198	323	98	0	122	0	5,5	SGMB formerly: MB 6
381	270	201	363	98	0	127	0	5,5	SGMB formerly: MB 7
402	278	213	386	98	0	126	0	5,5	SGMB formerly: MB 8
419	287	219	496	98	0	135	0	5,5	SGMB formerly: MB 9
465	303	251	452	98	0	123	0	5,5	SGMB formerly: MB 10
529	328	289	516	98	0	121	0	5,5	SGMB formerly: MB 11
546	337	292	625	98	0	129	0	5,5	SGMB formerly: MB 12
567	345	304	644	98	0	128	0	5,5	SGMB formerly: MB 13
589	353	317	665	98	0	128	0	5,5	SGMB formerly: MB 14
604	362	319	818	98	0	135	0	5,5	SGMB formerly: MB 15
623	365	342	644	98	0	122	0	5,5	SGMB formerly: MB 16
639	374	343	771	98	0	130	0	5,5	SGMB formerly: MB 17
660	382	355	789	98	0	129	0	5,5	SGMB formerly: MB 18
668	387	356	870	98	0	132	0	5,5	SGMB formerly: MB 19
695	394	381	759	98	0	124	0	5,5	SGMB formerly: MB 20
723	424	390	838	112	0	128	0	5,5	SGMB formerly: MB 21
744	432	402	857	112	0	127	0	5,5	SGMB formerly: MB 22
765	440	415	876	112	0	126	0	5,5	SGMB formerly: MB 23
794	453	428	964	112	0	129	0	5,5	SGMB formerly: MB 24
816	461	441	982	112	0	128	0	5,5	SGMB formerly: MB 25
837	470	454	1000	112	0	128	0	5,5	SGMB formerly: MB 26
859	478	467	1017	112	0	127	0	5,5	SGMB formerly: MB 27
888	491	479	1111	112	0	129	0	5,5	SGMB formerly: MB 28
909	499	492	1127	112	0	129	0	5,5	SGMB formerly: MB 29
930	507	505	1144	112	0	128	0	5,5	SGMB formerly: MB 30
952	515	518	1161	112	0	127	0	5,5	SGMB formerly: MB 31
973	524	531	1177	112	0	127	0	5,5	SGMB formerly: MB 32

continued on next page

Table A.1 – continued from previous page

W	H	r	r1	r2	r3	a	a1	WH	Remarks
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
1002	536	543	1274	112	0	129	0	5,5	SGMB formerly: MB 33
1023	545	556	1289	112	0	128	0	5,5	SGMB formerly: MB 34
1045	553	569	1305	112	0	128	0	5,5	SGMB formerly: MB 35
251	214	127	348	77	0	199	0	2	EW1
285	236	144	467	77	0	200	0	2	EW2
319	258	160	599	77	0	202	0	2	EW3
288	273	144	317	108	0	243	0	5,5	WA1
330	303	165	300	108	0	229	0	5,5	WA2
337	310	169	331	108	0	231	0	5,5	WA3
345	316	172	367	108	0	234	0	5,5	WA4
361	328	181	370	108	0	231	0	5,5	WA5
376	341	188	451	108	0	236	0	5,5	WA6
391	354	196	566	108	0	241	0	5,5	WA7
427	377	213	450	108	0	227	0	5,5	WA8
434	384	217	485	108	0	229	0	5,5	WA9
449	397	224	571	108	0	234	0	5,5	WA10
458	402	229	522	108	0	229	0	5,5	WA11
480	422	240	661	108	0	236	0	5,5	WA12
509	480	254	492	188	0	238	0	5,5	WA13
524	493	262	581	188	0	241	0	5,5	WA14
550	511	275	491	188	0	230	0	5,5	WA15
573	530	287	588	188	0	235	0	5,5	WA16
597	548	299	630	188	0	234	0	5,5	WA17
627	574	314	824	188	0	240	0	5,5	WA18
647	585	324	645	188	0	229	0	5,5	WA19
655	591	327	679	188	0	230	0	5,5	WA20
677	611	339	796	188	0	235	0	5,5	WA21
707	637	353	1012	188	0	240	0	5,5	WA22
720	641	360	758	188	0	228	0	5,5	WA23
745	659	372	795	188	0	228	0	5,5	WA24
757	673	378	961	188	0	235	0	5,5	WA25
769	677	384	831	188	0	228	0	5,5	WA26
781	692	390	1001	188	0	234	0	5,5	WA27
812	717	406	1091	188	0	235	0	5,5	WA28
832	728	416	980	188	0	230	0	5,5	WA29
849	740	424	978	188	0	228	0	5,5	WA30
861	754	430	1167	188	0	235	0	5,5	WA31
871	759	435	1104	188	0	232	0	5,5	WA32
892	779	446	1260	188	0	235	0	5,5	WA33
905	800	452	1191	215	0	235	0	5,5	WA34
929	819	464	1229	215	0	235	0	5,5	WA35
289	254	145	267	90	0	205	0	5,5	WB1

continued on next page

Table A.1 – continued from previous page

W	H	r	r1	r2	r3	a	a1	WH	Remarks
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
322	278	161	348	90	0	209	0	5,5	WB2
329	284	165	393	90	0	213	0	5,5	WB3
369	306	184	346	90	0	197	0	5,5	WB4
383	318	191	412	90	0	204	0	5,5	WB5
408	335	204	457	90	0	205	0	5,5	WB6
422	348	211	552	90	0	211	0	5,5	WB7
463	369	231	479	90	0	198	0	5,5	WB8
483	388	241	600	90	0	206	0	5,5	WB9
496	400	248	710	90	0	212	0	5,5	WB10
532	415	266	570	90	0	197	0	5,5	WB11
557	432	278	612	90	0	198	0	5,5	WB12
582	450	291	656	90	0	199	0	5,5	WB13
601	468	300	793	90	0	206	0	5,5	WB14
622	521	311	591	157	0	199	0	5,5	WB15
644	539	322	691	157	0	205	0	5,5	WB16
669	557	334	738	157	0	205	0	5,5	WB17
694	574	347	784	157	0	206	0	5,5	WB18
722	599	361	984	157	0	213	0	5,5	WB19
737	603	368	836	157	0	205	0	5,5	WB20
762	620	381	883	157	0	205	0	5,5	WB21
789	645	395	1086	157	0	211	0	5,5	WB22
812	655	406	977	157	0	206	0	5,5	WB23
830	666	415	980	157	0	204	0	5,5	WB24
855	684	428	1027	157	0	205	0	5,5	WB25
190	145	96	196,3	48,2	0	55,8	0	5,1	Old model, measures w.r.t. central axis
198	150	99,3	252,9	48,2	0	54,6	0	5,1	Old model, measures w.r.t. central axis
211	155	106,6	214,6	48,2	0	58,4	0	5,1	Old model, measures w.r.t. central axis
219	160	109,9	267,2	48,2	0	56,9	0	5,1	Old model, measures w.r.t. central axis
226	166	113	348,4	48,2	0	55,3	0	5,1	Old model, measures w.r.t. central axis
240	170	120,6	281,4	48,2	0	59,4	0	5,1	Old model, measures w.r.t. central axis
247	176	123,7	352,8	48,2	0	57,6	0	5,1	Old model, measures w.r.t. central axis
253	181	126,7	467,1	48,2	0	55,6	0	5,1	Old model, measures w.r.t. central axis
268	186	134,3	360,6	48,2	0	60,2	0	5,1	Old model, measures w.r.t. central axis
274	191	137,1	456,4	48,2	0	58,1	0	5,1	Old model, measures w.r.t. central axis

continued on next page

Table A.1 – continued from previous page

W	H	r	r1	r2	r3	a	a1	WH	Remarks
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
289	196	145,2	369,8	48,2	0	63	0	5,1	Old model, measures w.r.t. central axis
295	201	148	453,4	48,2	0	60,7	0	5,1	Old model, measures w.r.t. central axis
301	207	150,5	580,9	48,2	0	58,1	0	5,1	Old model, measures w.r.t. central axis
316	211	158,7	455,4	48,2	0	63,5	0	5,1	Old model, measures w.r.t. central axis
331	216	167,3	391,6	48,2	0	68,8	0	5,1	Old model, measures w.r.t. central axis
338	221	169,9	460,7	48,2	0	66,3	0	5,1	Old model, measures w.r.t. central axis
352	226	178,8	403,6	48,2	0	72,1	0	5,1	Old model, measures w.r.t. central axis
359	231	180,5	467,8	48,2	0	69,3	0	5,1	Old model, measures w.r.t. central axis
365	237	183,1	552,2	48,2	0	66,5	0	5,1	Old model, measures w.r.t. central axis
380	241	192,2	476,2	48,2	0	72,4	0	5,1	Old model, measures w.r.t. central axis
387	247	194,3	553,2	48,2	0	69,6	0	5,1	Old model, measures w.r.t. central axis
392	252	196,6	656,3	48,2	0	66,5	0	5,1	Old model, measures w.r.t. central axis
397	258	198,8	801,8	48,2	0	63,5	0	5,1	Old model, measures w.r.t. central axis
413	262	207,5	649,7	48,2	0	69,6	0	5,1	Old model, measures w.r.t. central axis
430	267	216,9	563,1	48,2	0	75,9	0	5,1	Old model, measures w.r.t. central axis
435	272	218,7	647,9	48,2	0	72,6	0	5,1	Old model, measures w.r.t. central axis
440	278	220,7	758,4	48,2	0	69,3	0	5,1	Old model, measures w.r.t. central axis
457	282	230,1	648,4	48,2	0	75,9	0	5,1	Old model, measures w.r.t. central axis
472	286	239,7	578,6	48,2	0	82,8	0	5,1	Old model, measures w.r.t. central axis
478	292	241,3	652	48,2	0	79,2	0	5,1	Old model, measures w.r.t. central axis
484	298	243	742,9	48,2	0	75,7	0	5,1	Old model, measures w.r.t. central axis
489	304	245,1	861,3	48,2	0	72,4	0	5,1	Old model, measures w.r.t. central axis
505	308	254,5	741,4	48,2	0	79	0	5,1	Old model, measures w.r.t. central axis

continued on next page

Table A.1 – continued from previous page

W	H	r	r1	r2	r3	a	a1	WH	Remarks
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
510	314	256,2	847,6	48,2	0	75,4	0	5,1	Old model, measures w.r.t. central axis
410	290	205,9	491,7	81,2	0	100,5	0	5,1	Old model, measures w.r.t. central axis
417	296	209	561,3	81,2	0	98,5	0	5,1	Old model, measures w.r.t. central axis
431	300	216,9	505,2	81,2	0	103,1	0	5,1	Old model, measures w.r.t. central axis
438	306	219,9	567,9	81,2	0	101,3	0	5,1	Old model, measures w.r.t. central axis
444	311	222,5	654,3	81,2	0	99	0	5,1	Old model, measures w.r.t. central axis
459	316	230,6	580,9	81,2	0	103,8	0	5,1	Old model, measures w.r.t. central axis
473	320	239	532,1	81,2	0	108,9	0	5,1	Old model, measures w.r.t. central axis
480	326	241,5	592	81,2	0	106,6	0	5,1	Old model, measures w.r.t. central axis
487	331	244,3	664,4	81,2	0	104,3	0	5,1	Old model, measures w.r.t. central axis
501	336	252,7	601,9	81,2	0	109,7	0	5,1	Old model, measures w.r.t. central axis
508	341	255,2	671	81,2	0	107,1	0	5,1	Old model, measures w.r.t. central axis
522	346	263,9	614,6	81,2	0	112,7	0	5,1	Old model, measures w.r.t. central axis
529	351	266,2	680,2	81,2	0	110,2	0	5,1	Old model, measures w.r.t. central axis
535	357	268,7	759,2	81,2	0	107,7	0	5,1	Old model, measures w.r.t. central axis
550	361	277,3	689,8	81,2	0	113,2	0	5,1	Old model, measures w.r.t. central axis
557	367	279,6	763,7	81,2	0	110,5	0	5,1	Old model, measures w.r.t. central axis
571	371	288,5	699,7	81,2	0	116,3	0	5,1	Old model, measures w.r.t. central axis
578	377	290,8	770,3	81,2	0	113,5	0	5,1	Old model, measures w.r.t. central axis
593	381	299,7	710,1	81,2	0	119,3	0	5,1	Old model, measures w.r.t. central axis
599	387	302	777,5	81,2	0	116,5	0	5,1	Old model, measures w.r.t. central axis
605	392	304,3	857,2	81,2	0	113,7	0	5,1	Old model, measures w.r.t. central axis
611	398	306,5	953,2	81,2	0	110,7	0	5,1	Old model, measures w.r.t. central axis

continued on next page

Table A.1 – continued from previous page

W	**H**	**r**	**r1**	**r2**	**r3**	**a**	**a1**	**WH**	**Remarks**
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
627	402	315,4	861,3	81,2	0	116,8	0	5,1	Old model, measures w.r.t. central axis
632	408	317,5	951,2	81,2	0	113,8	0	5,1	Old model, measures w.r.t. central axis
470	430	237,5	574,5	142,5	0	238	0	5,1	WU1
494	448	249,5	623,5	142,5	0	237	0	5,1	WU2
519	468	261,5	669,5	142,5	0	237	0	5,1	WU3
545	487	274,5	710,5	142,5	0	236	0	5,1	WU4
570	507	287,5	759,5	142,5	0	236	0	5,1	WU5
593	524	298,5	784,5	142,5	0	235	0	5,1	WU6
618	544	311,5	825,5	142,5	0	234	0	5,1	WU7
647	582	325,5	828,5	172,5	0	237	0	5,1	WU8
672	602	338,5	865,5	172,5	0	237	0	5,1	WU9
697	621	350,5	907,5	172,5	0	236	0	5,1	WU10
723	641	363,5	952,5	172,5	0	236	0	5,1	WU11
747	659	375,5	981,5	172,5	0	235	0	5,1	WU12
773	678	388,5	1025	172,5	0	235	0	5,1	WU13
796	696	400,5	1058	172,5	0	235	0	5,1	WU14
60	54	30	80	11	0	13,7	0	1,3	DM/2-1
80	55	40	116	11	0	15,1	0	1,3	DM/2-2
100	72	50	153	11	0	16,5	0	1,3	DM/2-3
120	89	60	189	11	0	17,9	0	1,3	DM/2-4
140	91	70	225	11	0	19,3	0	1,3	DM/2-5
160	108	80	262	11	0	20,7	0	1,3	DM/2-6
180	125	90	298	11	0	22,1	0	1,3	DM/2-7
200	126	100	334	11	0	23,5	0	1,3	DM/2-8
220	143	110	370	11	0	24,9	0	1,3	DM/2-9
240	160	120	407	11	0	26,3	0	1,3	DM/2-10
66	51	33,2	95,7	10,9	0	13	0	1,3	D-M.2-1
101	71	50,7	155,7	10,9	0	16	0	1,3	D-M.2-2
126	86	63,2	195,7	10,9	0	17	0	1,3	D-M.2-3
161	106	80,7	270,7	10,9	0	20	0	1,3	D-M.2-4
191	121	95,7	295,7	10,9	0	22	0	1,3	D-M.2-5
221	136	110,7	380,7	10,9	0	24	0	1,3	D-M.2-6
157	133	79	189	52	0	178	0	3,2	LA1
181	143	92	147	52	0	152	0	3,2	LA2
189	148	95	180	52	0	161	0	3,2	LA3
197	153	99	228	52	0	169	0	3,2	LA4
210	158	106	199	52	0	158	0	3,2	LA5
218	163	109	245	52	0	166	0	3,2	LA6
225	169	113	314	52	0	173	0	3,2	LA7

continued on next page

Table A.1 – continued from previous page

W	**H**	**r**	**r1**	**r2**	**r3**	**a**	**a1**	**WH**	**Remarks**
cm	*cm*	*cm*	*cm*	*cm*	*cm*	*deg*	*deg*	*cm*	
239	173	120	262	52	0	163	0	3,2	LA8
246	179	123	325	52	0	170	0	3,2	LA9
260	183	131	278	52	0	160	0	3,2	LA10
267	189	134	336	52	0	167	0	3,2	LA11
281	193	142	294	52	0	157	0	3,2	LA12
288	199	145	348	52	0	164	0	3,2	LA13
295	204	148	421	52	0	170	0	3,2	LA14
309	209	156	361	52	0	161	0	3,2	LA15
164	113	92	131	42	0	107	0	3,2	LB1
174	118	94	170	42	0	119	0	3,2	LB2
183	122	97	236	42	0	130	0	3,2	LB3
196	126	108	194	42	0	117	0	3,2	LB4
205	130	109	260	42	0	128	0	3,2	LB5
213	135	111	385	42	0	138	0	3,2	LB6
227	138	122	284	42	0	126	0	3,2	LB7
235	143	124	397	42	0	135	0	3,2	LB8
249	146	136	306	42	0	123	0	3,2	LB9
257	151	137	415	42	0	133	0	3,2	LB10
271	155	149	326	42	0	121	0	3,2	LB11
280	159	150	430	42	0	131	0	3,2	LB12
294	163	163	348	42	0	120	0	3,2	LB13
302	167	163	446	42	0	129	0	3,2	LB14
316	171	177	367	42	0	118	0	3,2	LB15
70	55	35	98	11	0	180	0	1,3	M1
80	63	40	116	11	0	180	0	1,3	M2
100	72	50	153	11	0	180	0	1,3	M3
120	89	60	189	11	0	180	0	1,3	M4
140	91	70	225	11	0	180	0	1,3	M5
160	108	80	262	11	0	180	0	1,3	M6
180	125	90	298	11	0	180	0	1,3	M7
200	126	100	334	11	0	180	0	1,3	M8
220	143	110	370	11	0	180	0	1,3	M9
240	160	120	407	11	0	180	0	1,3	M10
197	201	84	179	70	169	112	32	5,5	UF1
266	231	124	314	70	190	119	28	5,5	UF2
289	254	116	359	70	199	81	47	5,5	UF3
344	323	146	314	122	210	74	51	5,5	UF4
359	312	148	494	70	284	100	38	5,5	UF5
382	329	169	538	70	334	120	28	5,5	UF6
437	386	198	494	122	291	102	37	5,5	UF7
460	409	187	538	122	306	79	48	5,5	UF8

B River Flow controller options

This is a more detailed description of the use of the so called River Flow controllers. Following topics are discussed:

- ◇ General
- ◇ Controlling Procedure Applied in River Flow
- ◇ Overview of Controllers Available in River Flow
- ◇ Trigger Procedure Applied in River Flow
- ◇ Overview of Triggers Available in River Flow

General
A distinction is to be made between **a controller**, **a control parameter**, and **a controlled parameter**, viz:

- ◇ A controller is referred to as the computational procedure, which facilitates the user-defined way of controlling a particular structure;
- ◇ A control parameter refers to a parameter, which acts as the input to the controller in order to control another parameter, being the controlled parameter;
- ◇ A controlled parameter refers to the parameter, which the user likes to be controlled in a specific way during a computation.

Example 1:

Consider the fact that a user likes to maintain the discharge, flowing out of a small barrage constant in time by manipulating the crest level of a weir accommodated in this barrage. For this case the user can use the **interval controller.** The crest level of the weir will be the **control parameter,** while the outflow discharge is the **controlled parameter.**

Example 2:

Consider a weir operated by a **time controller,** which maintains the user-defined time-dependent crest levels of the weir. In this case the crest level is the **controlled parameter.** It will be obvious that there is no **control parameter** in this example.

Following parameters of hydraulic structures can be used for controlling purposes, viz:

- ◇ **River weir:** crest level can be adjusted.
- ◇ **Advanced weir:** crest level can be adjusted.
- ◇ **General structure:** crest level, crest width and gate lower edge level can be adjusted.
- ◇ **Database structure:** crest level can be adjusted.

In the River Flow module compound structures can be defined, where several different type of hydraulic structures can be defined next to each other at the same location. Each member of the compound structure may have its own triggers and controllers.

Controlling Procedure Applied in the River Flow module

The procedure, applied in SOBEK, for controlling hydraulic structures is as follows:

- ◇ The user has the option to define a trigger that is assigned to a particular controller. During a computation a trigger can either be "ON" or "OFF". In case a trigger is "ON", the corresponding controller will be activated, resulting in the fact that a particular hydraulic structure will be controlled in a used-defined way. In case a trigger is "OFF", the corresponding controller will be deactivated. In case no trigger is assigned to aparticular controller, this controller will be active during the entire computation. It is not fruitful to use a relative time controller and a relative from value (time) controller without triggers;
- ◇ A controller can be activated or de-activated by a trigger or by a combined trigger (maximum of four triggers can be combined to one combined trigger);
- ◇ A **maximum of four** controllers can be assigned to **one** particular hydraulic structure. It is to be mentioned that checks are not carried out for inconsistencies in the formulations of the four user-defined controllers.
- ◇ The controller is only activated **once during a computational time step (i.e. at the beginning of a time step execution),** meaning that the value of the controlled parameter is not updated during the Newton iteration process of solving the flow equations;
- ◇ The user has to define the **control frequency,** which refers to the time interval, after which the controlled parameter is to be updated again, in case the controller has not yet been de-activated. A default value for the control frequency is 1 (one), meaning that the controlled parameter will be updated at the beginning of each and every time step for which the controller has not yet been deactivated.

Overview of Controllers Available in the River Flow module

Six different types of controllers are available, viz:

1 Time controller,
2 Relative time controller,
3 Relative from value (time) controller,
4 Hydraulic controller,
5 Interval controller, and
6 PID controller.

B.1 Time controller

The **time controller** can be used for changing the settings of a hydraulic structure **(controlled parameter)** during a computation.

The controlled parameter can for instance be:

- ◇ the crest level of a hydraulic structure,
- ◇ the crest width of a hydraulic structure, or
- ◇ the gate lower edge level of a hydraulic structure

Please note that there is **no control parameter** in case of a time controller.

Following input is to be specified for a **time controller**, viz:

Regarding the controlled parameter

- ◇ Define whether it is the crest height, crest width or gate lower edge level of a particular hydraulic structure.

◇ Specify the time-dependent values of the controlled parameter.

Regarding the control parameter:

A control parameter is not applicable.

Regarding the controller and control procedure:

◇ **Update frequency:** This defines how often the controller function should be updated. Normally this will be every time-step, but can also be an interval of more time steps. When you enter 1, the controller will be updated every time-step; for 2 it will be updated every second time-step, and so on. Default is 1.
◇ **Control mechanism:** The procedure followed by the **time controller** is straightforward. Based on the computational time Δt, SOBEK determines in the user-defined time-table the requested value for the controlled parameter. After that SOBEK takes care that this value is given to the controlled parameter (e.g. the crest level, the crest width or the gate lower edge level of the hydraulic structure)
◇ **Control accuracy and applicability:** Using this option the user should pay much attention in specifying a realistic time-table for the controlled parameter in order to avoid instabilities in the computation. A realistic time-table for the controlled parameter means a time-table which can be met from the hydrodynamic point of view.

B.2 Relative time controller

The **relative time controller** can be applied for changing the settings of a hydraulic structure **(controlled parameter)** during a computation. The relative time controller is, therefore, a time controller type of controller.

The difference with the time controller option comprise of the fact that in the relative time controller option, the values for the controlled parameter have not to be specified for the entire computation time. When activated the relative time controller starts at the top of the user-defined time-table and from there on continues downward until the controller is de-activated by its trigger (see also control mechanism).

The controlled parameter can for instance be:

◇ the crest level of a hydraulic structure,
◇ the crest width of a hydraulic structure, or
◇ the gate lower edge level of a hydraulic structure.

Please note that there is **no control parameter** in case of a Relative time controller.

Following input is to be specified for a **relative time controller**, viz:

Regarding the controlled parameter:

◇ Define wether it is the crest height, crest width or gate lower edge level of a particular hydraulic structure.
◇ Specify the relative time-dependent values of the controlled parameter.

Regarding the control parameter:

◇ A control parameter is not applicable.

Regarding the controller and control procedure:

◇ **Update. Frequency** This defines how often the controller function should be updated. Normally this will be every time-step, but can also be an interval of more time steps. When you enter 1, the controller will be updated every time-step; for 2 it will be updated every second time-step, and so on. Default is 1.

◇ **Control mechanism:**

- For the first time that the controller is activated, the relative time table will be made absolute (= computation time + relative time) and the controller will start from the top of the table and will continue downwards untill the controller is deactivated by its trigger. The table will remain absolute during the user defined so called Start period. In other words in case the controller is de-activated and activated again within this Start period, the controller table remains absolute and at the point-in-time when the controller is activated again, the value coresponding to this absolute point-in-time will be obtained from the controller Table. In case the controller is activated after the so called Start period has passed, the controller table will be made absolute again and the controller will start from the top of the table and will continue downwards;
- Start period=0, means that each time the controller is triggered, the controller table will be made absolute;
- When the controller branches the end of the controller time table, the value of the controlled parameter is kept constant at the value found in the last row of the time table;
- In case the user defined value for $d(\text{value})/dt$ is too small to allow for the in the controller table defined changes in controlled parameter, SOBEK will divert from these defined controlled parameter values in such way as the best fit the overal defined controller table. d(value)/dt=0 means that there is no restriction in change in controlled parameter over one computational time step.

◇ **Control accuracy and applicability:** Using this option the user should pay much attention in specifying a realistic time-table for the controlled parameter in order to avoid instabilities in the computation. In addition the value of the controlled parameter at the moment that the controller is activated should be preferably be known in order to avoid large discontinuities and hence instabilities during the computation, which might even result in a termination of the program. A realistic time-table for the controlled parameter means a time-table which can be met from the hydrodynamic point of view.

B.3 Relative from value (time) controller

The **relative from value (time) controller** can be used for changing the settings of a hydraulic structure **(controlled parameter)** during a computation. The relative from value (time) controller is, therefore, a time controller type of controller. For the relative from value (time) controller, the values for the controlled parameter have not to be specified for the entire computation time. The relative from value (time) controller is usually applied when the actual values of the controlled parameter are unknown at the moment in which the relative from value (time) controller is activated. Therefore, when activated the relative from value (time) controller first determines the actual value of the controlled parameter. After that the relative from (time) value controller starts in the user-defined time-table at the actual value of the controlled parameter and continues downwards from this value onwards (see also control mechanism). In this way possible large discontinuities in the value of the controlled parameter are avoided and hence instabilities in SOBEK computations, leading to the possible termination of the program.

Following input is to be specified for a **relative from value controller**, viz:

Regarding the controlled parameter:

- ◇ Define wether it is the crest height, crest width or gate lower edge level of a particular hydraulic structure.
- ◇ Specify the relative time-dependent values of the controlled parameter.

Regarding the control parameter:

A control parameter is not applicable.

Regarding the controller and control procedure:

- ◇ **Update frequency:** This defines how often the controller function should be updated. Normally this will be every time-step, but can also be an interval of more time steps. When you enter "1", the controller will be updated every time-step; for "2" it will be updated every second time-step, and so on. Default is 1.
- ◇ **Control mechanism:** The procedure of the relative from value controller is as follows:
  - ◻ For the first time that the controller is activated, the relative time table will be made absolute (= computation time + relative time). The controller will determine the actual value of the controlled parameter. After that the controller searches for the line in the user-defined time table, which has the same value as the actual value of the controlled parameter. From there on the controller will continue downwards in the controller table untill it is de-activated. The controller table will remain absolute during the user defined so called Start period. In other words in case the controller is de-activated and activated again within this Start period, the controller table remains absolute and at the point-in-time when the controller is activated again, the value corresponding to this absolute point-in-time will be obtained from the controller table, which might lead to a discontinuity in controlled parameter value. In case the controller is activated after the so called Start period has passed, the controller table will be made absolute again and the controller willl start at the row coinciding with the actual value for the controlled parameter;
  - ◻ Start period=0, means that each time the controller is triggered the controller table will be made absolute;
  - ◻ When the controller branches the end of the controller time table, the value of the controlled parameter is kept constant at the value found in the last row of the time table;
  - ◻ In case the user defined value for $d(\text{value})/dt$ is too small to allow for the in the controller defined changes in controlled parameter, SOBEK will divert from these defined controlled parameter values in such way as to best fit the overall defined controller table. A value for $d(\text{value})/dt = 0$ means that there is no restriction in the allowable change in controlled parameter value over one computational time step.
- ◇ **Control accuracy and applicability:** Using this option the user should pay much attention in specifying a realistic time-table for the controlled parameter in order to avoid instabilities in the computation. A realistic time-table for the controlled parameter means a time-table which can be met from the hydrodynamic point of view.

B.4 Hydraulic controller

The hydraulic controller can be used to operate a hydraulic structure (i.e. the **controlled parameter)** based on the actual value of a specified hydraulic parameter (i.e. the **control parameter**).

The controlled parameter can for instance be:

◇ the crest level of a hydraulic structure,
◇ the crest width of a hydraulic structure, or
◇ the gate lower edge level of a hydraulic structure

The control parameter can be either:

◇ the actual value of a water level at a specific location,
◇ the actual value of the averaged flow velocity in the total flow section at a specific location,
◇ the actual direction of the flow at a specific location,
◇ the actual value of a discharge at a specific location (or the sum of the actual discharge at a maximum five locations),
◇ the actual head-difference over a hydraulic structure. The head-difference of another structure than the controlled structure can be used.
◇ the actual pressure-difference over a hydraulic structure or the member of a compound structure. The pressure-difference of another structure than the controlled structure can be used.

The following input is to be specified for a **Hydraulic controller**, viz:

Regarding the controlled parameter.

◇ Define whether it is the crest height, crest width or gate lower edge level of a particular hydraulic structure.
◇ Note that the requested values for the controlled parameter are determined from the controller Table (see below). It is, therefore, not necessary to specify the minimum and maximum value of the control parameter as well as the maximum possible adjustment of the control parameter over a time step

Regarding the control parameter.

◇ Definition of the type of hydraulic control parameter (i.e. whether it is a water level, a velocity, the flow direction, a discharge, a head difference over a structure, or a pressure difference over a structure).
◇ Definition of the location of the hydraulic control parameter (i.e. **either** the specific location (in case of a water level, a velocity, the flow direction or discharge) **or** the concerning structure in case of a head difference or pressure difference).
◇ Definition of the **controller-function** (or controller table) for the specified hydraulic control parameter. The controller-function contains the relation between the actual value of the hydraulic control parameter and the by the user requested value of the controlled parameter (e.g. the crest height, the crest width or the gate lower edge level of the controlled hydraulic structure).

Regarding the controller and control procedure.

◇ **Update frequency:** This defines how often the controller function should be updated. Normally this will be every time-step, but can also be an interval of more time steps.

When you enter “1”, the controller will be updated every time-step; for “2” it will be updated every second time-step, and so on. Default is 1.

- **Control mechanism:** The procedure followed by the **Hydraulic controller** is straightforward. First, the actual value of the hydraulic controller for $t = t$ is determined. Next the value of controlled parameter (e.g. the crest height, crest width or gate lower edge level of the hydraulic structure) is obtained by interpolation in the user-defined controller table. Finally SOBEK adjusts the actual value at $t = t + \Delta t$ of the crest height, crest width or gate opening of the hydraulic structure in accordance with the findings of the interpolation in the controller table.
- **Control accuracy and applicability:** Using this option the user should pay much attention in specifying a realistic controller-function (i.e. the relation between the value of the control parameter and the requested value of the controlled parameter) in order to avoid instabilities in the computation. A realistic controller-function means a function which can be met from the hydrodynamic point of view.

B.5 Interval controller

The **interval controller** can be used to operate a hydraulic structure (i.e. the **control parameter)** in such way that user-specified values of a hydraulic parameter (i.e. **controlled parameter**) are maintained.

The controlled parameter can be either:

- a water level at a specific location, or
- a discharge at a specific in the model.

Note:
The location of the to be controlled parameter should be under the control range of the concerning hydraulic structure.

The control parameter can for instance be either:

- the crest level of a hydraulic structure;
- the crest width of a hydraulic structure, or
- the gate lower edge level of a hydraulic structure.

The following input is to be specified for a **Interval controller**, viz:

Regarding the controlled parameter:

- Definition of the controlled parameter (i.e. water level or discharge at a particular location in the model).
- Setpoints for the controlled parameter. These are in fact the user-defined values for the controlled parameter. There are two possibilities in SOBEK, viz:
  - a constant water level or discharge, or
  - a time-dependent water level or discharge, defined by means of a table.
- Definition of the dead band type. On basis of the specified dead band type, SOBEK determines whether the value of the controller v (e.g. crest height, crest width or gate lower edge level of the hydraulic structure) are to be adjusted. The following dead band types are available in SOBEK, viz:
  - a constant dead band value D, or.
  - a dead band D as percentage of the discharge.

In this case except for the percentage, the minimum and maximum dead band values are to be specified as well.

Regarding the control parameter:

- ◇ Define wether it is the crest height, crest width or gate lower edge level of a particular hydraulic structure.
- ◇ The so called “$<$ Dead band” and “$>$ Dead band” control values, that defines the direction in which the control parameter (e.g. gate lower edge level) should move in order to maintain the defined set points for the controlled parameter (e.g. water level). In case actual value for controlled parameter is below set point + dead band, the control parameter should move towards the value specified for “$<$ Dead band”. In case the controlled parameter is above set point + dead band, the control parameter should move towards the value specified for “$>$ Dead band” (see also control mechanism below). Please note that only positive value for the control interval (dVs) can be defined.
- ◇ The “$<$ Dead band” and “$>$ Dead band” control values are also the range allowed for the control parameter (e.g. crest level) in order to try to maintain the controlled parameter within its defined set points.
- ◇ Definition of the control interval **(dvs) ,** which refers to the user-defined change in the value of the control parameter over 1 (one) time step. The reason why the user has to define a control interval **(dvs)** is the fact that in reality it takes time to adjust the crest height, crest width or gate lower edge level of hydraulic structure. In practical situations there is a limit to the change in the value of the control parameter over a time step. In SOBEK two types of control interval can be defined, viz:
  - □ *fixed control interval*, meaning that if required the value of the control parameter can be adjusted with a constant value only (for instance 0. 1 m' in case of a gated structure).
  - □ *variable control interval*, meaning that if required the value of the control parameter can be adjusted by the product of the actual computational time step Δt times the user-defined adaptation velocity v. For instance in case of a crest level as control parameter and an adaptation velocity v of 0.002 m/s, means that over a time step Δt = 10 s, the change in value of the control parameter (if activated by its trigger) will be $10 \times 0.002 = 0.02$ m.

Regarding the controller and control procedure:

- ◇ **The update frequency:** which refers to the time interval after which SOBEK updates the value of the to-be-controlled parameter in case the controller has not yet been de-activated by its trigger or combined trigger (see also point 3 under General aspects of controllers),
- ◇ **The control mechanism:** SOBEK applies the following algorithm for determining whether the value of the **control parameter (v.)** is to be updated, viz:
  if $-0.5D < e < +0.5D$ then $V_s = V_{s,old}$
  if $e < -0.5D$ then $V_s = V_{s,old} + d\Delta v_s$
  if $e > +0.5D$ then $V_s = V_{s,old} - d\Delta v_s$
  in which:

D	dead band (in $[m]$ in case of water level, and in $[m^3/s]$ in case of discharge),
d	control direction (1 or -1). Positive if “$>$ Dead band” control value larger than “$<$ Dead band” control value. Negative if “$>$ Dead band” control value smaller than “$<$ Dead band” control value.
e	deviation of the controlled parameter

= setpoint - actual value of the controlled parameter

ΔV_s control interval (fixed value or computed by $\Delta V_s = |v|\Delta t$,

in which:

v adaptation velocity, please note that v is always positive

$V_{s,old}$ value of the control parameter in the previous time step.

◇ **Controller accuracy and applicability:** The interval controller is not a very advanced type of controller. It is for instance, sensitive to instabilities, in particular if the adaptation velocity, control frequency or dead band are not selected properly. Also the control history (before the last time step is not ta$K_i$ng into account to determine the control parameter.

B.6 PID controller

As for the interval controller, the **PID controller,** can be used to operate a hydraulic structure (i.e. the **control parameter)** in such way (i.e. by adjusting its crest level, crest width or gate lower edge level) , that user-specified values of a hydraulic parameter (i.e. **controlled parameter)** are maintained. The difference with the interval controller option comprises of the fact that the PID controller option does take the control history into account.

The controlled parameter can be either:

◇ a water level at a specific location, or
◇ a discharge at a specific location in the model.

Note:
The location of the controlled parameter should be under the control range of the concerning hydraulic structure.

The control parameter can for instance be either:

◇ the crest level of a hydraulic structure,
◇ the crest width of a hydraulic structure, or
◇ the gate lower edge level of a hydraulic structure

The following input has to be specified for a **PID controller**:

Regarding controlled parameter:

◇ Definition of the controlled parameter (i.e. water level or discharge at a particular location in the model).
◇ Setpoints for the controlled parameter. These are in fact the user-defined values for the controlled parameter. There are two possibilities in SOBEK:

1 a constant water level or discharge, or
2 a time-dependent water level or discharge, defined by means of a table.

Regarding control parameter:

- ◇ Define whether it is the crest height, crest width or gate lower edge level of a particular hydraulic structure.
- ◇ Initial value of the control parameter (e.g. initial crest width).
- ◇ Maximum and minimum value of the control parameter (e.g. maximum and minimum values for the gate lower edge level).
- ◇ Maximum adaptation velocity of the control parameter (v_{max}). This refers to the maximum change in the value of the control parameter over one computational time step. For instance in case of a crest level as control parameter $v_{max} = 0.02$ m/s means that over a time step $\Delta t = 60$ s, the crest may be lowered or raised over $0.02 \times 60 = 1.2$ m.

Regarding controller and control procedure:

- ◇ **The control frequency,** which refers to the time interval after which SOBEK updates the value of the to-be-controlled parameter in case the controller has not yet been de-activated by its trigger or combined trigger,
- ◇ **Control mechanism:** Values for the following parameters are to be defined by the user:
  - $K_p^{'}$, proportional gain factor.
  - $K_i^{'}$, integral gain factor.
  - $K_d^{'}$, differential gain factor.

  The value of the control parameter $w(t)$ (i.e. user-defined water level or discharge) is computed as follows:

  1 First compute the correction value

$$u(t) = K_p^{'} e(t) + K_i^{'} \sum_{t=0}^{t} e(t) + K_d^{'}(e(t) - e(t-1)) \tag{B.1}$$

$$K_p^{'} = k_1 K_p, \qquad K_i^{'} = k_2 K_i, \qquad K_d^{'} = k_3 K_d \tag{B.2}$$

  2 Perform this correction value to the control parameter $w(t)$

$$w(t) = w(t-1) + u(t) \tag{B.3}$$

  in which:

  K_p, K_i, K_d Coefficient of the PID-controller (Åström and Hägglund, 1995).
  k_j $j = 1, 2, 3$. Conversion factor between the controll parameter (e.g. discharges [m^3/s]) and the controlled parameter (e.g. water levels [m])
  $e(t)$ deviation of the controlled parameter $y(t)$, $e(t) = y_{sp}(t) - y(t)$
  $u(t)$ correction factor on the control parameter, e.g. $e(t) \equiv 0 \Rightarrow u(t) = 0 \Rightarrow w(t)$ is unchanged.

  The above given equation for computing $w(t)$ on basis of $w(t-1)$, $K_p^{'}$, $K_i^{'}$, $K_d^{'}$, $e(t)$ and $e(t-1)$, refers to the fact that, for the PID controller method, the control history is taken into account.
  The following limitations are applied for the change in the control parameter $\Delta w (= w(t) - w(t-1))$ over one user-defined timestep (can be specified in Settings):

  - The value of the control parameter $w(t)$ cannot become larger than the user-defined maximum value.
  - The value of the control parameter $w(t)$ cannot become smaller than the user-defined minimum value.
  - The change in the control parameter cannot become larger than the user-defined maximum allowable change in the control parameter $dv_{\max} (= v_{\max} \Delta t)$ over one user-defined time-step.

If necessary the computed value for the controlled parameter $w(t)$ according to the formula above is overruled by a value for the controlled parameter that meets the limitations specified above.

- ◇ **Controller accuracy and applicability:** Since the PID controller is taking into account the history of the control parameter it is supposed to be more accurate than the interval controller.

Selection of gain factors $K_p^{'}$, $K_i^{'}$ and $K_d^{'}$

The gain factors $K_p^{'}$, $K_i^{'}$ and $K_d^{'}$ must be calibrated for optimal performance of the PID controller. For example the calibration can be carried out as follows:

- ◇ Take $K_i^{'}$, $K_d^{'}$ equal to zero, increase the value of $K_p^{'}$ gradually from zero until the solution starts to oscillate. The sign of $K_p^{'}$ must be chosen dependent of the type of structure and the chosen control parameter (see note below)
- ◇ Decrease $K_p^{'}$ until the controller is stable again.
- ◇ Increase the value of $K_d^{'}$ gradually. In case of small fast reacting channels start with $0.5K_p$, in case of slow reacting channels like big rivers start with $K_p^{'}$. When the controller starts to oscillate, decrease K_d.
- ◇ Tune $K_p^{'}$ and $K_d^{'}$ until the result is satisfactory.

In most cases $K_i^{'}$ can be kept zero. The $K_i^{'}$ factor can be used to smooth the movements of the controller, but is often not necessary, the water movements are already smoothing by it self. In case of non-constant target values the $K_i^{'}$ factor is even disturbing a proper functioning of the controller and should not be used at all.

A strict procedure for this calibration can not be presented, since the procedures and results are depending on the type of model. However after having some experience with tuning PID controllers the modeller will gain some 'Fingerspitzengefühl' as the Germans call it. In a model with several structures with PID controllers which have the same order of dimensions like channel section lengths, cross-section sizes, and structures, the K-factors tuned for one controller are a best guess for the other ones. Often they can be the same for all structures/controllers.

Note:
Gain factors may be positive or negative. The choice of the sign depends on the type of the control structure (e.g. crest level, crest width or gate lower edge level) and the location (upstream or downstream of structure) and type (e.g. water level or discharge) of the hydraulic parameter that is controlled by the PID controller. For example, consider a PID controller that at a bifurcation tries to maintain a constant discharge, $y_{sp}(t)$, flowing into one branch by manipulating the crest level of a River weir located in the branch that should receive this constant discharge. In case the discharge, $y(t)$, flowing into the branch of which its discharge is controlled is too large, then the deviation e is negative (i.e. $e(t) = y_{sp}(t) - y(t) < 0$). From a hydraulic point of view the crest level $w(t)$ of the river weir is to be raised in order to reduce the discharge flowing into the controlled branch. In order to achieve the $K_p^{'}$ gain factor should be negative.

Note:
The hydraulic parameter must be taken at a location which is directly influenced by the PID controller. The PID controller reacts also on the effect of its own actions. So it is not possible to properly calibrate a PID controller with a controlled (hydraulic) parameter which location is outside the influence area of the control structure.

C Deprecated functionality

C.1 Linkage node

Note: The *Flow - Linkage node* is considered deprecated functionality. It is currently only available for the purpose of backward compatibility. In SOBEK versions 2.12.004 and upwards, it is possible to add interpolation of *Flow - Cross Section* data between branches using the *Flow - Connection node*. It is recommended to use this functionality rather than the *Flow - Linkage node.*

The Flow -Linkage node is deprecated, see section C.1.2 for information on how to convert existing linkage nodes in the model to connection nodes.

C.1.1 Flow - Linkage node (deprecated)

Description

In this chapter, the *Flow - Linkage node* is described.

As described in more detail below, the *Flow - Linkage node* was intended to easily add and remove small tributaries to a parent river. Thus, the use of this node was limited to situations where inflows have discharges of at least an order of magnitude lower than the parent river. Please note that the Flow - Linkage Node option was not designed for modelling bifurcations.

- ◇ For a detailed description of this node's possible network configurations: see the "Flow - Linkage node topology" section from the Reference Manual

This type of node is a combination of a normal *Flow - Connection* node and a *Flow - calculation point* node. It has the following features:

- ◇ The linkage node it does not change if you use the option <Set calculation grid branch>. The location is fixed.
- ◇ The linkage node can be used as start or end node (like a normal connection node) for other branches.

It is very useful to have the main river as one long branch and to make the tributary rivers confluencing with linkage nodes. See the picture below, with the branch IDs displayed. Branch 1 is the main river, while all other branches are confluences of other rivers (the inflows have discharges of at least an order of magnitude lower than the parent river).

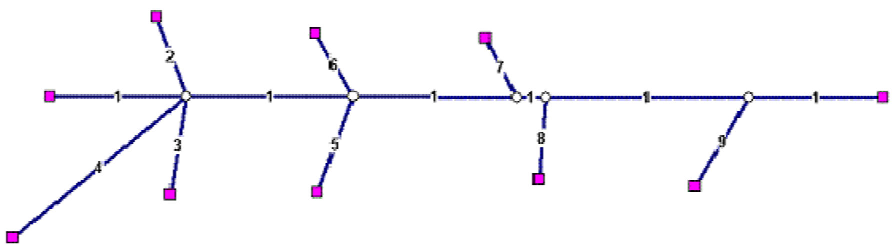


Figure C.1: *River confluences with linkage nodes*

The advantage of this method above using normal connection nodes for all the confluences is that the interpolation of cross-sectional data interpolation over the main river always continues. This is shown in the picture below. Although there are only two profile nodes at the main branch, the interpolation is done like there are no confluences. By using normal connection nodes for the confluences the interpolation only can be forced by added profile nodes with a definition that should be interpolated by the modeller.

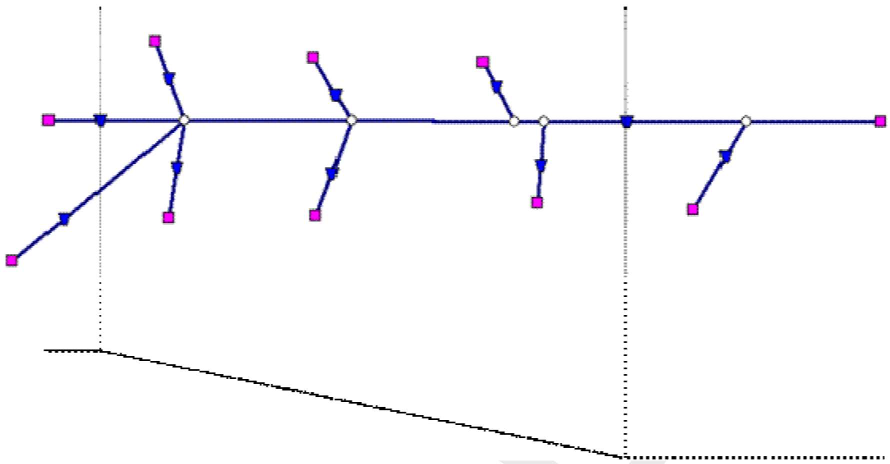


Figure C.2: *River confluences for linkage nodes showing interpolation over linkage node for main channel*

Topology
Note: The *Flow - Linkage node* is considered deprecated functionality. It is currently only available for the purpose of backward compatibility. In SOBEK versions 2.12.004 and upwards, it is possible to add interpolation of *Flow - Cross Section* data between branches using the *Flow - Connection node*. It is recommended to use this functionality rather than the *Flow - Linkage node.*

As described in more detail below, the *Flow - Linkage node* was intended to easily add and remove small tributaries to a parent river. Thus, the use of this node was limited to situations where inflows have discharges of at least an order of magnitude lower than the parent river. Please note that the Flow - Linkage Node option was not designed for modelling bifurcations.

The modeller can easily connect new branches to Linkage nodes. This works similar to normal connection nodes.

There is an extra functionality to join existing branches together as one branch, where the existing connection nodes are replaced by linkage nodes. This operation is very useful as many river systems are imported from GIS files, where each river section is defined as a single branch. By using the <join branches> edit action, it is possible to join all these single branches to one branch.

To make this option working correctly, it is required to have the option <Allow Edit for selected objects> enabled.

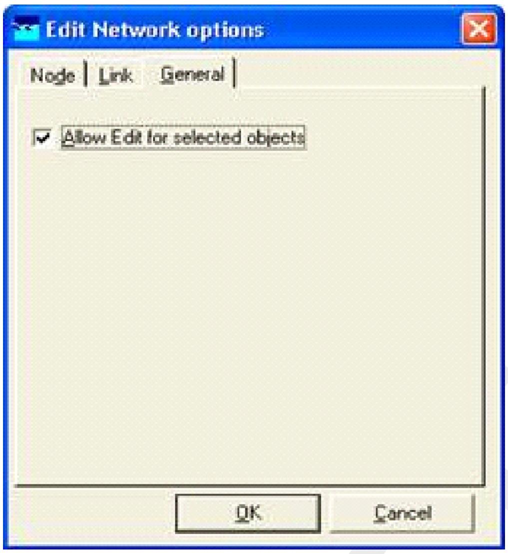


Figure C.3: *Check mark* Allow Edit for selected objects

Furthermore, the active node type should be the Flow - Linkage node.

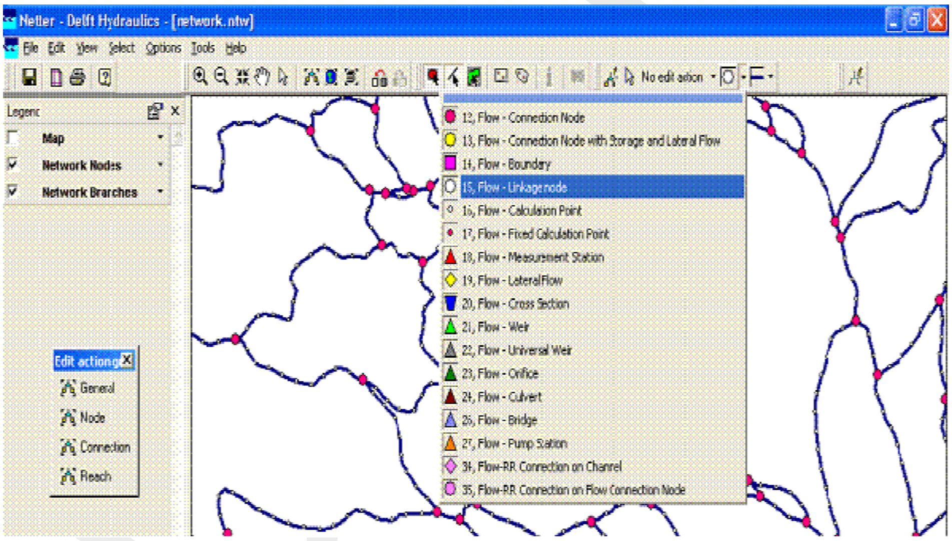


Figure C.4: *Select Flow - Linkage node*

The next step is to select the branches you want to join. This selection should include the connection nodes.

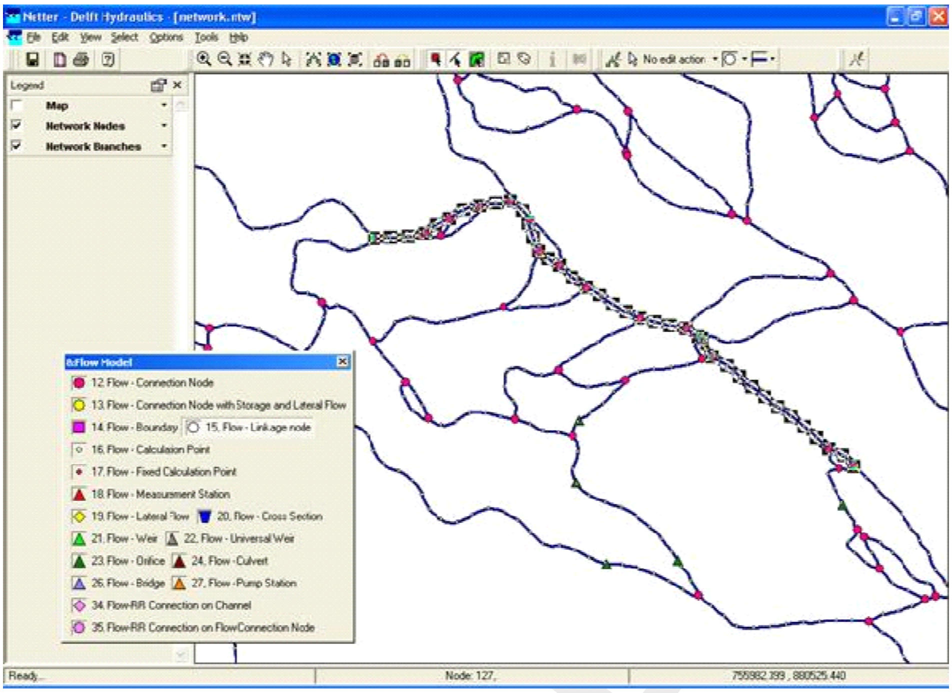


Figure C.5: Select branch to join

Now you have to select the action <Join branches>.

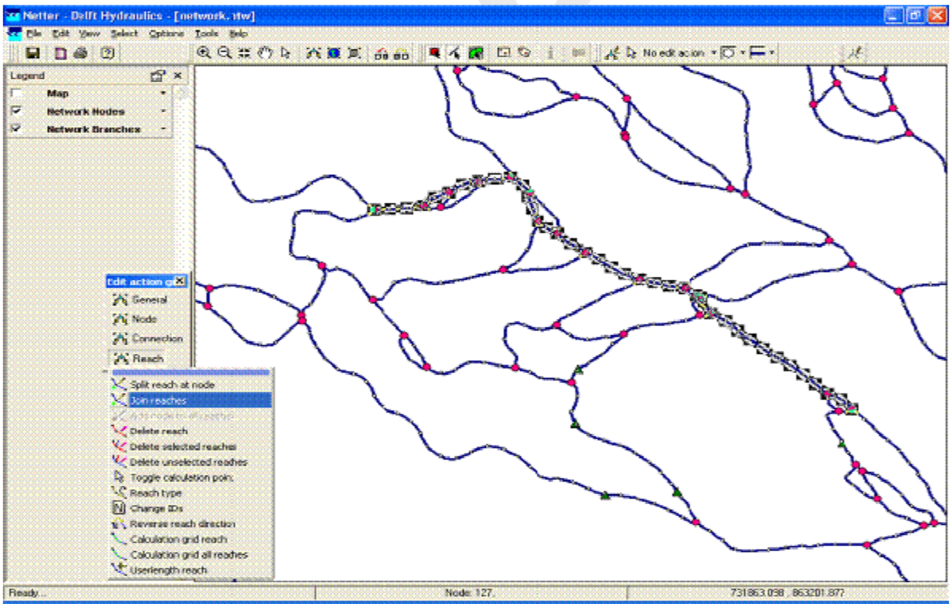


Figure C.6: Select action <Join branches>

You will be asked by the editor if you are sure. If so, you press *Yes* and the join operation will be performed.

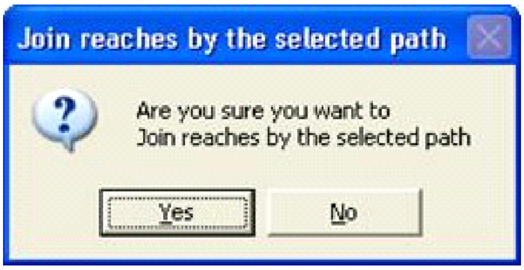


***Figure C.7: Join branches by the selectd path** window*

The existing connection nodes will be replaced by linkage nodes. Notice that also boundary nodes or flow connection nodes with lateral flow and storage will be replaced if they are in the selected path. You will lose the boundary, storage and/or flow definitions in that case.

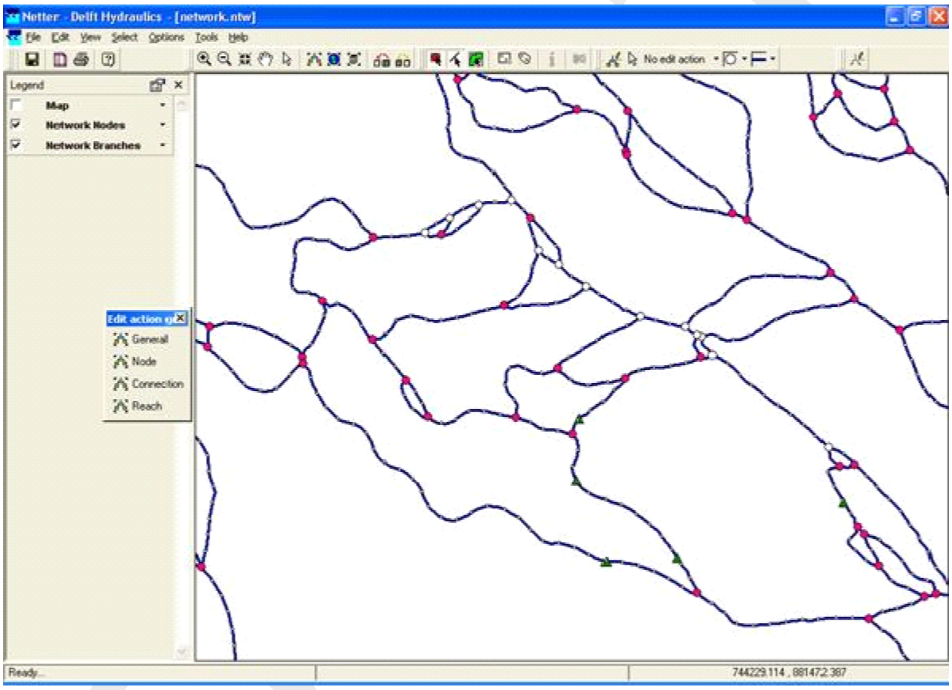


***Figure C.8:** Existing node is replaced by Linkage Node*

C.1.2 How to replace linkage nodes with connection nodes

This short tutorial will show you how to replace Flow - Linkage Nodes with Flow - Connection Nodes and how to enable interpolation of bathymetry data over a Flow - Connection Node. This tutorial is different from the other tutorials in that it is intended for users with existing SOBEK models containing linkage nodes. Linkage nodes are considered deprecated (abandoned) functionality and should not be used. For more information about linkage nodes, and why they were abandoned, see: Linkage node.

If you do not have any models with linkage nodes, this tutorial can be safely skipped as it does not provide any essential information not already covered by other tutorials.

- ◇ Start SOBEK.
- ◇ Click on the *Open Project* button.
- ◇ Select a project containing a linkage node you wish to replace.
- ◇ Open the case you want to edit.

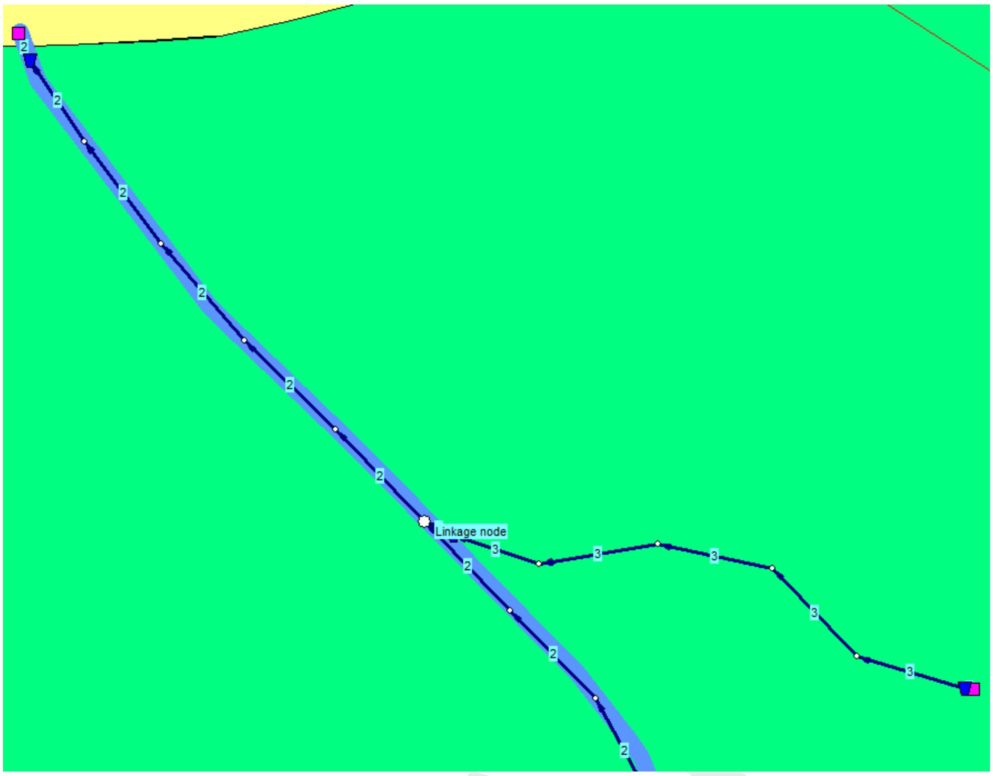


Figure C.9: *A schematisation with a Flow - Linkage Node.*

◇ Double click the 'Schematisation' task block.
◇ Click on the *Edit Model* button.

Your schematisation with a linkage node might look something like Figure C.9.

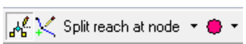
In the example shown in the screenshot, branch IDs have been visualised. This can be very useful when editing the model as described below. It is possible to visualise branch IDs by following these steps:

◇ Select 'Options' - 'Network Data...'.
◇ On the tab 'Link', select the radio button *Branch*.
◇ Click on *OK*.

In the screenshot of our example, you can see two branches in the schematisation. Branch 2 is the main branch, and branch 3 is the tributary.

First we will change the linkage node into a Connection Node. By definition a connection node is the start or end of a branch. So before we can place the connection node, it is necessary to split the two branches into three branches:

◇ Select the *Edit Network* button.
◇ Select the  node, Flow-Connection Node.
◇ Select the button *Split branch at node* in the 'Branch' toolbar.

Now your button bar should look like this: 

◇ Finally, left-click on the Flow - Linkage Node.

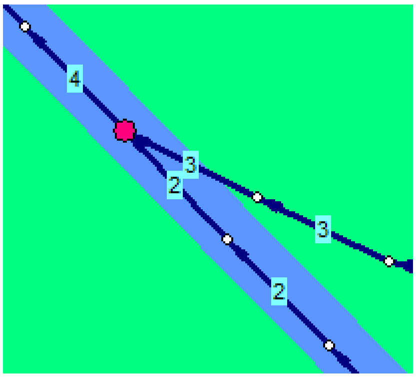


Figure C.10: *Example schematisation after the linkage node has been replaced by a connection node.*

The linkage has now been replaced by a connection node. In the example, the northern part of the main branch has now become branch 4. See Figure C.10.

Now all that is left is to define between which branches we want to interpolate bathymetry data.

◇ Select the connection node.
◇ Click on your right mouse button and select 'Model data' - 'Flow Model'.
◇ Turn on the option 'Interpolate Cross-section Bathymetry (only) over node'.
◇ Click on the listbox next to First Branch and select branch: '2'.
◇ Click on the listbox next to Second Branch and select branch: '4'.

See Figure C.11 for a filled in example using the above data.

◇ Finally, click on *OK*.

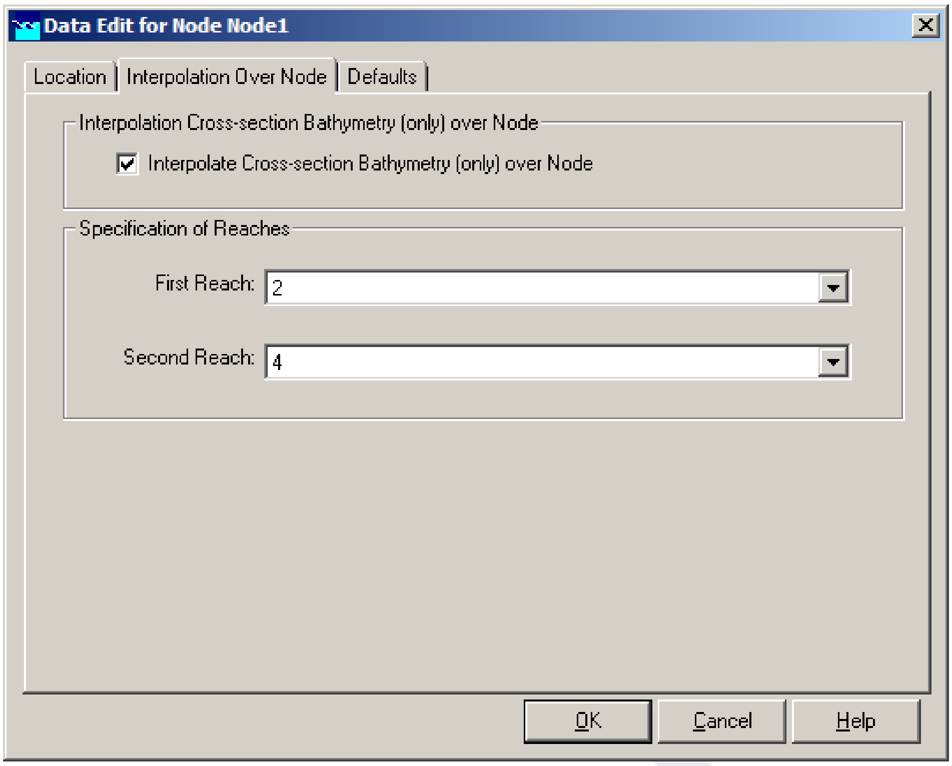


Figure C.11: *The filled in Data Edit screen for the example connection node.*

Using this method, linkage nodes can be replaced by connection nodes. In the above example, the Cross-section Bathymetry data is interpolated between branch 2 and branch 4.

Please note that every branch in SOBEK should contain at least one Cross-Section node. Depending on the schematisation performing the above steps might therefore require the user to add an extra cross-section after replacing the linkage node with a connection node.

Don't forget to save your network before leaving NETTER.

For more information about connection node interpolation, see also: Flow - Connection node.

C.2 Rainfall Runoff Friction

C.2.1 Flow - RR-Friction (deprecated)

Friction node description

The RR Friction node creates a water level difference between two nodes that calculate open water levels. The Friction node is designed to simulate the backwater effects of a narrow channel with certain length, friction and bank slopes.

Friction node input screens

When starting the model data editor for an RR Friction node type, the following tab will be available for input:

Friction tab:

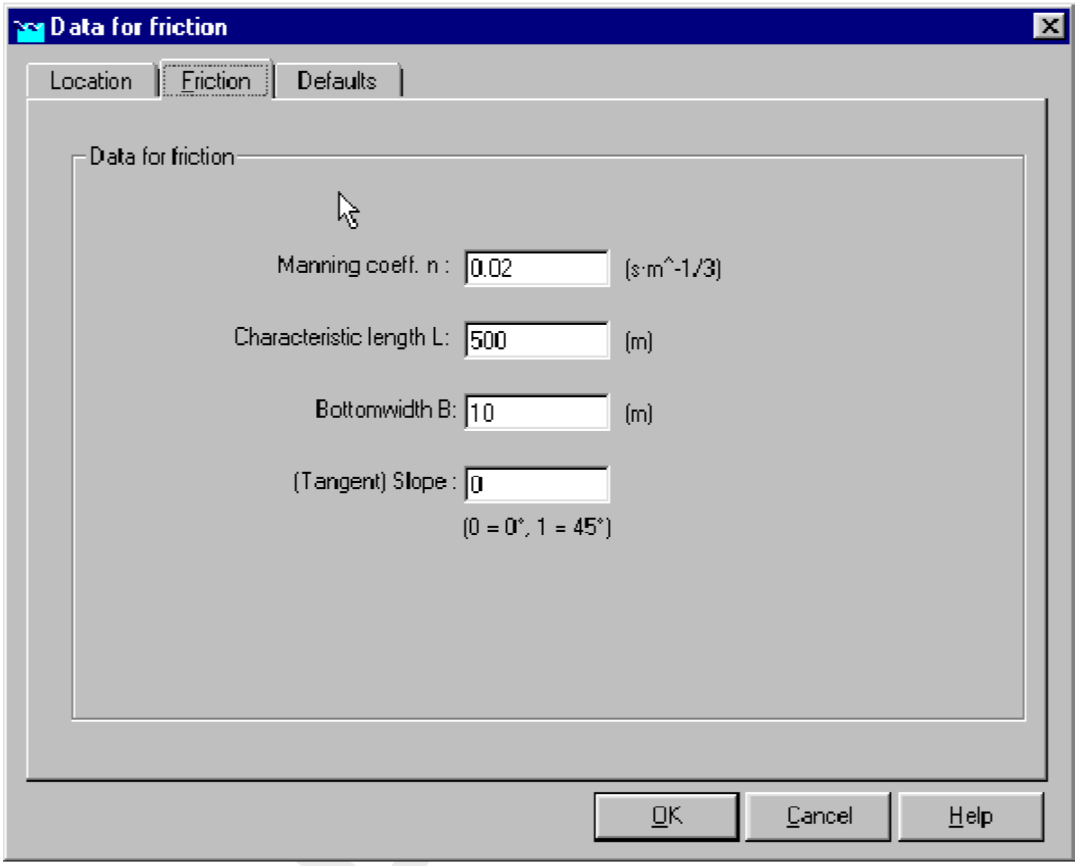


***Figure C.12: Data for friction**, the* friction *tab*

The following parameters should be entered here:

- **Manning coefficient n:** The hydraulic friction of the channel that the friction node represents
- **Characteristic length L:** The length of the channel that the friction node represents
- **Bottom width B:** The bed width of the channel that the friction node represents
- **Slope:** The tangent of the channel **bank sides** slope. (Not the slope of the channel bed itself!) 0 means that the banks are vertical. 1 means that the banks have a 45 degree slope.

See also:

The Friction chapter from the RR Technical Reference Manual

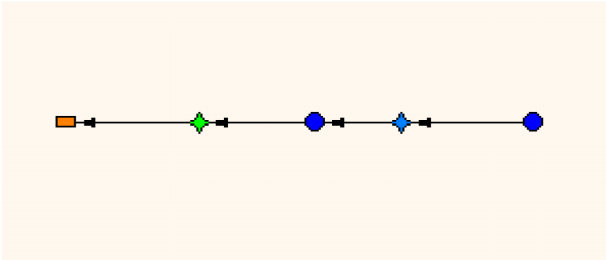


Figure C.13: Connecting two RR open water nodes through an RR friction node

Friction node connections

The RR Friction should always interconnect two of the following nodes:

- 

RR Open Water nodes:
- 

RR boundary nodes:
- 

CF RR on Flow connection nodes:
- 

CF RR on channel connections:

RR Friction nodes can be used to connect two RR Open water nodes (i.e. in polder systems):

But they can also be used to connect an RR Open Water node to a Channel Flow connection (i.e. in areas discharged by gravity where the second order water courses are modelled in an RR open water node, and the first order water courses are modelled in the SOBEK Channel Flow module).

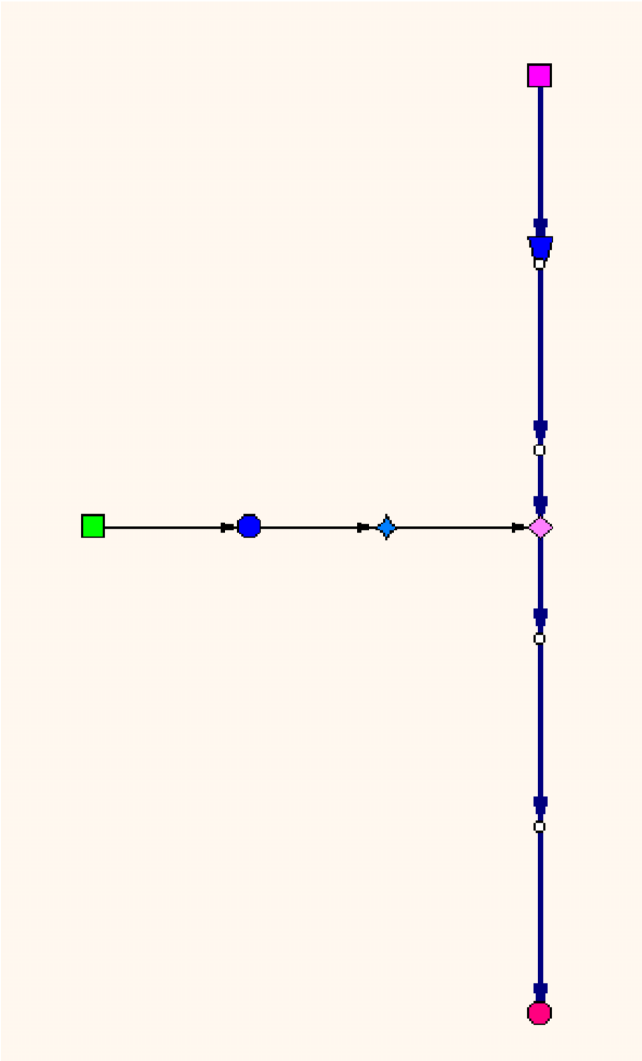


Figure C.14: Connecting an RR Open Water node to a Channel via an RR Friction node.

C.2.2 RR Friction node (deprecated)

A friction node can be put between two open water nodes to account for a significant hydraulic gradient. The discharge equation reads:

$$Q = \frac{1}{n} A R^{2/3} \sqrt{i} \tag{C.1}$$

Q	Discharge $[m^3/s]$
n	Manning coefficient $[s/m^{1/3}]$
A	Wetted area $[m^2]$
R	Hydraulic radius $[m]$
i	Average hydraulic gradient $[m/m]$

The wetted area is calculated by

$$A = DW \tag{C.2}$$

A	Wetted area $[m^2]$
D	Average depth $[m]$
W	Average width $[m]$

$$D = \frac{1}{2}(h_1 - h_2) - h_0 \tag{C.3}$$

h_1	Upstream water level [m wrt reference level]
h_2	Downstream water level [m wrt reference level]
h_0	Bed level [m wrt reference level]

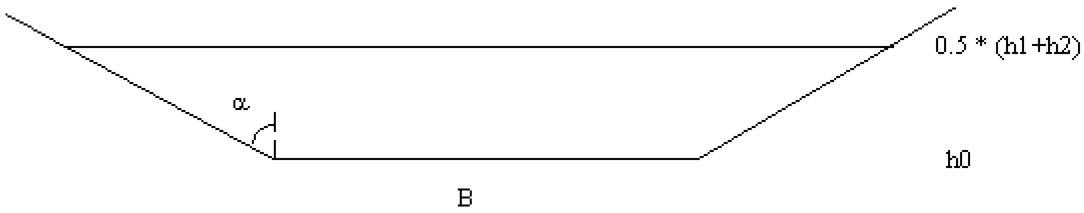


Figure C.15: *Explanation of symbols for calculating the wetted area*

When the friction node is put in between two RR-open water nodes, the bottom level h_0 is computed as the average of the bottom level of both open water nodes. When the friction node is put in between a RR-open water node and a RR- boundary node, the bottom level h_0 is set equal to the bottom level of the open water node.

The average width is calculated by

$$W = B + \tan(\alpha)D \tag{C.4}$$

B	Bottom width [m]

The hydraulic gradient is calculated by

$$i = \frac{h_1 - h_2}{L} \tag{C.5}$$

i	hydraulic gradient [m/m]
h_1	Upstream water level [m wrt reference level]
h_2	Downstream water level [m wrt reference level]
L	Characteristic length of water course [m]

The hydraulic radius R is defined as the wetted area of the cross section divided by the wetted perimeter.

$$R = \frac{A}{P} \tag{C.6}$$

A	Wetted area [m^2]
P	Wetted perimeter [m]

The wetted perimeter P is calculated by

$$P = B + 2D\sqrt{1 + \tan^2\alpha} \tag{C.7}$$

B	Bottom width [m]
D	Average depth [m]
α	angle of the side slope with vertical

D SOBEK input file formats

D.1 SOBEK Input file formats: the Model Database 4.00

D.1.1 Philosophy behind the Model Database

The combination of all SOBEK input files is called the Model Database.

The Model Database consists of a number of ASCII-files, which are logically separated into a number of layers; each layer covers a number of files. Each file contains the definition of one or more SOBEK-objects. In most cases the object has an attribute **id** which is the key to that particular object.

- ◇ For more information on the layers, see the chapter Subdivision into layers Database
- ◇ For more information on the design conventions, see the chapter Input conventions
- ◇ For exact description of certain input files, see the chapter of the module that has your interest:
  For the modules Channel Flow, River Flow and Sewer Flow: chapter 1DFLOW and Overland Flow(2D)
  For the modules Rainfall-Runoff: chapter RR (Rainfall Runoff)
  For the module Real Time Control: chapter RTC (Real Time control)

D.2 Structure of the Model Database: subdivision into layers

The following layers are distinguished in SOBEK:

1. Topography (TP)
2. Cross section (CR)
3. Structure (ST)
4. Friction (FR)
5. Condition (CN)
6. Initial conditions (IN)
7. Meteo (MT)
8. Dispersion (salt) (D)
9. Grid (GR)
10. Run time data (RT)
11. Transport (TR)
12. Substance (SB)
13. Measurements (ME)(cannot be edited by user)

In SOBEK, a distinction has been made between the objects as defined in the topography layer and the necessary data for these objects. Most of the objects in the topography layer are connected to the data objects via a so-called 'carrier id'.

There are two exceptions on this rule:

- ◇ the data from the run-time layer (layer 10) and
- ◇ the substances (layer 12)

Both of these data objects are not directly connected to an object from the topography layer. The data in every other layer is either specified for a specific location or as a 'global definition'.

Each layer consists of one or more files, each containing data stored in records. These records always start with a specific keyword in CAPITALS, and end with the same keyword in lower-case.

The following list gives an overview of the different layers and their files. It should be noted that not all files are necessary. For example, salt-related files are not necessary when the salt module is not used. The file containing the file names has a standard Windows-INI file structure. This means that the keywords are fixed and the file names behind the '=' can be chosen freely. Within a layer the keywords should be unique.

```
[SOBEK ASCII Layer definition]
Version=2.01

[General]
SettingsFile=settings.dat

[Topography layer]
ntw=network.ntw
net=network.tp
cpt=network.cp
dat=nodes.dat
xyc=network.d12

[Cross Section layer]
net=network.cr
def=profile.def
dat=profile.dat

[Structure layer]
net=network.st
def=struct.def
dat=struct.dat
con=control.def
trg=trigger.def
cmp=struct.cmp
vlv=valve.tab
dbs=struct.def

[Friction layer]
bed=friction.dat
glf=friction.dat
exr=friction.dat
wnd=friction.dat

[Grid layer]
net=network.gr

[Condition layer]
net=network.cn
flb=boundary.dat
fll=lateral.dat
sab=boundary.sal
sal=lateral.sal
fbd=boundlat.dat

[Initial Conditions layer]
igl=initial.dat
ifl=initial.dat
isa=initial.sal

[Runtime Data layer]
flt=..\work\runtime.dat
fln=..\work\runtime.dat
flm=..\work\runtime.dat
flh=..\work\runtime.dat
san=..\work\runtime.dat
```

```
sam=..\work\runtime.dat
sah=..\work\runtime.dat
inc=incclass.dat
net=network.me

[Meteo layer]
wnd=wind.#
bui=event.#
wlc=wnd_loc.dat

[Dispersion layer]
gld=globdisp.dat
lod=lokdisp.dat
fwt=freshwat.dat
mou=mouth.dat
brm=brmouth.dat
```

D.3 1DFLOW and Overland Flow(2D)

D.3.1 General principles of the model database

The SOBEK model database contains a number of ASCII-files. The database is logically separated into a number of layers; each layer consists of a number of files. Each file contains the definition of one or more SOBEK-objects. In most cases the object has an attribute id which is the key to that particular object.

The following layers are distinguished in SOBEK:

- ◇ Global definitions
- ◇ Topography (TP)
- ◇ Cross section (CR)
- ◇ Structure (ST)
- ◇ Friction (FR)
- ◇ Condition (CN)
- ◇ Initial conditions (IN)
- ◇ Meteo (MT)
- ◇ Dispersion (salt) (D)
- ◇ Grid (GR)
- ◇ Run time data (RT)
- ◇ Transport (TR)
- ◇ Substance (SB)
- ◇ Measurements (ME) (cannot be edited by user)

In SOBEK, a distinction has been made between the objects as defined in the topography layer and the necessary data for these objects. Most of the objects in the topography layer are connected to the data objects via a so-called 'carrier id'.

There are two exceptions on this rule:

- ◇ the data from the run-time layer (layer 10) and
- ◇ the substances (layer 12)

Both of these data objects are not directly connected to an object from the topography layer. The data in every other layer is either specified for a specific location or as a 'global definition'.

Each layer consists of one or more files, each containing data stored in records. These records always start with a specific keyword in CAPITALS, and end with the same keyword in lower-case.

The following list gives an overview of the different layers and their files. It should be noted that not all files are necessary. For example, salt-related files are not necessary when the salt module is not used. The file containing the file names has a standard Windows-INI file structure. This means that the keywords are fixed and the file names behind the '=' can be chosen freely. Within a layer the keywords should be unique.

```
[SOBEK ASCII Layer definition]
Version=2.01
[Global model] Rivers only
NrOfFiles=1
glb=defglb.1
[Topography layer]
NrOfFiles=7
ntw=network.ntw
net=network.tp
cpt=network.cp
nfl=network.fl
nsa=network.sa
nsm=network.sm
dat=nodes.dat
[Cross section layer]
NrOfFiles=3
net=network.cr
def=profile.def
dat=profile.dat
[Structure layer]
NrOfFiles=9
net=network.st
def=struct.def
dat=struct.dat
con=control.def
trg=trigger.def
sal=struct.sa
cmp=struct.cmp
cms=structcmp.sa
dbs=struct dbs
[Friction layer]
NrOfFiles=4
bed=bedfric.dat
wnd=windfric.dat
exr=exresist.dat
glf=globfric.dat
[Condition layer]
NrOfFiles=10
net=network.cn
flb=boundfl.dat
fll=laterfl.dat
wqb=boundwq.dat
wql=laterwq.dat
sab=boundsa.dat
sal=latersa.dat
mob=boundmo.dat
mon=nodesmo.dat
mol=latermo.dat
[Initial Conditions layer]
NrOfFiles=5
igl=incondgl.dat
ifl=incondfl.dat
iwq=incondwq.dat
isa=incondsa.dat
imo=incondmo.dat
```

```
[Meteo layer]
NrOfFiles=4
wnd=wind.mt
sun-sunshine.mt
wat=wattemp.mt
air=airtemp.mt
[Dispersion layer]
NrOfFiles=5
gld=globdisp.dat
lod=lokdisp.dat
fwt=freshwat.dat
mou=mouth.dat
brm=brmouth.dat
[Grid layer]
NrOfFiles=3
net=grid.tp
seg=wqsegm.tp
lim=segmlm.tp
[Runtime Data layer]
NrOfFiles=18
flt=flowtim.dat
fln=flownum.dat
flm=flowmap.dat
flh=flowhis.dat
san=saltnum.dat
sam=saltmap.dat
sah=salthis.dat
sen=sednum.dat
sem=sedmap.dat
seh=sedhis.dat
mon=mornum.dat
mom=mormap.dat
moh=morhis.dat
wqt=wqtim.dat
wqn=wqnum.dat
wqm=wqmap.dat
wqh=wqhis.dat
pwq=prepwq.dat
[Transport layer]
NrOfFiles=1
dat=transprt.dat
[Substance layer]
NrOfFiles=1
sub=substanc.def
```

D.3.2 Global definitions file

This file is only present in case of SOBEK-River. An example of this file is as follows:

ca 0
APAC fl sd mp apac
mv 0
LRVD tp cr st cn in fr rt ds tr mt sb gr lrvd
LMTS xi 0.00 xa 150000.00 yi 0.00 ya 150000.00 lmts
rh 0

Only the first 2 record are used by the Parser

record 1:

ca	area code: 0 = river 1 = estuary

record 2 (APAC):

APAC fl sd mp ..apac

This record contains the module identifiers. If an identifier is present the corresponding module will be activated. The identifiers are:

fl	flow module
sl	salt module
sd	sediment module
mp	morphology
m2	2D-morphology
pq	make water quality aggregation file
wq	water quality module
mz	Mozart coupling

record 3:

mv	validation flag (1=model has been validated) (only for UI)

record 4 (LRVD):

Contains layer information (only for UI)

record 5 (LMTS):

Contains the dimensions of the area of the model (only for UI)

record 6:

rh	refence level (only for UI)

D.4 2D Grid layer

Files in this layer

```
[2D grid layer]
NrOfFiles=1
net=..\work\network.d12
```

D.4.1 net-file (2D Grid layer)

This file describes the basic definition of 2D Grid [s] and the definitions of 1D Nodes (points) or 1D branches (lines) lying within the 2D Grid. This also holds for 2D Points (Boundary condition, Initial condition, Dam break structure) and 2D Line (combination of more than one Boundary node)

DOMN id 'Example Sobek-FLS, Use New' nm 'Example Sobek-FLS, Use New'
GFLS nc 61 nr 92 x0 101950 y0 463350 dx 100 dy 100 cp 0 fnm " gfls
ISCHILDOF ci 'parent'
CHILDBLOCK ltX 124100 ltY 437400 rbX 126200 rbY 435800 ltC 11 ltR 1 rbC 32 rbR 17 childblock
PARENTBLOCK ltC 1 ltR 11 rbC 22 rbR 27 parentblock
ischildof
PT12 id '92.31' nm '92.31' ci " lc 0 px 104994.2 py 454294.2 mc 31 mr 91 pt12

PT12 id ’2_1’ nm ’2_1’ ci ’2’ lc 300.9696 px 104994.9 py 454595.1 mc 31 mr 88 pt12
LI12 id ’1’ nm ’1’ ci ” lc 0 bx 105683.4 by 454213.5 ex 107991.5 ey 454213.5 bc 38 br 92 ec 61 er 92 li12
LI12 id ’l_bound’ nm ’bound’ ci ” lc 0 bx 257778.51 by 585767.76 ex 257774.22 ey 584358.75 bc 7 br 3 ec 7 er 17
TBLE cells 15
’7’ ’3’ <
..
’7’ ’17’ <
tble
li12
LM12 id ’l_meetraai’ nm ’meetraai’ ci ” lc 0 bx 257778.51 by 585767.76 ex 257774.22 ey 584358.75 bc 7 br 3 ec 7 er 17
TBLE cells 15
’7’ ’3’ <
..
’7’ ’17’ <
tble
TBLE crossings
PTCR cb ’10_7’ ci ’5’ lc 607.94 px 257776.34 py 585056.86 mc 7 mr 10 ptcr
tble
lm12
domn

where:

id	2D grid id
nm	2D grid name

GFLS...gfls = 2D grid definition

where:

nc	number of column (m) in X direction
nr	number of rows (n) in Y direction
x0	X coordinate
y0	Y coordinate
dx	distance between two grid points in X directions (m)
dy	distance between two grid points in Y directions (n)
cp 0	= 0 leftTop Corner (if grid definition file is *.dem, *.bil)
	= 3 leftBottom Centre (if grid definition file is ARC-INFO *.asc)
fnm	name of grid definition file (for example *.dem, *.asc etc.)

ISCHILDOF .. ischildof = definition of child-parent relation in case of multiple domain

where:

ci	identification of Parent domain

CHILDBLOCK .. childblock = definition of rectangular within Parent domain by the child domain.

where:

ltX	left top X coordinate (center of child 2D grid cell)
ltY	left top Y coordinate (center of child 2D grid cell)

rbX	right bottom X coordinate (center of child 2D grid cell)
rbY	right bottom Y coordinate (center of child 2D grid cell)
ltC	Left Column number of Parent domain cut by Child domain
ltR	Right Column number of Parent domain cut by Child domain
rbC	RightColumn number of Parent domain cut by Child domain
rbR	Right Column number of Parent domain cut by Child domain

PARENTBLOCK .. parentblock = definition of rectangular within Child domain by the Parent domain.

where:

ltC	Left Column number of Child domain cut by Parent domain
ltR	Right Column number of Child domain cut by Parent domain
rbC	RightColumn number of Child domain cut by Parent domain
rbR	Right Column number of Child domain cut by Parent domain

PT12 .. pt12 = definition of 1D and 2D Nodes (Points) lying within 2D grid

where:

id	point object id
nm	point object name
ci	branch id (carrier id)
lc	position relative to the branch origin (always 0)
px	X-coordinate
py	Y-coordinate
mc	column position within 2D grid (m) in X direction
mr	row position within 2D grid (n) in Y direction

LI12 .. li12 = definition of 2D Boundary Line or 1D2D Internal Boundary Line (lying within 2D Grid)

where:

id	line object id
nm	line object name
ci	”
lc	0
bx	begin X-coordinate
by	begin Y-coordinate
ex	end X-coordinate
ey	end Y-coordinate
bc	begin column position within 2D grid (m) in X direction
br	begin row position within 2D grid (n) in Y direction
ec	end column position within 2D grid (m) in X direction
er	end row position within 2D grid (n) in Y direction

LM12 .. lm12 = definition of 2D Measurement Line (meetraai)

where:

id	line object id
nm	line object name
ci	”
lc	0

bx	begin X-coordinate
by	begin Y-coordinate
ex	end X-coordinate
ey	end Y-coordinate
bc	begin column position within 2D grid (m) in X direction
br	begin row position within 2D grid (n) in Y direction
ec	end column position within 2D grid (m) in X direction
er	end row position within 2D grid (n) in Y direction

TBLE cells ncells tble = table of 2D grid cells and ncells is number of cells in table
TBLE crossings .. tble = table of 1D Channel/pipe crossing the 2D Measurement Line
PTCR .. ptcr = definition of 1D channel/pipe crossing the 2D Measurement Line

where:

cb	id of the branch
ci	if of the branch
lc	location on branch from beginning point
mc	column position within 2D grid (m) in X direction.
mr	row position within 2D grid (n) in Y direction.
px	X-coordinate of 2D grid point
py	Y-coordinate of 2D grid point

D.5 Condition layer

This layer contains data for boundary conditions and lateral discharges.

Files in this layer

```
[Condition layer]
NrOfFiles=10
net=network.cn
flb=boundfl.dat
fll=laterfl.dat
wqb=boundwq.dat
wql=laterwq.dat
sab=boundsa.dat
sal=latersa.dat
mob=boundmo.dat
mon=nodesmo.dat
mol=latermo.dat
```

D.5.1 flb-file (condition layer)

This file contains de boundary conditions for the FLOW modules.

For Channel Flow, Sewer Flow and River Flow:

FLBO id ’1’ st 0 ty 0 h_ wd 0 1.2 0 flbo (constant H)
or
FLBO id ’1’ st 1 ty 0 q_ wd 0 1.2 0 flbo (constant Q)
or (variable discharge)
FLBO id ’1’ st 0 ty 1 q_ dt 1 TBLE .. tble flbo (variable Q)
or
FLBO id ’1’ st 0 ty 1 h_ dt 1 TBLE .. tble flbo (variable H)
or (Fourier boundary)

FLBO id ’1’ st 1 ty 1 q_ df 2 TBLE .. tble h0 0.5 w0 0.1 flbo
or (tidal boundary)
FLBO id ’1’ st 1 ty 1 q_ di 2 TBLE .. tble h0 0.5 flbo
or tables library
FLBO id ’TH5’ st 0 ty 0 h_ wt 11 ’TH5’ flbo

where:

id	id
st	storage on boundary node (storage only possible in case of SOBEK Urban) 0 = no storage 1 = storage
ty	type boundary 0 = water level 1 = discharge
h_ wd 0	constant water level (only for ty 0)
h_ wt 1	variable water level as a function of time (TBLE ... tble)
h_ wt 11	use table from table library
h_ wf 2	only in SOBEK River: Fourier boundary for water level
h_ wi 3	only in SOBEK River: tidal boundary for water level
h_ wd 1	h_ wd 4
h_ wd 4	water level as a function of Q column 1 = Q column 2 = h
q_ dw 0	constant discharge
q_ dw 1	q_ dw 4
q_ dt 1	variable discharge as function of time (TBLE ... tble)
q_ df 2	SOBEK Estuarium model: Fourier boundary.

table with amplitude, phase

q_ di 3	SOBEK Estuarium model: tidal boundary

table with amplitude, frequency, phase.

q_ dw 4	Q = Q(H) according to Q-H table column 1 = h column 2 = Q
a0	only for Fourier and tidal boundary
w0	only for Fourier optie w0

Note: h_ wd 1, h_wd 4, q_ dw 1 and q_ dw 4 are handled by the Parser in the same manner.

For Overland Flow:

1D2DInternal Boundary Line:

D2LI id ’bound’ nm ’1D2D Internal Boundary Line’ ty 4 cp ’20’ d2li

2D Boundary Node:

D2PT id ’bound’ nm ’Boundary Node’ ty 0 h_wd 0 10.0 d2pt

2D Boundary Line:

D2LI id ’bound’ nm ’Boundary Line’ ty 1 q_ dw 0 0.5 d2li

where:

id	identification

ty	boundary type 0 = water level 1 = flow/Flux 2 = velocity 4 = 1D2D Boundary Line.
h_ wd 0	10.0 constant water level; only for ty 0
h_ wd 1	water level as a function of Q *** **not implemented yet** column 1 = discharge Q $[m^3/s]$ column 2 = water level h [m w.r.t. reference level] note: only for ty 0
h_ wt 1	variable water level as a function of time (TBLE ... tble) column 1= time in hours from start of simulation column 2 = water level h [m w.r.t. reference level] note: only for ty 0.
q_ dw 0	0.5 = constant discharge $[m^3/s]$; note: only for ty 1
q_ dw 1	Q=Q(H) according to Q-H table column 1 = water level h [m w.r.t. reference level] column 2 = discharge Q $[m^3/s]$ note: only for ty 1
q_ dt 1	= variable discharge as a function of time (TBLE ... tble) column 1 = time in hours from start of simulation column 2 = discharge Q $[m^3/s]$ note: only for ty 1
u_st 1	= velocity as a function of time (TBLE ... tble) column 1 = time in hours from start of simulation column 2 = velocity $[m/s]$ note: only for ty 2
cp	identification of the connected 1D2D Internal Boundary Node (node only available in Channel Flow)

```
BTBL id 'TH5' ty 0 h_ wt PDIN 0 0 pdin
TBLE
'2001/01/01;00:00:00' .33
'2001/01/01;01:00:00' .64
'2001/01/01;02:00:00' .83
'2001/01/01;03:00:00' .81

'2001/12/31;20:00:00' -.3
'2001/12/31;21:00:00' .27
'2001/12/31;22:00:00' .8
'2001/12/31;23:00:00' 1.1
tble btbl

BTBL id '107' sc 0 lt 0 PDIN 1 0 pdin
TBLE
'1990/01/01;00:00:00' 2102.4
'1990/01/02;00:00:00' 1716
'1990/01/03;00:00:00' 1954.3
'1990/01/04;00:00:00' 1734.9

'1990/09/26;00:00:00' 112.8
'1990/09/27;00:00:00' 130.24
'1990/09/28;00:00:00' 216.23
```

’1990/09/29;00:00:00’ 152.37
’1990/09/30;00:00:00’ 99.31
tble btbl

D.5.2 fll-file (condition layer)

This file contains the lateral discharges on nodes/branches for the FLOW module.

Lateral discharge on branch:

FLBR id ’3’ sc 0 lt 0 dc lt 0 ir 0 ms ’station 1’ ii 0.005 ar 600000 flbr
or
FLBR id ’Intensity from Meteostation’ sc 0 lt 0 dc lt 7 ir 0.003 ms ’meteostation’ ii 0.002 ar 1000 flbr
or
FLBR id ’Constant intensity’ sc 0 lt 0 dc lt 6 ir 0.003 ms ’meteostation’ ii 0.002 ar 1000 flbr
or
FLBR id ’1’ ci ’1’ sc 0 lt 0 dc lt 1 0 0TBLE .. tble flbr
or
FLBR id ’11’ sc 0 lt 0 dc lt 0 1 0flbr
or
FLBR id ’107’ sc 0 lt 0 dc lt 11 ’107’ flbr

lateral structures:

FLBR id ’1’ sc 0 lt 0 dc lt 4 0 0 sd ’S1’ wl ow 0 -1 0 flbr
or
FLBR id ’1’ sc 0 lt 0 dc lt 4 0 0 sd ’S1’ wl ow 1 TBLE .. tble flbr

where:

id	id
sc	section (for 2D morphology !) 0 = left (=main section; default) 1 = right
lt	length of discharge 0 = point discharge (m3/s) $>$0 = discharge over a certain length (m2/s) (**not in SOBEK-Urban**/**Rural**) -1 = discharge over the entire length of the branch (m3/s) (**new in SOBEK Urban**/**Rural**)
dc lt	table:dc lt 0= constant value,
dc lt 1	’real’ table (first column=time, second column=discharge)
dc lw 2	as a function of the waterlevel (**not in SOBEK Urban**/**Rural**) column 1 = h column 2 = Q
dc lt 3	linked to another lateral discharge (’2nd station’) (**not in SOBEK Urban**/**Rural**)
dc lt 4	indicates lateral structure on a branch
dc lt 5	retention
dc lt 6	rational method with constant intensity
dc lt 7	with intensity from the rainfall station
dc lt 11	from a table library
ir	constant intensity (mm/s)
ms	meteo-station

ii	seepage/infiltration intensity (mm/s)
ar	runoff area [m^2]
sd	id of structure definition (see STRUCT.DEF; only in case dc lt 4)
ci	id of second station (only in case of dc lt 3; **not in SOBEK Urban**/**Rural**)
wl ow	table with water levels outside the lateral structure
wl ow 0	constant as a table,
wl ow 1	'real' table

Sobek-lateral discharges on a node (Only for 2D-morphology):

```
FLND id '1' bi '1' '2' aw dd
TBLE ..
tble
dd
TBLE ..
tble
flnd
```

where:

bi	branch id's
aw dd	alpha w0 as a function of Q column 1 = table, discharge column 2 = alpha w
dd	alpha w1 as function of Q column 1 =table, discharge column 2 = alpha w

Only for SOBEK Urban/Rural

lateral discharge on node:

FLNO id '1' dc lt 0 -1.0 0 flno
or **lateral structure on node**:
FLNO id '1' lt 0 dc lt 4 0 0 sd 'S1' wl ow 1 TBLE .. tble flno

where:

id	id
dc lt	table with lateral discharge dc lt 0= constant as table, dc lt 1= 'real' table column 1 = time column 2 = discharge dc lt 4= indicates the structure being a lateral structure. dc lt 5 = retention
sd	id of structure description (see STRUCT.DEF)
wl ow	table with water levels outside the lateral structure wl ow 0= constant as table, wl ow 1= 'real' table

diffuse lateral discharge on the branch

FLDI id '15263' ci '15263' sc 0 lt -1 dc lt 0 2 0 fldi

Where:

id	id
ci	id of second station
sc 0 lt -1	diffuse
dc lt 0	constant value [$m^3/s/m$]
dc lt 1	= table
dc lt	table with lateral discharge dc lt 0= constant as table, dc lt 1= 'real' table

pipes with infiltration (trenches)

TRCH id '38' di '3' dl 1.05 dh 2.55 rt 22 pt 0.22 oi 2 il 0 pm 1 rg 2222 gl tt 0 1.45 trch
or
TRCH id '36' di '19' dl 0.1 dh 2 rt 10 pt 0.4 oi 2 il 0 pm 1 rg 1111 gl tt 1 PDIN 0 0 " pdin
TBLE '2011/11/29;00:00:00' 1.11 $<$
'2011/11/29;01:00:00' 1.22 $<$
'2011/11/29;02:00:00' 1.33 $<$
'2011/11/29;03:00:00' 1.44 $<$
tble trch

Where:

id	(branch carrier)ID of Pipe with Infiltration
di	ID of cross-section definition for trench (Id of cross-section of pipe is stored as normal.) Note: There is no difference in cross section definition of channel or trench or pipe.
dl	Distance invert level, the distance between the invert level of the pipe and the invert level of the trench.
dh	Trench height, if not (properly) specified the surface level is used as top of trench.
rt	Resistance factor pipe-trench
pt	Porosity trench
oi	option for initial water level trench 0 = Value specified by keyword il (default) 1 = Same as water level pipe 2 = Same as groundwater level
il	Initial water level trench (for oi = 0)
pm	Permeability of ground/soil 0 = Completely impermeable, no interaction between groundwater and trench 1 = Permeable
rg	Resistance factor trench-ground, only relevant when ground is permeable
gl tt	groundwater level 0 = Constant groundwater level, only relevant when ground is permeable 1 = Table with groundwater level varying in time column 1 date/time stamp column 2 groundwater level

D.5.3 mob-file (condition layer)

This file contains the boundary conditions on nodes for the sediment/ morphology modules.

MPBO id '1' cy 1 bl bt 1
TBLE ..
tble
mpbo

where:

id	id of the boundary condition
cy	condition type 0 = load (as a constant, f(time), of f(Q)) 1 = bed level (as a constant, of f(time)
lo	load type
lo sd 0	load as constant
lo sd 2	load as function of the discharge Q column 1 = Q column 2 = load column 3 = load right (only when using 2D morphology option)
lo st	load as function of time column 1 = time column 2 = load column 3 = load right
bl bt	bed level table bl bt 0 = constant bl bt 1 = table column 1 = time column 2 = bed level column 3 = bed level right

Note: The field **se** is generated internally by the user interface. `se 0` = no 2D morphology, `se 1` = 2D morphology on next branch. This field is not required.

D.5.4 mol-file (condition layer)

This file contains the descriptions of the lateral discharges of sediment on branches.

MPBR id '2' ty 0 le 0 lt dl 0 10.0 mpbr

or

MPBR id '2' ty 1 le 0 di '1' ct co 0 50.0 mpbr

where:

id	id
ty	type condition: 1 = concentration 0 = dry load
sc	section (only for 2D morphology option) 0 = left (default) 1 = right
le	length

lt dl	dry load table (only for ty 1) lt dl 0 = constant lt dl 1 = table function column 1 = time column 2 = load
di	id of accompanying lateral discharge (only for ty 1: concentration)
ct co	sediment load table (only for ty 0) ct co 0 = constant, ct co 1 = table column 1 = time column 2 =sediment load

D.5.5 mon-file (condition layer)

This file contains the description for morphology on nodes. For every node the distribution of sediment over the different branches needs to be given. When the module Morphology is used, there is a maximum of three branches for every node. This means that there are three combinations of two branches related to the node.

```
MPND id ’0’ ty 0 1 0 aa 9.9999e+009 2 9.9999e+009
ba 9.9999e+009 0 9.9999e+009 rt d0 ’Distribution Ratio’
TBLE
tble
rt d1
TBLE
tble
rt d2 bi ’0’ ’1’ ’2’ se 0 0 0 as m0 as m1 ex 9.9999e+009 9.9999e+009 ds 1 ta 0 mpnd
```

or

```
MPND id ’1’ ty 0 0 0 aa 9.9999e+009 9.9999e+009 9.9999e+009
ba 9.9999e+009 9.9999e+009 9.9999e+009 rt d0 rt d1 rt d2 bi ’2’ ’3’ ’4’
se 0 0 0 as m0 as m1 ex 9.9999e+009 9.9999e+009 ds 0 ta 0 mpnd
```

Where:

id	id
ty	type distribution per combination of two branches (max. 3 combinations) 0 =proportional (distribution according to the ratio of the discharges) 1 =linear (according to a linear relation) 2 =ratio (distribution according to interpolation relation of ratios of Q and Sediment load)
aa	alfa; only for type 1: linear distribution
ba	beta; only for type 1: linear distribution
rt d0	distribution ratio table first couple
rt d1	distribution ratio table second couple
rt d2	distribution ratio table third couple
bi	branch id’s of connection branches as m0=alpha sedredge values for branch-couple 1; only if the 2D-morphology option is selected. as m1=alpha sedredge values for branch-couple 2; only if the 2D-morphology option is selected.
ex	exponents of sediment-distribution according to Wang:
1e	exponent k for discharges

2e	exponent m for widths
ds	distribution used for each branch-couple or for all branches attached to the node: 0 = for all branches attached to the node 1 = for each branch-couple
ta	type of distribution in case ds = 0 (all branches of node) 0 = proportional 3 = exponential function according to Wang

Note: The field **se** is generated internally by the user interface. se 0 = no 2D morphology, se 1 = 2D morphology on next branch. This field is not required.

D.5.6 net-file (condition layer)

This file contains information about the locations in the network for which conditions are defined.

For Channel Flow, Sewer Flow and River Flow:

Locations flow conditions:

FLBO id '1' nm 'Bound_cond1' ci '4' flbo **Note: Fixed order**
FLND id '2' nm 'Flowatnode1' ci '1' flnd **Note: Fixed order**
FLBR id '1' nm 'Lateralflow op tak1' ci '1' lc 100. Flbr **Note: Fixed order**
FLDI id '1_1' ci '15263' fldi **Note: Diffuse lateral discharge whole branch**

The following 3 types are only defined in SOBEK Urban/Rural:

FLNO id '1' nm 'Lateralflow op knoop1' ci '4' flno
FLNX id '1' nm 'External Lateralflow knoop1' ci '3' flnx
FLBX id '1' nm 'External Lateralflow op tak1' ci '5' lc 50. flbx

Note: The 'Flow at node' (FLND) can only be used in the case when 2D-morfology branches connect to the node.

Locations conditions water quality:

WQBO id '1' nm ' Wqbound1' ci '4' wqbo
WQBR id '2' nm 'wq_at_branch' ci '5' lc 300. wqbr

Locations conditions salt:

STBO id '0' nm 'Saltbound1' ci '4' stbo
STBR id '1' nm 'Salt lat disch' ci '3' lc 50. Stbr
STBR id '1' nm 'Salt lat conc ci '-1' Stbr

Locations conditions sediment/morphology:

MPBO id '1' nm 'Sed_bound' ci '0' mpbo
MPND id '3' nm 'Sed_disch_node' ci '2'mpnd
MPBR id '5' nm 'Sed_disch' ci '4' lc 50. mpbr

where:

id	id of the condition (boundary or lateral discharge)
nm	name of the condition
ci	carrier id (node id in case of a boundary or node condition, branch id in case of a branch condition)
lc	location (only in combination with a branch-id)

For Overland Flow:

1D2D Internal Boundary Node (node is only available in Channel Flow):

FL2B id 'pnt1D2Dboundary' ci 'pnt1D2Dboundary' px 101991.8 py 458610.7 fl2b

2D Boundary Node:

D2PT id 'bound' nm " ci '1' px 101991.8 py 458610.7 d2pt

2D Boundary Line and 1D2D Internal Boundary Line:

D2LI id 'l_bound' nm " ci 'l_bound' bx 105683.4 by 454213.5 ex 107991.5 ey 454213.5 d2li

where:

id	boundary id
nm	boundary name
ci	2D grid id
px	X coordinate within 2D Grid
py	Y coordinate within 2D Grid
bx	begin X coordinate within 2D grid for Boundary Line
by	begin Y coordinate within 2D grid for Boundary Line
ex	end X coordinate within 2D grid for Boundary Line
ey	end Y coordinate within 2D grid for Boundary Line

Initial Condition Node:

D2IN id '1' ci '1' px 101991.8 py 458610.7 d2in

where:

id	Initial Condition Node id
ci	2D grid id
px	X coordinate within 2D Grid
py	Y coordinate within 2D Grid

D.5.7 sab-file (condition layer)

This file contains the boundary conditions for the saltmodule (SA)

STBO id '1' ty 1 co co 1
TBLE ..
tble
tl 0 tu 0 stbo

where:

id	id
ty	type of boundary
1	zero flux
0	concentration
co co	concentrations (only for type 0)
co co 0	as constant
co co 1	as table
column 1	time
column 2	concentrations
tl	Thatcher-Harleman time lag (only for SOBEK River, concentration boundary)
tu	time lag used (only for SOBEK River model, concentration boundary)

D.5.8 sal-file (condition layer)

This file contains the descriptions of the lateral discharges for the saltmodule (SA).

STBR id 0 ty 0 le 100 lo lt 0 10 0 co ct 0
TBLE ..
tble
stbr
STBR id 0 ty 1 le 100 co ct 1
TBLE ..
tble
stbr

where:

id	id of the lateral discharge of salt
ty	type condition 0 = dry substance 1 = concentration
le	length
lo lt	load table (for ty 0)
lo lt 0	= constant
lo lt 1	= table as function of time
di	id of accompanying lateral discharge (when ty 1: concentration)
co ct	concentration table (only for ty 1)
co ct 0	= constant
itemco ct 1	= table as function of the time

D.5.9 wqb-file (condition layer)

This file contains de boundary conditions for 1DWAQ-module

WQBO id '1' ct ct 0 as SLST .. slst wqbo

where:

id	id
ct ct	concentration type
ct ct 0	constant boundary concentration
ct ct 1	boundary concentration as table function of time:
column 1	time
column 2	values for all substances as function of time
as	between keywords SLST and slst a list:

- ◇ the active substances
- ◇ type (0=constant, 1=variable)
- ◇ the value

Note: : If a time-dependant boundary condition is used (**ct ct 1**) for one substance, it is still possible to enter constant values for other substances using the keyword **as**.

Only one timetable can be entered, so when more substances have a time dependant boundary concentration, they should be entered in the same table for every timestep.

Note: The order of the substances after the keyword **as SLST slst** is identical as the order in the table with concentrations as a function of time in the field **ct ct 1**

D.5.10 wql-file (condition layer)

This file contains the 1DWAQ conditions of the lateral discharges

SOBEK River FLBR

FLBR id '1' hs flbr

where:

id	id lateral discharge
hs	has substances
	0 no
	1 yes
as	active substances

Between the keywords SLST slst a list of substances, the type of discharge (0=constant, 1=variable) and the value. A variable concentration is only possible when the flow itself varies.

Note: : **Specification of water quality conditions; Keywords FLBR en WQBR**

In SOBEK River two keywords are used to specify these conditions, FLBR and WQBR. This depends on whether the condition is associated with lateral flow or with an input of a dry substance. It seems logical, however, to switch to the use of only one keyword, WQBR.

D.6 Cross Section layer

Files in this layer

```
[Cross section layer]
NrOfFiles=3
net=network.cr
def=profile.def
dat=profile.dat
```

D.6.1 dat-file (cross section layer)

This file contains a reference to the cross-section definition and information about the relative height of the cross-section.

CRSN id 'crossloc1' di 'crossdef1' rl -0.5 ll -0.4 rs 0.9 ls 0.9 crsn

Note: Fixed order

Items in one record; ll, rs and ls optional, separated by 1 or more blanks where:

id	id of cross-section location
di	id of cross-section definition
rl	reference level 1 (SOBEK River/Rural: level at the cross-section; SOBEK Urban/Rural: level at the end of the pipe)
ll	reference level 2 (SOBEK Urban/Rural: level at the beginning of the pipe)
fg	specification of flap gate (Pipe Only: 0 = None, 1 = Only Positive Flow, 2 = Only Negative Flow)
us	upstream slope (**not in SOBEK Urban**/**Rural**)
ds	Downstream slope (**not in SOBEK Urban**/**Rural**)
rs	Surface level right (same units as rl)
ls	Surface level left (same units as rl)

In SOBEK-River a bed-level (code **bl**) was already given for the cross-section descriptions (except for the tables, here it is given in the table itself). In addition, a local reference level **rl** at the cross-section needs to be given.

In SOBEK Urban/Rural however, it is not possible to give a bed level for every cross-section description. In this case, the bed level is given when placing the cross-section on a branch (pipe) by means of the keywords **rl** and **ll**. These keywords give the bed levels of the bottom of the pipe at the end and the beginning of the pipe, so that the slope is determined by the length of the pipe and the difference in height **rl-ll**.

In SOBEK-River, **rl** stands for the reference level at the location of the cross-section, while the slopes on the left and right side can differ. In the case of more cross-sections on one branch, the bed level slope between the cross-sections is assumed to be constant (based on the bed levels at the cross-sections corrected for local reference levels). For interpolation of the cross-sections both the up- and the downstream slopes are used.

In SOBEK-Rural for open water, only **rl** is used and interpreted the same way as in SOBEK-River.

D.6.2 def-file (cross section layer)

This file contains the descriptions of the cross sections. SOBEK River distinguishes 3 sections while SOBEK Urban/Rural only distinguishes 1 section.

The following types of cross sections are considered:

0 tabulated
1 trapezium
2 open circle
3 sedredge (2D morfology)
4 closed circle
5 5.
6 egg shaped (width)
7 egg shaped 2 (radius) **not implemented**
8 closed rectangular **not implemented**
9 9.
10 yz table
11 asymmetrical trapeziodal

D.6.3 Tabulated cross section

CRDS id ’Crdef1’ nm ’Tabel1’ ty 0 wm 86.23 w1 0 w2 0 sw 0 lt lw
TBLE
-2.5516.8016.80 <
-1.0030.1330.13 <
-0.6530.1330.13 <
0.0086.2386.23 <
tble
dk 0 dc 99999. db 99999. df 99999. dt 99999.
gl 0.5 gu 0
crds

where:

id	cross section definition id
nm	cross section definition name
ty	type cross section (0=table)
wm	width main channel
w1	width floodplain 1 (**used in River profile only, else value = 0**)
w2	width floodplain 2 (**used in River profile only, else value = 0**)
sw	sediment transport width (**not in SOBEK Urban/Rural**) Default 0. Only important for module sediment/morphology
lt lw	table for table profile between keywords TBLE and tble; the table contains height, total width en flowing width.
dk	summer dike (1 = active, 0 = not active) (in River profile only)
dc	dike crest level in River profile only()
db	floodplain base level behind dike (in River profile only)
df	flow area behind dike (in River profile only)
dt	total area behind dike (in River profile only)
gl	ground layer depth (meter relative to bed level). In case of River profile, always 0
gu	ground layer to be used within hydraulics calculation (1) or not (0). In case of River profile, always 0

In the Data Editor there are 5 other types of cross-sections available, however they are treated, stored and processed as tabulated cross-sections. These types are:

1 Rectangular
2 Elliptical
3 Arch
4 Cunette
5 Steel Cunette

To be able to represent these cross-sections in the Data Editor properly, the records for this cross-sections contain some extra keywords, which do not have any influence on the results and are used by the Data Editor only.

These keywords for the different types are:

Rectangular:

rw	width
rh	height

Elliptical:

ew	width
eh	height

Arch:

aw	width
ah	height
aa	height

Cunette:

cw	width

Steel Cunette:

sh	height
sr	radius r
sr1	radius r1
sr2	radius r2
sr3	radius r3
sa	angle a
sa1	angle a1

D.6.4 Trapezium cross section

CRDS id 'Crdef2' nm 'Trapezium' ty 1 wm 86.0 w1 0 w2 0 sw 0 bw 10 bs 0.5 aw 100 gl 0.5 gu 0 crds

Where:

id	cross section definition id
nm	cross section definition name
ty	type cross section (1=trapezium)
bl	bed level
wm	width main channel
w1	width floodplain 1 (in SOBEK Urban/Rural always 0)
w2	width floodplain 2 (in SOBEK Urban/Rural always 0)
bw	bottom width
bs	bank slope (horizontal/vertical)
aw	maximum flow width
gl	ground layer depth (meter relative to bed level)
gu	ground layer to be used within hydraulics calculation (1) or not (0).

Only important for module sediment/morphology (River)

sw	sediment transport width (not in SOBEK Urban/Rural)

D.6.5 Open circle cross section

CRDS id 'Crdef3' nm 'Opencirkel' ty 2 rd 5 gl 0.5 gu 0 crds

where:

id	cross section definition id
nm	cross section definition name
ty	type of cross section (2=open circle)
bl	bed level
rd	radius
gl	ground layer depth (meter relative to bed level)
gu	ground layer to be used within hydraulics calculation (1) or not (0).

Note: This profile cannot be used for sediment calculations (River). A table should be used instead.

D.6.6 Sedredge cross section

Only for 2D morphology calculations (SOBEK-River). 'left' means main channel, and 'right' is the floodplain.

CRDS id 'Crdef4' nm 'Sedredge' ty 3 ll 3 rl 2.5 lw 10 rw 5 crds

Where:

id	cross section definition id
nm	cross section definition name
ty	type cross section (3=2D morphology sedredge profile)
ll	left bed level
rl	right bed level
lw	left bed width

rw	right bed width

D.6.7 Closed circle cross section: (only SOBEK Urban/Rural)

CRDS id 'Crossdef5' nm 'Geslotencirkel' ty 4 rd 4 gl 0.5 gu 0 crds

Where:

id	cross section definition id
nm	cross section definition name
ty	type cross section (4=closed circle)
bl	bed level
rd	radius
gl	ground layer depth (meter relative to bed level)
gu	ground layer to be used within hydraulics calculation (1) or not (0).

D.6.8 Egg shaped cross section: (only SOBEK Urban/Rural)

CRDS id 'Crossdef6' nm 'Ei' ty 6 bo 0.5 gl 0.5 gu 0 crds

Where:

id	cross section definition id
nm	cross section definition name
ty	type cross section (6=egg shaped)
bl	bed level
bo	bottom width profile;

For this profile, the height is considered to be 1.5 * the width.

gl	ground layer depth (meter relative to bed level)
gu	ground layer to be used within hydraulics calculation (1) or not (0).

D.6.9 y-z table cross section: (only SOBEK Urban/Rural)

```
CRDS id 'Crdef' nm 'y-z table1' ty 10 lu 0 st 0 lt sw 0 12.0 lt yz
TBLE
0.0 12.0 <
1.0 10.0 <
2.0 9.0 <
3.0 9.5 <
4.0 10.5 <
5.0 11.0 <
tble
gl 0
gu 0
crds
```

or

```
CRDS id 'Crdef' nm 'y-z table1' ty 10 lu 0 st 0 lt sw
TBLE
12.0 0 <
20.0 1 <
tble
```

lt yz
TBLE
0.0 12.0 <
1.0 10.0 <
2.0 9.0 <
3.0 9.5 <
4.0 10.5 <
5.0 11.0 <
tble
gl 0
gu 0
crds

Where:

id	cross section (profile) definition identification
nm	cross section (profile) definition name
ty	type of cross section (10 = yz table)
lu	calculation of conveyance 0 = Open vertically segmented (default) 1 = Open lumped 2 = Closed lumped
st	storage type 0 = reservoir 1 = loss of water above the highest point in the cross-sectional profile (deprecated since SOBEK 2.12.002a)
lt sw 0	storage width on surface in m
lt sw	storage width on surface level as tabulated function of level above surface level in m (starting from zero).
lt yz	table for y-z values. Y horizontal distance increasing from the left to right, Z vertical distance increasing from bottom to top in m. In other words, use a coordinate system to define the Y-Z profile.
gl	ground layer depth (meter relative to bed level). Unused for y-z profiles and set to 0.
gu	ground layer to be used within hydraulics calculation (1) or not (0). Unused for y-z profiles and set to 0.

D.6.10 Asymmetrical trapeziodal cross section: (only SOBEK Urban/Rural)

CRDS id 'Crdef' nm 'Asymmetrical Trapezium1' ty 11 st 0 lt sw 0 12.0 lt yz
TBLE
0 .0 12.0 <
1.0 10.0 <
2.0 9.0 <
3.0 9.5 <
4.0 10.5 <
5.0 11.0 <
tble
gl 0
gu 0
crds

or

```
CRDS id 'Crdef' nm 'Asymmetrical Trapezium1' ty 11 st 0 lt sw
TBLE
12.0 0 <
20.0 1 <
tble
lt yz
TBLE
0 .0 12.0 <
1.0 10.0 <
2.0 9.0 <
3.0 9.5 <
4.0 10.5 <
5.0 11.0 <
tble
gl 0
gu 0
crds
```

Where:

id	cross section (profile) definition identification
nm	cross section (profile) definition name
ty	type of cross section (11 = asymmetrical trapezium)
st	storage type 0 = reservoir 1 = loss of water above the highest point in the cross-sectional profile (deprecated since SOBEK 2.12.002a)
lt sw 0	storage width on surface in m
lt sw	storage width on surface level as tabulated function of level above surface level in m (starting from zero).
lt yz	table for y-z values. Y horizontal distance increasing from the left to right , Z vertical distance increasing from bottom to top in m. In other words, use a coordinate system to define the Y-Z profile.
gl	ground layer depth (meter relative to bed level)
gu	ground layer to be used within hydraulics calculation (1) or not (0).

D.6.11 net-file (cross section layer)

This file contains the definitions of the cross-section locations.

If the file version is 1.1 or higher (CR_1.1 in the first line), also records indicating the nodes with storage are written to the file.

The cross-sections are defined using the CRSN record.

CRSN id 'c1' nm 'crossdef1' ci 10 lc 250.6 crsn **Note: Fixed order**

Where:

id	id of cross-section location
nm	name of cross-section definition

ci	carrier id (branch id)
lc	distance from the beginning of the branch

The storage nodes is defined using STON record.

STON id '2' ci '1' lc 0 st 'Normal' ston

Where:

id	node id
ci	dummy
lc	dummy
st	dummy

Only the node id is used, but all record keywords must be present and have a (dummy) value.

If no STON record is written for the node, then the storage as defined in the file nodes.dat should be skipped.

In the old definition of the file (version 1.0) all storage data on nodes is always used, although the actual node type in the User Interface does not suggest this functionality.

D.7 Dispersion layer

This layer contains the information for modelling the dispersion, which is used by the saltmodule.

Files in this layer

```
[Dispersion layer]
NrOfFiles=5
gld=globdisp.dat
lod=lokdisp.dat
fwt=freshwat.dat
mou=mouth.dat
brm=brmouth.dat
```

D.7.1 brm-file (dispersion layer)

This file contains the branch-mouth relations in SOBEK River, necessary for the Thatcher/Harleman formula and the empirical formula.

BRMT id '2' nm 'noname' ci '10' mi DLST '3' '4' '6' dlst brmt

Where:

id	branch-mouth id
nm	name of the branch-mouth relation
ci	carrier id (branch id)
mi	necessary estuary-mouth id's between the keywords DLST dlst (node id's)

D.7.2 gld-file (dispersion layer)

This file contains some global definitions concerning the dispersion: the type of dispersion formulation, and some other global parameters.

GLDS op 0 ty 0 f1 1 glds

or

GLDS op 1 ty 0 f1 1 f2 2 glds

or

GLDS op 1 ty 1 ds tt
TBLE ..
tble glds

or

GLDS op 2 ty 0 f1 1 f3 3 f4 4 glds

or

GLDS op 3 ty 0 f1 1 f3 3 f4 4 glds

or

GLDS op 2 ty 2 ds tt DSPN id '2' nm 'modelwide' ci '-1' ty 0 f1 1 f3 3 f4 4 dspn glds

Where:

op	option type formulation 0 = option 1 (dispersion coefficient as a function of location or time) 1 = option 2 (dispersion coefficient as a linear function of the concentration-gradient: $f1(x,t) + f2(x,t)dc/dx$) 2 = Thatcher-Harleman formula 3 = empirical formula, based on Thatcher-Harleman
ty	type: 0 = constant 1 = f(time) 2 = f(place) (only for 'model wide' definition)
f1	f1(x,t) in [m^2/s]
f2	f2(x,t) in [$m^6/(kg\,s)$]
f3	f3(x,t) [-]
f4	f4(x,t) [-]
dt tt	dispersion table as function of the time column 1 = time

the other columns are a selection of f1, f2, f3, f4, dependant of the type of dispersion chosen.

DSPN=keyword for the description of the model wide dispersion formulation (carrier id '-1'). Only relevant when dispersion is given as function of the location (type 2). The description of the DSPN record is given in the **lod**-file. In this case, constants will always be used for the model wide description.

Note: Functions f1 through f4 als defined as $f(x,t)$; this means that they can either depend of the place ($f(x)$) OR the time ($f(t)$), but NOT both.

Note: When defining global dispersion, f1 through f4 can be either a constant or a function of time. The **lod**-file contains descriptions of Dispersion as a function of place.

D.7.3 fwt-file (dispersion layer)

This file contains de fresh water discharge data necessary for using an empirical formulation (ty 3 in the **gld-file**) in SOBEK River.

FWTR id ’0’ nm ’freshwat1’ ci ’1’ lc 50 sg 0 fwtr

Where:

id	id of the sweet water discharge/ extraction
nm	name
vci	carrier id (branch id)
lc	location
sg	sign 0 = positive (discharge) 1 = negative (extraction)

D.7.4 lod-file (dispersion layer)

This file contains the location-dependant dispersion information. These are linked to a branch-id. This file is only necessary dispersion is entered as a function of place, in the **gld**-file.

DSPN id ’1’ nm ’Dispersietak1’ ci ’1’ ty 0 f1 11 f3 12 f4 13 dspn

or

DSPN id ’2’ nm ’Dispersietak2’ ci ’3’ ty 2 dl lt
TBLE ..
tble
dspn

Where:

id	id dispersion
nm	name
ci	carrier id (=branch id)
ty	type of dispersion function: 0 = constant 1 = f(time) 2 = f(place)
f1 t/m f4	see **gld-file**
l lt	dispersion table as function of the location on the branch column 1 = location the other columns any one of f1, f2, f3, f4 depending on the chosen dispersion formulation in the **gld-file**.

D.7.5 mou-file (dispersion layer)

This file contains the so-called mouth parameters; they are only necessary for the Thatcher/Harleman formulation and the empirical formulation in case of an estuary model.

For the Thatcher-Harleman dispersion formulation:

MDSP id '1' nm 'noname' ci '4' dm 1 cm 2 um 3 rm 1035 qm 5 mdsp

The empirical formula needs a couple of additional parameters:

MDSP id '1' nm 'noname' ci '4' dm 1 cm 2 um 3 rm 1035 qm 5 p0 0 p1 1 u0 2 u1 3 mdsp

Where:

id	id mouth id
nm	name
ci	carrier id of the node
dm	reference water depth $[m]$
cm	reference salt concentration $[kg/m^3]$
um	characteristic flood velocity $[m/s]$
rm	rho seawater (default 1035 kg/m^3)
qm	initial fresh water discharge $[m^3/s]$
p0	p0 $[m^3]$
p1	p1 $[m^2]$
u0	u0 $[m/s]$
u1	u1 [1/s]

D.8 Friction layer

Files in this layer

```
[Friction layer]
NrOfFiles=4
bed=bedfric.dat
wnd=windfric.dat
exr=exresist.dat
glf=globfric.dat
```

Friction information is connected to **branches**, and not to cross-sections. This means that no information is necessary about the place of the friction on the branch, as it is for the cross-sections. The friction itself can depend on for example the location on the branch (!), the discharge, or the water level. Friction can be specified for main- and subsections.

D.8.1 bed-file (friction layer)

This file contains the bed friction data per branch.

For Channel Flow, Sewer Flow and River Flow:

BDFR id '1' ci '1' mf 7 mt cp 0 20 0 mr cp 0 20 0 s1 6 s2 6 sf 7 st cp 0 20 0 sr cp 0 20 0 bdfr

Fixed number of items in one record, separated by 1 or more blanks

Where:

id	id of bed friction definition
nm	name of the bed friction definition (**not in SOBEK Urban**/**Rural**)
ci	carrier id = id of the branch
mf	main friction type (main = main channel) 0 = Chézy 1 = Manning 2 = Strickler Kn 3 = Strickler Ks 4 = White-Colebrook 5 = - 6 = - 7 = De Bos and Bijkerk
mt	friction in positive flow direction
mt fq	friction=f(Q)
mt fh	C=f(h)
mt cp	friction as a constant or as a function of the location on the branch For fq, fh, and cp: a constant (entered as a table) or a real table: 0 = constant 1 = variable The options fq and fh may have more dimensional tables: column 1 =Q or h value, column n = friction value on different locations along the branch for every Q or h Thus, the options fq and fh are a function of the location on the branch and Q (of h). The option cp (friction as function of the location) has a two-dimensional table: column 1 = location along the branch column 2 = friction-coefficient
mr	friction in negative direction: mr fq =friction=f(Q) mr fh =friction=f(h) mr cp =friction as a constant or as a function of the location on the branch) Option fq, fh, and cp may contain a constant given in a table or a table. 0 = constant 1 = variable
s1	friction for floodplain 1 (**not in SOBEK Urban**/**Rural**) can be either 'equal to main section', or Chézy/. . . /Nikuradse. (0=Chézy,. . . ,6=Equal to main section) **Note:** Engelund cannot be used for the floodplains.
s2	friction for floodplain 2 (**not in SOBEK Urban**/**Rural**)
sf	ground layer friction type (0 – 7) (for further details see description for mf)
st	friction in positive direction
st cp	(for all friction types, for further details see description for mt)
sr	ground layer friction in negative direction sr cp for all friction types (for further details see description for mt) c1 cp,fq,fh=floodplain 1 friction coefficients (friction can be defined as a function of Q, of h, of the location or as a constant. r1 cp,fq,fh=floodplain 1 reversed flow friction coefficients as a function of Q, of h, of the location or as a constant. c2 cp,fq,fh=floodplain 2 r2 cp,fq,fh=floodplain 2 reversed flow d9 f9=D90

Same friction for both directions (for SOBEK-River only)

This feature is optional. It is defined by the flags em, er, e1, e2, e3, e4. If these flags are present (in this order) they should follow the carrier id (ci ‘ ‘) immediately. These flags are only used by the user interface. For the parser bed friction is fully defined by the keys mf etc.

BDFR id ’33’ nm ’(null)’ ci ’31’ em 0 er 0 e1 0 e2 0 e3 0 e4 0 mf 0 mt fh 3 50 9.9999e+009 ’Chézy Coefficient Waterlevel’ PDIN 0 0 ” pdin CLTT ’H’ ’603.5’ cltt CLID ’(null)’ ’(null)’ clid
TBLE
-0.5 52.79 <
0 49.69 <
0.5 48.98 <
1 53.66 <
1.5 56.96 <
2 65.4 <
2.5 63.92 <
3 62.03 <
3.5 59.89 <
4 59.02 <
4.5 59.97 <
5 59.68 <
tble
mr fh 3 50 9.9999e+009 ’Chézy Coefficient Waterlevel’ PDIN 0 0 ” pdin CLTT ’H’ ’603.5’ cltt CLID ’(null)’ ’(null)’ clid
TBLE
0 53.13 <
0.5 49.9 <
1 49.3 <
tble s1 0 c1 cp 0 35 9.9999e+009 r1 cp 0 35 9.9999e+009 s2 6 c2 cp 0 9.9999e+009 9.9999e+009 r2 cp 0 9.9999e+009 9.9999e+009 d9 f9 0 9.9999e+009 9.9999e+009
bdfr

Where:

em	flag for main section 1 = bed friction for negative flow equals the definition for positive flow 0 = different friction definitions for both directions
er	flag for main section 1 = bed friction for positive flow equals the definition for negative flow 0 = different friction definitions for both directions
e1	flag for floodplain 1 1 = bed friction for negative flow equals the definition for positive flow 0 = different friction definitions for both directions
e2	flag for floodplain 1 1 = bed friction for positive flow equals the definition for negative flow 0 = different friction definitions for both directions
e3	flag for floodplain 2 Same meaning as e1
e4	flag for floodplain 2 Same meaning as e2

Cross section related Friction

CRFR id ’fricid’ nm ’Friction1’ cs ’Crdef’
lt ys
TBLE

0.0 3.0 <
3.0 5.0 <
tble
ft ys
TBLE
7 20 <
7 20 <
tble
fr ys
TBLE
7 20 <
7 20 <
tble
crfr

Where:

cs 'Crdef' = cross section definition id (only for yz profile and asymmetrical trapezium) to which this friction definition applies

lt ys= table for y values which defines the sections (in this case 2) within the profile for definition of friction, flow, etc.

Number of rows defines the number of defined sections and value per row defines the start of a section and end of a section (horizontal distance increasing from the left to right). For example, the first defined section starts at Y= 0.0 (including) till Y =3.0 (not including), and so on. Note: the defined sections should be based on the same coordinate system used in defining yz table.

ft ys= table for friction values in positive direction for (in this case 2) sections (division) of cross section with friction type (0-7), constant friction value, the number of rows should be the same as number of sections defined in the 'lt ys'.

fr ys =table for friction values in negative direction for (in this case 2) sections (division) of cross section with friction type (0-7), constant friction value the number of rows should be the same as number of sections defined in the 'lt ys'.

Structure Friction (for SOBEK Rural / Urban only)

STFR id '1' ci 'Culvert1' mf 7 mt cp 0 20 0 mr cp 0 20 0 s1 6 s2 6 sf 7 st cp 0 20 0 sr cp 0 20 0 stfr

Fixed number of items in one record, separated by 1 or more blanks

Where:

ci   id of structure definition
mf   main friction type (main = main channel)
      0 = Chézy
      1 = Manning
      2 = Strickler Kn
      3 = Strickler Ks
      4 = Nikuradse
      5 = Engelund

	6 = -
	7 = De Bos and Bijkerk
mt	friction in positive flow direction mt fq =friction=f(Q) mt fh =C=f(h) mt cp =friction as a constant or as a function of the location on the branch For fq, fh, and cp: a constant (entered as a table) or a real table: 0 = constant 1 = variable The options fq and fh may have more dimensional tables: column 1 =Q or h value, column n = friction value on different locations along the branch for every Q or h Thus, the options fq and fh are a function of the location on the branch and Q (of h). The option cp (friction as function of the location) has a two-dimensional table: column 1 = location along the branch column 2 = friction-coefficient
mr	friction in negative direction: mr fq =friction=f(Q) mr fh =friction=f(h) mr cp =friction as a constant or as a function of the location on the branch) Option fq, fh, and cp may contain a constant given in a table or a table. 0 = constant 1 = variable
s1	friction for floodplain 1 (**not in SOBEK Urban**/**Rural**) can be either 'equal to main section', or Chézy/../Nikuradse. (0=Chézy,..,6=Equal to main section) **Note:** Engelund cannot be used for the floodplains.
s2	friction for floodplain 2 (**not in SOBEK Urban**/**Rural**)
sf	ground layer friction type (0 - 7) (for further details see description for mf)
st	friction in positive direction st cp = (for all friction types, for further details see description for mt)
sr	ground layer friction in negative direction sr cp for all friction types (for further details see description for mt) c1 cp,fq,fh=floodplain 1 friction coefficients (friction can be defined as a function of Q, of h, of the location or as a constant. r1 cp,fq,fh=floodplain 1 reversed flow friction coefficients as a function of Q, of h, of the location or as a constant. c2 cp,fq,fh=floodplain 2 r2 cp,fq,fh=floodplain 2 reversed flow
d9 f9	D90

For Overland Flow:

D2FR id '1' nm 'Frictie1' ci '1' mf 0 mt cp 0 45 0 mw cp 0 45 0 d2fr

where:

id	bed friction definition id
nm	bed friction definition name (not available yet)
ci	carrier id = branch id
mf	main friction type (main = main channel) 0 = Chézy

	1 = Manning
	4 = White Colebrook
mt	bed friction
	mt cp 0 60 0 = all other cases
	(Chézy / Manning etc. as constant or dependent on the location)
	0 = constant
	mw cp 2 'c:\sobek\fls_files\bedfrict.asc' ' '
	2 = file
	mw = wall friction
	mw cp 0 60 0 = all other cases
	(Chézy / Manning etc. as constant or dependent on the location)
	0 = constant
	mw cp 2 'c:\sobek\fls_files\walfrict.asc' ' '
	2 = file

D.8.2 exr-file (friction layer)

This file contains the definition of 'extra resistance'.

XRST id '5' nm 'XRS05' ty 0 rt rs

TBLE
..
..
tble xrst

Where:

id	= id of the extra resistance definition
nm	= name of the resistance definition
ty	= type of extra resistance
	0 = extra resistance based on $\xi(\Delta H = \xi \times Q \times \|Q\|)$
rt rs	= table with extra resistance
	column 1 = water level
	column 2 = extra resistance

D.8.3 glf-file (friction layer)

This file contains two groups of global data:

- ◇ data necessary for the Engelund bed friction formula
- ◇ global bed friction parameters

As these parameters are defined globally, they are not linked to any specific location.

GLFR dd 1.65 s1 0.474 s2 0.55 p1 -6 p2 2.75 p3 5.5 p4 4.125 p5 -0.2 p6 2.447
a1 0.0005 a2 6e-005 ra 1.205
BDFR id '0' ci '0' mf 4 mt cp 0 0.003 0 mr cp 0 0.003 0 s1 6 s2 6 bdfr glfr

Where:

dd	delta D (global Engelund parameter)
s1	sigma1 (global Engelund parameter)
s2	sigma2 (global Engelund parameter)
p1	as11 (global Engelund parameter)
p2	as12 (global Engelund parameter)
p3	as21 (global Engelund parameter)
p4	as22 (global Engelund parameter)
p5	as31 (global Engelund parameter)
p6	as32 (global Engelund parameter)
a1	alpha1 (for wind friction)
a2	alpha2 (for wind friction)
ra	rho air (for wind friction)

BDFR .... bdfr contains the global/default bed friction data.

D.8.4 wnd-file (friction layer)

This file contains information for calculation of the wind friction. Other information like the wind speed and direction is stored in the meteo-layer.

WNDS id ’0’ nm ’null’ ci ’0’ lc 9.9E+009 sh ws 0 0 9.9E+009 wnds

or

WNDS id ’1’ nm ’null’ ci ’4’ lc 9.9E+009 sh ws 2
TBLE ...
tble
wnds

Where:

id	id of wind friction definition
nm	name of the wind friction definition
ci	carrier id (id of the branch)
lc	location on branch (**not used**)
sh ws	wind shielding table 0 constant 2 variable as a function of the location on the branch column 1 = location column 2 = wind shielding factor (1=no wind shielding).

Global wind friction is indicated by the extra keyword GLFR ... glfr, with carrier id **ci ’-1’**.

D.9 Grid layer

Files in this layer

```
[Grid layer]
NrOfFiles=3
net=grid.tp
seg=wqsegm.tp
lim=segmlm.tp
```

D.9.1 net-file (grid layer)

SOBEK automatically connects an id to a grid point. This is necessary to process the results per grid point via the HIS file format.

Grid table per branch

GRID id '1' ci '1' re 2 dc 2 gr gr
TBLE ..
tble
grid

Where:

id	id of the grid table
ci	carrier id (=branch id)
re	type of branch identifiers re 0 = default re 2 = identifier based on name and distance
dc	number of decimals
gr gr	grid table, with the distance of every grid point from the beginning of the branch. See 'Table'.

SOBEK-RE only:

GRID id '0' nm '(null)' ci '0' lc 9.9999e+009 se 0 oc 0 gr gr 'GridPoints on Branch <branch1> with length: 10000.0' PDIN 0 0 " pdin CLTT 'Location' '1/R' cltt CLID '(null)' '(null)' clid TBLE
...
tble
grid

D.9.2 seg-file (grid layer)

This file contains the definition of the segments for the 1DWAQ module.

SGMT id '1' nm 'Segment1' sl DLST 3 4 2 dlst uo 0 sgmt

Where:

id	segment id
nm	segment name
sl	segment limits, indicated by a list between the keywords DLST and dlst. This list refers to id's from the file with segment limits (**lim**-file)
uo	parallel 0 = no 1 = yes

D.10 Groundwater layer

Files in this layer:

```
[Groundwater layer]
NrOfFiles=1
gwm=groundwater.gwm
```

D.10.1 gwm-file (groundwater layer)

This file contains the groundwater parameters per branch (SOBEK River only)

GWPR id '1' nm 'gw' ci '-1' mu tm 0 45 9.9999e+009 kd tk 0 1400 9.9999e+009 gc tg 0 20 9.9999e+009 hb th 0 2.5 9.9999e+009 gwpr

With:

id	id of the groundwater definition
nm	name of the groundwater definition
ci	carrier id (= id of the branch)
mu tm	storage coefficient table 0 = constant 1= table (f(x)) TBLE ... (1st column=x, 2nd column= storage coefficient in [-])
kd tk	hydraulic conductivity table 0 = constant 1= table (f(x)) TBLE ... (1st column=x, 2nd column= hydraulic conductivity in [m2/d])
gc tg	entrance resistance table 0 = constant 1= table (f(x)) TBLE ... (1st column=x, 2nd column= entrance resistance in [d/m])
hb th	initial groundwater level table 0 = constant 1= table (f(x)) TBLE ... (1e column=x, 2e column= initial groundwater level in [m])

D.10.2 gwn-file (runtime-data layer)

The runtime-data layer (gwn-file) contains the numerical parameters. These consist of the following parameters:

GWNM dh 0.000000 lr 0 ll 0 av 0 ol 0 bl 0 gwnm

where:

dh	Δh
lr	length of the period containing recent water levels in $n\Delta t$
ll	length of the period containing less recent water levels in $n\Delta t$
av	length of the period containing less recent data, for which the average is calculated $n\Delta t$
ol	length of the period containing ancient water levels $n\Delta t$
bl	time between equilibrium water level and the first SOBEK water level $n\Delta t$

D.11 Initial Conditions layer

This layer contains the descriptions of the initial conditions. These can either be defined globally or per branch. Furthermore, there are the options of auto-start (only for Flow module) and restart.

Files in this layer

```
[Initial Conditions layer]
```

```
NrOfFiles=5
igl=incondgl.dat
ifl=incondfl.dat
iwq=incondwq.dat
isa=incondsa.dat
imo=incondmo.dat
```

D.11.1 ifl-file (initial conditions layer)

This file contains the description of initial conditions of the flow module. For global flow conditions the additional keywords GLIN glin (and carrier id -1) are used.

For Channel Flow, Sewer Flow and River Flow:

FLIN id '1' nm 'Initcond1' ci '1' q_ lq 0 0.01 ty 1 lv ll 1
TBLE ..
tble
flin

or

GLIN FLIN id '1' nm 'Initcond1' ci '-1' q_ lq 0 0.01 ty 0 lv ll 0 0.05 flin glin

where:

id	id
nm	name
ci	carrier id (branch id)
q_ lq	initial discharge q_ lq 0 = as a constant, q_ lq 2 = as a function of the location on the branch column 1 = location column 2 = discharge
ty	type water level/depth 1 = water level 0 = water depth
lv ll	value for depth or water level lv ll 0 = constant lv ll 2 = table as function of the location op de branch column 1 = location, column 2 = water depth or water level

For Overland Flow:

D2IN id '11' ci '-1' lc 0 ty 1 lv ll 0 9.88 d2in

where:

id	initial condition point id
ci	branch id
lc	0
ty	type water level 1=water level (always 1) lv ll 0 9.88 = constant water level [m w.r.t. reference level]

D.11.2 igl-file (initial conditions layer)

This file contains global definitions that have been entered in the UI in the settings screen.

```
GLIN fi 1 fr 'null' glin
```

Where:

fi	flow init 0 = user defined 1 = auto start
fr	restart filename

D.11.3 imo-file (initial conditions layer)

This file contains the initial conditions for the sediment/ morphology module (SA). Global WQ initial conditions start with GLIN, have a carrier id of -1, and end with glin.

```
MPIN id '1' ci '8' ty 2 dt gs
TBLE ..
tble
mpin
```

and

```
GLIN MPIN id '1' ci '-1' ty 0 c3 35 cm 40 c5 50 c9 90 mpin glin
```

Where:

id	id
nm	name
ci	carrier id (branch id)
ty	type 0 = constant 2 = as function of the location
c3	D35 constant
cm	Dmedium constant
c5	D50 constant
c9	D90 constant
r3	right D35 constant (only 2D morphology)
rm	right Dmedium constant (only 2D morphology)
r5	right D50 constant (only 2D morphology)
r9	right D90 constant (only 2D morphology)
dt gs	table met grain sizes: column 1 = location along the branch column 2 = D35 value column 3 = Dmedium value column 4 = D50 value column 5 = D90 value For 2D morphology: column 1 = location column 2 = D35 left column 3 = D35 right column 4 = Dmedium left column 5 = Dmedium right

column 6 = D50 left
column 7 = D50 right
column 8 = D90 left
column 9 = D90 right

Note: relation initial condition and bed-friction formulation

There is a relation between the type of initial condition and the type of bed friction on the branch; for example, the Engelund friction needs only the D50 and D90 constants, Nikuradse Dm and D90, Strickler Kn D50, Chézy f(Q) D35 and D50, and Chézy f(location) D50 and D90.

D.11.4 isa-file (initial conditions layer)

This file contains the initial conditions for the salt module (SA). Global WQ initial conditions start with GLIN, have a carrier id of -1, and end with glin.

STIN id '1' nm 'Init1' ci '5' ty 0 co co 0 90 stin

or

GLIN STIN id '0' nm 'Initglobal' ci '-1' ty 1 co co 0 50 stin glin

Where:

id	id of the initial condition
nm	name of the initial condition
ci	carrier id
ty	type concentration: 0 = salt 1 = chloride
co co	table with initial salt conditions co co 0 = constant co co 2 = table as a function of the location on branch column 1 = location column 2 = load

D.11.5 iwq-file (initial conditions layer)

This file contains the initial conditions for the WQ module (WQ). Global WQ initial conditions start with GLIN, have a carrier id of -1, and end with glin.

WQINid '1' nm 'Initcond1' ci '1'
ac 1 as SLST 'Surtemp' 0 11 'dTR1' 0 123 slst is SLST slst
wqin

or

GLIN WQINid '0' nm 'Initglob' ci '-1'
ac 1 as SLST 'Surtemp' 0 11 'dTR1' 0 123 slst is SLST slst
wqin glin

Where:

id	id of the initial condition
nm	name of the initial condition

ci	carrier id (branch id)
ac	active
as	between the keywords SLSR and slst:

- ◇ active substances
- ◇ name of substance,
- ◇ type initial condition (0=constant, 1/2= f(. . .),
- ◇ initial concentration

is	between SLST and slst: list with inactive substances

D.12 Measured Data layer

Files in this layer:

```
[Measurements]
NrOfFiles=1
net=..\work\network.me
```

In SOBEK Urban/Rural, it is possible to define measurement stations that can be used as target value locations for controllers.

D.12.1 net-file (measured data layer)

For Overland Flow:

History Node:

MEPT id '11' nm " px 102791.8 py 459887.6 mept

Measurement Line (meetraai):

MELM id 'l_meetraai' nm 'meetraai' ci " lc 0 bx 257778.51 by 585767.76 ex 257774.22 ey 584358.75 bc 7 br 3 ec 7 er 17 melm

where:

id	id of the history node or measurement line
nm	name of the history node or measurement line
ci	" (always blank)
lc	0 (always 0)
px	X-coordinate within 2D grid
py	Y-coordinate within 2D grid
bx	Begin X-coordinate within 2D grid for Measurement Line
by	Begin Y-coordinate within 2D grid for Measurement Line
ex	End X-coordinate within 2D grid for Measurement Line
ey	End Y-coordinate within 2D grid for Measurement Line
bc	begin column position within 2D grid (m) in X direction
br	begin row position within 2D grid (n) in Y direction
ec	end column position within 2D grid (m) in X direction
er	end row position within 2D grid (n) in Y direction

D.13 Meteo layer

Files in this layer

```
[Meteo layer]
NrOfFiles=4
wnd=wind.mt
sun-sunshine.mt
wat=wattemp.mt
air=airtemp.mt
```

This layer contains de descriptions of the meteo data which can be specified either globally or per branch. This layer can be compared to the friction layer in the sense that there is no separate file that contains references to locations, because the data is specified directly per branch. In all files the keywords GLMT and glmt (with carrier id -1) can be used to specify a 'model wide' meteo description.

D.13.1 air-file (meteo layer)

This file contains data about the air temperature. It can be used optionally by the 1DWAQ module. GLMT lmt and a carrier id of -1 indicate a global definition.

GLMT MTEO id '1' nm 'globalairtemp' ci '-1' au 0 mteo glmt

and

MTEO id '2' nm 'airtemp1'ci '1' au 1 at ta 0 10 mteo

Where:

id	id of the air temperature definition
ci	carrier id (= id of the branch)
au	air temperature used 0 = no 1 = yes
at ta	air temperature table 0 = constant 1 = table column 1 = time column 2 = air temperature

D.13.2 sun-file (meteo layer)

This file contains information about the sunshine per branch; this information can be used optionally in the water quality module. GLMT, glmt and a carrier id of -1 indicate a global definition.

GLMT MTEO id '1' nm 'globalsun' ci '-1' su 1 sh ts 1
TBLE ..
tble
mteo glmt

and

MTEO id '2' nm 'suntak4'ci '4' su 0 mteo

Where;

id	id of the sunshine definition

nm	name of the sunshine definition
ci	carrier id (= id of the branch)
su	sunshine used 0 = no 1 = yes
sh ts	sunshine table 0 = constant 1 = table column 1 = time column 2 = sunshine in W/m2

D.13.3 wat-file (meteo layer)

This file contains information about the water temperature, that can be used optionally in the salt module and water quality module. GLMT glmt and a carrier id of -1 indicate a global definition.

GLMT MTEO id ’1’ nm ’globalwattemp’ ci ’-1’ tu 1 tp tw 1
TBLE ..
tble
mteo glmt

and

MTEO id ’3’ nm ’wattemp3’ci ’3’ tu 0 mteo

Where:

id	id of the water temperature definition
ci	carrier id (= id of the branch)
tu	water temperature used 0 = no 1 = yes
tp tw	water temperature table 0 = constant 1 = table column 1 = time column 2 = water temperature

D.13.4 wnd-file (meteo layer)

This file contains de wind descriptions per branch; this information can be used optionally in the flow module. GLMT, glmt and a carrier id of -1 indicate a global definition.

GLMT MTEO id ’1’ nm ’globalwind’ ci ’-1’ wu 1 wv tv 0 3 wd td 0 270 mteo glmt

and

MTEO id ’2’ nm ’windtak1’ci ’1’ wu 0 mteo

Where:

id	id of the wind definition
nm	name of the wind definition
ci	carrier id (= id of the branch)

wu	wind used 0 = no 1 = yes
wv tv	wind velocity table 0 = constant 1 = table (f(time)) TBLE ... column 1 = time column 2 = wind velocity in m/s
wd td	wind direction table 0 = constant 1 = table (f(time)) TBLE ... column 1 = time column 2 = wind direction in degrees, North=0, East = 90, West=270

D.14 Run Time Data layer

This layer contains a number of different data elements for different modules. **Files in this layer**

```
[Runtime Data layer]
NrOfFiles=18
flt=flowtim.dat
fln=flownum.dat
flm=flowmap.dat
flh=flowhis.dat
san=saltnum.dat
sam=saltmap.dat
sah=salthis.dat
sen=sednum.dat
sem=sedmap.dat
seh=sedhis.dat
mon=mornum.dat
mom=mormap.dat
moh=morhis.dat
wqt=wqtim.dat
wqn=wqnum.dat
wqm=wqmap.dat
wqh=wqhis.dat
pwq=prepwq.dat
```

In this case there are 5*4-3+1=18 files (4 files per module, FL, SA, SE, MO, WQ, but no file with timeparameters for SA, SE and Mo, and an extra file for the link with WQ).

D.14.1 flh-file (SOBEK River) (run time data layer)

This file contains control parameters for the Flow History output.

```
FLHS it ITMS h\_ qt itms bi DLST ’1’ ’2’ dlst lc RLST 40 100 rlst
sl DLST ’0’ ’2’ dlst dl DLST ’0’ ’3’ dlst ts 1 flhs
```

Where:

it	List with History output items, between keywords ITMS and itms. The same items as for the MAP output can be selected. Extra available items are (only for structures):
gh	gate height
ch	crest height

cw	crest width
qs	flow at structure
bi	list with branch-id's, between keywords DLST dlst
lc	list with location on branch, between keywords RLST rlst
sl	list with structure id's, between keywords DLST dlst
dl	list with lateral discharge id's, between keywords DLST dlst
ts	output-interval in number of time steps

D.14.2 flm-file (run time data layer)

This file contains information for the map output of the flow module. Map output is generated for all locations.

```
FLMP it ITMS h\_ qt itms st '1994/01/01;00:00:00' et '1994/02/01;12:00:00' ts 10 flmp
```

Where:

it	List of map output items, between keywords ITMS itms.

In SOBEK River the following items can be chosen:

h	water level
at	area total
af	area flow section
am	flow area main section
a1	flow area floodplain1
a2	flow area floodplain2
qt	total discharge
qm	discharge main section
q1	discharge floodplain 1
q2	discharge floodplain 2
ct	total Chézy
cm	Chézy main section
c1	Chézy floodplain 1
c2	Chézy floodplain 2
ff	flow width section
fm	flow width main section
f1	flow width floodplain 1
f2	flow width floodplain 2
st	start time (yyyy/mm/dd;hh:mm:ss)
et	end time (yyyy/mm/dd;hh:mm:ss)
ts	report time step (number of time steps)

D.14.3 fln-file (run time data layer)

This file contains numerical parameters for the flow module.

FLNMg_ 9.81 th 0.55 ps 0.5 rh 1000 ur 0.5 mi 50 sw 0.01 sd 0.1 cm 1 er 0 us 1
in 0.001 pc 1000 xn 50 sm 0.01 dt 1 flnm

Where:

g	gravity acceleration (9.81 m/s2)
th	theta (default 0.55)
ps	psi (default 0.5)

rh	rho (density of water; default 1000)
ur	under relaxation (default 0.5)
mi	max. number iterations (default 50)
sw	stop criteria water level (default 0.01)
sd	stop criteria discharge (default 0.1)
sr	relative discharge stop criteria
cm	calculation mode 0 = steady 1 = unsteady (default = unsteady)
gm	continue after convergance 1 = no 0 = yes
er	extra resistance (default 0)
us	under relaxation structure (default 1)
in	increment numerical differences structures (default 0.001)
pc	pseudo-Courant number (default 1000)
xn	max. number of iteration nodal administration matrix (default 50)
sm	stop criteria, used in solving the nodal administration matrix
dt	transition height for summer dikes (default 1)
xr	type extra resistance 0 = eta 1 = ksi

D.14.4 flt-file (run time data layer)

This file contains the time related parameters for flow module.

For River model:

FLTM bt '1994/01/01;00:00:00' et '1994/02/01;12:00:00' ct '00:10:00' cd 0 fltm

For an estuary model:

FLTM bt '1994/01/01;00:00:00' et '1994/02/01;12:00:00' tp '12:00:00' tt 75 tf 1 nt 1 if 0 fltm

Where:

bt	begin time
et	end time
ct	computation time step (hh:mm:ss); only for river model
cd	computation time step days; only for river model
tp	tidal period: length of tidal period (hh:mm:ss, default 12 hour)
tt	time tidal: number of time steps per tidal period (default 75)
tf	tidal flow: number of tidal periods per flow period (default 1)
nt	number of flow periods per morphology time (default 1)
if	number of initial flow periods (default 0)
ri	restart interval (number of time steps) (0 = write information at the end of simulation)

D.14.5 lim-file (grid layer)

This file contains the segment boundaries for the WQ module.

SGLM id '2' nm 'Segmentlimit3' ci 5 lc 180. st 2 sg 1 10 sglm

id	id of segment boundary
nm	name of segment boundary
ci	carrier id (branch id)
lc	location
st	state 1 = ** 2 = **
sg	list of id's of the upper en lower segments

D.14.6 moh-file (run time data layer)

This file contains data for morphology History output:

MPHS it ITMS sd itms bi DLST '1' '2' dlst lc RLST 40 100 rlst mphs

Where:

it	List of map output items, between keywords ITMS itms.

Choice between:

lv	bed level
dz	increase of bed level
ab	mean bed level of main channel
cr	adapted cross sections
da	increase of cross-sectional area [m2]
sd	integrated sediment transportbi=list with branch-id's, between keywords DLST dlst.
lc	list with location on branch, between keywords RLST rlst. (gl=between DLST en dlst: a list with grid points id's; not yet implemented)

D.14.7 mom-file (run time data layer)

This file contains data for morphology map output.

MPMP it ITMS lv cr itms st '1994/01/01;00:00:00' et '1994/02/01;12:00:00' ts 1 mpmp

Where:

it	List of map output items, between keywords ITMS itms.

Choice between:

lv	bed level
dz	increase of bed level
ab	mean bed level of main channel
cr	adapted cross sections
da	increase of cross-sectional area [m2]
sd	integrated sediment transport
st	start time (yyyy/mm/dd;hh:mm:ss)
et	end time (yyyy/mm/dd;hh:mm:ss)
ts	report time step (number of calculation time steps)

D.14.8 pwq-file (run time data layer)

This file contains a parameter for the link between the flow module and the WQ module.

PRNM at 5 prnm

Where:

at	aggregation time step: indicates the number of time steps of the flow module that equal one time step of the WQ module.

D.14.9 mon-file (run time data layer)

This file contains the numerical parameters for the morphology module.

MPNM sf 1.01 me 0 mi 10 mpnm

Where:

sf	stability factor
me	method of adapting cross sections 0 = equal over transport width 1 = proportional to local water depth
mi	max. number of iterations

D.14.10 sah-file (run time data layer)

This file contains data for the History output of the salt modules.

STHS it ITMS st sh itms bi DLST '1' '2' dlst lc RLST 40 100 rlst ts 1 sths

Where:

it	list with History output items, between keywords ITMS and itms.
gl	between DLST and dlst: a list with grid point id's
bi	list with branch-id's, between keywords DLST dlst
lc	list with location on the branch, between keywords RLST rlst
ts	report time step (in number of time steps)

D.14.11 sam-file (run time data layer)

This file contains the control data for the map output of the salt module.

STMP it ITMS st sh itms st '1994/01/01;00:00:00' et '1994/02/01;12:00:00' ts 1 stmp

Where:

it	list with selected output items, between keywords ITMS itms.

Available items are:

sa	salt concentration
dp	dispersion coefficient
dn	density
st	start time (yyyy/mm/dd;hh:mm:ss)
et	end time (yyyy/mm/dd;hh:mm:ss)
ts	report time step (number of time steps)

D.14.12 san-file (run time data layer)

This file contains numerical parameters for the salt module.

STNM el 1000 fr 10 stnm

Where:

el	estuary length
fr	upper boundary Forester filter

D.14.13 seh-file (run time data layer)

This file contains data for the sediment history output.

SDHS it ITMS sd itms bi DLST '1' '2' dlst lc RLST 40 100 rlst ts 1 sdhs

Where:

it	list with History output items, between keywords ITMS and itms. See SDMP records.
bi	list with branch-id's, between keywords DLST dlst
lc	list with location on branch, between keywords RLST rlst
ts	report time step (in number of time steps)

D.14.14 sem-file (run time data layer)

This file contains data for the sediment map output.

SDMP it ITMS sd itms st '1994/01/01;00:00:00' et '1994/02/01;12:00:00' ts 10 sdmp

Where:

it	List of map output items, between keywords ITMS itms.

Choice between:

sd	sediment transport
lt	sediment transport left channel (only for 2D morphology)
rt	sediment transport right channel (only for 2D morphology)
ex	sediment exchange van left naar right (only for 2D morphology)
st	start time (yyyy/mm/dd;hh:mm:ss)
et	end time (yyyy/mm/dd;hh:mm:ss)
ts	report time step (number of time steps)

D.14.15 sen-file (run time data layer)

This file contains the numerical parameters for the sediment transport module.

SDNM kv 1e-006 rd 1.65 pk 0.4 al 0.2 rn 1 sdnm

Where:

kv	kinematic viscosity
rd	relative density
pk	packing factor
al	alluvial layer factor
	rn=reduction parameter for actual transport width

D.14.16 wqh-file (run time data layer)

This file contains description of the 1DWAQ module history output.

WQHS st '1994/01/01;00:00:00' et '1994/02/01;12:00:00' ts 1 se DLST dlst sb TLST tlst wqhs

Where:

st	start time (yyyy/mm/dd;hh:mm:ss)
et	end time (yyyy/mm/dd;hh:mm:ss)
ts	report time step (number of time steps)
se	list of selected segments, between keywords DLST dlst
sb	list of selected substances and process output items, between keywords TLST tlst.

D.14.17 wqm-file (run time data layer)

This file contains the control parameters for the map output of the 1DWAQ module.

WQMP st '1994/01/01;00:00:00' et '1994/02/01;12:00:00' ts 1 sb TLST tlst wqmp

Where:

st	start time (yyyy/mm/dd;hh:mm:ss)
et	end time (yyyy/mm/dd;hh:mm:ss)
ts	report time step (number of time steps)
sb	list of selected substances, between keywords TLST tlst

D.14.18 wqn-file (run time data layer)

This file contains numerical parameters for the 1DWAQ module.

WQNM mn 1 sn 0 wqnm

Where:

mn	main integration option .. = backward in space and time .. = modified 2nd order Runge Kutta .. = 2nd order Lax Wendroff .. = alternating direction implicit .. = Modified Flux Corrected Transport

	.. = fully implicit integration in time
sn	sub integration option option 1: .. = flows and dispersion as specified .. = dispersion only if flow is not zero .. = flows and dispersion as specified; no dispersion over open boundaries .. = dispersion only if flow is not zero; no dispersion over open boundaries option 2: .. = use flow concentration transport over open boundaries .. = use higher order approximation

D.14.19 wqt-file (run time data layer)

This file contains time parameters for the 1DWAQ module

WQTM ct '01:00:00' cd 0 wqtm

Where:

ct	computation time step (hh:mm:ss)
cd	computation time step days

D.15 Structure layer

Files in this layer

```
[Structure layer]
NrOfFiles=9
net=network.st
def=struct.def
dat=struct.dat
con=control.def
trg=trigger.def
sal=struct.sa
cmp=struct.cmp
cms=structcmp.sa
valve.tab
dbs=struct.dbs
```

D.15.1 cmp-file (structure layer)

(only SOBEK River)

This file contains de data of the compound structures.

STCM id '1' st DLST '1' '2' dlst stcm

Where:

id	id of the compound structure
st	list of structure id's of all structures in the compound, between keywords DLST dlst.

D.15.2 cms-file (structure layer)

(only SOBEK River)

This file contains the information of the compound structures for salt calculations, and is not necessary for calculations without the salt module or when no compound structures are being used.

The file is identical to the 'sal' file except for two differences:

◇ the **id** is now the identifier of the compound structure (instead of a normal structure id)
◇ there is no '**cm'**

STCM id '1' sy 1 el 1 er 2 l1 3 l2 4 r1 5 r2 6 sl 1 sr 2 stcm

D.15.3 con-file (structure layer)

This file contains the definition of the controllers for the structures.

There are 6 types of controllers:

0 time controller
1 hydraulic controller
2 interval controller
3 PID controller
4 relative time controller (**not in SOBEK Urban**/**Rural**)
5 relative from value controller; **not in SOBEK Urban**/**Rural**)

0. time controller:

River Controller:

CNTL id '24' nm 'RivCntrl' ct 0 ca 2 ac 1 cf 1 ta 1 1 0 0 gi
'2"3"-1"-1' ao 1 1 1 1 mc 0.0046 ti tv PDIN 0 0 pdin
TBLE
'1999/01/02;04:20:00' 0
'1999/01/02;04:28:00' 0.57
'1999/01/02;13:14:00' 0.57
'1999/01/02;13:17:00' 0
'1999/01/02;17:01:00' 0
tble cntl

Urban/Rural Controller:

CNTL id '60' nm 'UrbRur' ct 0 ac 1 ca 2 cf 1 mc 0 bl 1 ti tv
TBLE
'2004/11/26;00:00:00' 2.2
'2004/11/26;01:00:00' 2.33
'2004/11/26;02:00:00' 2.12
'2004/11/26;03:00:00' 2.2
'1999/01/02;17:01:00' 0
tble cntl

2D Breaking – Dam controller:

```
CNTL id '##9' nm '2DdambrkContr' ct 0 ca 5 ac 1 cf 1 ta 0 0 0 0 gi '-1' '-1' '-1' '-1' ao 1 1 1 1
mc 1110 ti tv PDIN 0 1 '365;00:00:00' pdin
TBLE
'2000/01/01;00:00:00' 5 <
'2000/01/01;01:00:00' 4 <
'2000/01/01;03:30:00' 3 <
'2000/01/01;06:00:00' 1 <
tble cntl
```

Where:

id	id of the controller definition
nm	name of the controller definition
ct	controller type 0 = time controller
ca	controlled parameter 0 = crest level 1 = crest width; **(not in Urban/Rural Controller)** 2 = gate height 3 = pump capacity (SOBEK Urban/Rural; not implemented) 4 = 5 =bottom level of 2D grid cell
ac	controlled active yes/no 1 = active 0 = inactive
cf	update frequency (number of timesteps)
ta	trigger active (not in Urban/Rural Controller, 4 in River Controller) 0 = not active 1 = active
gi	id of trigger description, -1 in case of non-active triggers **(no triggers available for Urban/Rural Controller, maximum of 4 triggers for a River Controller)**
ao	and(=1)/or(=0) relations when using more triggers **(not in Urban/Rural Controller, maximum of 4 triggers for a River Controller)** *Example: [tr1 AND tr2] OR [tr3 AND tr4];* *The combined trigger comprising of the four individual triggers tr1, tr2, tr3 and tr4 is evaluated as follows:* *1) the combined trigger is "true" in case trigger tr1 is "true" and trigger tr2 is "true";* *2) the combined trigger is "true" in case trigger tr3 is "true" and trigger tr4 is "true";* *3) the combined trigger is "false" for all situations except for the ones described under point 1) and 2) above.*
mc	dValue/dt, denotes max. change velocity in controlled structure parameter (always 0 for Urban/Rural Controller)
bl	interpolation method table (only for Urban/Rural Controller) 0 = no interpolation, a block function 1 = linear interpolation
ti tv	time table

PDIN . . . . pdin = Characteristics of Time controller table

1st : 0/1 = Lineair function/Block function

2st : 0/1 = No periodicity/Use periodicity of

3st : periodicity in ddd;hh;mm;ss (only in case 2st =1)

TBLE . . . tble = Time controller table

Note :

For structure type 13 (SOBEK Urban/Rural 1D Dambreak Node, and for Formula 2 i.e:

td 2 = Verheij-vdKnaap (2002), the time table only has two rows

for example:

TBLE
'2000/01/06;10:00:00' 0 < ('Start Time' Opening area in sq.m)
'2000/01/06;11:00:00' 44.9 < ('Time to branch lowest Level' Opening area in sq m)
tble

1. hydraulic controller:

River Controller:

CNTL id 'P_1' nm 'P_Amerongen' ct 1 ca 0 ac 1 cf 1 ta 1 0 0 0 gi '5' '-1' '-1' '-1'
ao 1 1 1 1 cp 1 mp 0 ml 'l_88' '61' '-1' '-1' '-1' hc ht PDIN 0 0 pdin
TBLE
-999 5.71<
25 5.71 <
261 4.61<
504 3.48<
641 -1<
6000 -1<
tble cntl

Urban/Rural Controller:

CNTL id '60' nm 'ExampleCntrl' ct 1 ca 2 cf 1 ml 'l_88' cp 1 bl 1 hc ht 1
TBLE
1 3.2<
2 4.2 <
2.5 4.45<
3 4.5<
tble cntl

Where:

id	id of the controller definition
nm	name of the controller definition
ta	trigger active (**not in Urban/Rural Controller, 4 in River Controller)**
gi	id of trigger description (**not in Urban/Rural Controller, 4 in River Controller)**
ao	and/or relations when using more triggers **(not in Urban/Rural Controller, 4 in River Controller)**
ct	controller type 1 = hydraulic controller
ac	controller is active 1 = active

	0 =not active
ca	controlled parameter 0 = crest level 1 = crest width; **(not in Urban/Rural Controller)** 2 = gate height 3 = pump capacity **(SOBEK Urban/Rural; not implemented)**
cf	control frequency (number of timesteps)
ml	id of measurement node **(5 locations in River Controller, at present 1 in Urban/Rural Controller)**
mp	time lag between controlling parameter and controlled parameter
cb	id of branch used for measuring (control branch) **(5 locations in River Controller, not in Urban/Rural Controller)**
cl	location (relative to beginning of branch) used for measuring (control location) **(5 locations in River Controller, not in Urban/Rural Controller)**
cp	type of measured parameter 0 = water level (on branch cb location cl) 1 = discharge (on branch cb location cl) The following types of control parameters are available in River but **not in Urban/Rural Controller**: 2 = head difference (at a structure) 3 = velocity (on branch cb location cl) 4 = flow direction 5 = pressure difference
b1	Interpolation method table (**only SOBEK Urban/Rural**) 0 = none (block function) 1 = linear
hc ht	control table with relation between measured and controller parameter column 1 = measured parameter or summons of measured parameters column 2 = settings of the controlled parameter
bl	branch location used (**not in Urban/Rural Controller**) 0 = no 1 = yes whether the 5 possible branch locations are being used by the hydraulic controller; the table contains de value of the controlled parameter for the total of the selected parameter. Note: this is sensitive to the branch direction.
si	structure id (only for controlling head difference or pressure difference; **not in Urban/Rural Controller**)
ps	positive stream (only for controlling flow direction: control parameter 4=stream direction; **not in Urban/Rural Controller**)
ns	negative stream (only when using control parameter 4=stream direction; **not in Urban/Rural Controller**)

2. Interval controller:

River Controller:

```
CNTL id ’18066’ nm ‘Arjan’ ct 2 ca 1 ac 1 cf 1 ta 1 0 1 0 gi ’5’ ’-1’ ’4’ ’-1’ ao 1 1 1 1
cp 1 ml ‘89’ ui 1.5 ua 2.5 cn 1 cv 0.05 dt 1 pe 20 di 200 da 205 sp tc 1 PDIN 0 0 pdin
TBLE
‘1999/11/30;00:00:00’ 1.5
‘1999/11/30;01:00:00’ 1
‘1999/11/30;02:00:00’ 1.25
tble cntl
```

Rural Controller:

CNTL id '60' nm 'IntervCntrl' ct 2 ac 1 ca 2 cf 1 ml '89' cp 0 ui 3.2 ua 2.7 cn 1 du 0 cv 0.025 dt 0 d_0.05 bl 1 sp tc 0 1.23 0 cntl

Where:

id	id of the controller definition
nm	name of the controller definition
ta	trigger active **(not in Urban/Rural Controller, 4 in River Controller)**
gi	id van trigger description **(not in Urban/Rural Controller, 4 in River Controller)**
ao	and/or relations when using more triggers **(not in Urban/Rural Controller, 4 in River Controller)**
ct	controller type 2 = interval controller
ac	controller is active 1 = active 0 =not active
ca	controlled parameter 0 =crest level 1 =crest width; **not in Urban/Rural Controller** 2 =gate height 3 =pump capacity (**Urban/Rural Controller)**
cf	control frequency (number of time steps)
cb	id of branch used for measuring (control branch) **(not in Urban/Rural Controller)**
cl	location (relative to beginning of branch) used for measuring (control location)) **(not in Urban/Rural Controller)**
ml	id of measurement node
cp	type of measured parameter 0 = water level (on branch cb location cl) 1 = discharge (on branch cb location cl)
ui	Us minimum
ua	Us maximum
cn	control interval type 0 = fixed interval 1 = variable
du	d(U) (fixed interval)
cv	control velocity (variable interval)
dt	dead band type 0 = fixed 1 = as percentage of the discharge (**Not in Urban/Rural Controller**)
d_	dead band step size (fixed)
pe	dead band percentage D (**not in Urban/Rural Controller**)
di	minimum dead band value (**not in Urban/Rural Controller**)
da	maximum dead band value (**not in Urban/Rural Controller**)
bl	interpolation method table (**only in Urban/Rural Controller**) 0 = none (block function) 1 = linear
sp tc 0	constant set point
sp tc 1	table with set point varying in time: column 1 = date/time stamp, column 2 = set points of the controlled parameter

3. PID controller:

River Controller

CNTL id '18067' nm 'PIDCntrl' ct 3 ca 0 ac 1 cf 1 ta 1 1 1 0 gi '2' '7' '4' '-1' ao 1 1 1 1 cp 0 ml '116' u0 1.5 ui 1 ua 2.5 va 0.5 pf 1.5 if 0.05 df 0.5 sp tc 0 1.5 0 cntl

Urban/Rural Controller

CNTL id '60' nm 'PIDCntrl' ct 3 ac 1 ca 2 cf 1 ml '116' cp 0 ui 0.7 ua 1.7 u0 1.25 pf 0.56 if 0.04 df 0.25 va 0.01 bl 1 sp tc 1

TBLE
'1999/11/30;00:00:00' 1.25
'1999/11/30;01:00:00' 1.25
'1999/11/30;02:00:00' 1.75
tble cntl

Where:

id	id of the controller definition
nm	name of the controller definition
ta	trigger active **(not in Urban/Rural Controller, 4 in River Controller)**
gi	id van trigger description **(not in Urban/Rural Controller, 4 in River Controller)**
ao	and/or relations when using more triggers **(not in Urban/Rural Controller, 4 in River Controller)**
ct	controller type 3 = PID controller
ac	controller is active 1 = active 0 = not active
ca	controlled parameter 0 = crest level 1 = crest width; **(not in Urban/Rural Controller)** 2 = gate height 3 = pump capacity **(Urban/Rural Controller)**
cf	control frequency (number of time steps)
cb	id of branch used for measuring (control branch) **(not in Urban/Rural Controller)** Only one location can be used here, as opposed to the hydraulic controller, where 5 locations can be entered.
cl	control location (relative to the beginning of the branch) **(not in Urban/Rural Controller)**
ml	id of measurement node
cp	type of measured parameter 0 = water level (on branch cb location cl) 1 = discharge (on branch cb location cl)
ui	Us minimum
ua	Us maximum
u0	Us initial (not used, initial value is taken from structure definition)
pf	K factor proportional
if	K factor Integral
df	K factor differential

va	maximum speed of change (i.e. m/s for the crest of a movable weir)
bl	interpolation method table **(only Urban/Rural Controller)**
	0 = none (block function)
	1 = linear
sp tc 0	constant set point
sp tc 1	table with set point varying in time:
	column 1 = date/time stamp
	column 2 = set points of the controlled parameter

4. 'relative time' controller'/ 5. 'relative from value' controller (only **River Controller**) is the definition:

Relative Time Controller

CNTL id 'P_0' nm 'P_Driel' ct 4 ca 0 ac 1 cf 1 ta 0 1 0 0 gi '-1"3"-1"-1' ao 1 1 1 1 mc 5 mp 0 ti vv PDIN 0 0 pdin
TBLE
0 1.5
240 2
600 1.5
tble cntl

Relative from Value Controller

CNTL id 'P_0' nm 'P_Driel' ct 5 ca 0 ac 1 cf 1 ta 0 1 0 0 gi -1 3 -1 -1 ao 1 1 1 1 mc 5 mp 0 ti vv PDIN 0 0 pdin
TBLE
0 1.5
240 2
600 1.5
tble cntl

Where:

id	id of the controller definition
nm	name of the controller definition
ta	trigger active
gi	id of the trigger description
ao	and/or relations when using more triggers
ct	Controller type
	4 = relative time controller
	5 = relative from value controller
ac	controller active yes/ no
	1 = active
	0 =not active
ca	controlled parameter
	0 = crest level
	1 = crest width
	2 = gate height
cf	control frequency (number of time steps)
mc	max. change of dValue / dT
mp	minimum period between two active periods of the time controller
ti vv	table with set point varying in time:
	column 1 = relative time

column 2 = set point of the controlled parameter

Note: Relative time stamps
In a River Controller, the relative time is indicated by the number of time steps. So when the size of the time steps changes, the reaction speed of the controller will change as well.

Note: Relative controllers
The 'relative time controller' and the 'relative from value' controller are derived from the time controller. The time controller uses the absolute time while the **relative time controller** uses the time from the moment the controller is active. This type of controller is used by BOS-Nieuwe Waterweg to control the sluices with a fixed procedure, but variable starting time.The **'relative from value controller'** uses the actual value of the controlled parameter from the moment the controller is active. Thus, the relative time is not set to zero, but set to the value found in the given table.

D.15.4 dat-file (structure layer)

This file contains information about every structure in the network. This information consists of a number of parameters of the structure and a reference to the description used for the structure in the "**def**" file.

For Channel Flow, Sewer Flow and River Flow:

Urban/Rural structure:

STRU id '59' nm " dd '59' ca 1 cj '59' stru

or

Single River Structure:

STRU id '123' nm " dd '17130' df 1.5 ca 0 1 1 0 cj '-1' '18070' 'P_1' '-1' cm 0 stru

or

River Structure which is member of Compound Structure:

STRU id '105##1' nm 'Member01' dd '17129' ca 1 0 0 0 cj '18042' '-1' '-1' '-1' cm 1 mo '105' stru

Where:

id	id of the structure
nm	name of the structure
dd	id of the structure definition (refers to the def-file)
ca	indicates whether a controller is used (controller is active) 1 = active 0 = inactive
cj	controller id's
cm	compound structure **(Only in River Structure)** 0 = the structure is not part of a compound structure 1 = the structure is part of a compound structure
mo	member of compound structure (in case cm= 1) **(Only in River Structure)**
df	structure inertia damping factor

if not present a default (setting) value will be used

Note: Controllers
In SOBEK Urban/Rural only one controller can be used for every structure; in SOBEK River the maximum is 4; when less controllers are being used, the list of 'active controllers' is filled with 1 or more '0', and the list of controller id's with 1 or more '-1'.

For Overland Flow:

D2ST id '49' nm 'Houtrib2' dd '##4' ou 0 ca 0 0 0 0 cj '-1' '-1' '-1' '-1' d2st

D2ST id '56' nm 'Houtrib1' dd '##3' ou 1 ca 1 0 0 0 cj '##5' '-1' '-1' '-1' d2st

Where:

id	id of the structure
nm	name of the structure
dd	id of the "Decrease in Height Table" (refers to the struct.def file)
ou	denotes if a "Decrease in Height Table" or a "Controller" is active 0 = Decrease in Height Table is active 1 = Controller is active
ca	indicates if controllers are active, applicable only in case ou=1, max. four controllers 1 = active 0 = inactive
cj	controller id's -1 = no controller is active ##5 = controller id (refers to the control.def file)

D.15.5 dbs-file (structure layer in SOBEK RE)

This file is only used in SOBEK-RE, see also the description of the definition of the Database Structure in the structure Def-File.

This file contains the data bases of data base structures

STDS id '6' db qt 5 TBLE .
0 1 2 3 4 5 6 7 8 9 < gate, h2/dh values.
1 0 -11 -12 -14 -15 U U U U < h1, Q-values
3 9 7 0 -11 -13 -16 U U U < h1, Q-values
5 15 12 11 5 0 -12 -15 -16 U < h1, Q-values
7 U U U 10 9 8 0 -7 -9 < h1, Q-values
9 U U U U 11 9 3 1 0 < h1, Q-values
tble
db fi 5 TBLE
5 9 <
1 5 <
1 6 <
1 8 <
4 9 <
5 9 <
tble
stds

id	identification of structure

db qt	table containing the data base.
db fi	table that defines the part of the data base that has been filled in by the user.

Description of data base table (db qt).

◇ The data base table contains a number of matrices (in this release 1 matrix). Every matrix corresponds to one gate value. The matrices in the table are in the order of increasing gate heights. Instead of gate height also a crest level or crest width can be used.
◇ The first row contains a gate height and then H2- or dH-values.
◇ The next rows contain discharges, but the first column contains a H1-value.
◇ Not all discharges need to be defined by the user. The remaining discharges are determined by an extrapolation process in the UI. In this example the extrapolated values are identified by U.
◇ The H1, H2 and dH-values must increase along rows and columns.

D.15.6 def-file (structure layer)

This file contains general definitions of structures. The following types of structures are distinguished:

0 River weir (River module only)
1 River advanced weir (River module only)
2 General structure (River module only)
3 River pump (River module only)
4 Database structure(River module only)
5 -
6 Weir
7 Orifice
8 -
9 Pump
10 Culvert, Siphon and Inverse siphon
11 Universal weir
12 Bridge
13 Bbranch growth 1D Dam break node
112 Bbranch growth 2D Dam break node

SOBEK River weir:

STDS id ’17130’ nm ’Weir’ ty 0 cl 1.5 cw 3 cs 0 po 1 ps 0.82 pt pr PDIN 0 0 “ pdin
TBLE
0.82 1.00
0.86 0.95
0.90 0.90
0.94 0.80
0.96 0.70
0.97 0.60
1.00 0.00
tble no 1 ns 0.82 nt nr PDIN 0 0 “pdin
TBLE
0.82 1.00
0.86 0.95
0.90 0.90
0.94 0.80
0.96 0.70

0.97 0.60
1.00 0.00
tble stds

Where:

id	id of the structure definition
nm	name of the structure definition
ty	type structure (0=SOBEK weir)
cl	crest level
cs	crest shape 0 = broad 1 = triangular 2 = round 3 = sharp
cw	crest width
po	correction coefficient for positive flow direction
ps	submergence limit for positive direction
pt pr	reduction table for positive flow direction first column = (h2 - z) / (h1 - z) second column = reduction factor for positive flow direction
no	correction coefficient for negative flow direction first column = (h2 - z) / (h1 - z) second column = reduction factor for negative flow direction
ns	submergence limit for negative flow direction
nt nr	reduction table for negative flow direction

SOBEK River advanced weir:

STDSid '17131' nm 'advanced Weir' ty 1 cl 1.5 sw 5 ni 7
ph 10 nh 10 pw 3nw 3pp 0.01np 0.01 pa 0.1na 0.1
stds

Where:

id	id of the structure definition
nm	name of the structure definition
ty	type structure (1] SOBEK advanced weir)
cl	crest level
sw	sill width
ni	number of piers
ph	upstream face height for positive flow direction
nh	upstream face height for negative flow direction (height of the weir relative to the bed level at the upstream side)
pw	weir design head for positive flow direction
nw	design head for negative flow direction
pp	pier contraction coefficient for positive flow direction
np	pier contraction for negative flow direction
pa	abutment contraction coefficient for positive flow direction
na	abutment contraction coefficient for negative flow direction

Note: ph, pw, pp and pa are coefficients for positive flow;

nh, nw, np and na are coefficients for negative flow.

SOBEK River general structure:

STDS id ’17129’ nm ’General’ ty 2
w1 75 wl 46.8 ws 46.8 wr 46.8 w2 75
z1 -6.5 zl -6.5 zs -6.5 zr -6.5 z2 -6.5 gh 20
pg 0.71 pd 0.64 pi 1 pr 0.8 pc 1
ng 0.71 nd 0.64 nf 1 nr 0.8 nc 1 er 1
stds

Where:

id	id of the structure definition
nm	name of the structure definition
ty	type of structure (2] SOBEK general structure)
w1	w1: width upstream side of structure
wl	wSdl: width structure upstream side
ws	wS: width structure centre
wr	wSdr: width structure downstream side
w2	w2: width downstream side of structure
z1	zb1: bed level upstream side of structure
zl	zbSl: bed level upstream side structure
zs	zbS: bed level at centre of structure
zr	zbSr: bed level downstream side structure
z2	zb2: bed level downstream side of structure
gh	gate lower edge level
pg	free gate flow in positive flow direction
pd	drowned gate flow in positive flow direction
pi	free weir flow in positive flow direction
pr	drowned weir flow in positive flow direction
pc	contraction coefficient for positive flow direction
ng	free gate flow in negative flow direction
nd	drowned gate flow in negative flow direction
nf	free weir flow in negative flow direction
nr	drowned weir flow in negative flow direction
nc	contraction coefficient for negative flow direction
er	extra resistance if not present a default (setting) value will be used

SOBEK River pump:

STDSid ’17127’ nm ’Pomp Example’ ty 3 dn -1 rt cr 1 PDIN 0 0 pdin
TBLE
0 1
1 .8
2 .6
3 .2
tble ct lt 1 PDIN 0 0 “ pdin
TBLE
12 1.7 1.5 0 0
tble stds

Where:

id	id of the structure definition
nm	name of the structure definition

ty	type of structure (3] SOBEK River Pump)
dn	control direction 1 = suction side control, pumps in positive branch direction. -1 = suction side control, pumps in negative branch direction. 2 = delivery side control, pumps in positive branch direction. -2 = delivery side control, pumps in negative branch direction.
rt cr	reduction table for the pump capacity, as a function of the water level difference. first column = level difference, second column = reduction factor
rt cr 0	constant
rt cr 1	table
ct lt	(one row) table with pump capacity and start and stop levels. The first item is the pump capacity, the second and the third item are the start and stop levels at the suction side of the pump and the fourth and the fifth item are the start and stop levels at the delivery side of the pump.

SOBEK-River Database structure:

The Database Structure consists of a set of one STDS-record and a MATR-record:

```
STDS id '##3' nm 'DB-Struc Example' ty 4 cl 2.34 di 0 dm 1 d2 0stds
MATR id '##3' db qt 5
TBLE
0 0 .5 1 2 3
0 0 -150.77 -212.77 -300.77 -335.77
.5 150.77 0 -150.77 -259.77 -335.77
1 212.77 150.77 -212.77 0 -300.77
2 300.77 259.77 212.77 0 -212.77
3 335.77 335.77 300.77 212.77 0
tble
db fi 5
TBLE
5 5
1 4
1 5
1 5
1 5
2 5
tble matr
```

Where in the STDS record:

id	id of the structure definition
nm	name of the structure definition
ty	type of structure (4] databse structure)
cl	crest level or reference level for values h1, h2 in database
di	interpolation type 0 = linear 1 = spline
dm	third dimension of data base i.e. number of gate values. In this release 1.
d2	value at second axis 0 : h2 1 : dh = h1 - h2 and where in the MATR record:

id	identification of structure
db qt	table containing the data base
db fi	table that defines the part of the data base that has been filled in by the user.

Description of data base table (db qt).

◇ The data base table contains a number of matrices (in this release 1 matrix). Every matrix corresponds to one gate value. The matrices in the table are in the order of increasing gate heights. Instead of gate height also a crest level or crest width can be used.
◇ The first row contains a gate height and then H2- or dH-values.
◇ The next rows contain discharges, but the first column contains a H1-value.
◇ Not all discharges need to be defined by the user. The remaining discharges are determined by an extrapolation process in the UI. In this example the extrapolated values are identified by U.
◇ The H1, H2 and dH-values must increase along rows and columns.

Note:
In SOBEK-RE the data in the MATR-record is written as an STDS-record with the same format in a separate file, thd dbs-file.

SOBEK Urban/Rural weir:

STDS id 'S003' nm 'stuw3' ty 6 cl -1.5 cw 2.5 ce 1 sc 1 rt 0 stds

Where:

id	id of the structure definition
nm	name of the structure definition
ty	type of structure (6] SOBEK Urban/Rural weir)
cl	crest level
cw	crest width (-1 : look at profile)
ce	discharge coefficient (depends on crest shape)
sc	lateral contraction coefficient
rt	possible flow direction (relative to the branchdirection): 0 : flow in both directions 1 : flow from begin node to end node (positive) 2 : flow from end node to begin node (negative) 3 : no flow

SOBEK Urban/Rural orifice:

STDS id 'S003' nm 'test onderlaat' ty 7 cl -2.0 cw 1.0 gh 0.5 mu 0.63 sc 1 rt 0 mp 1 0.96 mn 0 0 stds

Where:

id	id of the structure definition
nm	name of the structure definition
ty	type of structure (7] SOBEK Urban/Rural orifice)
cl	crest level
cw	crest width
gh	gate height
mu	contraction coefficient
sc	lateral contraction coefficient
rt	possible flow direction (relative to the branch direction):

	0 : flow in both directions
	1 : flow from begin node to end node (positive)
	2 : flow from end node to begin node (negative)
	3 : no flow
mp	maximum flow in positive direction
	mp <switch> <value>, where switch
	0 : do not use
	1 : use max. flow
mn	maximum flow in negative direction
	mp <switch> <value>, where switch
	0 : do not use
	1 : use max. flow

SOBEK Urban/Rural pump:

STDS id '4' nm 'Pomp 4' ty 9 dn 2 rt cr 1
TBLE
2 0.5 <
5 0.7 <
8 0.9 <
tble ct lt 1
TBLE10 0.05 -0.20 0.05 1.50 <
10 0.10 -0.10 0.10 1.25 <
5 0.20 -0.05 0.20 1.00 <
tble stds

Where:

id	id of the pomp definition
nm	name of the pomp definition
ty	type of structure (9 = pump)
dn	flow direction/ control
	1 = upward control
	2 = downward control
	3 = downward + upward control
	-1, -2, -3 : same as positive, but flow direction opposite to branch direction. Or, in the case of a lateral structure, for an extraction of water.
rt cr 1	reduction function of the water level difference, in a table
	column 1 = water level difference (suction side - pressure side)
	column 2 = reduction factor for capacity
rt cr 0 1 0	constant reduction factor (in this case, the reduction factor is 1).
ct lt 1	Table with 5 columns.
	column 1 = extra capacity;
	column 2 and 3 =start- en stop level for suction side
	column 4 and 5 =start- en stop level for pressure side

SOBEK Urban/Rural Culvert, Siphon and Inverted Siphon:

STDS id 'culvert1' nm 'culvert' ty 10 tc 1 ll -2.0 rl -1.0 si 'Crdef' li 0.63 lo 0.63 lb 0 ov -2.2 tv 1 'Table1' rt 0 dl 10.0 hs 7.6 he 8.8 stds

Where:

ty	type of structure
	10 = culvert or siphon or inverted siphon

tc type of culvert
1 = culvert
2 = siphon
3 = inverted siphon
rl bed level (right)
ll bed level (left)
si id of cross section definition (profile.def), only closed profiles
li inlet loss coefficient
lo outlet loss coefficient
lb bend loss coefficient
ov initial opening level of valve
tv table of loss coefficient
0 no table, no valve
1 valve present, reference to table in file valve.tab. See detailed decription of this file below.
rt possible flow direction (relative to the branch direction):
0 : flow in both directions
1 : flow from begin node to end node (positive)
2 : flow from end node to begin node (negative)
3 : no flow
dl length of culvert, siphon or inverted siphon
hs start level of operation of siphon
he end level of operation of siphon

SOBEK Urban/Rural Universal weir:

STDS id 'weir1' nm 'UniversalWeir' ty 11 cl 1.5 si 'trapezium1' ce 1.0 sv 0.667 rt 0 stds

where:

ty type of structure
11 = Universal Weir
cl crest level
si id of cross section definition (profile.def), only YZ Table Profile and Asymmetrical Trapezium Profile
ce coefficient in discharge formulation
sv water level based modular limit for rectangular sections, default 0.667 not to be edited
rt possible flow direction (relative to the branch direction):
0 : flow in both directions
1 : flow from begin node to end node (positive)
2 : flow from end node to begin node (negative)
3 : no flow

SOBEK Urban/Rural Bridge:

STDS id 'bridge1' nm 'bridge' ty 12 tb 2 si 'trapezium1' pw 0.5 vf 1.15 li 0.63 lo 0.63 dl 10.0 rl -1.0 stds

where:

ty type of structure
12 = bridge
tb type of bridge
2 = pillar bridge

	3 = abutment bridge
	4 = fixed bed bridge
	5 = soil bed bridge
si	id of cross section definition (profile.def), only open profiles (if tb = 3,4, or 5)
pw	total width of pillars in direction of flow (if tb] 2)
vf	shape factor (if tb] 2)
li	inlet loss coefficient
lo	outlet loss coefficient
dl	length of bridge in flow direction.
rl	bottom level

SOBEK Urban/Rural 1D Dam break node:

STDS id 'dambreak' nm 'Sobekdambreak' ty 13 cl 1.5 cs 1 cw 10 ml -0.5
stds

where:

id	id of the structure definition
nm	name of the structure definition
ty	structure type (13] SOBEK dambreak)
cl	crest level] Initial top level w.r.t. reference level [m]
cs	crest shape
	0 = broad crest (default)
cw	crest width - initial width/gap [m]
ml	minimum level w.r.t. reference level [m]
td	type of dambreak-formula
	1 = vdKnaap (2000) [default for backward compatibility]
	2 = Verheij-vdKnaap(2002)
	3 = ... not defined yet
t1	flag for using once hydraulic trigger i.s.o. start date/time
	0 = use start date/time [default for backward compatibility]
	1 = use once hydraulic trigger
f1	alpha constant for Verheij-vdKnaap(2002) (note: it is f1] F + one)
f2	beta constant for Verheij-vdKnaap(2002)
uc	critical flow velocity sediment/soil [m/sec] for Verheij-vdKnaap(2002)
ce	coefficient of discharge **(not used)**
rt	possible flow direction (relative to the branch direction):
	0 : flow in both directions (default and only possible value)

The following parameters are only used to generate the controller table

eq 0	sand for vdKnaap (2000) formula (if type of dambreak formula = vdKnaap; thus td = 1)
eq 1	clay for vdKnaap (2000) i.e td = 1 (if type of dambreak formula = vdKnaap; thus td = 1)
ts	time start in 'yyyy/mm/dd;hh:mm:ss'
dt	elasped time after the time start to branch lowest level ml in 'dd:hh:mm:ss'
ec	maximum bbranch width in m - for vdKnaap (2000)
	0 200 = constant width of 200 m

Initial top level w.r.t. reference level [m] - maximum initial opening depth [m] = minimum level w.r.t. reference level [m]

SOBEK Urban/Rural 2D Dam break node:

D2SD id 'dam' nm 'breakstruct' ty 112 t0 0.0 t1 4.0 dh lt 1
TBLE .... tble
d2sd

where:

id	id of the structure definition
nm	name of the structure definition
ty	structure type (112] 2D Dam break node)
t0	start time in hours from start of simulation
t1	end time (not if if table) in hours
dh lt 1	decrease in height with time - table with time in hours and height decrease in m
dh lt 0 10.0	constant value - linear decrease in height m start at start time
wg 0 0	width growth at rate of 5% of grid size *** **not yet implemented**
wq 1 0.6	width growth at rate specified in m/sec *** **not yet implemented**
wd 0 0	no increase in width growth with time (assumed total grid size at beginning of break) *** **not yet implemented** wd 10.5 = increase in width with time in m till grid size *** **not yet implemented**

D.15.7 net-file (structure layer)

This file contains information about the placement of structures in the network. For every place in the network that contains a structure, there is a definition:

For Channel Flow, Sewer Flow and River Flow:

STRU id 'S003' nm 'Sluis3' ci '5' lc 5007.6 stru

or

STCM id '1' nm 'compound23' ci '3' lc 100 stcm

where:

id	id of the structure
nm	name of the structure
ci	id of the branch (carrier id)
lc	position relative to the beginning of the branch

In SOBEK Rural, the keyword STRU is used. For compound structures in SOBEK River, the keyword STCM is used.

For Overland Flow:

D2ST id '7' nm ' ' ci '-1' lc 0 px 106075.3 py 458695.3 d2st

where:

id	id of the structure
nm	name of the structure
ci	id of the branch (carrier id)
lc	position relative to the beginning of the branch (always 0)
px	X coordinate within 2D grid
py	Y coordinate within 2D grid

D.15.8 sal-file (structure layer)

(only SOBEK River)

This file contains the data for salt calculations using single (non-compound) structures, and is not necessary for calculations without the salt module. The information for the compound structures is contained in another file, **structcmp.sa**.

STRU id sy 1 el 1 er 2 l1 3 l2 4 r1 5 r2 6 sl 1 sr 2 cm 0 stru

Where:

id	structure id
sy	type of salt modelling (salt type) 0 = equal salt concentrations on both sides of the structure 1 = different, complex formulation 2 = different, simple formulation
el	effective length Left (if sy=1)
er	effective length Right (if sy=1)
l1	gamma1Left (if sy=1)
l2	gamma2Left (if sy=1)
r1	gamma1Right (if sy=1)
r2	gamma2Right (if sy=1)
sl	gammaSLeft (if sy=2)
sr	gammaSRight (if sy=2)
cm	compound structure: 0=structure is not part of a compound structure 1=structure is part of a compound structure

D.15.9 trg-file (structure layer)

Only in SOBEK River.

This file contains the definition of the triggers.

A trigger can be defined as:

Time Trigger:

TRGR id '##12' nm 'TimTrig' ty 0 t1 0 tp 0 tt tr PDIN 1 0 pdin
TBLE
'1986/07/04;00:00:00' 0 0 0 0 <
'1986/07/04;01:00:00' 1 0 0 0 <
'1986/07/04;02:00:00' 0 0 0 0 <
'1986/07/04;03:00:00' 1 0 0 0 <
'1986/07/04;04:00:00' 0 0 0 0 <
tble trgr

or

Hydraulic Trigger:

TRGR id '##14' nm 'HydTrig2' ty 1 t1 0 tp 4 ts '12' ch 0 tt tr PDIN 1 0 pdin
TBLE
'1988/08/11;01:00:00' 0 0 1 4.5 <

'1988/08/11;02:00:00' 0 0 0 5 <
'1988/08/11;03:00:00' 0 0 0 4.5 <
tble trgr

or

Combined Trigger:

TRGR id '##15' nm 'CombiTrig' ty 2 t1 0 tp 0 ml '61' tt tr PDIN 1 0 pdin
TBLE
'1991/04/06;00:00:00' 1 1 0 2 <
'1991/04/06;01:00:00' 1 0 1 5 <
'1991/04/06;02:00:00' 0 1 0 3.3 <
'1991/04/06;03:00:00' 1 0 1 4.5 <
'1991/04/06;04:00:00' 0 1 0 3.55 <
tble trgr

Where:

- id — id of the trigger
- nm — name of the trigger
- ty — trigger type
  0 = time
  1 = hydraulic
  2 = combined (both time and hydraulic)
- t1 — once trigger
  0 = normal trigger (default)
  1 = once trigger (only possible for hydraulic trigger)
- tp — trigger parameter (for hydraulic /combined triggers only)
  0 = water level at branch location
  1 = head difference over structure
  2 = discharge at branch location
  3 = gate lower edge level of structure
  4 = crest level of structure
  5 = crest width of structure
  6 = waterlevel in retention area
  7 = pressure difference over structure

Note:
These number differ from the ones used in the Nefis file, which uses 1,2,3,4,5,6,7,8.

A trigger refers to exactly 1 location (branch-location or structure)

- ml — measurement station id
- ts — structure id (for hydraulic/combined triggers only)
- ch — check on (only relevant if trigger parameter=3,4,5)
  0 = value (default)
  1 = direction
- tt tr — trigger table containing 5 columns:
  column 1 = time (all types of triggers, including the hydraulic trigger; in this case, the setpoint can de defined in time)
  column 2 = on/off (for time trigger and combined trigger),
  column 3 = and/or (for combined trigger),
  column 4 = operation (hydraulic trigger: < or >)
  column 5 = trigger parameter (for hydraulic and combined trigger)

D.15.10 Valve.tab (structure layer)

Table for loss coefficients of valve. Each table refers to valves which is set in different position with relation to the structure. (for example before the culvert, within the culvert, etc. etc.)

VLVE id 'Table1' nm 'Valve for culvert' lt lc
TBLE
0 2.1 <
0.1 1.96 <
0.2 1.80 <
0.3 1.74 <
0.4 1.71 <
0.5 1.71 <
0.6 1.71 <
0.7 1.64 <
0.8 1.51 <
0.9 1.36 <
1.0 1.19 <
tble
vlve

Where:

nm	name of the table which specifies the position of valve with relation to the structure
lt lc	table for opening height of valve in percentage versus loss coefficient

D.16 Substance layer

This layer is different from the other layers. Firstly, there is no field **id** or **nm** starting a record. Secondly, all processes are stored in one record and not in one record per process.

Files in this layer

```
[Substance layer]
NrOfFiles=1
sub=substanc.def
```

D.16.1 sub-file (substance layer)

sub=substanc.def

This file contains information about (i) the list of substances for SOBEK WQ, (ii) active processes for SOBEK WQ, (iii) constants for SOBEK WQ and (iv) list of output variables for SOBEK WQ.

SBST id 'cTR1'
md 1
nm 'Conservative tracer 1'
un 'mg/l'
as 1
sp 0
ct 0.1
ou 1
sbst

Where:

id	id (Character)
md	type of list (Integer) 1 = substance 2 = constant 3 = active process 0 = output for SOBEK WQ yes/no
nm	name (only relevant when md=1)
un	unit (only relevant when md=1)
as	(only relevant as md=1) 1 = active substance 0 = inactive substance
sp=0	constant (only relevant when md=2)
ct	constant (only relevant when md=2)
ou	(relevant for all values of md) 1 = write as output 0 = don't write as output

D.16.2 Tables

The type table indication: (general)

0 = constant

1 = table as function of the time

2 = table as function of the location on the branch

3 = table as function of h for more locations on a branch, distances in CLTT

4 = table as function of Q for more locations on a branch, distances in CLTT

The name of the table (generated by the UI) (Optional)

The period and interpolation method:

PDIN
0 0 '' =interpolation continuous, no periodicity
1 0 '' =interpolation block, no periodicity
1 1 '1;00:00:00'=interpolation block, periodicity on, defined as "d;hh:mm:ss", so in this example 1 day
0 1 '365;00:00:00'=interpolation continuous, periodicity on, so in this example 1 year
pdin

The header per column between quotes

CLTT 'kop1' 'kop2' cltt

The labels id

CLID 'null' 'null' clid

The table

Every record is separated with $<$ sign. In case of an array with 1 column, 1 number per record; in case of an arry with 2 columns, 2 numbers,.........

TBLE
$<$
$<$
tble

Example of bed-friction as function of water level at one location for flow- and reverse-flow direction:

BDFR id ’33’ nm ’(null)’ ci ’31’ em 0 er 0 e1 0 e2 0 e3 0 e4 0 mf 0 mt fh 3 50 9.9999e+009 ’Chézy Coefficient Waterlevel’ PDIN 0 0 ” pdin CLTT ’H’ ’603.5’ cltt CLID ’(null)’ ’(null)’ clid
TBLE
-0.5 52.79 $<$
0 49.69 $<$
0.5 48.98 $<$
1 53.66 $<$
1.5 56.96 $<$
2 65.4 $<$
2.5 63.92 $<$
3 62.03 $<$
3.5 59.89 $<$
4 59.02 $<$
4.5 59.97 $<$
5 59.68 $<$
tble
mr fh 3 50 9.9999e+009 ’Chézy Coefficient Waterlevel’ PDIN 0 0 ” pdin CLTT ’H’ ’603.5’ cltt CLID ’(null)’ ’(null)’ clid
TBLE
0 53.13 $<$
0.5 49.9 $<$
1 49.3 $<$
tble
s1 0 c1 cp 0 35 9.9999e+009 r1 cp 0 35 9.9999e+009 s2 6 c2 cp 0 9.9999e+009 9.9999e+009 r2 cp 0 9.9999e+009 9.9999e+009 d9 f9 0 9.9999e+009 9.9999e+009 bdfr

D.16.3 Tables in network layer

Example:

’GridPoint Table’ PDIN 0 0 ” pdin CLTT ’Location’ ’1/R’ cltt CLID ” ” clid

TBLE
0 0 ” ’2’ ’2’ $<$
298.250949765087 0 ’2’ ’287_1’ ’3’ $<$
596.501899530173 0 ’4’ ’287_2’ ’7’ $<$
894.75284929526 0 ’7’ ’287_3’ ’5’ $<$
1193.00379906035 0 ’5’ ’287_4’ ’8’ $<$
1491.25474882543 0 ’8’ ’287_5’ ’6’ $<$
1789.50569859052 0 ’6’ ’287_6’ ’287_8’ $<$
2087.75664835561 0 ’287_8’ ’287_7’ ’287_9’ $<$
2386.00759812069 0 ’287_9’ ’287_8’ ’287_10’ $<$
2684.25854788578 0 ’287_10’ ’287_9’ ’287_11’ $<$

2982.50949765087 0 ’287_12’ ’1_30’ ” <
tble grid

D.17 Topography layer

Files in this layer:

```
[Topography layer]
NrOfFiles=6
net=network.tp
cpt=network.cp
nfl=network.fl
nsa=network.sa
nsm=network.sm
dat=nodes.dat
```

D.17.1 cpt-file (topography layer)

This file contains the data for the wind curving points for every branch. It contains information about the direction of parts of the branch relative to the north; this information is necessary for the calculation of the wind friction.

BRCH id ’1’ cp 1 ct bc **Note: Fixed order**
TBLE ....
tble
brch

Where:

id	branch id,
cp	number of curving points
ct bc	curving point table TBLE ... tble = Table with ’curving points: column 1 = location on the branch in meters column 2 = angle (0 = north, 90= east)

D.17.2 dat-file (topography layer)

This file contains data for the modelling of different kinds of storage for nodes in sewer calculations and for the specification of interpolation of cross-section bathymetry over the node along two branches.

Node with storage:

NODE id ’1’ ty 0 ws 1.0 ss 100.0 wl -4.05 ml -1.0 node

Where:

id	node id
ty	type water on street 1 = reservoir 2 = closed 3 = loss
ws	storage area (manhole)
ss	street storage area

wl   bed level storage reservoir (manhole)
ml   street level

NODE id ’0-62’ ty 1 ct sw PDIN 1 0 ’ ’ pdin
TBLE
16.28 2 <
20.84 2 <
tble ct ss PDIN 0 0 ’ ’ pdin
TBLE
20.85100 <
21.9150 <
tble
node

Where:

ct sw TBLE .. tble = table for storage in well
ct ss TBLE .. tble = table for storage at street

PDIN 1 0 ’ ’ pdin = block function
PDIN 0 0 ’ ’ pdin = linear interpolation

Interpolation of cross-section bathymetry over the node along specified branches:

NODE id ’ND-4’ ty 0 ni 1 r1 ’rch-1’ r2 ’rch-2’ node

Where:

id   node id
ty   Node type, always 0, interpolation only for Connection Node
ni   interpolation over Node
     1 = On
     0 = Off (default, also if record not present)
r1   branch id
r2   branch id, r1 and r2 cannot be the same. In case of ni = 0, r1 and r2 are ignored.

D.17.3 net-file (topography layer)

This file contains the date for the topography: the definitions and positions of the nodes and branches.

The node definition:

NODE id ’1’ nm ’Node1’ px 11404.2 py 123768.5 node**Note: Fixed order**

Where:

id   node id
nm   name of the node
px   position X (X coordinate)
py   position Y (Y coordinate)

The branch definition:

BRCH id ’1’ nm ’Tak1’ bn ’1’ en ’2’ al ’1233.4’ brch**Note: Fixed order**

Where:

id	branch id
nm	branch name
bn	id begin node
en	id end node
al	actual length

Linkage nodes are described in NDLK records. A linkage node is handled as a normal calculation at the actual branch. The linkage node can be used as a start or end connection node for another branch. For the actual branch, this linkage node should also be decribed in the grid layer, at the same position at the branch.

NDLK id '125' ci 'RIV_350' lc 25261.4 ndlk

id	id of linkage node
ci	branch id, where the linkage node is handled as calculation point
lc	location at the branch

D.17.4 nfl-file (topography layer)

This file contains the information for the 'dry bed procedure' in the flow module of SOBEK River (NOT for SOBEK Urban/Rural). The data is necessary for each branch.

BRCH id '1' db 1 th 0.01 sh 10.0 brch

Where:

id	branch id,
db	dry bed procedure active (1) or not (0)
th	threshold
sh	slot depth (depth Preismann)

D.17.5 nsa-file (topography layer)

This file contains extra data for the boundary-nodes in a estuarium-model, necessary for the salt module. Only SOBEK River, NOT for SOBEK Urban/Rural.

NODE id '1' mt 0 node

Where:

id	node id
mt	mouth
	0 = boundary node does not have estuarium mouth
	1 = boundary node does have estuarium mouth

This information is necessary for the choice of the formulation of the dispersion, when the Thatcher-Harleman of the 'user defined' option have been chosen. See dispersion layer for more information.

D.17.6 nsm-file (topography layer)

This file contains the extra information necessary for the sediment/morphology module of SOBEK. Only SOBEK-River, NOT for SOBEK Urban/Rural.

BRCH id '1' sd 0 fl 1 ft fl
TBLE
....
tble
rf 0 brch

Where:

id	branch id
sd	sedredge option 0 = no 2D morphology on this branch (default) 1 = 2D morphology on this branch
fl	fixed layer (1=active, 0=not active)
ft fl	fixed layer table (between keywords TBLE en tble)

The first column in the table contains the locations along the branch and the second column contains the layer.

rf	reduction function 0 = sinus 1 = linear

D.18 Transport formula layer

The descriptions of this layer are defined either globally or for a branch (linked to branch-id's).

Files in this layer

```
[Transport layer]
NrOfFiles=1
dat=transprt.dat
```

D.18.1 dat-file (transport formula layer)

This file contains the parameters of the transport formulas used. It is only necessary when the sediment/ morphology modules are used. A transport formula can be defined for every branch. GLTR gltr and a carrier id of -1 indicate a global definition.

TRNS id '1' nm 'trans1' ci '3' ty 0 mu 1 e1 0.4 e2 0.1 e3 0.5 e4 0.3 e5 0.1 e6 0.01 e7 1 trns

or

TRNS id '3' nm 'trans2' ci '1' ty 5 mu au bu gu tc rf trns

or

GLTR TRNS id '2' nm 'globaltrans' ci '-1' ty 1 mu 1.1 trns gltr

Where:

id	id transport
nm	name
ci	carrier id (branch id, -1 for a global definition)
ty	type of transport formula 0 = Engelund/Hansen 1 = Meyer-Peter/Muller 2 = Ackers/White 3 = van Rijn 4 = Parker/Klingemann 5 = User formula
mu	multiplication
au	alpha_u (only for type 5: user formula)
bu	beta_u (only for type 5: user formula)
gu	gamma (only for type 5: user formula)
tc	theta_c (only for type 5: user formula)
rf	ripple factor (only for type 5: user formula) 0 = calculated by the model, (calculate) 1 = to be defined by user (define)
va	value ripple factor (only for type 5, with switch rf=1)
e1 .. e7	7 extra parameters, only when using 2D morphology

D.19 RR (Rainfall Runoff)

Introduction

This document describes the model database (MDB) of Sobek-RR available in SOBEK version 2.08 and higher.

RR datafiles

Control layer

- ◇ Ini file (Delft_3B.Ini)

Topography layer

- ◇ node file (3b_nod.tp)
- ◇ link file (3b_link.tp)
- ◇ runoff file (3b_runoff.tp)

Paved area layer

- ◇ data file (paved.3b)
- ◇ storage definition file (paved.sto)
- ◇ dry weather flow file (paved.dwa)
- ◇ table file (paved.tbl)

Unpaved area layer

- ◇ data file (unpaved.3b)
- ◇ storage definition file (unpaved.sto)
- ◇ alfa definition file (unpaved.alf)
- ◇ infiltration definition file (unpaved.inf)
- ◇ seepage definition file (unpaved.sep)

◇ table file (unpaved.tbl)

Greenhouse layer

◇ data file (greenhse.3b)
◇ roof storage file (greenhse.rf)
◇ silo definition file (greenhse.sil)

Open water layer

◇ data file (openwate.3b)
◇ seepage definition file (openwate.sep)
◇ table file (openwate.tbl)

Structure layer

◇ data file (struct3b.dat)
◇ definition file (struct3b.def)
◇ controller file (contr3b.def)
◇ table file (struct3b.tbl)

Boundary layer

◇ data file (bound3b.3b)
◇ table file (bound3b.tbl)

NWRW layer

◇ data file (pluvius.3b)
◇ dry weather flow file (pluvius.dwa)
◇ general data (pluvius.alg)
◇ table file (pluvius.tbl)

WWTP layer

◇ data file (wwtp.3b)
◇ table file (wwtp.tbl)

Industry layer

◇ data file (industry.3b)
◇ table file (industry.tbl)

Sacramento layer

◇ data file (sacrmnto.3b)
◇ capacities and contents definition file (sacrmnto.cap)
◇ unit hydrograph definition file (sacrmnto.uh)
◇ other data file (sacrmnto.oth)

Meteo and fixed files

◇ storage coefficients soil types <..\..\fixed\bergcoef>
◇ crop factors agricultural crops <..\..\fixed\cropfact>

- ◇ Capsim soil types <..\..\fixed\bergcoef.cap>
- ◇ Capsim rootzone data <..\..\fixed\root_sim.inp>
- ◇ Capsim unsaturated zone data <..\..\fixed\unsa_sim.inp>
- ◇ greenhouse class file <..\..\fixed\kasklasse>
- ◇ greenhouse initialisation file <..\..\fixed\kasinit>
- ◇ greenhouse water use file <..\..\fixed\kasgebr>
- ◇ crop factors open water <..\..\fixed\crop_ow>
- ◇ rainfall file (1 or more events)
- ◇ evaporation file
- ◇ Default defition file <..\fixed\3bEdit.def>
- ◇ Optional: New format greenhouse class file <..\fixed\3B\NewKasKlasData.dat>
- ◇ Optional: New format greenhouse initialisation file <..\fixed\3B\NewKasInitData.dat>
- ◇ Optional: New format greenhouse water use file <..\fixed\3B\NewKasGebrData.dat>
- ◇ Optional: New format crop data <..\fixed\3B\NewCropData.dat>
- ◇ Optional: New format open water crop factor data <..\fixed\3B\NewCropOWData.dat>
- ◇ Optional: New format soil data <..\fixed\3B\NewSoilData.dat>

The files of the Topography layer are generated by the SOBEK Network Editor (Netter). The other files are generated by other user interface programs like Settings, ModelEdit and Meteo, or generated using a conversion tool. At the moment, there are conversion tools to convert SUF-HYD files (Dutch standard files for sewer systems) and Diwa-Hydra files (an old Dutch stationary open channel flow model). In the conversion of SUF-HYD files, the SOBEK-RR input files for the NWRW layer are automatically generated.

D.19.1 Boundary layer

Data file: Bound3B.3B

This file contains the data for the nodes of model type 6 (boundary)

BOUN id ’1’ bl 0 -0.5 is 100. boun

With

id	node identification
bl	boundary level type bl 0 0.5 = fixed boundary level, of 0.5 m NAP bl 1 ’bound_1’ = variable boundary level with table identification ‘bound_1’ (data in Bound3B.Tbl file) bl 2 ’3’ = variable boundary level, with on-line coupling SOBEK. Data taken from id ‘3’ in the SOBEK-HIS-file.
is	initial salt concentration (mg/l)

Boundary table file: Bound3b.Tbl

Boundary water levels and salt concentrations as a function of time, in case of option 1. Data is known before the simulation starts.

```
BN_T id 'bnd_level' nm 'IJsselmeer' PDIN 1 0 ' ' pdin
TBLE
1997/01/01;00:00:00 -1.5 200<
1997/05/01;00;00;00 -1.45 220<
1997/10/01;00:00:00 -1.50 200<
tble
```

bn_t

With

id	table identification
nm	name of table
	PDIN ..pdin = period and interpolation method
	0 0 ' ' = interpolation continuous, no period
	1 0 ' ' = interpolation block, no period
	0 1 '365;00:00:00' = interpolation continuous, period in DDD;HH:mm:ss
	1 1 '1;00:00:00' = interpolation block, period in DDD;HH:mm:ss

TBLE .. tble contains the table, first column is the date (year month day hour minute second), the second column contains the boundary level in meters w.r.t. reference level (NAP), and the third column contains the salt concentration (only used if salt computation option is switched on).

D.19.2 Control layer

Ini file: DELFT_3B.Ini

This file is set-up with a Windows INI type of filestructure. If records are not present, defaults will be used.

```
[System]
CaseName= Test
DebugFile=0
DebugTime=1 1
DebugTime2=900 905
DebugTimeCapsim=0 0
Version=2.04
SkipBinfile=-1       ! In 2.07 Altijd =-1;
            ! In 2.05 en 2.06 optioneel inlezen van data uit ASCII ipv Bin file.
CaseSensitive=-1     ! -1=yes; 0=no. Default -1. (ARS 4988)
TimeMessage=-1       ! -1=yes.Default 0=no. Shows time on screen after reading input
            ! data and just before starting actual calculations, and immediately
            ! after finishing actual calculations
[OutputOptions]
OutputTimestep=1     *1=yes, 0=no;
OutputEvent=1     *1=yes,0=no
OutputOverall=1      * not used?
OutputDetail=47      *number event
OutputScreen=0       *1=yes, 0=no
OutputOpenwater=1       *1=output w.r.t. NAP, 0=w.r.t. referemce level
OutputGroundwater=1     *1=w.r.t. NAP, 0=w.r.t. surface level
OutputBoundary=2     *0=no, 1=yes, Rijnland format; 2=yes, Pluvius format
OutputAtTimestep=1      *Output every xx timestep
OutputAtTimestepOption=1  *1=Current value, 2=Average value, 3=Maximum value
Reduced Output=0     !-1=yes: only output at open water, structures.
                     ! Also balances and link flows.
                     ! 0=no (default)
ExtendedBalance=0       ! 0=no, -1=yes. Default=0
OutputRRPaved=0         ! 0=no, -1=yes. Default=-1.
OutputRRUnpaved=1       ! 0=no, -1=yes. Default=-1.
OutputRRGreenhouse=0    ! 0=no, -1=yes. Default=-1.
OutputRROpenWater=1     ! 0=no, -1=yes. Default=-1.
OutputRRStructure=0     ! 0=no, -1=yes. Default=-1.
OutputRRBoundary=0      ! 0=no, -1=yes. Default=-1.
OutputRRNWRW=0          ! 0=no, -1=yes. Default=-1.
OutputRRWWTP=0          ! 0=no, -1=yes. Default=-1.
OutputRRIndustry=0      ! 0=no, -1=yes. Default=-1.
```

```
OutputRRSacramento=0    ! 0=no, -1=yes. Default=-1.
OutputRRBalance=1       ! 0=no, -1=yes. Default=-1.
OutputRRSalt=0          ! 0=no, -1=yes. Default=-1.
OutputRRLinkFlows=1     ! 0=no, -1=yes. Default=-1.
[Options]
Dirlisting=1      (1=yes,0=no; within CMT always 1; indicator use @ in filenames)
WeirSettingTargetLevel=0 (Weir setting yes/no adjusted to be $<$= upstream target level; 0=n
UnsaturatedZone=0       (0=no,1=Capsim,)
InitGwlOption=-1     ! This option is relevant when the initial groundwater level is
            ! specified with respect to surface level
            ! If no Scurve is used, initialisation is done w.r.t. the specified
            ! constant surface level.
            ! If an Scurve is used, it depends on the switch InitGwlOption
            ! 0 =when using SCurve: initialisation of groundwaterlevel w.r.t
            ! specified constant surface level
            ! Default=-1= in case of using Scurve: initialisation of
            ! groundwaterlevel relative to lowest point of the Scurve
InitBcOption=0       !How to initialize storage coefficients unpaved area
            ! 0 = using open water target level (default)
            ! 1 = using initial groundwater level
InitCapsimOption=1      ! 0 = default old initialisatie,
            ! 1 = at equilibrium moisture,
            ! 2 = at moisture content for pF=2
            ! 3 = at moisture content for pF=3
CapsimPerCropArea=0     ! 0 = call Capsim with crop area averaged data once per
            ! unpaved area
            ! Default= -1=call Capsim for each crop area seperately
KvdlVariatieOpenwater=0 !0 = groundwater level follows changes open water level (default)
            !1 = treat changes open water level as rainfall/evaporation
KvdLInitOpenwater=0     !1=treat initial level difference open water - groundwater
            !as rainfall/evaporation; see KvdLVariatieOpenwater  option.
            !default = 0
KvdLDimensionInDays=30  !Number of days history to be taken into account in Krayenhoff
            ! van de Leur calculations. If missing, backwards compatibility is
            ! accomplished by assuming 10 days, with a maximum of 999
            ! timesteps.  If the record is specified, there is no maximum
            ! anymore. Only a warning is given if the number of timesteps
            ! exceeds 1000.
UnpavedScurveAreas=100  ! Number of sub areas for SCurve; default=100.
UnpavedScurveAlfaOption=0 ! 0 = absolute, 1=relative w.r.t. surface level; Default=0.
DrainageDeltaH=-1       ! -1=parallell systems (default), 0=stacked system;
MinimumDepthCF=0.0      ! Minimum depth in Channel Flow boundaries
            ! Below this level RR-unpaved infiltration from CF are stopped
ControlModule=-1     (-1=with control module, 0=without)
MaalstopModule=-1       (-1=with RTC module, 0=without)
ModFlowModule=0      (-1=with Modflow module, 0=without) \textbf{not yet operational}
WLMModule=0       !( -1=with WLM, 0=without) \textbf{not yet operational }
WQModule=0        !( -1=with WQ module, 0=without) \textbf{not yet operational}
SaltComputation=1       ! -1=yes, 0=no.
SaltConcOutput=0     ! -1=yes, 0=no.
SaltConcRainfall=8      (8 mg/l)
MinFillingPercentage=10 (below 10\% no extractions for greenhouse from storage basins)
RestartIn=1       (Use binary restart input file; 1=yes, 0=no)
RestartOut=0      (write binary restart file)
LowestRestartGwl=       ! laagste toegestane restart grondwaterstand in m - maaiveld
SkipBoundaryLevelFromRestartFile=0 !0 or -1, default=0;
            !-1 = do not use boundary levels from restart file
MaxIterations=5      ! maximum aantal iteraties binnen 1 tijdstap (default=5)
DetailedGwlComputation=-1 ! --1 = yes (default)
StepGwlStorageCoefficient=0.01 !Default 0.01 m.
CheckRootzoneGwl=-1     !-1=yes, 0=no (default).Checks whether groundwater levels
            ! is in the rootzone and sets storage coefficient to 0.01
BinaryInput=0     ! -1=yes, 0=no
BinaryOutput=0       ! -1=yes, 0=no
Defaultdataset=0     !1=yes,0=no;  option from rainfall rainfall file
            (=1 default dataset, 0 for the 44-year series)
```

```
EvaporationYear=-1      !-1 = check on year = rainfall year (default),
            !xxxx=use data from year xxxx
GreenHouseYear=1951     !-1 = check on year = rainfall year (default),
            !xxxx=use data from year xxxx
VolumeCheckFactorToCF=1 ! A multiplier on the computed maximum volume to be exchanged.
            ! Default=1. A large value effectively means that the volume check ! is not active.
VolumeCheckFactorToOW=1.0 ! A multiplier on the computed maximum volume to be exchanged.
            ! Default=1.0. A large value effectively means that the volume check ! is not activ
DetailedVolumeCheckMessages=-1 ! Enables extra information in Sobek\_3B log file in order to sh
CoefRz=0.1        ! 4 Interpolation coefficients for interpolation in Capsim tables
CoefGwl=100.      ! 2 coefficients en 2 power coefficients, voor root zone and
            !groundwater level
PowerRz=2.        !Default power coefficients PowerRz=PowerGwl=2
PowerGwl=2.       ! weight coefficients CoefRz=0.1 and CoefGwl=100.
CheckBalance=0       ! 0 = no detailed output balance error per timestep, -1=yes
OpenWaterLevelComputations=Advanced
            ! Simple or Advanced. Default =Advanced
            ! Advanced = take into account storage on land of neighbouring
            ! nodes in computing open water levels
OpenWaterPrecipitation=0 ! 0 = rainfall at area at actual level, (Default)
            ! -1=at area at lowest surface level of connected nodes,
            ! if no paved/unpaved/greenhouse nodes connected, the area at
            ! maximum allowable level is used.
StructureOperation=     ! 0 = Operate structure depending on initial level. (=default)
            ! 1 = Use SetFractionTime to switch on/off.
            ! (Sept 1999, ARS xxxx) For internal use only.
DrownedWeirDepth=       ! Switch to set which depth is used in weir formula. Default=0.
            ! 0 = lowest depth; 1 = maximum depth; 2=average depth;
EmulateUnixOnPc=        ! default=false; for testing Sobek-Parallell functionality only
```

For internal use only.

```
HeaderRunoffOutAlways=  ! default=false; for testing Sobek-Parallell functionality only.
```

For internal use only.

```
FixARS5176=       ! default=true, if =0 then false.
            !(ARS5176=surface runoff bug fixed in April/May 2000)
FixARS8842=       ! default=true, if =0 then false.
            !(ARS8842=added volume check to unpaved module drainage to
            ! open water, January 2002)
FixARS11610=      ! default=true, if =0 then false.
            ! (ARS11610=added volume check for drainage from unpaved area to boundary (CF node)
FixARSControllerLvlCheck=   ! default=true, if =0 then false.
            !(Extra check that the computed weir controller flow satisfies the
            ! possible flow directions)
FixARS10084=      ! default=.false., if=-1 then true.    ! if gw on surface, include water laye
FixARS12253=      ! default=.false., if=-1 then true
            ! NWRW runoff delay correction (1s-1min) (in temporary version this switch was call
CumulativeGroundwaterExceedance=0 ! 0=Sum, 1=Sum of squares above threshold; default=0
MessageInundation=      ! 1 = detailed inundation message per node and timestep,
            ! 0 = no message
MessageVolumeCheck=     ! 1 = yes, 0 = no message
MessagePerTimestep=     ! 1 = 1 message per inundation timestep, 0 = no message
DailyRainfallStartHour= ! Option to specify start hour of daily rainfall.
            ! Default=0. 8=daily rainfall starts at 8 o'clock in the morning
EstimateRemainingDuration=      ! 0 = no, -1=give estimate remaining runtime on screen
GenerateAanvoerAbr=     ! 1= yes, 0=no; default=0
LargeBalanceErrorPercentage=    ! Percentage ($>$0).
            ! An error message will be given if the total balance error for the
            ! RR simulation exceeds this percentage. Default value = 1.0
NewFormatCropFactors=0  ! 0=no, -1=yes. Option to use new format crop factors.
```

```
CropDefinition='Default'    ! If NewFormatCropFactors=-1, CropDefinition= is used to specify
            ! the set of crop definitions used. Default set is named 'Default'.
            ! In the crop definition, crop names and crop factors are specified.
OpenWaterCropDefinition='Default'    ! Cf. CropDefinition, but now for open water crop facto
NewFormatSoilData=0     ! 0=no, -1=yes. Option to use new format soil data.
SoilDefinition='Default'    ! If NewFormatSoilData=-1, SoilDefinition= is used to specify
            ! the set of soil data used. Default set is named 'Default'.
            ! In the soil definition, soil names and soil data like storage
            ! coefficients are defined
NewFormatKasdata = 0    ! 0=no, -1=yes. Option to use new format greenhouse data.
KasDefinition='Default' ! If NewFormatKasdata is used, then KasDefinition= is used to
            ! specify the set of fixed greenhouse data used. This set contains
            ! greenhouse initialisation data and greenhouse water use data.
ParseTokenMethod=0      ! 0=no (default), -1=yes. Alternative method of parsing input files;
            ! at the moment only available for NWRW nodes.
HisConvergence=0     ! 0=no (default), -1=yes. Option to generate an extra His file called
            ! Convergence.His containing convergence info
            ! (compare with FlowAnal.His file of Sobek-CF-SF)
StructComp=0      ! 0=no (default, backwards compatible), -1=yes.
            ! If Structcomp=.true., then the same internal iteration criteria are
            ! used for weir, orifice and friction nodes.
RunOffOutHis=0       ! --1 = Runoff.Out in HIS format, 0=no.
SeparateRWA\_DWA     ! --1=yes,separate DWA and RWA in Runoff.Out file, 0=no
DWAString='DWA'      ! String to add to NWRW location for DWA
RWAString='RWA'      ! String to add to NWRW location for RWA
NWRWContinuous=-1       !-1=yes, 0=no. Simulate NWRW model continuously for series of
            ! events, using an additional continuous bui file covering all events.
ConstantHdeZBergC=-1    !0=(default) no; --1=constant Hellinga-deZeeuw coefficient used.
WriteRestartFileWithAdimC=0     ! 0=no, -1=yes (default); includes AdimC Sacramento in resta
VnotchWeirNrInterpolationpoints=14  ! number of interpolation points for Vnotch weir.
VnotchWeirh2h1Ratio=0.8 0.85 0.885 0.9 0.925 0.945 0.95 0.96 0.97 0.978  0.985 0.99 0.995 1.
            ! Default values for H2H1 ratio Vnotch weir.
            ! Should be given for specified number of interpolation points.
VnotchWeirDrownedFlowReductionfactor= 1.0 0.96 0.9 0.87 0.8 0.7 0.67 0.6 .5 0.4 0.3 0.2 0.1
            ! Default values for drowned weir flow reduction factor
RestartOrderStrict=-1   !-1=true, 0=false. Default =0.
ReduceRROpenWaterInfiltrationAtNegativeVolume=-1 !0=false, -1= true, default value = false
[TimeSettings]
TimestepSize=3600       (computation timestepsize in seconds)
Timeweightfactor=0.5    ! weight factor of t=t0 and t=t + delta t.
TimestepExchange=3600   ! exchange timestepsize in seconds,
            ! should be a multiple of the computation timestepsize
            ! Default = equal to computation timestep
            ! At the moment, only for Modflow coupling a multiple is allowed.
ExchangeOption=Detail   ! Detail, Average or Current;
```

! **At the moment only option Detail is impemented**!.

```
NightFromHrs=23      ! night yours from  -  to
NightToHrs=7
WeekHrsasNight=1     ! Weekend hours (Saturday, Sunday) to be considered as night?
            ! 1=yes, 0=no
EvaporationFromHrs=7    ! daily evaporation from - to
EvaporationToHrs=19
PeriodFromEvent=0       ! period derived from rainfall event file. -1=yes (default);0=no.
StartTime=1995/01/01;00:30:00   ! Start date and time of simulation (within rainfall event)
EndTime=1995/01/01;01:55:00     ! End date and time of simulation period
```

D.19.3 General layer

Storage coefficients soil types <..\..\fixed\bergcoef>

The fixed input file with soil type names and storage coefficients as a function of the soil type and drainage basis. 12 soil types, 12 values for the drainage basis, followed by the table with 12 lines (1 line per drainage basis) with the storage coefficients for all soil types for that particular value of the drainage basis.

```
*Comment lines start with an * in column 1
*Bergingscoefficient, afh. ontw.diepte en grondsoort
*aantal grondsoorten (=12)
 1  'loamy, humous fine sand (mu  = 0.115 per m)'
 2  'peat (mu = 0.103 per m)'
 3  'heavy clay (mu = 0.089 per m)'
 4  'humous clay and peat (mu = 0.085 per m)'
 5  'light loamy, medium coarse sand (mu = 0.084 per m)'
 6  'loamy silt (mu = 0.072 per m)'
 7  'humous clay and peat with silty top layer (mu = 0.069 per m)'
 8  'clay and light clay (mu = 0.062 per m)'
 9  'loamless, medium coarse and coarse sand (mu = 0.060 per m)'
10  'silt (mu = 0.058 per m)'
11  'very light clay (mu = 0.048 per m)'
12  'sand with a silty top layer (mu = 0.044 per m)'
*aantal ontw.diepten (=12) (tov maaiveld)
 -0.1 -0.2  -0.3  -0.4  -0.5  -0.6  -0.7  -0.8  -0.9  -1.0  -1.2 -1.5
*Grond1   2      3      4      5      6      7      8      9     10     11   12
0.0074 0.0077 0.0094 0.0063 0.0046 0.0049 0.0049 0.0048 0.0027 0.0029 0.0025 0.0013
0.0183 0.0153 0.0166 0.0134 0.0124 0.0103 0.0118 0.0092 0.0098 0.0066 0.0052 0.0032
0.0302 0.0226 0.0226 0.0206 0.0220 0.0158 0.0181 0.0132 0.0162 0.0105 0.0079 0.0052
0.0419 0.0305 0.0278 0.0279 0.0323 0.0211 0.0237 0.0171 0.0228 0.0145 0.0107 0.0074
0.0532 0.0387 0.0323 0.0350 0.0427 0.0262 0.0289 0.0206 0.0294 0.0186 0.0134 0.0096
0.0664 0.0467 0.0363 0.0420 0.0528 0.0310 0.0339 0.0240 0.0359 0.0226 0.0160 0.0120
0.0805 0.0545 0.0399 0.0486 0.0625 0.0364 0.0386 0.0271 0.0422 0.0265 0.0185 0.0143
0.0938 0.0621 0.0431 0.0551 0.0715 0.0416 0.0430 0.0300 0.0484 0.0303 0.0210 0.0167
0.1061 0.0702 0.0461 0.0613 0.0801 0.0466 0.0472 0.0328 0.0542 0.0341 0.0235 0.0191
0.1173 0.0784 0.0489 0.0673 0.0880 0.0514 0.0512 0.0355 0.0598 0.0377 0.0258 0.0214
0.1372 0.0939 0.0538 0.0786 0.1024 0.0605 0.0586 0.0404 0.0704 0.0446 0.0303 0.0261
0.1614 0.1158 0.0600 0.0941 0.1208 0.0729 0.0685 0.0470 0.0845 0.0541 0.0366 0.0329
```

Crop factors agricultural crops <..\..\fixed\cropfact>

The fixed input file with crop names and crop factors as a function of time.

1 year of data is enough, since it is assumed that the crop factors are constant over the years; the variation is taken into account in the reference evaporation data and not in the crop factors.

The header of the file contains the number of crops and crop names.

```
*Aantal gewassen
     16
*Namen
'grass           '
'corn            '
'potatoes        '
'sugarbeet       '
'grain           '
'miscellaneous      '
'non-arable land       '
'greenhouse area       '
'orchard         '
'bulbous plants     '
'foliage forest     '
'pine forest        '
'nature          '
'fallow          '
'vegetables         '
'flowers         '
```

```
*Gemiddelde  1     2     3     4     5     6     7     8     9     10    11    12    13    1
1996   1  1  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1  2  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1  3  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1  4  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1  5  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1  6  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1  7  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1  8  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1  9  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1 10  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0.00  0
1996   1 11  0.35  0.40  0.40  0.40  0.40  0.40  0.40  0.00  0.35  0.40  0.35  0.40  0.35  0
1996   1 12  0.35  0.40  0.40  0.40  0.40  0.40  0.40  0.00  0.35  0.40  0.35  0.40  0.35  0
1996   1 13  0.35  0.40  0.40  0.40  0.40  0.40  0.40  0.00  0.35  0.40  0.35  0.40  0.35  0
etc.
```

Capsim soil types <..\..\fixed\bergcoef.cap>

File containing the names of the 21 soil types of the Staring reeks used in Capsim.

1 'Veengrond met veraarde bovengrond'
2 'Veengrond met veraarde bovengrond, zand'
3 'Veengrond met kleidek'
4 'Veengrond met kleidek op zand'
5 'Veengrond met zanddek op zand'
6 'Veengrond op ongerijpte klei'
7 'Stuifzand'
8 'Podzol (Leemarm, fijn zand)'
9 'Podzol (zwak lemig, fijn zand)'
10 'Podzol (zwak lemig, fijn zand op grof zand)'
11 'Podzol (lemig keileem)'
12 'Enkeerd (zwak lemig, fijn zand)'
13 'Beekeerd (lemig fijn zand)'
14 'Podzol (grof zand)'
15 'Zavel'
16 'Lichte klei'
17 'Zware klei'
18 'Klei op veen'
19 'Klei op zand'
20 'Klei op grof zand'
21 'Leem'

The list of 21 Capsim soil types in English:

Soil1	Decomposed clayey peat over eutrophic peat: peat soil with decomposed topsoil (hVb, hVc)
Soil2	Decomposed mesotrophic peat over a coarse textured, sandy subsoil: peat soil with decomposed topsoil (aVz, hVz)
Soil3	Humic very fine textured, clay topsoil over eutrophic peat: peat soil with a clay cover (pVb, kVb)
Soil4	Humic very fine textured, clay topsoil over coarse textured, sandy subsoil: peat soil with a clay cover (kVz)
Soil5	Humic, medium textured, sandy topsoil over coarse textured, sandy subsoil: peat soil with sand cover (iWz, iWp)
Soil6	Decomposed clayey peat over unripened clay: peat soil with decomposed topsoil (W0)
Soil7	Eolian, coarse textured sandy soil: sandy soil (Zd20, Zd21)

Soil8	Podzolic, coarse textured sandy soil: sandy soil (Hd21)
Soil9	Podzolic, medium textured sandy soil: sandy soil (Hn21)
Soil10	Podzolic, medium textured sandy soil over coarse textured sand:sandy soil (Hn21g)
Soil11	Podzolic, medium textured sandy soil over boulder clay: sandy soil (Hn23x)
Soil12	Plaggen, coarse textured sandy soil: sandy soil (zEZ21)
Soil13	Humic gleysol, coarse textured sandy soil: sandy soil (pZg23)
Soil14	Podzolic, coarse textured sandy soil: sandy soil (gHd30)
Soil15	Calcareous, medium textured, clay soil: alluvial soil (Mn25A)
Soil16	Medium textured clay soil: alluvial soil (Mn35A, Rd90A, Rd90C)
Soil17	Fine textured clay soil: alluvial soil (Rn44C, gMn83C, kMn48C, Rn47C)
Soil18	Fine textured clay over mesotrophic peat: alluvial soil (RvO1C, Mv41C)
Soil19	Medium textured clay over sand: alluvial soil (Mn22A)
Soil20	Medium textured clay over coarse textured sand: alluvial soil(n52A)
Soil21	Eolian, medium textured loam: lvss soil (BLd6)

Capsim rootzone data <..\..\fixed\root_sim.inp>

This file contains the rootzone data. The lay-out of the file is column-wise, with:

column 1 = soil type
column 2 = crop type
column 3 = root zone thickness (mm)
column 4 = relative root zone storage for h1
column 5 = relative root zone storage for h2
column 6 = relative root zone storage for h3l
column 7 = relative root zone storage for h3h
column 8 = relative root zone storage for h4

The soil water pressure heads h1, h2, h3l, h3h, h4 are used to define evaporation reduction (see technical reference for more explanation).

```
     1     1    0.40   1.000   1.000   0.760   0.669   0.479
     1     2    0.60   1.000   1.000   0.757   0.695   0.479
     1     3    0.60   1.000   1.000   0.760   0.695   0.434
     1     4    0.60   1.000   1.000   0.760   0.695   0.434
     1     5    0.60   1.000   1.000   0.713   0.658   0.434
     1     6    0.60   1.000   1.000   0.713   0.658   0.434
     1     7    0.50   1.000   1.000   0.713   0.658   0.434
     1     8    0.50   1.000   1.000   0.757   0.695   0.479
     1     9    0.60   1.000   1.000   0.760   0.669   0.479
     1    10    0.50   1.000   1.000   0.760   0.669   0.479
     1    11    0.60   1.000   1.000   0.760   0.669   0.479
     1    12    0.60   1.000   1.000   0.760   0.669   0.479
     1    13    0.50   1.000   1.000   0.760   0.669   0.479
     1    14    0.20   1.000   1.000   0.760   0.669   0.479
     1    15    0.40   1.000   1.000   0.760   0.713   0.513
     1    16    0.60   1.000   1.000   0.760   0.713   0.548
     2     1    0.40   1.000   1.000   0.628   0.486   0.249
     2     2    0.50   1.000   1.000   0.673   0.580   0.302
     2     3    0.50   1.000   1.000   0.676   0.580   0.254
     2     4    0.50   1.000   1.000   0.676   0.580   0.254
     2     5    0.50   1.000   1.000   0.607   0.527   0.254
     2     6    0.50   1.000   1.000   0.607   0.527   0.254
     2     7    0.50   1.000   1.000   0.607   0.527   0.254
     2     8    0.50   1.000   1.000   0.673   0.580   0.302
     2     9    0.50   1.000   1.000   0.676   0.542   0.302
     2    10    0.50   1.000   1.000   0.676   0.542   0.302
etc.
```

Capsim unsaturated zone data <..\..\fixed\unsa_sim.inp>

This file contains the unsaturated zone data. The lay-out of the file is column-wise, with:

column 1 = soil type
column 2 = rootzone thickness (cm)
column 3 = groundwater level (meter below surface)
column 4 = root zone soil moisture storage in equilibrium conditions (mm)
column 5 = potential capillary rise (mm/day)
column 6 = storage coefficient (m/m)
1 10 0.00 61.6 5.000 0.000
1 10 0.10 61.0 5.000 0.010
1 10 0.20 59.8 5.000 0.018
1 10 0.30 58.7 5.000 0.029
1 10 0.40 57.7 3.546 0.038
1 10 0.50 56.7 2.392 0.049
1 10 0.60 55.9 1.780 0.062
1 10 0.70 55.1 1.397 0.070
1 10 0.80 54.4 1.134 0.081
1 10 0.90 53.8 0.942 0.091
1 10 1.00 53.2 0.797 0.101
1 10 1.10 52.6 0.684 0.110
1 10 1.20 52.1 0.595 0.119
1 10 1.30 51.6 0.521 0.126
1 10 1.40 51.2 0.461 0.135
1 10 1.50 50.8 0.410 0.142
1 10 1.60 50.4 0.367 0.149
1 10 1.70 50.0 0.330 0.156
1 10 1.80 49.6 0.298 0.162
1 10 1.90 49.3 0.271 0.168
1 10 2.00 49.0 0.246 0.174
1 10 2.10 48.7 0.225 0.180
1 10 2.20 48.4 0.206 0.185
1 10 2.30 48.1 0.190 0.190
1 10 2.40 47.8 0.175 0.195
1 10 2.50 47.6 0.162 0.199
1 10 3.00 46.4 0.112 0.220
1 10 4.00 44.6 0.060 0.251
1 10 5.00 43.2 0.035 0.275
1 10 10.00 39.1 0.002 0.341
1 20 0.00 123.2 5.000 0.000
1 20 0.10 122.6 5.000 0.010
1 20 0.20 120.8 5.000 0.010
1 20 0.30 118.5 5.000 0.018
1 20 0.40 116.4 5.000 0.028
1 20 0.50 114.4 3.916 0.039
1 20 0.60 112.6 2.749 0.052
1 20 0.70 111.0 2.074 0.063 etc.

Greenhouse class file <..\..\fixed\kasklasse>

This file contains the definition of the greenhouse classes; the name, the maximum storage (in m3/ha), the maximum depth (m), and en evaporation indicator (0=no evaporation,1=yes)

```
*Kasklassen
*Code  Name  Max\_berging in m3/ha, Max.diepte, wel/geen verdamping uit bassins
1  '$<$500 m3/ha'   0      0.       0
2  ' 500-1000'     500     0.5      0
3  '1000-1500'    1000       1.       0
4  '1500-2000'    1500       1.5      1
5  '2000-2500'    2000       2.       1
6  '2500-3000'    2500       2.5      1
7  '3000-4000'    3000       3.       1
8  '4000-5000'    4000       4.       1
9  '5000-6000'    5000       5.       1
10 '$>$6000-m3/ha'  6000       5.       1
```

Greenhouse initialisation file <..\..\fixed\kasinit>

The greenhouse initialisation file defines the free space (available storage) at the start of the simulation for each greenhouse class. It contains data from 1951 up to 1999. This data is only used for defining the initial storage in the greenhouse storage basins.

```
*       Grootte bassin  (m3/ha)
*           0       500      1000     1500     2000     2500     3000     4000     5000     6000
*Jaar   Maand   Dag     Ruimte  voor    waterberging    (m3)
1951    1       1       0       0       0       0       31      100      215      519      829
1951    1       2       0       0       0       0       8       77       191      495      803
1951    1       3       0       5       5       5       13      82       196      500      808
1951    1       4       0       0       0       0       0       67       181      484      792
1951    1       5       0       0       0       0       0       5        118      419      725
1951    1       6       0       0       0       0       0       0        108      409      715
1951    1       7       0       3       3       3       3       3        112      412      718
1951    1       8       0       0       0       0       0       0        85       384      689
1951    1       9       0       0       0       0       0       0        63       362      666
1951    1       10      0       0       0       0       0       0        32       330      633
1951    1       11      0       0       0       0       0       0        0        204      503
1951    1       12      0       0       0       0       0       0        0        181      479
1951    1       13      0       0       0       0       0       0        0        152      449
1951    1       14      0       0       0       0       0       0        0        127      424
```

Greenhouse water use file <..\..\fixed\kasgebr>

This file defines the actual water use (m3/ha) from the greenhouse storage basins by the greenhouse crops for each day. Values are depending on year and date, but are assumed independant of the size of the greenhouse storage basins (so independant of the greenhouse class).

```
*         Voor alle kasklassen hetzelfde
*Jaar Maand Dag    EACT
 1951   1     1       8.36
 1951   1     2       7.04
 1951   1     3       8.04
 1951   1     4       7.04
 1951   1     5       7.04
 1951   1     6       7.04
 1951   1     7       7.72
 1951   1     8       7.04
 1951   1     9       9.43
 1951   1    10       7.53
 1951   1    11       7.31
 1951   1    12      10.94
 1951   1    13       7.31
 1951   1    14       7.51
```

Crop factors open water <..\..\fixed\crop_ow>

The fixed input file with open water evaporation factors as a function of time. 1 year of data is enough; it is assumed that the evaporation factors are same for all years.

```
*Oppervlaktewater
     1
*Namen
'0.0 Oppervlaktewater        '
*Gemiddelde       1
    1994     1     1     0.00
    1994     1     2     0.00
    1994     1     3     0.00
    1994     1     4     0.00
    1994     1     5     0.00
    1994     1     6     0.00
    1994     1     7     0.00
    1994     1     8     0.00
    1994     1     9     0.00
    1994     1       10     0.00
    1994     1       11     0.50
    1994     1       12     0.50
    1994     1       13     0.50
etc.
```

Rainfall file (1 or more events)

This file contains the rainfall data: the number of stations, the names (id's) of the rainfall stations, the number of rainfall events, the rainfall data timestep size (may be different from the computation timestep size), the start date and duration of each event and the rainfall data for each event.

```
*Name of this file: \SOB_LITE\FIXED\DEFAULT.BUI
*Date and time of construction: 17-04-1997   17:13:50
*Enige algemene wenken:
*Gebruik de default dataset voor overige invoer (altijd 1)
1
*Aantal stations
 1
*Namen van stations
'Station1'
*Aantal gebeurtenissen (omdat het 1 bui betreft is dit altijd 1)
*en het aantal seconden per waarnemingstijdstap (10800 = 3x3600)
1  10800
*Elke commentaarregel wordt begonnen met een * (asteriks).
*Eerste record bevat startdatum en -tijd, lengte van de gebeurtenis in dd hh mm ss
*Het format is: yyyymmdd:hhmmss:ddhhmmss
*Daarna voor elk station de neerslag in mm per tijdstap.
 1996 1 1 0 0 0 1 3 0 0
 0.2
 0.2
 0.2
 0.2
 0.2
 0.2
 0.2
 0.2
 0.2
```

Evaporation file

This file contains the evaporation data. It is assumed that meteo stations occur in the same order as in the rainfall file. If the evaporation file contains less stations, missing stations will use data from the first station in the file.

```
*Name of this file: \SOB\_LITE\FIXED\DEFAULT.EVP
*Date and time of construction: 10-03-2000   10:04:25
*Verdampingsfile
*Meteo data: evaporation intensity in mm/day
*First record: start date, data in mm/day
*Datum (year month day), verdamping (mm/dag) voor elk weerstation
*jaar maand dag verdamping[mm]
 1996    1    1    0
 1996    1    2    0
```

Default defition file <..\fixed\3bEdit.def>

This file contains some default values for records in the Sobek-RR MDB files for all types of 3B-nodes. The file is used by ModelEdit.

Optional: New format greenhouse class file file..\fixed\3B\NewKasKlasData.dat

This file contains the greenhouse class specification in a new format, which allows extensions by the user.

The file contains two type of records: the KASD records and KLAS records. In the KASD records the global definitions are placed, including references to the greenhouse initialisation and greenhouse water use definitions. The KLAS record contains the greenhouse class definition. The file can contain multiple KASD and KLAS records. Selection of the actual records used is done per case in the <Delft_3B.Ini> file (see description of the NewFormatKasData= and KasDefinition= records).

An example input file:

KASD id ’Default’ nm ’Default kas initialisation’ nc 10 kk ’Default kasklassen’ ki ’Default kas initialisation data’ kg ’Default kasgebruik data’ kasd

KASD id ’Low’ nm ’Low kas initialisation’ nc 10 kk ’Default kasklassen’ ki ’Low kas initialisation data’ kg ’Low kasgebruik data’ kasd

KASD id ’Periodic test’ nm ’Periodic test’ nc 10 kk ’Default kasklassen’ ki ’Periodic kas initialisation data’ kg ’Periodic kasgebruik data’ kasd

KASD id ’My own mix’ nm ’My own mix’ nc 10 kk ’Default kasklassen’ ki ’Low kas initialisation data’ kg ’Periodic kasgebruik data’ kasd

with

id	greenhouse definition id
nm	greenhouse definition name
nc	number of greenhouse classes
kk	reference to greenhouse class definition
ki	reference to greenhouse initialisation definition
kg	reference to greenhouse water use definition

The KLAS record contains the greenhouse class definition.

KLAS id ’Default kasklassen’

nm ’<500 m3/ha’ ’ 500-1000’ ’1000-1500’ ’1500-2000’ ’2000-2500’ ’2500-3000’ ’3000-4000’

’4000-5000’ ’5000-6000’ ’>6000-m3/ha’

mxstorage 0 500 1000 1500 2000 2500 3000 4000 5000 6000

mxdepth 0. 0.5 1. 1.5 2. 2.5 3. 4. 5. 5.

evap 0 0 0 1 1 1 1 1 1 1

KLAS

with

id	greenhouse class definition id
nm	names of greenhouse classes
mxstorage	maximum storage per greenhouse class (m3/ha)
mxdepth	maximum depth per greenhouse class (m)
evap	0=no evaporation, 1=evaporation from greenhouse class storage basins (per class)

Optional: New format greenhouse initialisation file <..\fixed\3B\NewKasInitData.dat>
This file contains the greenhouse initialisation data in a new format, which allows extensions by the user. The file contains INIT records. The INIT record contains a time table of greenhouse initial free storage. The table is a standard SOBEK time table with options for interpolation and periodicity. The file can contain multiple INIT records; selection of the actual record used is based on the selected greenhouse definition from the greenhouse class file.

An example input file:

INIT id ’Default kas initialisation data’ PDIN 1 0 pdin TBLE
’1951/01/01;00:00:00’ 0 0 0 0 31 100 215 519 829 1078 <
’1951/01/02;00:00:00’ 0 0 0 0 8 77 191 495 803 1052 <
’1951/01/03;00:00:00’ 0 5 5 5 13 82 196 500 808 1057 <
’1951/01/04;00:00:00’ 0 0 0 0 0 67 181 484 792 1040 <
’1951/01/05;00:00:00’ 0 0 0 0 0 5 118 419 725 971 <
’1951/01/06;00:00:00’ 0 0 0 0 0 0 108 409 715 960 <
’1951/01/07;00:00:00’ 0 3 3 3 3 3 112 412 718 964 <
’1951/01/08;00:00:00’ 0 0 0 0 0 0 85 384 689 934 <
’1951/01/09;00:00:00’ 0 0 0 0 0 0 63 362 666 910 <
’1951/01/10;00:00:00’ 0 0 0 0 0 0 32 330 633 876 <
’1951/01/11;00:00:00’ 0 0 0 0 0 0 0 204 503 743 <
’1951/01/12;00:00:00’ 0 0 0 0 0 0 0 181 479 718 <
’1951/01/13;00:00:00’ 0 0 0 0 0 0 0 152 449 687 <
’1951/01/14;00:00:00’ 0 0 0 0 0 0 0 127 424 661 <
’1951/01/15;00:00:00’ 0 0 0 0 0 0 0 86 382 617 <
’1951/01/16;00:00:00’ 0 8 8 8 8 8 8 95 390 626 <
’1951/01/17;00:00:00’ 0 0 0 0 0 0 0 0 212 442 <
’1951/01/18;00:00:00’ 0 0 0 0 0 0 0 0 98 325 <
’1951/01/19;00:00:00’ 0 0 0 0 0 0 0 0 60 286 <
’1951/01/20;00:00:00’ 0 0 0 0 0 0 0 0 34 259 <
’1951/01/21;00:00:00’ 0 0 0 0 0 0 0 0 0 219 <
’1951/01/22;00:00:00’ 0 0 0 0 0 0 0 0 0 209 <
...
...
’1995/10/10;00:00:00’ 0 125 361 799 1216 1668 2135 2994 3346 3419 <

```
'1995/10/11;00:00:00' 0 135 370 808 1226 1677 2145 3004 3356 3429 <
'1995/10/12;00:00:00' 0 150 385 823 1241 1693 2161 3020 3372 3446 <
'1995/10/13;00:00:00' 0 159 394 833 1250 1702 2170 3030 3382 3456 <
'1995/10/14;00:00:00' 0 176 412 851 1269 1721 2189 3049 3402 3476 <
'1995/10/15;00:00:00' 0 190 426 865 1283 1735 2204 3064 3417 3492 <
'1995/10/16;00:00:00' 0 203 439 878 1297 1749 2218 3079 3432 3507 <
'1995/10/17;00:00:00' 0 212 448 887 1305 1758 2227 3088 3441 3516 <
'1995/10/18;00:00:00' 0 224 460 899 1318 1770 2239 3100 3454 3530 <
'1995/10/19;00:00:00' 0 237 473 913 1331 1784 2253 3115 3469 3544 <
'1995/10/20;00:00:00' 0 240 477 916 1334 1787 2257 3118 3472 3548 <
'1995/10/21;00:00:00' 0 257 494 933 1352 1805 2274 3136 3491 3567 <
'1995/10/22;00:00:00' 0 274 511 951 1370 1823 2293 3155 3510 3586 <
'1995/10/23;00:00:00' 0 292 529 969 1388 1841 2311 3174 3529 3606 <
'1995/10/24;00:00:00' 0 309 546 987 1406 1860 2330 3193 3549 3626 <
'1995/10/25;00:00:00' 0 324 562 1002 1422 1876 2346 3209 3565 3643 <
'1995/10/26;00:00:00' 0 324 562 1002 1421 1875 2345 3209 3565 3642 <
'1995/10/27;00:00:00' 0 310 547 987 1406 1860 2330 3192 3548 3625 <
'1995/10/28;00:00:00' 0 325 562 1002 1422 1876 2346 3209 3564 3642 <
'1995/10/29;00:00:00' 0 342 579 1019 1439 1893 2363 3226 3583 3660 <
'1995/10/30;00:00:00' 0 357 594 1035 1455 1909 2379 3243 3599 3677 <
'1995/10/31;00:00:00' 0 371 609 1050 1470 1924 2395 3259 3616 3694 <
'1995/11/01;00:00:00' 0 275 511 950 1368 1821 2290 3150 3504 3579 <
'1995/11/02;00:00:00' 0 254 490 928 1346 1798 2267 3127 3480 3554 <
'1995/11/03;00:00:00' 0 262 498 936 1354 1806 2275 3135 3488 3562 <
'1995/11/04;00:00:00' 0 274 510 949 1367 1819 2288 3148 3502 3576 <
'1995/11/05;00:00:00' 0 289 525 964 1382 1834 2303 3164 3517 3592 <
'1995/11/06;00:00:00' 0 299 535 974 1392 1845 2314 3175 3528 3603 <
tble
init
INIT id 'Low kas initialisation data' PDIN 1 0 pdin TBLE
'1951/01/01;00:00:00' 0. 0. 0. 0. 0. 24. 74. 223. 322. 372. <
'1951/01/02;00:00:00' 0. 0. 0. 0. 0. 0. 45. 193. 292. 341. <
'1951/01/03;00:00:00' 0. 0. 0. 0. 0. 0. 44. 192. 291. 340. <
'1951/01/04;00:00:00' 0. 0. 0. 0. 0. 0. 24. 172. 270. 318. <
'1951/01/05;00:00:00' 0. 0. 0. 0. 0. 0. 0. 101. 197. 243. <
'1951/01/06;00:00:00' 0. 0. 0. 0. 0. 0. 0. 86. 182. 227. <
'1951/01/07;00:00:00' 0. 0. 0. 0. 0. 0. 0. 84. 179. 225. <
'1951/01/08;00:00:00' 0. 0. 0. 0. 0. 0. 0. 50. 145. 190. <
'1951/01/09;00:00:00' 0. 0. 0. 0. 0. 0. 0. 21. 115. 159. <
'1951/01/10;00:00:00' 0. 0. 0. 0. 0. 0. 0. 0. 77. 120. <
'1951/01/11;00:00:00' 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. <
'1951/01/12;00:00:00' 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. <
'1951/01/13;00:00:00' 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. <
...
...
'1995/10/10;00:00:00' 0. 72. 73. 408. 842. 1334. 1629. 2348. 2772. 2893. <
'1995/10/11;00:00:00' 0. 76. 78. 412. 847. 1339. 1634. 2353. 2778. 2898. <
'1995/10/12;00:00:00' 0. 83. 85. 420. 855. 1347. 1642. 2361. 2786. 2907. <
'1995/10/13;00:00:00' 0. 88. 89. 424. 859. 1351. 1646. 2366. 2791. 2912. <
'1995/10/14;00:00:00' 0. 96. 98. 433. 868. 1360. 1656. 2376. 2802. 2923. <
'1995/10/15;00:00:00' 0. 102. 105. 440. 875. 1368. 1663. 2384. 2810. 2932. <
'1995/10/16;00:00:00' 0. 109. 111. 446. 882. 1375. 1670. 2391. 2817. 2940. <
'1995/10/17;00:00:00' 0. 111. 113. 448. 884. 1377. 1673. 2394. 2820. 2943. <
'1995/10/18;00:00:00' 0. 116. 119. 454. 890. 1383. 1679. 2400. 2827. 2950. <
```

'1995/10/19;00:00:00' 0. 122. 125. 461. 897. 1390. 1686. 2407. 2834. 2957. <
'1995/10/20;00:00:00' 0. 122. 124. 460. 896. 1389. 1685. 2406. 2834. 2957. <
'1995/10/21;00:00:00' 0. 126. 129. 464. 900. 1394. 1690. 2412. 2839. 2963. <
'1995/10/22;00:00:00' 0. 130. 133. 469. 905. 1399. 1695. 2418. 2845. 2969. <
'1995/10/23;00:00:00' 0. 134. 138. 474. 910. 1404. 1701. 2423. 2852. 2976. <
'1995/10/24;00:00:00' 0. 139. 143. 479. 916. 1410. 1707. 2430. 2859. 2983. <
'1995/10/25;00:00:00' 0. 143. 147. 483. 920. 1415. 1712. 2435. 2864. 2989. <
'1995/10/26;00:00:00' 0. 137. 140. 477. 914. 1408. 1705. 2428. 2857. 2982. <
'1995/10/27;00:00:00' 0. 116. 119. 456. 892. 1386. 1683. 2406. 2834. 2958. <
'1995/10/28;00:00:00' 0. 120. 123. 459. 896. 1390. 1687. 2410. 2839. 2964. <
'1995/10/29;00:00:00' 0. 124. 127. 464. 901. 1395. 1692. 2416. 2845. 2970. <
'1995/10/30;00:00:00' 0. 127. 131. 468. 905. 1399. 1697. 2420. 2850. 2975. <
'1995/10/31;00:00:00' 0. 131. 135. 472. 909. 1404. 1701. 2425. 2855. 2981. <
'1995/11/01;00:00:00' 0. 29. 32. 367. 802. 1295. 1591. 2312. 2739. 2861. <
'1995/11/02;00:00:00' 0. 0. 2. 337. 772. 1265. 1560. 2280. 2706. 2827. <
'1995/11/03;00:00:00' 0. 0. 1. 336. 772. 1264. 1560. 2280. 2706. 2827. <
'1995/11/04;00:00:00' 0. 1. 3. 338. 773. 1266. 1562. 2282. 2708. 2830. <
'1995/11/05;00:00:00' 0. 3. 5. 340. 776. 1269. 1564. 2285. 2711. 2834. <
'1995/11/06;00:00:00' 0. 4. 6. 341. 777. 1270. 1566. 2287. 2713. 2836. <
tble
init
INIT id 'Periodic kas initialisation data' PDIN 1 1 '31536000' pdin TBLE
'1994/01/01;00:00:00' 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. <
'1994/02/01;00:00:00' 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. <
'1994/03/01;00:00:00' 0. 10. 10. 10. 10. 10. 10. 10. 10. 10. <
'1994/04/01;00:00:00' 0. 20. 20. 20. 20. 20. 20. 20. 20. 20. <
'1994/05/01;00:00:00' 0. 30. 30. 30. 30. 30. 30. 30. 30. 30. <
'1994/06/01;00:00:00' 0. 40. 40. 40. 40. 40. 40. 40. 40. 40. <
'1994/07/01;00:00:00' 0. 50. 50. 50. 50. 50. 50. 50. 50. 50. <
'1994/08/01;00:00:00' 0. 50. 50. 50. 50. 50. 50. 50. 50. 50. <
'1994/09/01;00:00:00' 0. 40. 40. 40. 40. 40. 40. 40. 40. 40. <
'1994/10/01;00:00:00' 0. 30. 30. 30. 30. 30. 30. 30. 30. 30. <
'1994/11/01;00:00:00' 0. 20. 20. 20. 20. 20. 20. 20. 20. 20. <
'1994/12/01;00:00:00' 0. 10. 10. 10. 10. 10. 10. 10. 10. 10. <
'1994/12/31;23:59:00' 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. <
tble
init

Optional: New format greenhouse water use file <..\fixed\3B\NewKasGebrData.dat>
This file contains the greenhouse water use data in a new format, which allows extensions by the user. The file can contain several GEBR records, in which typical water uses of greenhouses are specified in a standard SOBEK time table. Selection of the GEBR record to be used is done by the selection of the greenhouse definition from the greenhouse class file. An example input file is given below.

GEBR id 'Default kasgebruik data' PDIN 1 0 pdin TBLE
'1951/01/01;00:00:00' 8.36 <
'1951/01/02;00:00:00' 7.04 <
'1951/01/03;00:00:00' 8.04 <
'1951/01/04;00:00:00' 7.04 <
'1951/01/05;00:00:00' 7.04 <
'1951/01/06;00:00:00' 7.04 <
'1951/01/07;00:00:00' 7.72 <
'1951/01/08;00:00:00' 7.04 <

'1951/01/09;00:00:00' 9.43 <
'1951/01/10;00:00:00' 7.53 <
'1951/01/11;00:00:00' 7.31 <
'1951/01/12;00:00:00' 10.94 <
'1951/01/13;00:00:00' 7.31 <
'1951/01/14;00:00:00' 7.51 <
'1951/01/15;00:00:00' 9.81 <
'1951/01/16;00:00:00' 10.61 <
'1951/01/17;00:00:00' 7.31 <
'1951/01/18;00:00:00' 9.17 <
'1951/01/19;00:00:00' 7.31 <
'1951/01/20;00:00:00' 7.31 <
...
...
'1995/11/10;00:00:00' 9.88 <
'1995/11/11;00:00:00' 12.61 <
'1995/11/12;00:00:00' 12.88 <
'1995/11/13;00:00:00' 12.18 <
'1995/11/14;00:00:00' 9.65 <
'1995/11/15;00:00:00' 7.41 <
'1995/11/16;00:00:00' 7.81 <
'1995/11/17;00:00:00' 6.70 <
'1995/11/18;00:00:00' 8.21 <
'1995/11/19;00:00:00' 7.15 <
'1995/11/20;00:00:00' 13.20 <
tble
gebr
GEBR id 'Low kasgebruik data' PDIN 1 0 pdin TBLE
'1951/01/01;00:00:00' 2.17 <
'1951/01/02;00:00:00' 1.84 <
'1951/01/03;00:00:00' 2.09 <
'1951/01/04;00:00:00' 1.84 <
'1951/01/05;00:00:00' 1.84 <
'1951/01/06;00:00:00' 1.84 <
'1951/01/07;00:00:00' 2.01 <
'1951/01/08;00:00:00' 1.84 <
'1951/01/09;00:00:00' 2.45 <
'1951/01/10;00:00:00' 1.96 <
'1951/01/11;00:00:00' 2.67 <
'1951/01/12;00:00:00' 3.97 <
...
...
'1995/12/07;00:00:00' 2.11 <
'1995/12/08;00:00:00' 2.86 <
'1995/12/09;00:00:00' 2.24 <
'1995/12/10;00:00:00' 1.82 <
'1995/12/11;00:00:00' 1.65 <
'1995/12/12;00:00:00' 1.94 <
'1995/12/13;00:00:00' 1.73 <
'1995/12/14;00:00:00' 2.23 <
'1995/12/15;00:00:00' 1.93 <
'1995/12/16;00:00:00' 2.38 <
'1995/12/17;00:00:00' 2.11 <
'1995/12/18;00:00:00' 1.56 <

'1995/12/19;00:00:00' 1.48 <
'1995/12/20;00:00:00' 2.18 <
'1995/12/21;00:00:00' 1.69 <
'1995/12/22;00:00:00' 1.43 <
'1995/12/23;00:00:00' 1.60 <
'1995/12/24;00:00:00' 2.75 <
'1995/12/25;00:00:00' 2.42 <
'1995/12/26;00:00:00' 1.83 <
'1995/12/27;00:00:00' 3.30 <
'1995/12/28;00:00:00' 3.79 <
'1995/12/29;00:00:00' 3.16 <
'1995/12/30;00:00:00' 2.56 <
'1995/12/31;00:00:00' 1.61 <
tble
gebr
GEBR id 'Periodic kasgebruik data' PDIN 1 1 '31536000' pdin TBLE
'1995/01/01;00:00:00' 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 <
'1995/02/01;00:00:00' 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 <
'1995/03/01;00:00:00' 5.00 5.00 5.00 5.00 5.00 5.00 5.00 5.00 5.00 5.00 <
'1995/05/01;00:00:00' 6.00 6.00 6.00 6.00 6.00 6.00 6.00 6.00 6.00 6.00 <
'1995/07/01;00:00:00' 7.00 7.00 7.00 7.00 7.00 7.00 7.00 7.00 7.00 7.00 <
'1995/09/01;00:00:00' 6.00 6.00 6.00 6.00 6.00 6.00 6.00 6.00 6.00 6.00 <
'1995/10/01;00:00:00' 5.00 5.00 5.00 5.00 5.00 5.00 5.00 5.00 5.00 5.00 <
'1995/11/15;00:00:00' 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 <
'1995/12/31;23:59:00' 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 2.00 <
tble
gebr

Optional: New format crop data <**..\fixed\3B\NewCropData.dat**> This file contains the crop data (names and crop factors) in a new format, which allows extensions by the user. The file contains 3 type of records:

- **CRPD records,** containing the general crop definitions (id, name, number of crops, and references to the crop name definition and crop factor definition)
- **NAME records,** containing the crop name definitions.
- **CRF records**, containing a time table with crop factors**.**

An example file is given below,

with

id	id
nm	name
nc	number of crops
cn	crop name definition
cf	crop factor definition

PDIN pdin = as in other SOBEK time tables: interpolation and periodicity options

TBLE tble = as in other SOBEK time tables

BBB2.2

CRPD id 'Default' nm 'Nederlandse gewasdefinitie' nc 16 cn 'Dutch Crops' cf 'Dutch Crop

factors' crpd
CRPD id 'Example' nm 'Example crop definition' nc 1 cn 'Example Crops' cf 'Example Crop factors' crpd
CRPD id 'Taiwan' nm 'Taiwan crop definition' nc 7 cn 'Taiwan Crops' cf 'Taiwan Crop factors' crpd
NAME id 'Dutch Crops' nm
'grass '
'corn '
'potatoes '
'sugarbeet'
'grain '
'miscellaneous '
'non-arable land'
'greenhouse area'
'orchard '
'bulbous plants'
'foliage forest'
'pine forest '
'nature '
'fallow '
'vegetables '
'flowers '
name
NAME id 'Taiwan Crops' nm 'paddy1' 'paddy2' 'fishpond' 'upland1' 'upland2' 'upland3' 'fallow' name
NAME id 'Example Crops' nm 'all crops' name
CRF id 'Dutch Crop factors' nm 'Dutch Crop factors' PDIN 1 1 '31536000' pdin TBLE
'1994/01/01;00:00:00' 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 <
'1994/01/11;00:00:00' 0.35 0.40 0.40 0.40 0.40 0.40 0.40 0.00 0.35 0.40 0.35 0.40 0.35 0.40 0.35 0.00 <
'1994/01/21;00:00:00' 0.49 0.56 0.56 0.56 0.56 0.56 0.56 0.00 0.49 0.56 0.49 0.56 0.49 0.56 0.49 0.00 <
'1994/02/01;00:00:00' 0.56 0.64 0.64 0.64 0.64 0.64 0.64 0.00 0.56 0.64 0.56 0.64 0.56 0.64 0.56 0.00 <
'1994/02/11;00:00:00' 0.70 0.80 0.80 0.80 0.80 0.80 0.80 0.00 0.70 0.80 0.70 0.80 0.70 0.80 0.70 0.00 <
'1994/02/21;00:00:00' 0.70 0.70 0.70 0.70 0.70 0.80 0.70 0.00 0.70 0.70 0.70 0.80 0.70 0.80 0.70 0.00 <
'1994/03/01;00:00:00' 0.84 0.72 0.72 0.72 0.72 0.96 0.72 0.00 0.84 0.48 0.84 0.96 0.84 0.96 0.84 0.00 <
'1994/03/11;00:00:00' 0.91 0.65 0.65 0.65 0.65 1.04 0.65 0.00 0.91 0.65 0.91 1.04 0.91 1.04 0.91 0.00 <
'1994/03/21;00:00:00' 0.91 0.52 0.52 0.52 0.52 1.04 0.52 0.00 0.91 0.78 0.91 1.04 0.91 1.04 0.91 0.00 <
'1994/04/01;00:00:00' 1.04 0.52 0.52 0.52 0.52 1.04 0.52 0.00 1.04 0.91 1.04 1.04 1.04 1.04 1.04 0.00 <
'1994/04/11;00:00:00' 1.04 0.39 0.39 0.39 0.65 1.04 0.39 0.00 1.04 0.91 1.04 1.04 1.04 1.04 1.04 0.00 <
'1994/04/21;00:00:00' 1.04 0.26 0.26 0.26 0.78 1.04 0.26 0.00 1.04 0.91 1.04 1.04 1.04 1.04 1.04 0.00 <
'1994/05/01;00:00:00' 1.04 0.52 0.52 0.52 0.91 1.04 0.13 0.00 1.04 1.04 1.04 1.04 1.04 1.04 1.04 0.00 <
'1994/05/11;00:00:00' 1.04 0.65 0.65 0.65 1.04 1.04 0.13 0.00 1.04 1.04 1.04 1.04 1.04 1.04

1.04 0.00 <
'1994/05/21;00:00:00' 1.04 0.78 0.78 0.78 1.17 1.04 0.13 0.00 1.04 1.04 1.04 1.04 1.04 1.04
1.04 0.00 <
'1994/06/01;00:00:00' 1.05 0.92 0.92 0.92 1.31 1.05 0.13 0.00 1.05 1.05 1.05 1.05 1.05 1.05
1.05 0.00 <
'1994/06/11;00:00:00' 1.05 1.05 1.05 1.05 1.31 1.05 0.13 0.00 1.05 1.05 1.05 1.05 1.05 1.05
1.05 0.00 <
'1994/06/21;00:00:00' 1.05 1.18 1.18 1.18 1.18 1.05 0.13 0.00 1.05 0.92 1.05 1.05 1.05 1.05
1.05 0.00 <
'1994/07/01;00:00:00' 1.03 1.16 1.16 1.16 1.16 1.03 0.13 0.00 1.03 0.77 1.03 1.03 1.03 1.03
1.03 0.00 <
'1994/07/11;00:00:00' 1.02 1.14 1.14 1.14 1.14 1.02 0.13 0.00 1.02 0.64 1.02 1.02 1.02 1.02
1.02 0.00 <
'1994/07/21;00:00:00' 0.99 1.12 1.12 1.12 0.99 0.99 0.12 0.00 0.99 0.50 0.99 0.99 0.99 0.99
0.99 0.00 <
'1994/08/01;00:00:00' 0.97 1.09 1.09 1.09 0.73 0.97 0.12 0.00 0.97 0.12 0.97 0.97 0.97 0.97
0.97 0.00 <
'1994/08/11;00:00:00' 0.95 1.07 0.95 1.07 0.36 0.95 0.12 0.00 0.95 0.12 0.95 0.95 0.95 0.95
0.95 0.00 <
'1994/08/21;00:00:00' 0.94 1.06 0.83 1.06 0.24 0.94 0.12 0.00 0.94 0.12 0.94 0.94 0.94 0.94
0.94 0.00 <
'1994/09/01;00:00:00' 0.94 1.05 0.70 1.05 0.23 0.94 0.23 0.00 0.94 0.23 0.94 0.94 0.94 0.94
0.94 0.00 <
'1994/09/11;00:00:00' 0.94 1.05 0.59 1.05 0.35 0.94 0.35 0.00 0.94 0.35 0.94 0.94 0.94 0.94
0.94 0.00 <
'1994/09/21;00:00:00' 0.94 1.05 0.47 1.05 0.47 0.94 0.47 0.00 0.94 0.47 0.94 0.94 0.94 0.94
0.94 0.00 <
'1994/10/01;00:00:00' 0.70 0.40 0.40 0.40 0.40 0.80 0.40 0.00 0.70 0.40 0.70 0.80 0.70 0.80
0.70 0.00 <
'1994/10/11;00:00:00' 0.63 0.45 0.45 0.45 0.45 0.72 0.45 0.00 0.63 0.45 0.63 0.72 0.63 0.72
0.63 0.00 <
'1994/10/21;00:00:00' 0.56 0.48 0.48 0.48 0.48 0.64 0.48 0.00 0.56 0.48 0.56 0.64 0.56 0.64
0.56 0.00 <
'1994/11/01;00:00:00' 0.56 0.56 0.56 0.56 0.56 0.64 0.56 0.00 0.56 0.56 0.56 0.64 0.56 0.64
0.56 0.00 <
'1994/11/11;00:00:00' 0.49 0.56 0.56 0.56 0.56 0.56 0.56 0.00 0.49 0.56 0.49 0.56 0.49 0.56
0.49 0.00 <
'1994/11/21;00:00:00' 0.42 0.48 0.48 0.48 0.48 0.48 0.48 0.00 0.42 0.48 0.42 0.48 0.42 0.48
0.42 0.00 <
'1994/12/01;00:00:00' 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00
0.00 0.00 <
'1994/12/31;23:59:00' 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.00
0.00 0.00 <
tble
crf
CRF id 'Taiwan Crop factors' nm 'Taiwan Crop factors' PDIN 1 1 '31536000' pdin TBLE
'1994/01/01;00:00:00' 0.60 0.00 1.20 0.20 0.00 0.80 0.40 <
'1994/01/11;00:00:00' 0.80 0.60 1.20 0.20 0.40 1.00 0.40 <
'1994/01/21;00:00:00' 1.00 0.80 1.20 0.20 0.40 1.10 0.40 <
'1994/02/01;00:00:00' 1.10 1.00 1.20 0.50 0.60 0.00 0.40 <
'1994/02/11;00:00:00' 1.00 1.10 1.20 1.10 0.80 0.00 0.40 <
'1994/02/21;00:00:00' 1.10 1.10 1.20 1.10 1.00 0.00 0.40 <
'1994/03/01;00:00:00' 1.15 1.10 1.20 1.10 1.10 0.00 0.40 <
'1994/03/11;00:00:00' 1.15 1.15 1.20 0.90 1.10 0.00 0.40 <

’1994/03/21;00:00:00’ 1.15 1.15 1.20 0.90 0.80 0.00 0.40 <
’1994/04/01;00:00:00’ 1.00 1.00 1.20 0.70 0.80 0.00 0.40 <
’1994/04/11;00:00:00’ 1.00 1.00 1.20 0.60 0.60 0.00 0.40 <
’1994/04/21;00:00:00’ 1.00 0.80 1.20 0.60 0.60 0.00 0.40 <
’1994/05/01;00:00:00’ 1.00 0.80 1.20 0.40 0.60 0.00 0.40 <
’1994/05/11;00:00:00’ 0.80 0.60 1.20 0.40 0.40 0.00 0.40 <
’1994/05/21;00:00:00’ 0.80 .04 1.20 0.40 0.00 0.00 0.40 <
’1994/06/01;00:00:00’ 0.80 .05 1.20 0.40 0.00 0.00 0.40 <
’1994/06/11;00:00:00’ 0.80 .00 1.20 0.40 0.00 0.00 0.40 <
’1994/06/21;00:00:00’ 0.80 .00 1.20 0.40 0.00 0.00 0.40 <
’1994/07/01;00:00:00’ .00 .00 1.20 0.00 0.00 0.00 0.40 <
’1994/12/31;23:59:00’ .00 .00 1.20 0.00 0.00 0.60 0.40 <
tble
crf
CRF id ’Example Crop factors’ nm ’Example Crop factors’ PDIN 1 1 ’31536000’ pdin TBLE
’1994/01/01;00:00:00’ 0.90 <
’1994/06/11;00:00:00’ 0.90 <
’1994/12/31;23:59:00’ 0.90 <
tble
crf

Optional: New format open water crop factor data <..\fixed\3B\NewCropOWData.dat>
This file contains the open water crop factors in a new format, which allows extensions by the user. This file is similar to the new format crop data file, but somewhat simpler because there is only 1 ‘crop’, i.e. open water. There are two type of records, the CROW and the CRFO records. The file can contain several CROW and CRFO records. Selection of the actual records used is done by using the options NewFormatCropFactors=-1 and OpenWaterCropDefinition= in the Delft_3B.Ini file. An example input file:

CROW ’Default’ nm ’Nederlandse openwaterverdamping’ nc 1 cn ’Open water Netherlands’ cf ’Dutch Open water crop factors’ crow
CROW id ’Example’ nm ’Example open water evaporation’ nc 1 cn ’Open water Example’ cf ’Example Open water crop factors’ crow
CROW id ’Taiwan’ nm ’Taiwan open water evaporation’ nc 1 cn ’Open water Taiwan’ cf ’Taiwan Open water crop factors’ crow
CRFO id ’Dutch Open water crop factors’ nm ’Dutch Open water Crop factors’ PDIN 1 1 ’31536000’ pdin TBLE
’1994/01/01;00:00:00’ 0.00 <
’1994/01/11;00:00:00’ 0.50 <
’1994/01/21;00:00:00’ 0.70 <
’1994/02/01;00:00:00’ 0.80 <
’1994/02/11;00:00:00’ 1.00 <
’1994/02/21;00:00:00’ 1.00 <
’1994/03/01;00:00:00’ 1.20 <
’1994/03/11;00:00:00’ 1.30 <
’1994/05/21;00:00:00’ 1.30 <
’1994/06/01;00:00:00’ 1.31 <
’1994/06/21;00:00:00’ 1.31 <
’1994/07/01;00:00:00’ 1.29 <
’1994/07/11;00:00:00’ 1.27 <
’1994/07/21;00:00:00’ 1.24 <
’1994/08/01;00:00:00’ 1.21 <
’1994/08/11;00:00:00’ 1.19 <
’1994/08/21;00:00:00’ 1.18 <

'1994/09/01;00:00:00' 1.17 <
'1994/09/21;00:00:00' 1.17 <
'1994/10/01;00:00:00' 1.00 <
'1994/10/11;00:00:00' 0.90 <
'1994/10/21;00:00:00' 0.80 <
'1994/11/01;00:00:00' 0.80 <
'1994/11/11;00:00:00' 0.70 <
'1994/11/21;00:00:00' 0.60 <
'1994/12/01;00:00:00' 0.00 <
'1994/12/31;23:59:00' 0.00 <
tble
crfo
CRFO id 'Taiwan Open water crop factors' nm 'Taiwan Open water Crop factors' PDIN 1 1 '31536000' pdin TBLE
'1994/01/01;00:00:00' 1.00 <
'1994/02/01;00:00:00' 1.05 <
'1994/03/01;00:00:00' 1.10 <
'1994/04/01;00:00:00' 1.15 <
'1994/05/01;00:00:00' 1.10 <
'1994/06/01;00:00:00' 1.05 <
'1994/07/01;00:00:00' 1.00 <
'1994/08/01;00:00:00' 0.95 <
'1994/09/01;00:00:00' 0.95 <
'1994/10/01;00:00:00' 1.00 <
'1994/12/31;23:59:00' 1.00 <
tble
crfo
CRFO id 'Example Open water crop factors' nm 'Example Open water Crop factors' PDIN 1 1 '31536000' pdin TBLE
'1994/01/01;00:00:00' 1.2 <
'1994/12/31;23:59:00' 1.2 <
tble
crfo

Optional: New format soil data <..\fixed\3B\NewSoilData.dat>

This file contains the soil data in a new format, which allows extensions by the user. The file contains several type of records, and is used to specify data on the soil types (both with or without Capsim).

- **SLDF records:** containing general definitions and references to other records.
- **NAME records**, containing the names of the soil types (both with or without Capsim) (the names are actually only relevant for the user interface)
- **SDEF records**, containing the drainage depths (for use without Capsim)
- **STAB records**, containing the tables with storage coefficients for use without Capsim; values are specified for the number of specified soil types (from the SLDF record) and the number of drainage depths (from the STAB record)
- **SCNV records**, containing conversion tables from soil types to use without Capsim to Capsim soil types.

An example is given below.

SLDF id 'Default' nm 'Dutch default soils no Capsim' ns 12 sn 'Default soil names' nd 12 sd 'Default soil depths' st 'Default table soil storage coefficients' nsc 21 snc 'Capsim soil names' cv ’Default conversion’ sldf

SLDF id ’BergcoefARS4633’ nm ’Dutch default soils no Capsim’ ns 12 sn ’Default soil names’ nd 12 sd ’Default soil depths’ st ’Soil storage coefficients ARS4633’ nsc 21 snc ’Capsim soil names’ cv ’Default conversion’ sldf

with

id	id of soil definition
nm	name of soil definition
ns	number of soil types, no Capsim
nsc	aantal bodemtypen, Capsim
sn	soil names definition, no Capsim
snc	soil names definition Capsim
nd	number of drainage depths in table SDEF record
st	soil storage coefficient table for ns soil types and sd drainage depths (no Capsim)
cv	id conversion tables

NAME id ’Default soil names’ nm

’loamy, humous fine sand (μ=0.115 per m)’
’peat (μ=0.103 per m)’
’heavy clay (μ=0.089 per m)’
’humous clay and peat (μ=0.085 per m)’
’light loamy, medium coarse sand (μ=0.084 per m)’
’loamy silt (μ=0.072 per m)’
’humous clay and peat with silty top layer (μ=0.069 per m)’
’clay and light clay (μ=0.062 per m)’
’loamless, medium coarse and coarse sand (μ=0.060 per m)’
’silt (μ=0.058 per m)’
’very light clay (μ=0.048 per m)’
’sand with a silty top layer (μ=0.044 per m)’

name

NAME id ’Capsim soil names’ nm
’Veengrond met veraarde bovengrond’
’Veengrond met veraarde bovengrond, zand’
’Veengrond met kleidek’
’Veengrond met kleidek op zand’
’Veengrond met zanddek op zand’
’Veengrond op ongerijpte klei’
’Stuifzand’
’Podzol (Leemarm, fijn zand)’
’Podzol (zwak lemig, fijn zand)’
’Podzol (zwak lemig, fijn zand op grof zand)’
’Podzol (lemig keileem)’
’Enkeerd (zwak lemig, fijn zand)’
’Beekeerd (lemig fijn zand)’
’Podzol (grof zand)’
’Zavel’
’Lichte klei’

’Zware klei’
’Klei op veen’
’Klei op zand’
’Klei op grof zand’
’Leem’
name

with

id	name definition id
nm	crop names for all crops (the number of names should match with the number specified in the SLDF record for this name definition id)

SDEF id ’Default soil depths’ dp -0.1 -0.2 -0.3 -0.4 -0.5 -0.6 -0.7 -0.8 -0.9 -1.0 -1.2 -1.5 SDEF

with

id	soil drainage depth definition id
dp	drainage depths (corresponding with nd drainage depths as specified in the SLDF record)

STAB id ’Default table soil storage coefficients’ nm ’Default ns*nd storage coefficient values’ sc
0.0050 0.0080 0.0230 0.0070 0.0030 0.0070 0.0060 0.0130 0.0020 0.0050 0.0070 0.0030
0.0145 0.0195 0.0385 0.0175 0.0100 0.0155 0.0150 0.0235 0.0085 0.0125 0.0140 0.0080
0.0243 0.0313 0.0497 0.0273 0.0190 0.0243 0.0243 0.0310 0.0157 0.0200 0.0200 0.0130
0.0342 0.0433 0.0585 0.0370 0.0290 0.0325 0.0338 0.0375 0.0225 0.0268 0.0252 0.0178
0.0486 0.0544 0.0654 0.0462 0.0390 0.0400 0.0414 0.0430 0.0292 0.0330 0.0300 0.0226
0.0635 0.0652 0.0713 0.0548 0.0490 0.0472 0.0480 0.0477 0.0360 0.0390 0.0342 0.0273
0.0777 0.0754 0.0766 0.0630 0.0580 0.0540 0.0540 0.0519 0.0424 0.0444 0.0380 0.0317
0.0910 0.0851 0.0810 0.0709 0.0678 0.0604 0.0596 0.0556 0.0486 0.0495 0.0415 0.0360
0.1034 0.0944 0.0850 0.0783 0.0763 0.0663 0.0649 0.0590 0.0547 0.0543 0.0447 0.0401
0.1148 0.1031 0.0887 0.0852 0.0845 0.0719 0.0697 0.0621 0.0605 0.0588 0.0476 0.0440
0.1350 0.1193 0.0950 0.0983 0.0992 0.0820 0.0786 0.0676 0.0713 0.0670 0.0529 0.0513
0.1595 0.1409 0.1027 0.1155 0.1182 0.0951 0.0902 0.0745 0.0858 0.0777 0.0597 0.0611
stab

STAB id ’Soil storage coefficients ARS4633’ nm ’Default ns*nd storage coefficient values’ sc
0.0074 0.0077 0.0094 0.0063 0.0046 0.0049 0.0049 0.0048 0.0027 0.0029 0.0025 0.0013
0.0183 0.0153 0.0166 0.0134 0.0124 0.0103 0.0118 0.0092 0.0098 0.0066 0.0052 0.0032
0.0302 0.0226 0.0226 0.0206 0.0220 0.0158 0.0181 0.0132 0.0162 0.0105 0.0079 0.0052
0.0419 0.0305 0.0278 0.0279 0.0323 0.0211 0.0237 0.0171 0.0228 0.0145 0.0107 0.0074
0.0532 0.0387 0.0323 0.0350 0.0427 0.0262 0.0289 0.0206 0.0294 0.0186 0.0134 0.0096
0.0664 0.0467 0.0363 0.0420 0.0528 0.0310 0.0339 0.0240 0.0359 0.0226 0.0160 0.0120
0.0805 0.0545 0.0399 0.0486 0.0625 0.0364 0.0386 0.0271 0.0422 0.0265 0.0185 0.0143
0.0938 0.0621 0.0431 0.0551 0.0715 0.0416 0.0430 0.0300 0.0484 0.0303 0.0210 0.0167
0.1061 0.0702 0.0461 0.0613 0.0801 0.0466 0.0472 0.0328 0.0542 0.0341 0.0235 0.0191
0.1173 0.0784 0.0489 0.0673 0.0880 0.0514 0.0512 0.0355 0.0598 0.0377 0.0258 0.0214
0.1372 0.0939 0.0538 0.0786 0.1024 0.0605 0.0586 0.0404 0.0704 0.0446 0.0303 0.0261
0.1614 0.1158 0.0600 0.0941 0.1208 0.0729 0.0685 0.0470 0.0845 0.0541 0.0366 0.0329
stab

with

id	soil storage coefficient table id
nm	name
sc	storage coefficient vales (for nd drainage depths and ns soil types)

SCNV id 'Default conversion' nocap 'Default soil names' cap 'Capsim soil names'

c1 10 1 17 18 9 11 6 16 8 21 15 12 c2 2 2 7 7 7 7 9 9 1 1 1 12 1 5 11 8 3 4 12 12 6 scnv

with

id	soil conversion tabel id
nocap	soil names definitie zonder Capsim
cap	soil names definitie met Capsim
c1	conversion table of soil type (no Capsim) to soil type Capsim
c2	conversion table of soil type Capsim to soil type (no Capsim)

D.19.4 Greenhouse layer

Data file: Greenhse.3b

This file contains the data for the nodes of model type 3 (greenhouse or glasshouse area)

GRHS id '1' na 10 ar 1000. 0. 0. 3000. 0. 0. 0. 0. 0. 0. sl 1.0 as 0. sd 'roofstor_1mm' si 'silo_typ1' ms 'meteostat1' is 50.0 grhs

With

id	node identification
na	number or areas (default=10)
ar	area (in m2) as a table with areas for all greenhouse classes (na values)
as	greenhouse area connected to silo storage [m^2]
sl	surface level in m NAP
sd	storage definition on roofs
si	silo definition
ms	identification of the meteostation
is	initial salt concentration ([mg/l])

Roof storage file: Greenhse.rf

This file contains the roof storage definitions for the nodes of model type 3 (greenhouse area)

STDF id 'roofstor_1mm' nm 'roof_1mm' mk 1. ik 0. stdf

Where:

id	storage identification
nm	name (optional)
mk	maximum storage on roofs (in mm). Default 1 mm. Initial value is zero by default.
ik	initial storage on roofs (in mm). Default 0. **NOT READ. Default value zero used.**

Silo definition file: Greenhse.sil

This file contains the silo definitions for the nodes of model type 3 (greenhouse or glasshouse area)

SILO id 'silo_typ1' nm 'silo_200m3' sc 200.0 pc 0.2 silo

With:

id	silo identification
nm	name (optional)
sc	silo capacity (m3/ha). Default 0.
pc	silo pump capacity (m3/s). Default 0.

D.19.5 Industry layer

dat=industry.dat

This file contains the data for the nodes of model type 15: industrial demands and/or discharges.

INDU id '1' dm 'Qdemand' ds 'Qdis' rf 90. sc. 100. co 1 indu

With

id	node identification
dm	name of demand table (refer to tbl file)
ds	name of discharge table (refer to tbl file)
rf	return flow percentage
sc	salt concentration of discharge
co	computation option: discharge as a table, of a percentage of allocation 1 = discharge and salt concentration as a table (ds field) 2 = fixed percentage and fixed salt concentration (rf and sc fields)

Optional fields:

redow	reduction level open water specified as absolute level (0) or relative to target level (1)
redlv	level at open water node below which industrial abstraction is reduced
redbnlv	level at boundary node below which industrial abstraction is reduced
redrf	return flow reduction option (0=same ratio as demand reduction; otherwise no reduction of return flow)

tab=industry.tbl

Contains demand flow and discharge flow and salt concentration tables

DISC id 'Qdis' PDIN 1 1 '1;00:00:00' pdin
TBLE
1997/01/01;00:00:00 0.50 1000.0 <
1997/12/31;00:00:00 0.50 1000.0 <
tble
disc

DEMD id 'Qdemand' PDIN 1 1 '1;00:00:00' pdin
TBLE
1997/01/01;00:00:00 0.60 <
1997/12/31;00:00:00 0.60 <

tble
demd

With

id	table identification
nm	table name

PDIN ..pdin = period and interpolation method

0 0 ' ' = interpolation continuous, no period

1 0 ' ' = interpolation block, no period

0 1 '365;00:00:00' = interpolation continuous, period in DDD;HH:mm:ss

1 1 '1;00:00:00' = interpolation block, period in DDD;HH:mm:ss

After the date and time fields in the table of the DISC record, the first value is the discharge (m3/s), the second value is the salt concentration (g/m3).

The table of the DEMD records only contains a demand (m3/s).

D.19.6 NWRW layer

Data file: Pluvius.3B

This file contains the data for the nodes of model type 7 (NWRW rainfall-runoff nodes for Sobek-Urban)

NWRW id '1' sl 2.0 ar 1. 2. 3. 4. 5. 6. 7. 8. 9. 10. 11. 12. np 3 dw '125_lcd' ms 'meteostat1' nwrw

or

NWRW id '1' sl 2.0 na 2 aa 123 456 nw 'special1' 'special2' ar 1. 2. 3. 4. 5. 6. 7. 8. 9. 10. 11. 12. np 3 dw '125_lcd' ms 'meteostat1' nwrw

With

id	node identification
sl	surface level (in m) (optional input data)
ar	area (12 types) as combination of 3 kind of slopes (with a slope, flat, flat stretched) and 4 types of surfaces (closed paved, open paved, roofs, unpaved) a1=closed paved, with a slope a2=closed paved, flat a3=closed paved, flat stretched a4=open paved, with a slope .. a7=roofs, with a slope .. a10=unpaved, with a slope
np	number of people
dw	dry weather flow identification
ms	identification of the meteostation

na	number of special areas with special inflow characteristics
aa	special area in m2 (for number of areas as specified after the na keyword)
nw	reference to definition of special inflow characteristics (for na special areas)

Remarks:

- Number of areas (3 type of slopes, 4 types of surfaces, 12 possible combinations) is hard-coded.

- Salt concentrations are not implemented for this type of node.

DWA file: Pluvius.Dwa

DWA id ’125_lcd’ nm ’125_liters per day’ do 1 wc 12. wd 125. wh 1.5 1.5 1.5 1.5 1.5 3.0 4.0 5.0 6.0 6.5 7.5 8.5 7.5 6.5 6.0 5.0 5.0 5.0 4.0 3.5 3.0 2.5 2.0 2.0 dwa

id	dwa identification
nm	name
do	dry weather flow computation option 1 = nr. of people * constant DWA per capita per hour 2 = nr. of people * variable DWA per capita per hour 3 = 1 * constant DWA per hour 4 = 1 * variable DWA per hour 5 = using a table
wc	water use per capita as a constant value per hour (l/hour)
wd	water use per capita per day (l/day)
wh	water use per capita per hour (24 percentages, total should be 100%)
dt	dwa table id

NWRW table file: Pluvius.Tbl

This file contains the DWA discharges as a function of time. The lay-out is similar to other time tables.

DW_T id ’DWA_table’ nm ’DWA_table’ PDIN 1 0 ’ ’ pdin
TBLE
1997/01/01;00:00:00 0.5 <
1997/05/01;00;00;00 0.55 <
1997/10/01;00:00:00 0.5 <
1997/12/31;23:59:00 0.50 <
tble
dw_t

With

id	table identification
nm	name of table

PDIN ..pdin = period and interpolation method

0 0 ’ ’ = interpolation continuous, no period

1 0 ’ ’ = interpolation block, no period

0 1 ’365;00:00:00’ = interpolation continuous, period in DDD;HH:mm:ss

1 1 '1;00:00:00' = interpolation block, period in DDD;HH:mm:ss

TBLE .. tble contains the table, first column is the date (year month day hour minute second), second column the DWA discharge in m3/s.

General data: Pluvius.Alg

This file contains the general data for the nodes of model type 7 (NWRW rainfall-runoff nodes)

PLVG id '-1' nm ' ' ru 0.5 0.2 0.1 ms 0.0.5 1.0 0 0.5 1.0 0. 2. 4. 2. 4. 6. ix 0. 2.0 0. 5.0

im 0. 0.5 0. 1.0 ic 0.0 0.1 0.0 0.1 dc 0.0 3.0 0.0 3.0 od 0 or 0 plvg

With:

id	identification (default id –1; this record is the default general definition)
nm	name
ru	runoff-delay factor for 3 types of slopes (with a slope, flat, flat stretched)
ms	maximum storage (in mm per m2) for 12 types of area (see plv file)
ix	maximum infiltration capacities in mm/hour for 4 types of surface (closed paved, open paved, roofs, unpaved)
im	minimum infiltration capacities in mm/hour for 4 types of surface (closed paved, open paved, roofs, unpaved)
ic	decrease in infiltration capacity (1/hour) for 4 types of surface
dc	increase in infitration capacity (1/hour) for 4 types of surface
od	option for infiltration from depressions (1=yes, 0=no)
or	option for infiltration from runoff (1=yes, 0=no)

PLVA id 'Special1' nm 'Special1' ru .5 ms 1 ix 3 im 1 ic .5 dc .2 ef 1 od 1 or 0 plva

With:

id	identification definition for special inflow areas
nm	name
ru	runoff-delay factor
ms	maximum storage (in mm per m2)
ix	maximum infiltration capacities in mm/hour
im	minimum infiltration capacities in mm/hour
ic	decrease in infiltration capacity (1/hour)
dc	increase in infitration capacity (1/hour)
ef	evaporation factor (default=1.0)
od	option for infiltration from depressions (1=yes, 0=no)
or	option for infiltration from runoff (1=yes, 0=no)

D.19.7 Open water layer

Data file: Openwate.3b

This file contains the data for the nodes of model type 4 (open water area)

OPWA id ’1’ ml 0.0 rl 0.0 al 2 na 6 ar 10000. 110000. 120000. 130000 140000. 150000.

lv -1. -0.8 -0.6 -0.4 -0.2 0. bl -2.0 tl 0 -0.9 sp ’seep_1’ ms ’meteostat1’ is 75.0 opwa

With

id	node identification
al	area-level relation (only used by ModelEdit) 1 = constant area, 2=interpolation, 3=lineair
na	number of area/level combinations. Default 6. NOT READ.
ar	6 values of area (in m2)
lv	6 corresponding values of level (m NAP) in increasing order
ml	maximum allowable level (m NAP)
rl	reference level (m NAP)
bl	bottom level (m NAP) Default 1 meter below lowest value from area-level relation.
sp	seepage definition.
ms	identification of the meteostation
is	initial salt concentration (mg/l)
tl	target level; constant or reference to a table. tl 0 -0.9 = initial groundwater level as a constant, with value -0.9 m NAP. tl 1 ’Tlv-Table’ = target open water levels as a table, with table id Tlv-Table.

Seepage definition file: Openwate.sep

This file contains the seepage definitions for the nodes of model type 4 (open water)

SEEP id ’seep_1’ nm ’constant seepage_1mm’ co 1 sp 1. ss 500. seep
SEEP id ’seep_2’ nm ’simple variable seepage’ co 2 cv 3.0 h0 ‘H0Table’ ss 500. seep
SEEP id ’seep_3’ nm ’variable seepage using coupling with Modflow’ co 3 cv 3.0 ss 500. seep

With:

id	seepage identification
nm	name
co	computation option seepage 1 = constant seepage (default) 2 = variable seepage, using C and a table for H0 3 = variable seepage, using C and H0 from Modflow If the co field is missing, co 1 will be assumed.
sp	Seepage or percolation (mm/day) Positive numbers represent seepage, negative numbers represent percolation. Default 0.
ss	salt concentration seepage (mg/l). Default 500 mg/l. This value is only important for positive seepage values.
cv	Resistance value C for aquitard
h0	reference to a table with H0 values

Remark: File could be combined with similar file from unpaved layer.

Open water table file: Openwate.tbl

This file contains the tables for target levels and variable H0 values for the open water nodes.

Table input of the target open water level as function of time.

```
OW_T id 'targetlevel table1' PDIN 1 0 '1;00:00:00' pdin
TBLE
1995/01/01;00:00:00 0.3 <
1995/04/15;00:00:00 0.4 <
1995/09/14;23:00:00 0.4 <
1995/09/15;00:00:00 0.3 <
1995/12/31;23:59:00 0.3 <
tble
ow_t
```

With

id	table identification
nm	name of table

PDIN ..pdin = period and interpolation method

0 0 ' ' = interpolation continuous, no period

1 0 ' ' = interpolation block, no period

0 1 '365;00:00:00' = interpolation continuous, period in DDD;HH:mm:ss

1 1 '1;00:00:00' = interpolation block, period in DDD;HH:mm:ss

TBLE .. tble contains the table, first column is the date (year month day hour minute second), second column the target level in m NAP. The $<$ sign is a separator.

H0 table definitions:

```
H0_T id 'H0table' PDIN 1 1 '1;00:00:00' pdin
TBLE
1995/01/01;00:00:00 0.4 <
1995/02/01;00:00:00 0.5 <
1995/03/01;00:00:00 0.5 <
1995/04/01;00:00:00 0.4 <
1995/05/01;00:00:00 0.3 <
1995/06/01;00:00:00 0.2 <
1995/07/01;00:00:00 0.1 <
1995/08/01;00:00:00 0.0 <
1995/09/01;00:00:00 0.0 <
1995/10/01;00:00:00 0.1 <
1995/11/01;00:00:00 0.2 <
1995/12/01;00:00:00 0.3 <
tble
h0_t
```

With:

id	table identification

PDIN .. pdin = period and interpolation method (for description see above)

D.19.8 Paved area layer

Data file: Paved.3B

This file contains the data for the nodes of model type 1 (paved or impervious area)

The keyword PAVE is used to mark the beginning of a record, and in lower case characters it marks the end of a record. The keywords within the record may appear in any order (same for other files). Some keywords within the records may be missing; in that case defaults will be used. If essential keywords (e.g. the id) are missing, error message will follow.

PAVE id '1' ar 100000. lv 1.0 sd 'stor_1mm' ss 0 qc 0 0.2 0 qo 1 1 so 0.5 0.5 si 0 0 ms 'meteostat1' is 100. np 5000 dw '1' ro 1 ru 0.5 qh 'Qhrelation' pave

With

id	node identification
ar	area (in m2)
lv	street level (m NAP)
sd	storage identification
ss	sewer system type (0=mixed, 1=separated, 2=improved separated)
qc	capacity of sewer pump (m3/s) qc 0 0.2 0.0 = capacity as a constant, with value 0.2 (mixed/rainfall sewer) and 0.0 (DWA in separated or improved separated systems). **So, first value is for mixed/rainfall sewer, second value for the dry weather flow (DWA) sewer.** qc 1 'qctable1' = capacity as a table, with table identification qctable1.
qo	1 1 = both sewer pumps discharge to open water (=default) 0 0 = both sewer pumps discharge to boundary 0 1 = rainfall or mixed part of the sewer pumps to open water, DWA-part (if separated) to boundary 1 0 = rainfall or mixed part of the sewer discharges to boundary, DWA-part (if separated) to open water 2 2 = both sewer pumps discharge to WWTP 2 1 = rainfall or mixed part of the sewer pumps to open water, DWA-part (if separated) to WWTP etc. **Note: first position of record is allocated to DWA sewer, second position is allocated to mixed/rainfall sewer; 0=to boundary, 1= to openwater, 2=to WWTP. In all other keywords the order is just the other way around!!!!**
so	sewer overflow level (first value for RWA/Mixed sewer, second value for DWA sewer). If missing, the surface level will be used. The level is used to verify whether sewer overflows can occur (no overflows can occur if the related boundary or open water level is higher)
si	sewer inflow from open water/boundary possible yes/no (1=yes,0=no); first value for RWA/Mixed sewer, second value for DWA sewer). Default value is 0, meaning that no external inflow is possible.
s	identification of the meteostation by a character id If this id is missing in the rainfall file, data from the first station in the rainfall file will be used.
is	initial salt concentration (g/m3). Default 0.

np	number of people
dw	dry weather flow identification
ro	runoff option 0 = default, no delay (=previous situation) 1 = using runoff delay factor (as in NWRW model) 2 = using Qh relation
ru	runoff delay factor in (1/min) (as in NWRW model) (only needed and used if option ro 1 is specified)
qh	reference to Qh-relation (only needed and used if option ro 2 is specified)

Spilling of paved area sewers occurs to the downstream open water node (if existing), otherwise to the downstream boundary. In case no downstream open water node and no downstream boundary exists, an error message will be given.

Storage Definition file: Paved.Sto

This file contains the storage definitions for the nodes of model type 1 (paved or impervious area)

STDF id 'stor_1mm' nm 'storage 1mm' ms 1. is 0. mr 7.0 0.0 ir 0.0 0.0 stdf

With

id	storage identification
nm	name (optional)
ms	maximum storage on streets (mm). Default 1 mm.
is	initial storage on streets (mm). Default 0.
mr	maximum storage sewer (mm). Default 7 mm.
ir	initial storage in sewer (mm). Default 0.

For **mr** and **ir** different sewer systems are distinghuished (mixed systems, separated systems, improved separated system). The first value is for mixed and rainfall sewer, the second value for DWA sewer.

DWF definition file: Paved.dwa

This file defined the dry weather flow for paved area. It is similar to the DWA file for the NWRW rainfall-runoff model.

DWA id '125_lcd' nm '125 liter per capita per day' do 1 wc 12. wd 125. wh 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24 sc 50. dwa

id	dwa identification
nm	dwa name
do	dwa computation option see NWRW dwa description.
wc	water use per capita as a constant value per hour (l/hour)
wd	water use per capita per day (l/day)
wh	water use per capita per hour (24 percentages, total should be 100%)
sc	salt concentration (mg/l) of DWA. Default 400 mg/l. sc keyword is **NOT READ. Default value always used.**

Table definition file: Paved.tbl

For table input of the sewer pump capacity as function of time.

```
QC_T id 'qctable1' nm 'Test' PDIN 1 0 ' ' pdin
TBLE
1997/01/01;00:00:00 1.0 0.0 <
1997/12/31;00:00:00 1.0 0.0 <
tble
qc_t
```

With

id	table identification
nm	table name

PDIN ..pdin = period and interpolation method

0 0 ' ' = interpolation continuous, no period

1 0 ' ' = interpolation block, no period

0 1 '365;00:00:00' = interpolation continuous, period in DDD;HH:mm:ss

1 1 '1;00:00:00' = interpolation block, period in DDD;HH:mm:ss

TBLE .. tble contains the table, first column is the date (year month day hour minute second), second and third column the capacity in m3/s. Third column is default zero, can only be used for separated sewer systems. The $<$ sign is used to separate the rows.

Period can be indicated in DDD;HH:mm:ss indicates a yearly period, for example 365;00:00:00. This option is especially useful for open water target levels as a table.

The period is indicated within quotes; this is also done in the SOBEK-CF modeldatabase.

For a Qh-table the following lay-out is used:

```
QHTB id 'Qhrelation' nm 'Qhrelation table' TBLE
0.0 0.0 <
0.01 0.5 <
0.03 1.0 <
0.05 1.5 <
0.07 2.0 <
tble qhtb
```

With the keywords:

id	reference to Qh-relation
nm	name of Qh-relation

TBLE . . . tble = table, with

1st column = Q in mm/s,

2nd column = h in mm.

D.19.9 Runoff layer

Data file (sacrmnto.3b)

This file contains the data for the nodes Sacramento, HBV, SCS, NAM and external timeseries

SACR id '5' ar 319000000 ms '1' ca 'Capacities&Contents' uh 'UnitHydrograph' op 'OtherParameters' sacr

EXTR id 'nodeid' ar 7010324 rs 'rs7001' ms 'ms7001' Extr

HBV id 'nodeid' ar 7010324 sl 300 snow '1' soil '1' flow '2' hini'3' ts 'Moezeltemperature' ms 'Moezel' aaf 0.9 hbv

SCS id 'nodeid' ar 7010324 slp 0.530683 le 2650.826 cn 50 ms '7001' uh 1 amc 1 scs

SCS id 'nodeid2' ar 7010324 slp 0.530683 le 2650.826 cn 50 ms '7001' uh 0 tl 1.0 scs

SCS id 'nodeid3withbaseflow' ar 7010324 slp 0.530683 le 2650.826 cn 50 ms '7001' uh 0 tl 1.0 amc 1 bu 1 gsmax 100 gsinit 50 kr 0.01 scs

With

id	node identification
ar	area in m2
ca	name of capacity and content definition (refer to sacrmnto.cap file)
uh	name of unit hydrograph definition (refer to sacrmnto.uh file)
op	name of definition of other parameters (refer to sacrmnto.oth file)
rs	runoff station
ms	meteo station
ts	temperature station
sl	surface level (altitude)
snow	reference to snow definition in SNOW record
soil	reference to soil definition in SOIL record
flow	reference to flow definition in FLOW record
hini	reference to initial water contents definition in HINI record
slp	average basin slope (m/m)
le	flow path length to outlet (m)
cn	SCS curve number
uh	SCS unit hydrograph option (optional) default value uh 0 = HEC-HMS unit hydrograph; other option uh 1 = SCS dimensionless unit hydrograph.
tl	SCS time lage (hours), optional input (if not specified, computed using SCS time lag formula from SCS curve number, average basin slope and flow path length)
amc	antecedent moisture condition. 1=dry, 2=average, 3=wet. The keyword is optional, if missing a default value of 2 is assumed.
aaf	area adjustment factor. Optional keyword, default value 1. Value should be $>=$0.
bu	baseflow used; 0=no, 1=yes. Default 0
gsmax	groundwater storage, maximum value (mm)
gsinit	groundwater storage, initial value (mm)
kr	recession coefficient Kr (1/day)

In the SACR, HBT and SCS record also the aaf keyword is optionally present to specify area adjustment factor. There are no other changes in the SACR, HBV, EXTR, SNOW, SOIL, FLOW, HINI records in the Sacrmnto.3B file.

Capacities and contents definition file (sacrmnto.cap)

This file contains the definitions of the capacities, initial contents, and reservoir coefficients of the Sacramento soil storage reservoirs.

CAPC id 'Capacities&Contents' uztwm 50 uztwc 50 uzfwm 150 uzfwc 100 lztwm 150 lztwc 150 lzfsm 200 lzfsc 100 lzfpm 150 lzfpc 150 uzk .08 lzsk .05 lzpk .003 CAPC

With

id	identification
uztwm	upper zone tension water storage capacity (maximum content)
uztwc	upper zone tension water initial content
uzfwm	upper zone free water storage capacity (maximum content)
uzfwc	upper zone free water initial content
lztwm	lower zone tension water storage capacity
lztwc	lower zone tension water initial content
lzfsm	lower zone supplemental free water storage capacity
lzfsc	lower zone supplemental free water initial content
lzfpm	lower zone primary free water storage capacity
lzfpc	lower zone primary free water initial content
uzk	upper zone drainage rate
lzsk	lower zone secondary drainage rate
lzpk	lower zone primary drainage rate

Unit hydrograph definition file (sacrmnto.uh)

UNIH id 'UnitHydrograph' uh 1 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 dt 1 unih

With

id	identification
uh	unit hydrograph components
dt	step size to be applied on unit hydrograph components

other data file (sacrmnto.oth)

Other Sacramento parameters:

OPAR id 'OtherParameters' zperc 5.0 rexp 9.0 pfree 0.2 rserv 0.95 pctim 0 adimp 0.5 sarva 0.0 side 0.0 ssout 0.0 pm 0.1 pt1 500 pt2 500 opar

With

id	identification
zperc	proportional increase in percolation from saturated to dry conditions
rexp	exponent in percolation equation
pfree	fraction of percolated water directly to lower zone free water
rserv	fraction of lower zone free water unavailable for transpiration
pctim	permanently impervious fraction of basin
adimp	fraction of basin which becomes impervious as all tension water

	requirements are met
sarva	fraction of basin covered by streams, channels
side	portion of baseflow not observed in the stream channel
ssout	sub-surface outflow
pm	time interval increment parameter
pt1	rainfall threshold 1
pt2	rainfall trheshold 2

For an explanation of the Sacramento concepts and parameters, reference is made to the HYMOS manual.

D.19.10 NAM rainfall runoff model

The SOBEK graphical user interface writes all D-NAM input parameters (see section 5.4.13) into six different type of NAM database records, that are stored in the <SACRMNTO.3B> database file.

1 **NAM** database record (Main record)

NAM id '1' nm '1' ar 5000000 sp '##1' lwd '##1' sr '##1' bf '##1' gw '##1' ms 'Station1' aaf 1 nam

where:

id	ID of D-NAM node
nm	NAME of D-NAM node (optional)
ar	Catchment area $[m^2]$
sp	Reference to NAMS database record with id=##1, containing the soil parameters definition
lwd	Reference to NAML database record with id=##1, containing the levels and initial water depth definition
sr	Reference to NAMR database record with id=##1, containing the surface runoff parameters definition
bf	Reference to NAMB database record with id=##1, containing the base flow parameters definition
gw	Reference to NAMG database record with id=##1, containing groundwater pump definition
ms	Meteo-station
aaf	Area adjustment factor [-]

2 **NAMS** database record (Soil parameters definition)

NAMS id '##1' nm 'SoilDef01' infcap 11 sfc 0.8 syrz 0.9 syss 0.4 capo 0 capc 12 capt PDIN 0 0 '' pdin
TBLE
.2 15 <
.5 12 <
1 10 <
2 10 <
tble
pero 0 perc 8 pert PDIN 0 0 '' pdin
TBLE
0 1 <
1 5 <
2 8 <

4 16 <
tble
nams

where:

id	ID of this NAMS database record, containing a particular soil parameter definition
nm	Name of this NAMS database record, containing a particular soil parameter definition (shown in GUI)
infcap	Maximum infiltration capacity ($INFCap$) [mm/hr]
sfc	Field capacity of root zone layer (s_{fc}) [-]
syrz	Specific yield of root zone layer ($s_{y,rz}$) [-]
syss	Specific yield of subsoil layer ($s_{y,ss}$) [-]
capo	Option for capillary rise (CR), 0 = Constant, 1 = Table
capc	Capillary rise (CR) [mm/day] as a Constant
capt	Capillary rise (CR) [mm/day] as Function of groundwater table depth in default SOBEK format
pero	Option for Percolation capacity (G), 0 = Constant, 1 = Table
perc	Percolation Capacity (G) as a Constant [mm/day]
pert	Percolation Capacity (G) as Function of groundwater table depth [mm/day] in default SOBEK format

3 **NAML** database record (Levels and initial water depth definition)

NAML id ’##1’ nm ’IniValDef01’ sl 6.66 rzbl 4 gwsbl 3 ui 25 li 400 gwsdi 45 naml

where:

id	ID of this NAML database record, containing a particular level and initial water depth definition
nm	Name of this NAML database record, containing a particular level and initial water depth definition (shown in GUI)
sl	Surface level (SL) [m AD]
rzbl	Bed level of the root zone layer ($RZBL$) [m AD].
gwsbl	Bed level of the subsoil layer or groundwater storage ($GWSBL$) [m AD]
ui	Initial water depth in the surface storage (U_i) [mm]
li	Initial water depth in lower zone storage (L_i) [mm]
gwsdi	Initial water depth in groundwater storage ($GWSD_i$) [mm]

4 **NAMR** database record (Runoff parameters definition)

NAMR id ’##1’ nm ’Runoff’ cl 5000 ss 0.001 man 0.02 utof 5 ckif 1000 utif 1 ltif 5 namr

where:

id	ID of this NAMR database record, containing a particular runoff parameters definition
nm	Name of this NAMR database record, containing a particular runoff parameter definition (shown in GUI)
cl	Catchment length (CL) [m]
ss	Averaged slope of the catchment surface area towards its overland outflow point (S) [-]
man	Manning value (n) for the roughness of the surface catchment area [$s/m^{1/3}$]
utof	Threshold, used in determining if water in the surface storage may flow out

	as overland flow [mm]
ckif	Reservoir coefficient used in routing interflow [days]
utif	Threshold, used in determining if water in the surface storage may flow out as interflow [mm]
ltif	Lower-zone-storage-threshold, used in determining if water in the surface storage may flow out as interflow [mm]

5 **NAMB** database record (Base flow parameters definition)

```
NAMB id '##1' nm 'BaseFlow' ckfastbf 1500. ckslowbf 2500. ckgwinf 3000. ltp 10. tfastbf
-1.0 tslowbf -5.0 namb
```

where:

id	ID of this NAMB database record, containing a particular base flow parameter definition
nm	Name of this NAMB database record, containing a particular base flow parameter definition (shown in GUI)
ckfastbf	Reservoir coefficient used in routing fast base flow ($CKFastBF$) [days]
ckslowbf	Reservoir coefficient used in routing slow base flow ($CKSlowBF$) [days]
ckgwinf	Reservoir coefficient used for the inflow of external (ground)water ($CKGWInflow$) [days]
ltp	Threshold, used in determining if water in the lower zone storage may percolate to the groundwater storage (L_{TP}) [mm]
tfastbf	Threshold for fast base flow (T_{FastBF}) [m AD]. Only water in the groundwater storage above T_{FastBF} can flow out as fast base flow]
tslowbf	Threshold for slow base flow (T_{SlowBF}) [m AD]. Only water in the groundwater storage above T_{SlowBF} can flow out as slow base flow

Note: $CKGWInflow \geq CKSlowBF \geq CKFastBF \geq 1$.

6 **NAMG** database record (Groundwater pump definition)

```
NAMG id '##1' nm 'GWPump' po 0 pu 0.1 pt PDIN 0 1 '365;00:00:00' pdin
TBLE
'2001/01/01;00:00:00' 0.0 <
'2001/03/31;23:59:00' 0.0 <
'2001/04/01;00:00:00' 0.1 <
'2001/09/30;23:59:00' 0.1 <
'2001/10/01;00:00:00' 0.0 <
'2001/12/31;00:00:00' 0.0 <
tble
namg
```

where:

id	ID of this NAMG database record, containing a particular groundwater pump definition
nm	Name of this NAMG database record, containing a particular groundwater pump definition (used in GUI)
po	Pumping option, 0 = Constant, 1 = Time Series
pu	Discharge of the groundwater pump ($GWPump$) as a Constant [m^3/s]
pt	Discharge of the groundwater pump ($GWPump$) as a Time series [m^3/s] in default SOBEK format

D.19.11 Walrus rainfall runoff model

The <delft_3b.ini> file:

The <delft_3b.ini> file contains switch (UseWalrusorWagmod=ModelName):

- ◇ SOBEK 2: Switch set as UseWalrusorWagmod=WALRUS, means that the Walrus model is run. Switch set as UseWalrusorWagmod=WAGMOD, means that the WAGMOD model is run. Limitation, the Walrus and Wagmod model cannot jointly be used during a simulation.
- ◇ SOBEK 3: Switch is default set as UseWalrusorWagmod=WALRUS, since the Wagmod model cannot be run in SOBEK 3.

Following five numerical parameters can be specified in the <delft_3b.ini> file, which control the Walrus internal timestep (i.e. possible reductions of the user-defined timestep):

1 Walrus_min_deltime: minimum timestep in seconds (default=60.0)
2 Walrus_maxhchange: maximum change in the water depth of the surface water reservoir and/or quickflow reservoir in 1 internal timestep in mm (default =10.0).
3 Walrus_minh: minimum water depth in the the surface water reservoir and/or quickflow reservoir in mm (default =0.001).
4 Walrus_max_pstep: maximum precipitation in 1 internal Walrus timestep in mm (default =10.0).
5 Walrus_max_substeps: maximum number of internal timesteps in 1 computation timestep (default=288)

Note: For more information on the Walrus time step reduction procedure, reference is made to section "Possible time step reductions and applied threshold values" in the technical reference manual.

Walrus database records in the <sacrmnto.3b> file

DeltaShell (graphical user interface) writes the input parameters of a Walrus catchment into six different type of Walrus database records, that are stored in the <sacrmnto.3b> database file. These Walrus database records are:

1 The **WALR** database record (main record):

WALR id ‘node id’ nm ‘node name’ ar 1000 sl 20.1 wa 0 cw 200 wit ‘wetness index table’ cv 4 cg 5000000 cq 10 va 0 vit ‘Veq table’ ba 0 bit ‘evaporation reduction table’ qa 0 qit ‘Qh interpolation table’ cs 4 xs 1.5 as 0.01 cd 1500 hst 0 hsmin 0 hstable ‘1’ st 21 b 4.05 psi_ae 121 theta_s 0.395 hs0 0 hq0 0 dg0 0 dv0 0 q0 1 ms ‘meteostation’ msevap ‘evap’ fxs ‘sw supply/extraction’ fxg ‘seepage’ aaf 1.0 walr

where:

id	Walrus node id
nm	Walrus node name
ar	Catchment surface area [m^2]
sl	Surface level of the catchment area (optional) [m AD]
wa	-1 (wetness index is determined, applying the Walrus analytical formulation

	for wetness index and value for parameter cw), or 0 (wetness index is determined, applying the user-defined interpolation table for wetness index).
cw	Wetness index parameter [mm] (default=200). Value for cw is only used if wa=-1.
wit	id of a user-defined interpolation table for the wetness index, which is to be specified on a WETN database record. This interpolation table is only used if wa=0.
cv	Vadose zone relaxation time constant [hours] (default=4).
cg	Constant parameter applied in computing groundwater drainage or infiltration [mm hour] (default=5000000).
cq	Quickflow constant reservoir coefficient [hours] (default=10).
va	-1 (equilibrium storage deficit is determined, applying the Walrus analytical formulation for equilibrium storage deficit), or 0 (equilibrium storage deficit is determined, applying the user-defined interpolation table for equilibrium storage deficit).
vit	id of a user-defined interpolation table for equilibrium storage deficit, which is to be specified on a VEQD database record. This interpolation table is only used if va=0.
ba	-1 (transpiration reduction factor β is determined, applying the Walrus analytical formulation for β), or 0 (transpiration reduction factor β is determined, applying the user-defined interpolation table for β).
bit	id of a user-defined interpolation table for the transpiration reduction factor β, which is to be specified on a BETA database record. This interpolation table is only used if ba=0.
qa	-1 (discharge Q flowing out of the surface water reservoir is determined, applying the Walrus analytical formulation for Q), or 0 (discharge Q flowing out of the surface water reservoir is determined, applying the user-defined interpolation table for Q).
qit	id of a user-defined interpolation table for discharge Q flowing out of the surface water reservoir, which is to be specified on a WQH database record. This interpolation table is only used if qa=0.
cs	Bankfull discharge of the surface water reservoir [mm/hour] (default=4). Value for cs is only used if qa=-1.
xs	Exponent in Walrus analytical formulation for computing discharge (Q) flowing out of the surface water reservoir (default=1.5). Value for xs is only used if qa=-1.
as	Surface water area fraction, ratio of the area of the surface water reservoir and the catchment surface area [-] (default=0.1)
cd	Channel depth or bed level of the surface water reservoir [mm below SL] (default=1500).
hst	-1 (value of $h_{S,min}$ in the Walrus analytical formulation for discharge (Q) flowing out of the surface water reservoir is defined as a time table), or 0 (a constant value is defined for $h_{S,min}$ in the Walrus analytical formulation for discharge (Q) flowing out of the surface water reservoir).
hsmin	For a water depth in the surface water reservoir equal to $h_{S,min}$, the outflow of the surface water reservoir Q equals zero. Value for hsmin is only used if qa=-1.
hstable	id of the user-defined time table for hsmin, which is to be specified on a HSTT database record. This time table is only used if qa=-1 and hst=-1.
st	Soil type (default; st=22 loamy sand). For soil types st=21 up to st=33 yields that values for b, ψ_{ae} and θ_S are read from Table D.1. For soil type st=34 (custom) yields that the values for b, ψ_{ae} (or psi_ae) and θ_S (or theta_s) are

	applied, that are specified hereunder.
b	Pore size distribution parameter [-]. This value for b is only used if soil type st=34 (custom) is selected.
psi_ae	Air entry pressure [mm]. This value for psi_ae is only used if soil type st=34 (custom) is selected.
theta_s	Soil moisture content at saturation [-]. This value for theta_s is only used if soil type st=34 (custom) is selected.
hs0	Initial water depth in surface water reservoir [mm]
hq0	Initial water level in quickflow reservoir [mm]
dg0	Initial groundwater table depth [mm]
dv0	Initial water-storage deficit (or availabe water storage capacity in vadose zone of soil water reservoir [mm]
q0	Initial discharge flowing out of the surface water reservoir [mm/hour]
ms	Reference (or id) to meteo-station with rainfall time-series in [mm/timestep]
msevap	Reference (or id) to meteo-station with potential evapotranspiration time-series in [mm/timestep] (to give the option to specify evaporation not only as daily values, but in detail)
fxs	Reference (or id) to surface water supply or extraction time-series in [mm/-timestep] (from rainfall file)
fxg	Reference (or id) to groundwater seepage or extraction time-series in [mm/-timestep] (from rainfall file)
aaf	Area adjustment factor for rainfall (default 1.0)

Please note that:

- ◇ Input parameters ar, cw, cv, cg, cq, cs, cd, xs, hsmin, as, hs0, hq0, dg0, dv0 and q0 should be larger than or equal to zero.
- ◇ For input parameters hsmin and cd should yield that hsmin < cd. Since, the $q = 0$ level (coinciding with hsmin) should be smaller than the channel depth (cd).
- ◇ The groundwater reservoir area fraction (a_G) is not defined in the WALR database record, since a_G= 1-as.
- ◇ If a restart file is used, the values for hs0, hq0, dg0, dv0 and q0 are read from the restart file. So the values specified in the WALR database record are neglected.
- ◇ Time-series for precipitation (P), potential evapotranspiration (ET_{pot}), groundwater seepage and extraction (fxg) and external surface water supply or extraction (fxs) are assigned to a particular meteo-station (ms). The id of this meteo-station as well as the time-series data is read from a particular rainfall file.

Table D.1: *Soil types and related Walrus parameter values.*

Walrus soil type **Nr.**	**Soil type** **Description**	**Pore-size distribution parameter** **b [-]**	**Air entry pressure** Ψ_{ae} **[-]**	**Soil moisture content at saturation** θ_s **[-]**
21	sand	4.05	121	0.395
22	loamy_sand	4.38	90	0.410
23	sandy_loam	4.9	218	0.435
24	silt_loam	5.3	786	0.485
25	loam	5.39	478	0.451
26	sandy_clay_loam	7.12	299	0.42
27	silt_clay_loam	7.75	356	0.477
28	clay_loam	8.52	630	0.476

29	sandy_clay	10.4	153	0.426
30	silty_clay	10.4	490	0.492
31	clay	11.4	405	0.482
32	cal_Hupsel	2.63	90	0.418
33	cal_Cabauw	16.77	9	0.639
34	custom	User defined	User defined	User defined

Note: Soil type descriptions "cal_Hupsel" and "cal_Cabauw" are respectively the calibrated dataset for the Hupselse Beek catchment and the Cabauw catchment in the Netherlands.

2 The **WETN** database record (Wetness index (W) interpolation table)

WETN id 'wetness index table' nv 4 dv 0 100 200 300 wi 1.0 0.7 0.1 0.0 wetn

where:

id	Table id (matching with id given after keyword wit in the WALR database record)
nv	Number of values (length of table)
dv	Water-storage deficit (d_V) values [mm]. For dv should yield dv ≥ 0; consecutive values for dv should be increasing.
wi	Wetness index (W) [-] For wi should yield $0 \leq$ wi ≤ 1; consecutive values for wi should be non-increasing.

3 The **BETA** database record (Transpiration reduction (β) interpolation table)

BETA id 'evaporation reduction table' nv 5 dv 0 100 200 300 400 er 1.0 0.99 0.9 0.75 0.0 beta

where:

id	Table id (matching with id given after keyword bit in the WALR record)
nv	Number of values (length of table)
dv	Water-storage deficit (d_V)values [mm]. For dv should yield dv ≥ 0; consecutive values for dv should be increasing.
er	Transpiration reduction factor (β) [-]. For er should yield $0 \leq$ er ≤ 1; consecutive values for er should be non-increasing.

4 The **VEQD** database record (Equilibrium water-storage deficit ($d_{V,eq}$) interpolation table)

VEQD id 'Veq table' nv 6 dg 0 250 500 1000 1500 2000 veq 0.0 0.0 100 250 500 900 veqd

where:

id	Table id (matching with id given after keyword vit in the WALR record)
nv	Number of values (length of table)
dg	Groundwater table depth (d_G) [mm]. For dg should yield dg ≥ 0; consecutive values for dg should be increasing.
veq	Equilibrium water-storage deficit ($d_{V,eq}$) [mm]. For veq should yield veq ≥ 0; consecutive values for veq should be non-decreasing.

5 The **WQH** database record (Discharge relationship ($Q = f(h_S)$ interpolation table)

WQH id ‘Qh interpolation table’ nv 4 hs 0 1000 1500 2000 q 0.0 1.0 2.0 3.0 wqh

where:

id	Table id (matching with id given after keyword qit in the WALR record)
nv	Number of values (length of table)
hs	Surface water depth (h_S) [mm]. For hs should yield hs ≥ 0; consecutive values for hs should be increasing.
q	Discharge (Q) flowing out of the surface water reservoir [mm/hour]. For q should yield q ≥ 0; consecutive values for q should be non-decreasing.

6 The **HSTT** database record (Time-series for $h_{S,min}$)

```
HSTT id ’1’ nm ’table hsmin’
PDIN 1 0 ’365;00:00:00’ pdin
TBLE
’2016/01/01;00:00:00’ 0.0 <
’2016/04/01;00:00:00’ 20.0 <
’2016/10/01;00:00:00’ 0.0 <
’2016/12/31;23:59:00’ 0.0 <
tble
hstt
```

where:

id — Table id (matching with id given after keyword hstable in the WALR database record)

nm — Name of table

PDIN 1 0 ‘365;00:00:00’ pdin

First value: 0 (linear interpolation) / 1(block interpolation)
Second value: 0 (no periodicity) / 1 (periodic)
Third value: period in ‘days;hh:mm:ss’ (only used if second value equals 1)

TBLE . . . tble

Free number of sub-records allowed. Each sub-record consecutively contains: date ‘YYYY/MM/DD;hh:mm:ss’, value and $<$, denoting the end of a sub-record.

D.19.12 Structure layer

Data file: Struct3b.dat

STRU id ’pomp1’ dd ’pomp1’ ca 0 0 0 0 cj ’-1’ ’-1’ ’-1’ ’-1’ STRU

With:

id	node identification
dd	structure definition
ca	controller active (1=yes, 0=no.). To be consistent with the SOBEK modeldatabase 4 controllers are allowed. **Only the first is used** at the moment.
cj	controller id’s (-1 for no controller). **Only the first is used.**

Structure table file: Struct3b.Tbl

Contains the tables of switch on/off levels for pumps (including inlet pumps), weir or gate settings as a function of time, switch-on/off levels for inlet weirs and gates.

Switch-on and -off levels for a pump:

```
SWLV 'on-off_levels' nm ' pump1' PDIN 0 0 '1;00:00:00' pdin
TBLE
1997/01/01;00:00:00 0.03 -0.03 0.10 0. 0.03 -0.05 0.10 0.02 <
1997/04/01;00:00:00 0.04 -0.03 0.10 0. 0.03 -0.05 0.10 0.02 <
1997/05/01;00:00:00 0.05 -0.03 0.10 0. 0.03 -0.05 0.10 0.02 <
1997/08/01;00:00:00 0.05 -0.03 0.10 0. 0.03 -0.05 0.10 0.02 <
1997/10/01;00:00:00 0.04 -0.03 0.10 0. 0.03 -0.05 0.10 0.02 <
1997/11/01;00:00:00 0.03 -0.03 0.10 0. 0.03 -0.05 0.10 0.02 <
tble
swlv
```

The first column in the table indicates the date, the second 4 values are the switch-on and off levels relative to the related open water target levels during the day (on low, off low, on high, off high), the next 4 values are the switch-on and off levels relative to the related open water target levels during the night (on low, off low, on high, off high)

For a normal pump the switch-on levels will be higher than the switch-off levels; for an inlet pump it will be just the other way around.

Switch-on and -off levels for a weir or gate:

For a weir or gate the table contains only 2 values per line, viz. the switch-on and off level relative to the related open water target levels. The switch-on level for inlet weirs or gate will be smaller than the switch-off level.

```
SWLV 'on-off_levels' nm ' weir1' PDIN 0 0 '1;00:00:00' pdin
TBLE
1997/01/01;00:00:00 -0.05 0.03 <
1997/05/01;00:00:00 -0.03 0.05 <
1997/08/01;00:00:00 -0.05 0.03 <
1997/12/31;00:00:00 -0.05 0.03 <
tble
swlv
```

Consistency checks for normal pumps: (dn 1)

- ◇ switch on levels >=switch off levels,
- ◇ switch-on level low capacity <= switch-on level high capacity
- ◇ switch-off level low capacity <= switch-off level high capacity

for dn 2 (inlet pump) just the other way around.

Initial weir or gate settings:

```
INST 'init_weir_setting' nm ' name1' PDIN 0 0 '1;00:00:00' pdin
TBLE
1997/01/01;00:00:00 -1.5 <
1997/04/01;00:00:00 -1.45 <
1997/05/01:00:00:00 -1.40 <
```

1997/09/01:00:00:00 -1.45 <
1997/10/01:00:00:00 -1.50 <
tble
inst

In previous versions of Sobek-RR always block interpolation was assumed, but since version **2.06.000.35g** the data specified for PDIN .. pdin is used.

Controller definition file: Contr3B.Def

This file contains the definition of the controllers of RR-structures.

The following type of controllers are distinguished (type numbering starts at 11, to prevent confusion with Sobek-controllers):

11 = 3B -PID controller (**not yet available**)

12 = 3B -fixed upstream level controller

13 = 3B -fixed level difference controller

14 = 3B -minimum level difference controller

15 = 3B -time controller

PID controller not yet defined and implemented

CNTL id ’1’ nm ’3B_PID’ ty 11 cntl

....

All controllers can at the moment only be defined for normal weirs, except for the time controller. The time controller is also available for normal RR-pumps and RR-orifices.

Fixed upstream level controller with flow limitation:

CNTL id ’1’ nm ’target level controller’ ty 12 mf 1.0 ml -0.5 md -0.7 zmin -2.75 zmax -2.0 cntl

with

id	controller definition identification
nm	name
ty	controller type (12=fixed upstream level controller)
mf	maximum flow
ml	maximum level above which the maximum flow may be exceeded
md	maximum downstream level
zmin	minimum crest level in m w.r.t. reference level (optional keyword); (if this is not specified, the lowest crest level is taken equal to the bottom level of the boundary or the RR-open water; this keyword provides a way to specify a higher lowest level)
zmax	maximum crest level in m w.r.t. reference level (optional keyword); (if this is not specified, the maximum crest level is taken equal to 9999.99; this keyword provides a way to specify a lower maximum crest level)

This controller tries to maintain the upstream target level as long as the flow over the structure does not exceed the maximum flow; if the flow equals the maximum flow, the level may increase up to the maximum level; at that time the maximum flow may also be exceeded. If the downstream level exceeds the specified maximum downstream level, a more strict flow limitation is used.

Firstly, it is verified that this flow can be physically realised by lowering the weir to the specified lowest value zmin (either bottom level, or a higher user defined level).

Secondly, there may be a maximum crest level zmax which limits the operation of the weir. If not specified, the controller operates such that negative weir flows are not allowed (this is equivalent with specifying zmax very high). If a zmax is specified, negative flows may occur (if the downstream level is higher than the upstream level; and also either the downstream level is above the zmax level or the upstream level is below target level)

Fixed level difference controller:

CNTL id ’1’ nm ’3B_fixed level difference’ ty 13 cntl

With:

id	controller definition identification
nm	name
ty	controller type (13=fixed level difference controller)

This controller requires no extra input data; the fixed level difference is taken as the difference of the upstream and downstream target levels.

Minimum level difference controller:

CNTL id ’1’ nm ’3B_minimum level difference’ ty 14 ml 0.1 cntl

with

id	controller definition identification
nm	name
ty	controller type (14=minimum level difference controller)
ml	minimum level difference. **NOT READ; minimum level difference is defined as difference in upstream and downstream target levels.**

Time controller:

CNTL id ’1’ nm ’3B_time controller’ ty 15 ts ’TimeTable’ cntl

With:

id	controller definition identification
nm	name
ty	controller type (15=time controller)
tb	name of controller table

This controller requires a table with the time settings of the crest level (weir), opening height (gates), or capacity (pumps). All tables are specified in the Structure Table using the INST records. This has the advantage that the table for the initial settings of weirs/gates and the tables for the time controller can be the same.

Equal filling controller:

** To be completed **

CNTL id '1' nm '3B_equal filling controller' ty 16 mf 1.0 mf2 5.0 zmin 0. zmax 1. cntl

with

id	controller definition identification
nm	name (optional)
ty	controller type (16=equal filling controller)
mf	maximum flow at target level (from upstream open water)
mf2	maximum flow at maximum level (from upstream open water)
zmin	minimum crest level in m w.r.t. reference level (optional keyword); (if this is not specified, the lowest crest level is taken equal to the bottom level of the boundary or the RR-open water; this keyword provides a way to specify a higher lowest level)
zmax	maximum crest level in m w.r.t. reference level (optional keyword); (if this is not specified, the maximum crest level is taken equal to 9999.99; this keyword provides a way to specify a lower maximum crest level)

Structure definition file: Struct3b.def

This file contains the data for the nodes of Sobek-RR-model type 8 to 12 (various types of structures: pump, weir, gate, Manning resistance, Q-h relation). The following type of structures are present in Sobek-RR:

8=pump

9=weir

10=gate

11=Manning resistance

12= Q-h relation

Where possible, the keywords are chosen equal as in the Sobek-modeldatabase, to enable using the same conventions and routines.

The definition for a pump:

STDS id '1' nm '3B-pomp' ty 8 in 0 dn 1 nc 2 pc 0.5 0.2 so 'on-off_levels' stds

With:

id	id van de structure definition definition
nm	name of pump (optional)
ty	type of structure (8=3B-pomp)
in	inlet pump (0=no, 1=yes). Default 0.
dn	Direction and type of control 1 : Check upstream level only (normal pump). Default. 2 : Check downstream level only (inlet pump)
nc	Number of capacities. Default = 2. **NOT READ. Default value 2 used.**

pc	Pump capacities (in m3/s). First value low capacity, second value extra capacity at high capacity. In the future maybe more values, depending on nc.
so	switch-on and switch-off levels. Reference to a table.

The definition for a weir:

STDS id '1' nm '3B-overlaat' ty 9 in 0 wt 1 dc 1.1 cl -1.5 cw 3.0 cp 1.5 rt 0 ws 'init_weir_setting' fl 0. stds

or

STDS id '1' nm '3B-overlaat' ty 9 in 1 wt 2 sl 1.0 dc 1.1 cl -1.5 cw 3.0 cp 2.5 rt 0 ws 'init_weir_setting' so 'table2' fl 0. stds

or

STDS id '23' nm '3B-overlaat' ty 9 wt 3 dc 1 cw 1 cw2 9 cl 1.1 cl2 1.3 cp 1.5 ws '' in 0 rt 0 stds

With:

id	structure definition identification
nm	name
ty	structure type (9=3B-weir)
in	inlet weir (0=no, 1=yes). Default 0.
wt	weir type (1=rectangular weir, 2=V-shape weir, 3=2-stage weir)
sl	slope (only for V-shape type of weir)
dc	discharge coefficient. Default 1.0
cl	crest level (m NAP)
cw	crest width (m)
cw2	(additional) crest width 2nd stage (m)
cl2	crest level 2nd stage (m NAP) The keywords cl2 and cw2 are only needed for 2-stage weirs. The second crest level (cl2) should be higher than the first crest level (cl).
cp	power coefficient. Default 1.5, for V-notch 2.5.
rt	0 = flow in both directions possible (default for normal weirs) 1 = only positive flow possible (default for inlet weirs) 2 = only negative flow possible 3 = no flow possible
ws	Initial weir setting table as a function of time; if not present the initial weir setting will default be equal to crest level;
fl	Inlet flushing flow. Default 0. Only used for inlet weirs.
so	table of switch-on and off levels. Only for inlet weirs.

For a gate:

STDS id '1' nm '3B-onderlaat' ty 10 in 0 dc1 cl -1.5 cp 0.5 cw 2.0 gh 1.0 mu 0.63 rt 0 gs 'init_setting' fl 0. stds

With:

id	structure definition identification
nm	name
ty	structure type (10=3B-gate)
in	inlet gate (0=no, 1=yes). Default 0.

mu	contraction coefficient. Default 0.63
dc	discharge coefficient. Default 1.0
cl	(bottom) crest level (m NAP); default=bottom level of upstream open water node, if available.
cw	crest width (m)
cp	power coefficient; default 0.5 **NOT READ. Always value 0.5 used.**
gh	gate height (opening height) in m NAP.
rt	0 = flow in both directions possible (default for normal weirs) 1 = only positive flow possible (default for inlet weirs) 2 = only negative flow possible 3 = no flow possible
gs	Initial gate setting table as a function of time; if not present the initial gate setting will default be equal to crest level;
fl	Inlet flushing flow. Default 0. Only used for inlet gates.
so	table of switch-on and off levels. Only for inlet gates.

For a Manning resistance:

STDS id ’1’ nm ’3B- weerstand’ ty 11 mn 0.02 cl 1000.0 dp 1.0 bw 3.0 ss 1.0 stds

With:

id	structure definition identification
nm	name
ty	structure type (11=3B-Manning resistance)
mn	Manning coefficient n
cl	characteristic length (m)
dp	depth wrt upstream reference level (in m)
bw	bottom width (m)
ss	slope talud (0=rectangular cross section, 1=slope of 45 degrees)

For a Q-h relation:

STDS id ’1’ nm ’3B-Q-h_relatie’ ty 12 qv -1.0 0. 0.5 1. 1.5 2. hv -0.1 0. 0.1 0.2 0.3 0.4 0.5 hr 0 stds

id	structure definition identification
nm	name
ty	structure type (12=3B-Q-h relation)
qv	6 values for Q (m3/s)
hv	6 corresponding values of h (m NAP)
hr	0 = use upstream level in Q-h relation 1 = use downstream level in Q-h relation 2 = use level difference in Q-h relation

D.19.13 Topography layer

[Topography layer]

The topography layer consists of three files which describe the topography of the network. The first file contains the list of all the nodes in the schematisation, the second file the list of links (connections) between these nodes. The third file is only used in case of Sobek-Urban models using the NWRW node type. The Topography layer in fact defines all the network objects of which the attribute data are described in the other layers.

Node file: 3b_nod.tp

This file contains the node definitions for Sobek-Rural applications. The header line of this file contains in the first positions the string ‘BBB2.2’. The keyword NODE is used to mark the beginning of a record, and in lower case characters it marks the end of a record.

NODE id ’3B72’ nm ’Node1’ ri ’2’ mt 1 ‘6’ nt 23 ObID ‘SBK_SBK-3B-REACH’ px 11404.2 py 123768.5 node

With

id	node identification
nm	name of the node
ri	branch identification
mt	model nodetype
nt	netter nodetype
ObID	Object id
px	position X (X coordinate)
py	position Y (Y coordinate)

Remarks :

Remark 1: no weather station in this file!

Remark 2: For aggregation purposes it is very well possible that this file should be extended with fields such as

au = administrative unit

rb = river basin

so that e.g. total areas or total waste loads can be determined later on by administrative unit and/or river basins. Also a ‘multi-select’ option of changing some input data for all nodes within one administrative unit, or setting default values per river basin, could be nice options for a user.

Remark 3: Another suggestion is to include a second name in the file, to be able to have the node names in two languages.

Remark 4: the following model types are available:

1=Paved area

2=Unpaved area

3=Greenhouse

4=Open water

(5=Internally reserved for all structures)

6=Boundary

7=NWRW

8=pump

9=weir

10=orifice

11=Manning resistance

12=Q-h relation

14=WWTP (RWZI)

15=Industry

16=Sacramento (ObId '3B_SACRAMENTO')

23= Wagmod/Walrus

Remark 5: the Object id is used by RR to distinguish RR-boundaries from RR-CF connection nodes. The RR model type of these different objects is the same (mt 1 '6') , but there Netter types and Object id's are different.

Link file: 3B_Link.Tp

This file contains the link definitions for Sobek-Rural applications. The header line of this file contains in the first positions the string 'BBB2.2'.

BRCH id '1' nm 'Tak1' ri '-1' mt 1 '0' bt 11 ObID '3B-LINK' bn '1' en '2' brch

With:

id	link identification
nm	name of the link
ri	branch identification
mt	model type
bt	branch type
ObID	Object identification
bn	identification of begin node ('from' node)
en	identification of end node ('to' node)

The branch identification is only used for special (aggregation) purposes of the NWRW rainfall-runoff model which is used in Sobek-Urban applications.

The branch type and Object Id are not used by the RR-computational core, but are used by user-interface programs (Netter). The model type of the links is used to check whether RR-routing links are used or not.

Runoff file: 3B_Runoff.Tp

This file contains the NWRW node definitions for Sobek-Urban applications.

NODE id '0-28' ri '1' mt 1 '7' nt 3 ObID 'SBK_CONN&RUNOFF' px 198002.5 py 457582.5 node

With

id	node identification
nm	name of the node
ri	branch identification
mt	model nodetype
nt	netter nodetype
ObID	Object id
px	position X (X coordinate)
py	position Y (Y coordinate)

The branch identification is only used for special (aggregation) purposes of the NWRW rainfall-runoff model which is used in Sobek-Urban applications.

D.19.14 RR-Routing link layer

Data file (3B_Rout.3b)

This file contains the data for the links of model type 30: RR-Routing links.

The file contains two type of records, ROUT and RDEF records. The ROUT records contain for each routing link a reference to the routing definition, while the RDEF records contain the routing definition.

ROUT id 'l_8' di '1' rout

ROUT id 'link_id' di 'def-id' rout

With

id	link identification
di	definition of routing id to be used

RDEF id '1' nm 'Donau' nl 2 x 0.3 0.12 k 2.7 4.06 qm 4000 qi 2400 qo 2750 rdef

With

id	routing definition id
nm	routing definition name
rt	routing type, type 0 = Muskingum
nl	number of layers
x	x coefficient per layer; there are as many coefficients as layers
k	k coefficient per layer; there are as many coefficients as layers
qm	maximum discharge Q per layer, except for the last layer
qi	initial link inflow

qo	initial link outflow

D.19.15 Unpaved area layer

Data file: Unpaved.3B

This file contains the data for the nodes of model type 2 (pervious or unpaved area)

UNPV id ’1’ na 16 ar 1. 0. 3. 0. 0. 6. 0. 0. 0. 10. 11. 12. 13. 14. 15. 16. lv 1.0 ga 110.
co 1 su 0 sd ’ovhstor_1mm’ ad ’alfa_1’ sp ’seepage_1’ ic ’infcap_5’ bt 1 0 ig 0 0.5
mg 0.8 gl 1.5 ms ’meteostat1’ is 100. Unpv

With

id	node identification
na	number of areas (at the moment fixed at 16)
ar	area (in m2) for all crops. In the user interface either the total area can be specified, or the different areas per crop. In case the total area is specified, it is put at the first crop (grass).
ga	area for groundwater computations. Default = sum of crop areas.
lv	surface level (=ground level) in m NAP
co	computation option (1=Hellinga de Zeeuw (default), 2=Krayenhoff van de Leur, 3=Ernst)
rc	reservoir coefficient (for Krayenhoff van de Leur only);
su	Indicator Scurve used su 0 = No Scurve used (Default) su 1 ‘Scurve-id’ = Scurve used; Unpaved.Tbl contains defniition of table with id ‘Scurve-id’.
sd	storage identification
ad	alfa-level identification (for Hellinga de Zeeuw drainage formula only)
ed	Ernst definition (for Ernst drainage formula only)
sp	seepage identification.
ic	infiltration identification
bt	soil type (from file BERGCOEF or BergCoef.Cap) Indices $>$100 are from Bergcoef.Cap.
ig	initial groundwater level; constant, or as a table ig 0 0.2 = initial groundwater level as a constant, with value 0.2 m below the surface. ig 1 ’igtable1’ = initial groundwater level as a table, with table identification igtable1.
mg	maximum allowed groundwater level (in m NAP)
gl	initial depth of groundwater layer in meters (for salt computations)
ms	identification of the meteostation
is	initial salt concentration (mg/l) Default 100 mg/l

Alfa definition file: Unpaved.alf

This file contains the definitions of alfa-factors and related levels for the nodes of model type 2 (unpaved or pervious area). Also the Ernst definitions are included in this file.

ALFA id ’alfa_1’ nm ’set1 alfa factors’ af 5.0 0.9 0.7 0.6 0.3 0.03 lv 0. 1.0 2.0 alfa

ERNS id ’Ernst_1’ nm ’Ernst definition set1’ cvi 300 cvo 30 30 30 cvs 1 lv 0. 1.0 2.0 erns

With

id	alfa-factors identification
nm	name
af	alfa factors (say a1 to a6) for Hellinga-de Zeeuw formula (1/day). a1 = alfa factor surface runoff a2 = alfa factor drainage to open water, top soil layer a3 = alfa factor drainage to open water, second layer a4 = alfa factor drainage to open water, third layer a5 = alfa factor drainage to open water, last layer a6 = alfa factor infiltration
cvi	Resistance value (in days) for infiltration from open water into unpaved area
cvo	Resistance value (in days) for drainage from unpaved area to open water, for 3 layers
cvs	Resistance value (in days) for surface runoff
lv	three levels below surface (say lv1, lv2, lv3), separating the zones with various alfa-factors (or Ernst resistance values) for drainage. a2 is used between surface level and lv1 m below the surface. a3 is used between lv1 and lv2 m below the surface. a4 is used between lv2 and lv3 m below the surface a5 is used below lv3 m below surface.

Storage definition file: Unpaved.sto

This file contains the storage definitions for the nodes of model type 2 (unpaved or pervious area)

STDF id ’ovhstor_1mm’ nm ’1 mm storage’ ml 1. il 0. STDF

With:

id	storage identification
nm	name
ml	maximum storage on land (mm). Default 1 mm.
il	initial storage on land (mm). Default 0.

Seepage definition file: Unpaved.sep

This file contains the seepage definitions for the nodes of model type 2 (unpaved or pervious area)

SEEP id ’seep_1’ nm ’constant seepage_1mm’ co 1 sp 1. ss 500. seep

SEEP id ’seep_2’ nm ’simple variable seepage’ co 2 cv 3.0 h0 ‘H0Table’ ss 500. seep

SEEP id ’seep_3’ nm ’variable seepage using coupling with Modflow’ co 3 cv 3.0 ss 500. seep

With:

id	seepage identification
nm	name
co	computation option seepage 1 = constant seepage (Default) 2 = variable seepage, using C and a table for H0 3 = variable seepage, using C and H0 from Modflow

	If the co field is missing, co 1 will be assumed.
sp	Seepage or percolation (mm/day) Positive numbers represent seepage, negative numbers represent percolation. Default 0.
ss	salt concentration seepage (mg/l). Default 500 mg/l. This value is only important for positive seepage values.
cv	Resistance value C for aquitard
h0	reference to a table with H0 values Remark 1: This file has similar structure as a file of the open water layer.

Infiltration file: Unpaved.inf

This file contains the infiltration definitions for the nodes of model type 2 (unpaved or pervious area)

INFC id 'infcap_5' nm 'inf_5mm' ic 5. INFC

With:

id	infiltration identification
nm	name
ic	infiltration capacity of the soil, constant. (mm/hour) Remark: no variable infiltration capacity implemented yet.

Table file: Unpaved.tbl

This file can contain three kinds of tables:

- ◇ initial groundwater level as a function of time
- ◇ S curve definitions
- ◇ H0 definitions for variable seepage

Table input of the initial groundwater level as function of time.

```
IG_T id 'igtable1' PDIN 1 1 '0;10:00:00' pdin
TBLE
1995/01/01;00:00:00 0.3 <
1995/03/01;00:00:00 0.2 <
1995/05/01;00:00:00 0.3 <
1995/07/01;00:00:00 0.4 <
1995/09/01;00:00:00 0.5 <
1995/11/01;00:00:00 0.4 <
tble
ig_t
```

With

id	table identification

PDIN ..pdin = period and interpolation method

0 0 ' ' = interpolation continuous, no period

1 0 ' ' = interpolation block, no period

0 1 '365;00:00:00' = interpolation continuous, period in DDD;HH:mm:ss

1 1 ’1;00:00:00’ = interpolation block, period in DDD;HH:mm:ss

TBLE .. tble contains the table, first column is the date (year month day hour minute second), second column the initial groundwater level in m below surface. The < sign is a separator.

S-curve definitions:

This table is not time depending, so different from the other types of tables in this file.

SC_T id ’ScurveTable’ nm ’ScurveTable1’ PDIN 1 1 ’0’ pdin
TBLE
0 -5.1 <
10 -5.0 <
80 -4.9 <
90 -4.2 <
100 -4.0 <
tble
sc_t

H0 table definitions:

H0_T id ’H0table’ PDIN 1 1 ’1;00:00:00’ pdin
TBLE
1995/01/01;00:00:00 0.4 <
1995/02/01;00:00:00 0.5 <
1995/03/01;00:00:00 0.5 <
1995/04/01;00:00:00 0.4 <
1995/05/01;00:00:00 0.3 <
1995/06/01;00:00:00 0.2 <
1995/07/01;00:00:00 0.1 <
1995/08/01;00:00:00 0.0 <
1995/09/01;00:00:00 0.0 <
1995/10/01;00:00:00 0.1 <
1995/11/01;00:00:00 0.2 <
1995/12/01;00:00:00 0.3 <
tble
h0_t

With:

id	table identification

PDIN ..pdin = period and interpolation method (for description see above at IG_T record)

D.19.16 WWTP layer

The RWZI layer represents nodes of model type 14 (WWTP's or waste water treatment plants).

Table file (WWTP.Tbl)

Contains measured WWTP discharge flow (m3/s)

```
MEAS id 'WWTPTable' nm 'Test' PDIN 0 0 ' ' pdin
TBLE
1997/01/01;00:00:00 -1.0 <
1997/01/01;00:00:01 0.5 <
1997/01/02;00:00:00 0.55 <
1997/01/06;00:00:00 0.55 <
1997/01/06;00:00:01 0.5 <
1997/01/08;00:00:00 0.6 <
1997/01/10;00:00:00 0.6 <
1997/01/11;00:00:00 -1.0 <
1997/01/20;00:00:00 -1.0 <
1997/01/20;00:00:01 0.5 <
1997/01/30;00:00:00 0.54 <
1997/02/01;00:00:00 -1.0 <
tble
meas
```

With

id	table identification
nm	table name

PDIN ..pdin = period and interpolation method

0 0 ' ' = interpolation continuous, no period

1 0 ' ' = interpolation block, no period

0 1 '365;00:00:00' = interpolation continuous, period in DDD;HH:mm:ss

1 1 '1;00:00:00' = interpolation block, period in DDD;HH:mm:ss

Convention: Missing data is indiciated with -1. In that case the discharge of the WWTP is equal to the computed inflow of the WWTP.

No salt concentration included in input table, is derived from inflow.

Data file (WWTP.3b)

This file contains the data of the WWTP node (RR model type 14).

```
WWTP id '1' tb 0 wwtp
```

With

id	node identification
tb	table used yes/no

tb 0 = no table of measured data; the WWTP outflow is equal to the sum of the inflows.
tb 1 ‘WWTPTable’ = table of measured data with id ‘WWTPTable’

D.20 RTC (Real Time Control)

General

This document only describes the files of Real-time Control which can be edited by the user. These files are indicated with an asterisk in the table.

The rainfall and wind files other input files which are editable by the user. These files are described in the Rainfall-Runoff and Water Flow model database description.

Filename	**File description**	**Input/ Output file**	**User editable**
RTC.FNM	Filename file	I	
RTC.INI	INI file with options	I	✓
SBK_LOC.RTC	File with Water Flow data locations available for Real-time Control	I	✓
3B_LOC.RTC	File with Rainfall-Runoff data locations available for Real-time Control	I	✓
PREC_LOC.RTC	File with data locations for rainfall prediction	I	✓
EXT_LOC.RTC	File with other data locations (e.g. wind prediction)	I	✓
Not used anymore		I	
Not used anymore		I	
DECISPAR.RTC	Decision parameter definitions	I	✓
SBK_MEAS.RTC	Water Flow measure definitions	I	✓
3B_MEAS.RTC	Rainfall-Runoff measure definitions	I	✓
SBK_RTC.HIS	Communication file with results from Water Flow data locations to Real-time Control	I	
3B_RTC.HIS	Communication file with results from Rainfall-Runoff data locations to Real-time Control	I	
PRE_RTC.ASC	Communication file with results from Sobek-Predict data locations to Real-time Control.	I	

Filename	File description	Input/ Output file	User editable
EXT_RTC.ASC	Communication file with results from other data locations to Real-time Control	I	
MONSTAT.WQ	File with available WQ monitoring locations	I	✓
MONWQ.HIS	Communication file with results from WQ monitoring locations to Sobek-RTC	I	
RTC_3B.OUT	Communication file with results from Real-time Control to Rainfall-Runoff	O	
RTC_SBK.HIS	Communication file with results from Real-time Control to Water Flow	O	
RTC_OUT.MSG	Communication file with results from Water Flow data locations to Real-time Control	O	
RTC3BALL.HIS	All output to Rainfall-Runoff	O	
RTCSBALL.HIS	All output to Water Flow	O	
RTCPARAL.HIS	All values of decision parameters	O	
RTC.DBG	Debug file of Real-time Control	O	
*.BUI or *.RKS	Rainfall file	I	Meteo task block
*.WDC or *.WND	Wind file	I	Meteo task block
PRE_RAIN.HIS	Rainfall prediction HIS file	O	
PRE_WND.HIS	Wind prediction HIS file	O	
CTRL.INI	Ini file for control module to enable on-line coupling of Real-time Control with other SOBEK modules.	I	

rtc.ini-file (Control layer)

This file contains some general options in a Windows-ini file type structure. The file contains general options such as version number, date, options indicating how Real-time Control is used in combination with other modules, the time control of Real-time Control, etc. An example file looks like this:

```
[General]
Version=1.00
```

```
Date = June 1997
[Options]
3B=-1 (with SOBEK-Rural RR)
 -1=On;
 0=Off;
 input either by 0,-1,or On or Off.
Precipitation=-1
Extern=-1
Sobek=-1 (-1 = with a SOBEK 1DFLOW module)
WQ = -1 (-1 = with the SOBEK-Rural 1DWAQ module)
Control=INI
 !available options:
 3B=Event dates/times from Rainfall file;
 INI = time control via this INI file
3Bformat=ASC
 !options:
 HIS = HIS format,
 ASC = ASCII format .
Sbkformat=HIS
WQformat=HIS
ModePrecipitation=Actual
 Available option:
 !Actual = prediction is equal to actual precipitation according to rainfall file
ModeWind=Actual
 Available option:
 Actual = prediction is equal to actual wind data
PrecipTimeHorizon=5
 Max. number of time steps ahead precipitation prediction
WindTimeHorizon=5
 Max. number of time steps ahead for wind prediction
DecisionHorizon=720
 Max. number of time steps memory for decision
 parameters. Default=720
Debug=0
 0=no debug file;
 -1=debug file is generated.
DebugTime=1 2
 DebugFromTimestep, DebugToTimestep
DebugTime2=101 102
 DebugFromTimestep2, DebugToTimestep2
OutputTimestep=1
 Output every x timesteps; default OutputTimestep=1
WriteRtcHisFiles=0
 0=only writes overall HIS files for event 1
 -1=writes overall his files for all events (default)
SetSequenceDecisionParameters=0
 0 = (default) = by order in input file
 -1 = determined by RTC
WindUseTableModule=-1
 0=no (default),
 -1=yes
ReduceWindTable=-1
 0=no,
 -1=yes (default); only reads part of the wind table within the specified
 simulation period + time horizon prediction.
NLocHis=99
 Maximum number of locations in on-line HIS files, default 9999.
NTimHis=99
 Maximum number of timesteps from external HIS files, default  365*50*24.
NParQ=99
 Maximum number of on-line available Water Quality parameters, default value 500.
FormatMeasureFiles209003=-1
 0=False (Default), measure files are read according to the new format
 -1=true, measure files are read according to the old 2.09.003 format
ReservoirMaxIterations=10
 Maximum number of reservoir iterations, default value 10.
```

```
ReservoirVolumeConvergenceCriterion=0.1
 Volume convergence criterion for reservoir, default value 0.1 m3.
ReservoirFlowConvergenceCriterion=0.001
 Flow convergence criterion for reservoir, default value 0.001 m3/s.
[Control]
 Time step size etc.\ always through this data block;
 Event data either here or through 3B Rainfall file.
NumberofEvents=1
 Number of events
NumberofStepsperkeer=1
 Should be 1
Deltat=0.010000
 (Time step size; cf. definition of Time start below)
Timestart=19940901.000000
 Start time per event
 (year*10000+month*100+day+ ihour/100.+iminute/10000.+isecond/100000.)
LastTimeStep=8
 Last time step of event
[CouplingMatlab]
MatlabMFileDir=Matlabdir
 Matlab directory containing the input file
MatlabMFileName=matlabfile
 Matlab input file name
MatlabDebugMode=1
 Debug option (1=true)
MatlabRRData=0
 0 = False (default);
 1 = also RR open water levels are passed to MATLAB.
MatlabRainData=0
 0 = False (default)
 1 = also Rainfall data are passed to MATLAB.
MatlabWQData=0
 Default 0=False.
 If =1 then a selection of WQ data are passed to MATLAB.
MatlabNrWqPar=
 Number of WQ parameters passed to MATLAB.
 MatlabWQPar1= id of WQ parameter 1 to be passed to MATLAB
 MatlabWQPar2= id of WQ parameter 2 to be passed to MATLAB
 etc.\ up to the number of WQ parameters passed to MATLAB.
```

MatlabWithSobekCString=0

0= False. In each RTC time step the MATLAB set points will be used and transferred to the SOBEK Water Flow and Rainfall-Runoff modules (default).

1= True. RTC will also request the SobekC en RRC strings from MATLAB. SobekC and RRC can be used to specify whether the MATLAB values should be used or not, giving the option for controlling at other time intervals. In RTC versions before March 2001 this option was default.

MaxCountGeneral=100

Maximum number of items (RR data, WQ data, precipitation data) to be passed in 1 string to Matlab.

MaxCountStructures=15

Maximum number of Flow-structures for which Sobek-Flow data is passed in 1 string to Matlab.

MaxCountBranchSegments=50

Maximum number of Sobek-branchsegments for which data is passed in 1 string to Matlab.

MaxCountNodes=50

Maximum number of Sobek-nodes for which data is passed in 1 string to Matlab.

MaxCountBiLcLocations=30

Maximum number of Sobek-branch id point locations for which data is passed in 1 string to Matlab.

MaxCountMeasLocations=30

Maximum number of Sobek-Measurement stations for which data is passed in 1 string to Matlab.

D.20.1 Data Locations layer

sbk_loc.rtc-file (Data locations layer)

This file specifies the locations containing SOBEK data. These locations can subsequently be used in specifying decision parameters in the RTC module.

The general format of available data records are:

SBKO id 'Sobek-id' nm 'Sobekname' bi 'branch-id' lc 25.5 sbko
SBKO id 'meetid' nm 'name' in 'Node_id' sbko
SBKO id 'meetid' nm 'name' ir 'Branchsegment_id' sbko
SBKO id 'meetid' nm 'name' is 'Structure_id' sbko
SBKO id 'meetid' nm 'name' ml 'MeasLoc_id' sbko
SBKO id 'meetid' nm 'name' hl '2DHistory location' sbko
SBKO id 'meetid' nm 'name' db '2D Breaking dam location' sbko where:

id	id of data location
nm	name, which can be used as additional comment
bi	SOBEK-branch id
lc	location on the branch (in meters)
in	node id
ir	branch-id
is	] structure-id
ml	measurement location-id
hl	2D-history location-id
db	2D-breaking dam location-id

Notice that 'in' can only be used to measure a water level (on nodes or calculation points), 'ir' can only be used to measure a discharge (on branch segments), 'is' can only be used to measure the control structure parameter of a structure (crest level for weir, opening height for orifice, discharge for pump and valve opening for culvert) and 'ml' can be used te measure a discharge and/or water level, depending on the type of measurement station.

However, the user should still specify the correct variable (1=water level, 2=discharge, 3=structure parameter) in the definition of the decision parameter (see file description decision parameter file).

Notice that 'ir' is focused on the id of the first part of the branch segment.

Optionally, the data file may contain a header record. A header record either starts with an asterisk '*' or with the 3 characters 'RTC'.

An example file:

RTC1.0
SBKO id 'Sbk_meas_id1' nm 'Sobek_lokatie_1' bi 'branch_id_1' lc 25.5 sbko
SBKO id 'Sbk_meas_id2' nm 'Sobek_lokatie_2' bi 'branch_id_2' lc 0. sbko
SBKO id 'meetid' nm 'name' is 'Structure_id' sbko

3b_loc.rtc-file (Data locations layer)

This file contains the data-locations of Rainfall-Runoff module which are available in Real-time Control. These locations can be used in the definition of Real-time Control decision parameters.

The general format of a record is as follows:

3BO id '3B-id' nm '3B-name' 3bo

where:

id	id of data location
nm	name

Optionally, the data file may contain a header record. A header record either starts with an asterisk '*' or with the 3 characters 'RTC'.

An example file:

RTC1.0
3BO id '3B_id1' nm '3B_naam_1' 3bo
3BO id '3B_id2' nm '3B_naam_2' 3bo
3BO id '3B_id3' nm '3B_naam_3' 3bo

prec_loc.rtc-file (Data locations layer)

This file contains the data-locations of SOBEK-RR which are available in SOBEK-RTC. These locations can be used in the definition of SOBEK-RTC decision parameters.

The general format of a record is as follows:

PREC id 'precipitation-id' nm 'precipitation-name' prec where:

id	id of precipitation location
nm	name

Optionally, the data file may contain a header record. A header record either starts with an asterisk '*' or with the 3 characters 'RTC').

An example file:

RTC1.0

PREC id 'Neerslag1' nm 'Neerslag1' prec
PREC id 'Neerslag2' nm 'Neerslag2' prec

ext_loc.rtc-file (Data locations layer)

This file contains the other external data locations. The file can contain the wind predictions computed in the Real-time Control module, and locations related to external HIS files. Wind information concerns wind direction and wind velocity.

These external data locations can be used to define decision parameters.

The general format of a data record is:

EXT id 'external-id' nm 'external-name' ext

HEXT id 'BoezemlastDZV_Sacr' nm 'BoezemlastDZV_Sacr' hf '..\fixed\Sacrmnto.His' hl '1427' hp 'ChannelInflow [m^3/s]' hext where:

id	id of external data location
nm	name
hf	external HIS filename
hl	location from external HIS file
hp	parameter from external HIS file

Optionally, the data file may contain a header record. A header record either starts with an asterisk '*' or with the 3 characters 'RTC'.

An example file:

RTC1.0
EXT id 'Global wind' nm 'global wind data' ext
EXT id 'Extern2' nm 'Extern serie2 for temperature' ext

monstat.wq-file (Data locations layer)

This file contains the monitoring data locations defined in the water quality module. The file may be empty or non-existent. The format is different from other data location files, since an already existing input file of the water quality module is used.

The file contains a header with the lines:

MON1.0

Monitoring area's

Branch = 0

The next line contains the number of monitoring locations. Then, for each monitoring location some lines are included. First, a line with:

Area nr Name: id

where:

nr	number of the monitoring location

id	id of WQ monitoring location

and then some lines with additional data, not used by RTC.

An example file:

```
MON1.0
# Monitoring area's
Branch = 0
5
Area 1 Name: WVO006
1,"l_307"
0
Area 2 Name: WVO023
1,"l_317"
0
Area 3 Name: BLV002
1,"l_255"
0
Area 4 Name: GHC002
1,"l_159"
0
Area 5 Name: WVO001
1,"l_389"
0
```

D.20.2 Decision layer

The decision parameter file contains the definition of the decision parameters. There are four possible ways of defining decision parameters:

- ◇ A first possible definition of a decision parameter is a linear combination of Water Flow results, Rainfall-Runoff results, predicted rainfall or other external (wind) data, on-line water quality data, and other decision parameters.
- ◇ A second option is to define decision parameters as a non-linear function of other decision parameters.
- ◇ A third option is to define decision parameters as SOBEK tables.
- ◇ The fourth option is to define SOBEK reservoirs.

Furthermore, there are some standard predefined decision parameters which can be used to define other decision parameters or measures. These predefined decision parameters are:

- ◇ Year
- ◇ Month
- ◇ Day
- ◇ Hour (an integer between 0 and 23)
- ◇ Minute (an integer between 0 and 59)
- ◇ Second (an integer between 0 and 59)
- ◇ Date (a number defined as 10000*year+100*month+day)
- ◇ Time (a number defined as 10000*hour+100*minute+second)
- ◇ Date_Time (the number defined as date + time/10000)
- ◇ Day of Week
  0=Sunday;
  1=Monday;

..
6=Saturday;

DECISPAR.RTC-file (Decision layer)

Function Decision Parameters

The value of decision parameters is based on data locations and also may be dependent of other decision parameters. These decision parameters are specified in the PAR2 records. In the User Interface these records are presented under Decision Parameters as Function Parameters.

These data records may look like:

PAR2 id 'beslispar_1' nm 'para_1' iv 1 do 'None'

DATA ty 'ExtLoc' lo 'Extern_id' va 1 ca 0.1 cb -1 cn 0 data par2

or

PAR2 id 'beslispar_2' nm 'para_2' do 'Interpolate'

DATA ty '2DFlowLoc' lo '2D_Weurt' va 1 ca 1 cb 0 cn 0 data

DATA ty 'InterpolationTable' lo 'InpTable_1' va 1 ca 0 cb 0 cn 0 data par2

or

PAR2 id 'beslispar_3' nm 'para_3' do 'multiply'

DATA ty 'FlowLoc' lo 'Sobek-id1' va 1 ca 1. cb 0. cn 0 data

DATA ty 'RRLoc' lo '3b-stuw' va 3 ca 1. cb 0. cn 0 data

DATA ty 'PrecipLoc' lo 'Neerslag-id' va 1 ca 1. cb 0. cn 0 data

DATA ty 'RRLoc' lo '3B-ow' va 1 ca 0.5 cb 0. cn 0 data

DATA ty 'ParLoc' lo 'beslispar_1' va 1 ca 0 cb -1 cn -1 data

DATA ty 'ExtLoc' lo 'Extern_id' va 1 ca 0.1 cb -1 cn 0 data

DATA ty 'FlowLoc' lo 'Sobek-id2' va 2 ca. 2. cb. 1. cn -1 data par2 where:

id	id of decision parameter
nm	name of decision parameter
iv	initial value to be used at beginning of simulation when parameter is used in computations with negative time shift; the default initial value is 0 for all parameters
do	mathematical actions or functions on location value[s]

The following functions (or mathematical actions) can be defined in PAR2 records:

Functions that can be applied using one argument only:

- ◇ Arccosine (result in degrees)
- ◇ Arccosine (result in radians)
- ◇ Arcsine (result in degrees)
- ◇ Arcsine (argument in radians)
- ◇ Arctangent (result in degrees)
- ◇ Arctangent (result in radians)
- ◇ Ceiling (smallest integer $>=$ x)
- ◇ Cosine (input argument in degrees)
- ◇ Cosine (argument in radians)
- ◇ Exponent
- ◇ Floor (largest integer $<=$ x)
- ◇ Hyperbolic cosine
- ◇ Hyperbolic sine
- ◇ Hyperbolic tangens
- ◇ Logarithm with base 10
- ◇ Natural logarithm
- ◇ Nearest integer
- ◇ None (no function is applied)
- ◇ Sine (input argument in degrees)
- ◇ Sine (argument in radians)
- ◇ Square
- ◇ Square root
- ◇ Tangent (input argument in degrees)
- ◇ Tangent (argument in radians)

Functions that can be applied for two arguments only:

- ◇ Interpolate (determine values using an Interpolation table)

Functions that can be applied for two or more arguments:

- ◇ Add
- ◇ Average
- ◇ Divide
- ◇ Max
- ◇ Min
- ◇ Multiply
- ◇ Power
- ◇ Subtract

The PAR2 record contains DATA sub-records with data for the specified locations. The data items will appear in the same order as specified in the PAR2-record, so the order may influence the results, depending on the specified action function.

The DATA sub-record contains the following keywords:

ty	Type of location, which can be: ‘FlowLoc’ = Water Flow Location in 1D schematization ‘RRLoc’ = Rainfall-Runoff Location ‘ExtLoc’ = External data from file ‘PrecipLoc’ = Precipitation Prediction Location ‘DateTimLoc’ = Date Time ‘ParLoc’ = (Other) Decision Parameter ‘WQLoc’ = Sobek-WQ Monitoring Location ’2DFlowLoc’ = Water Flow Location in 2D schematization ’InterpolationTable’ = Interpolation Table
lo	Location-id (defined in RTC on the “Data Location” Tab)
va	Selected variable, which has different values for the different data types:

Water Flow Location in 1D schematization:
Node:
1 = Water Level
4 = Water Depth
Reach Segment:
2 = Discharge
Structure:
5 = Crest Level
6 = Crest Width
7 = Gate Lower Level
8 = Opening Height
9 = Structure Flow Area
10 = Discharge Structure
11 = Structure Velocity
12 = Water Level Up
13 = Water Level Down
14 = Head
15 = Pressure Difference
16 = Pump Capacity
Measurement Location:
1D Flow Measurement Station:
1 = Water level
2 = Discharge
3 = Storage Surface Area
4 = Water Depth
Manhole Measurement Station:
1 = Water level
3 = Storage Surface Area
4 = Water Depth
Pipe Measurement Station:
2 = Discharge
Reach Location:
1 = Water level
2 = Discharge
3 = Storage Surface Area
4 = Water Depth
Water Flow Location in 2D schematization:
2D History Station & 2D Breaking - Dam:
1 = Water level
4 = Water depth
17 = Bed level
18 = U-velocity
19 = V-velocity
20 = C-Abs. flow velocity $[= \sqrt{U^2 + V^2}]$
Rainfall-Runoff Location
1 = Open Water Level
2 = Groundwater Level
External data
From HIS-file
1 = Value
In case of Public Wind
1 = Wind Direction
2 = Wind Velocity
Precipitation Prediction Location

1 . . . n = Prediction period in number of time steps
Decision Parameter
1 = Parameter Value
Date Time
1 = Actual Value, that can be
Integer value of:

- ◇ Year(yyyy),
- ◇ Month(mm),
- ◇ Day(dd),
- ◇ Hour(hh),
- ◇ Minute(mm),
- ◇ Second(ss),
- ◇ Date(yyyymmdd),
- ◇ Time(hhmmss),
- ◇ Day_of_week(day no)

Real value of

- ◇ Date+Time(yyyymmdd.hhmmss)
- ◇ CompTimeStep (Timestep as defined in RTC settings)

Sobek-WQ Monitoring Location
SOBEK WQ variable (series id as character string)

ca	coefficient for multiplication
cb	constant for addition
cn	time shift: integer values 1 represents: t+1 0 represents: t -1 represents: t-1 -2 represents: t-2 etc. When using negative time shifts, the value used for the first time steps will be equal to the initial value of that parameter (see the iv keyword).

The value of the decision parameter is determined by executing the specified action over the calculated values of all locations.

The time indexes are t, $t-1$, $t-2$. At the moment the time shift is limited to at most 720 time steps (based on 5 days memory of a computation time step of 10 minutes). This can be specified in the INI file (keyword `DecisionHorizon`).

By using different time indices, a moving average or a trend can be determined. For instance, by defining as a decision parameter: the water level at time step t minus the water level in time step $t-1$, in fact the increase in water level will be determined. If the water level rises quickly, you may want to take some measure. Real-time Control will remember the data values of all data locations of the previous time steps; the communication with the other modules is such that it only receives the values for the current time steps.

When you are using SOBEK-RTC in combination with WQ, you could define the decrease in O_2 concentration, or the (increase) in fraction of WWTP water in the system as decision parameters, and define measures to set structure set points in the SOBEK Water Flow modules, based on these (water quality based) decision parameters.

The coefficients for multiplication and addition can be positive, negative or zero. This gives many possibilities in the definition of decision parameters. Even the definition of a constant decision parameter is possible (specify zero for the multiplication value, and set the proper addition value)

The time shifts can be zero or negative, meaning that the decision parameter is depending on current or previous values of variables at data locations. If a decision parameter is depending on other decision parameters, it should depend on values of previous time steps (since the values for the current time step may not yet be defined). In that case only negative values for the time shift are allowed.

PAR2 records may refer to decision parameter values for the current timesteps or for previous timesteps. When defining decision parameters depending on values of other decision parameters in the same timesteps, dependency loops should be avoided.

The first set of functions can accommodate multiple arguments; in case of 1 argument (only 1 input decision parameter), the result is equal to that decision parameter. In case of 2 input decision parameters a and b, the result is obvious. In case of 3 input decision parameters a,b, and c, the result for the maximum operation will be: max (max(a,b),c)) which is the same as max(a,b,c); the result of the power operation will be ((a**b)**c).

The result of the computations is as the name suggests; using the short notation d_1 for decision parameter 1, being equal to $di_1 * dc_1 + da_1$, the results are:

multiply: d1*d2
divide : d1 / d2
add: d1 + d2
subtract: d1 - d2
max: max (d1, d2)
min: min (d1, d2)
average: average (d1, d2) = add (d1, d2) / 2
power: power(d1,d2) = d1 ** d2

The generalisation to more than 2 arguments is obvious, e.g.

subtract (d1, d2, d3, d4) = d1- d2 - d3 - d4
divide (d1, d2, d3, d4) = (((d1 / d2) / d3) / d4)
average (d1, d2, d3) = add (d1, d2, d3) / 3 = (d1+d2+d3) / 3

Some example records:

PAR2 id 'PreviousLvlShimen' nm " do 'none'
DATA ty 'FlowLoc' lo 'Lvl Lower Shimen' va 1 ca 1 cb 0 cn -1 data par2
PAR2 id 'ExpectedRain402001_7' nm " do 'add'
DATA ty 'PrecipLoc' lo '402001_7' va 1 ca 0.166 cb 0 cn 0 data
DATA ty 'PrecipLoc' lo '402001_7' va 2 ca 0.166 cb 0 cn 0 data
DATA ty 'PrecipLoc' lo '402001_7' va 3 ca 0.166 cb 0 cn 0 data
DATA ty 'PrecipLoc' lo '402001_7' va 4 ca 0.166 cb 0 cn 0 data
DATA ty 'PrecipLoc' lo '402001_7' va 5 ca 0.166 cb 0 cn 0 data
DATA ty 'PrecipLoc' lo '402001_7' va 6 ca 0.166 cb 0 cn 0 data par2
PAR2 id 'ExpectedInflowShimen' nm " do 'max'
DATA ty 'ParLoc' lo 'PreviousInflowShimen' va 1 ca 1 cb 0 cn 0 data
DATA ty 'ParLoc' lo 'ExpectedRunoffShimen' va 1 ca 0.5 cb 0 cn 0 data par2
PAR2 id 'Downstream demands Shimen' nm " do 'add'
DATA ty 'ParLoc' lo 'DemandTaoYuan' va 1 ca 1 cb 0 cn 0 data

DATA ty 'ParLoc' lo 'PWS Shimen' va 1 ca 1 cb 0 cn 0 data
DATA ty 'ParLoc' lo 'Irr Demand Shimen' va 1 ca 1 cb 0 cn 0 data par2

As the example indicates, records may be more than one line long. The beginning of a record is indicated by PAR2 in capitals (upper case), and the end of a record is indicated by par2 in lower case.

Time Decision Parameters

A third way for defining decision parameters is by defining SOBEK time-tables. In this way, a decision parameter is simply defined as a standard SOBEK time-table. SOBEK-RTC allows tables for one decision parameter at a time only. The table can contain interpolation and periodicity switches.

The tables are defined using the PAR3 record.

PAR3 id 'MaxFlowOutlet2TimeTable' PDIN 1 1 365;00:00:00 pdin
TBLE
'1995/01/01;00:00:00' 9999. <
'1995/02/15;00:00:00' 500. <
'1995/03/01;00:00:00' 9999. <
'1995/06/01;00:00:00' 750. <
'1995/12/31;23:59:00' 750. <
tble par3

PAR3 id 'Cons.DemandOutlet1TimeTable' PDIN 1 1 365;00:00:00 pdin
TBLE
'1995/01/01;00:00:00' 0. <
'1995/03/01;00:00:00' 60. <
'1995/12/31;23:59:00' 60. <
tble par3

Where:

id id of decision parameter

PDIN .. pdin= option for interpolation and periodicity

= 1 1 365;00:00:00 means block functions, periodicity one year

= 0 0 means linear interpolation, no periodicity

= 0 1 365;00:00:00 means linear interpolation, period one year

= 1 0 means block function, no periodicity

TBLE .. tble = the table, containing date; time string, value, and

Interpolation Tables

Interpolation tables can be applied by an “Interpolate” function, that determines the values of a particular “Interpolate Function Decision Parameter”. Values for the independent variable are contained in the first column of an interpolation table, while values for the dependent variable are contained in the second column. For the interpolation table given below yields that an input (independent) value of 0.15 results in an output (dependent) value of 0.025. Independent values may be contained in any data series defined in RTC. The dependent values will be assigned to the concerning “Interpolate Function Decision Parameter.

Note that:

1 Independent values in the Interpolation Table (i.e. the values in the first Table column) should be given in ascending order,
2 If the independent value is less than the independent value on first Table row, than the dependent value will be equal to the dependent value on the first Table row,
3 If the independent value is larger than the independent value on the last Table row, than the dependent value will be equal to the dependent value on the last Table row.

```
INTP id 'InpTable1' nm 'InterpolationTable1' v1 'Heading_v1' v2 'Heading_v2'
TBLE
0.0 0.0 <
0.1 0.0 <
0.2 0.05 <
0.3 0.15 <
0.4 0.25 <
0.5 0.25 <
tble intp
```

where:

id	id of interpolation table
nm	name of interpolation table
v1	heading of column 1 (Independent input variable)
v2	heading of column 2 (Dependent variable)

SOBEK reservoir

SOBEK offers functionality of modelling reservoirs. This is done by defining a RSVP record in the decision parameter file, and detailed reservoir information in the reservoir input file. An example of the RSVP record is described in this subsection and the reservoir file (Reservoi.Rtc) is described in the next subsection.

An example of the RSVP record:

RSVP id 'Res1' nm ' ' nb 2 nq 5 nt 2 ns 1 hav 'HeadVolumeReservoir1'

rule 'RuleCurveRsv1' hedg 'HedgingRule'

bg 'Bottom gate Definition' tg 'Turbine gate Definition' sg 'Spillway gate Definition'

dp 'RsvRes1BotGate1' 'RsvRes1BotGate2' 'RsvRes1TurbGate1'

'RsvRes1TurbGate2' 'RsvRes1SpillGate1'

no 2 gb 1 2 gt 1 2 gs 2 dm 'Cons.DemandOutlet1TimeTable'

'Cons.DemandOutlet2'

mf '9999' 'MaxFlowOutlet2TimeTable' il 'Initial RsvLvl1' ei 'Expected Inflow1' rsvp

Where:

id	reservoir id
nm	name of the reservoir
nq	total number of gates
nb	number of bottom gates
nt	number of turbine gates
ns	number of spillway gates SOBEK will check that nq=nb+nt+ns
dp	id's of decision parameters (flows over individual gates)
hav	reference to head-area-volume curve definition
rule	reference to rule curve definition
hedg	reference to hedging rule definition
il	reference to decision parameter representing initial reservoir level in m (initial= at begin of time step)
ei	reference to decision parameter representing expected inflow (in m3/s)
no	number of outlet links (check: no $<=$ ng)
gb	assignment of bottom gates to outlet links
gt	assignment of turbine gates to outlet links
gs	assignment of spillway gates to outlet links
mf	reference to maximum flow definition for each outlet link
dm	reference to demand definition for each outlet link

Combination of different type of decision parameters
In the decision parameter file, all type of decision parameters (PARA, PAR2, PAR3, and RSVP) records can be used. There is no specific order of specification required. It is possible to use decision parameters inRSVP records (i.e. the discharges over the different gates) as 'input' decision parameters in PAR2 records.

Also the RSVP decision parameters may depend on other parameters (initial reservoir level, expected inflow, consumptive demands on each downstream outlet) which itself may be defined as decision parameters in PARA, PAR2, PAR3, or even other RSVP records.

Default RTC will compute the decision parameters in the order in which they are specified in the input file. However, SOBEK-RTC can also determine the order of computations itself (use the option SetSequenceDecisionParameters=-1 in the Ini file RTC.DAT). RTC will check whether cyclic definitions of decision parameters occur. It is not allowed to define decision parameter A depending on decision parameter B, decision parameter B depending on C, and C on A for the same time step. Such cyclic chains of dependencies are only allowed if somewhere a time delay is introduced (like the value of decision parameter A at time t is depending on the value of decision parameter B at time $t - 1$).

reservoi.rtc-file (Decision layer)

This file contains the detailed information of the SOBEK-reservoirs. The information is related to the RSVP records in the decision parameter file (DECISPAR.RTC). The information in the reservoir input file consists of:

- ◇ level-area-volume curve of the reservoir (HAVC record)
- ◇ reservoir rule curves (RULE record)
- ◇ bottom gate data (BOTG record)
- ◇ turbine data (TURB record)
- ◇ spillway data (SPIL record)
- ◇ Q-h relation data for individual gates (QHRE record)

- ◇ maximum flow data for individual gates (MAXF record)
- ◇ flow demands (for energy generation) for individual turbine gates (ENGD record)
- ◇ hedging rule (HEDG record)

Each of these records is described below.

An example of each record with a description:

```
HAVC id 'HeadVolumeReservoir1'
TBLE
0100000000
1010000000100000000
2010000000200000000
3010000000300000000
4010000000400000000
5010000000500000000
tble havc
```

Where:

id	id of the head-area-volume curve relation

TBLE .. tble contains the relation;

the first column is the level (in m with respect to reference level),

the second column is the area [m^2]

the third column is the volume (m3)

```
RULE id 'RuleCurveRsv1' PDIN 0 1 365;00:00:00 pdin
TBLE
'1995/01/01;00:00:00' 50. 45. 40.
'1995/02/01;00:00:00' 50. 45. 40.
'1995/03/01;00:00:00' 50. 45. 40.
'1995/04/01;00:00:00' 50. 45. 35.
'1995/05/01;00:00:00' 50. 40. 30.
'1995/06/01;00:00:00' 45. 35. 25.
'1995/07/01;00:00:00' 40. 30. 20.
'1995/08/01;00:00:00' 35. 30. 20.
'1995/09/01;00:00:00' 35. 30. 20.
'1995/10/01;00:00:00' 40. 35. 25.
'1995/11/01;00:00:00' 45. 40. 30.
'1995/12/01;00:00:00' 50. 45. 35.
'1995/12/30;00:00:00' 50. 45. 40.
tble rule
```

Where:

id	id of the rule curve

TBLE .. tble contains the rule curve time table;

the first column is the flood control level;

the second column is the target level;

the third column is the firm level;

(all levels in m with respect to reference level)

the flood control level $>=$ the target level $>=$ the firm level

BOTG id 'Bottom gate Definition' lv 0 10 qh 'Q-NetHead bottom gate' 'Q-NetHead bottom gate'

mf 'Max.Flow 999' 'Max.Flow 999' botg

Where:

id	id of the bottom gate definition
lv	intake levels for the number of bottom gates (this number is defined in the RSVP record in DECISPAR.RTC)
qh	reference to Q-h relations for all bottom gates
mf	maximum flow definition for each individual bottom gate

TURB id 'Turbine gate Definition' lv 20 15 qh 'Q-NetHead turbine gate' 'Q-NetHead turbine gate'

ed 'Energy demands' 'Energy demands 2nd turbine' mf 'Max.Flow turbine gate' 'Max.Flow turbine gate' turb

Where:

id	id of the turbine gate definition
lv	intake levels for the number of turbines (this number is defined in the RSVP record in DECISPAR.RTC)
qh	reference to Q-h relations for all turbine gates
mf	maximum flow definition for each individual turbine gate
ed	energy demand definition for each individual turbine gate (energy demands can of course only be defined for the turbine gates, not for bottom gates or spillway gates)

SPIL id 'Spillway gate Definition' lv 50 qh 'Q-NetHead spillway gate' mf 'Max.Flow 999' spil

Where:

id	id of the spillway gate definition
lv	intake levels for the number of spillways (this number is defined in the RSVP record in DECISPAR.RTC)
qh	reference to Q-h relations for all spillways
mf	maximum flow definition for each individual spillway gate

MAXF id 'Max.Flow turbine gate' PDIN 1 1 365;00:00:00 pdin
TBLE
'1995/01/01;00:00:00' 31.25
'1995/01/15;00:00:00' 27.
'1995/02/01;00:00:00' 24.5
'1995/02/15;00:00:00' 19.
'1995/03/01;00:00:00' 50.

’1995/05/01;00:00:00’ 50.5
’1995/06/01;00:00:00’ 59.5
’1995/06/15;00:00:00’ 59.5
’1995/07/01;00:00:00’ 50.
’1995/07/15;00:00:00’ 50.
’1995/08/01;00:00:00’ 50.
’1995/08/15;00:00:00’ 50.
’1995/09/01;00:00:00’ 50.
’1995/10/01;00:00:00’ 50.
’1995/11/01;00:00:00’ 50.
’1995/12/01;00:00:00’ 50.
’1995/12/15;00:00:00’ 50.
’1995/12/31;23:59:00’ 50.
tble maxf

Where:

id	id of the maximum flow definition

PDIN .. pdin = option for interpolation and periodicity

= 1 1 365;00:00:00 means block functions, periodicity one year

= 0 0 means linear interpolation, no periodicity

= 0 1 365;00:00:00 means linear interpolation, period one year

= 1 0 means block function, no periodicity

TBLE .. tble = the table, containing date;time string, value, and

QHRE id ’Q-NetHead bottom gate’

TBLE
0 999
10 999
20 999
30 999
40 999
45 999
50 999 tble qhre

Where:

id	id of Q-h relation

TBLE .. tble contains the relation;

the first column is the flow (m3/s)

the second column is the water level h (m)

ENGD id ’Energy demands’ PDIN 1 1 365;00:00:00 pdin
TBLE
’1995/01/01;00:00:00’ 31.25

’1995/01/15;00:00:00’ 27.
’1995/02/01;00:00:00’ 24.5
’1995/02/15;00:00:00’ 19.
’1995/03/01;00:00:00’ 12.5
’1995/04/01;00:00:00’ 12.5
’1995/05/01;00:00:00’ 12.5
’1995/06/01;00:00:00’ 12.5
’1995/10/01;00:00:00’ 12.5
’1995/11/01;00:00:00’ 12.5
’1995/12/31;23:59:00’ 12.5
tble engd

Where:

id id of the energy demand definition

PDIN .. pdin = option for interpolation and periodicity

= 1 1 365;00:00:00 means block functions, periodicity one year

= 0 0 means linear interpolation, no periodicity

= 0 1 365;00:00:00 means linear interpolation, period one year

= 1 0 means block function, no periodicity

TBLE .. tble = the table, containing date;time string, value, and <

The values specified in the table are flows in m3/s.

HEDG id ’HedgingRule’

TBLE
100 100
90 100
70 75
50 50
30 25
10 0 tble hedg

Where:

id id of hedging rule relation

TBLE .. tble contains the relation;

the first column is the level percentage (between 0 and 100)

(0 corresponds with the level at dead storage (=the level at the lowest outlet);

100 corresponds with the firm level).

the second column is the release percentage (between 0 and 100)

D.20.3 Measures layer

sbk_meas.rtc-file (Measures layer)

In this file, the user can specify the measures related to structures/controllers in the Flow module.

The measures are specified in an FLCM record for each controller which is controlled by RTC. Each FLCM record contains one or more SBMS sub-records in which the measures for the regarding controller are specified.

The general format of a data record is:

FLCM id ’Ctrl_id’ nm ’Comment’ na 0 iv 41.
SBMS pr 1 ty 5 nv 1 bp ’Decispar1’ cv 71.35 ch ’>’ sp 41.05 sbms
flcm

or

FLCM id ’Ctrl_id’ nm ’Comment’ na 0 iv 1.0
SBMS pr 1 ty 2 bp ’Decispar1’ nv 3 cv 1.0 2.0 sp 0.0 1.0 2.0 sbms
flcm

or

FLCM id ’Sbk-Cntrl-id’ nm ’Comment’ na 0 iv 41.
SBMS pr 1 ty 7 nv 1 bp ’Decispar1’ cp ’Decispar2’ ch ’>’ sp 41.05 sbms flcm
SBMS pr 2 ty 9 mi ‘Matlabid’ dv 18.0 sbms flcm

or

FLCM id ’Ctrl_1’ nm ’Comment’ na 0 iv 41.
SBMS pr 1 ty 8 nv 1 bp ’Decispar1’ cp ’Decispar2’ ch ’>’ psp ’Decispar3’ sbms
flcm

or

FLCM id ’Sobek1D2D_BottomLevel controller location X’ nm ’Comment’ na 0 iv 12.75
SBMS pr 1 ty 10 psp ’Bottom Level at location X’ sbms
flcm

where: ***in the FLCM record:***

id	Water Flow controller-id on which the measure is working
nm	name of the controller, which can be used as additional comment
na	option not active, 0 = active, <> 0 means not active. This option can be used to switch off RTC for this pump while keeping the definition
iv	initial set point value, and

in the SBMS sub-record:

pr	priority
ty	type of measure; the format of the record is depending on the type 2 = table with n check values, n set points 5 = record with n decision parameters, n check values and 1 set point (number);

6 = record with n decision parameters, n check values and 1 set point as a decision variable
7 = record with n decision parameters, n check parameters, 1 set point (number)
8 = record with n decision parameters, n check parameters and a set point as a decision parameter
9 = MATLAB measure
10 = record with 1 setpoint decision parameter only
12 = external TCN measure

bp	id of decision parameter
nv	number of values or decision parameters (only for type ty 2, 5,6,7,8)
cv	check value (only for type 2,5,6; nv values for type 2,5,6)
ch	check to be carried on on the check value (not for type 2)
cp	check decision parameter (only for measure type 7, 8)
sp	set point of the controller (PID or interval controller) or the controlled parameter (hydraulic controller or time controller) controlling the Flow-structure.
psp	parameter set point (only for ty 6, 8, 10)
mi	Matlab communication id, the id which is used in Matlab for this measure
ti	TCN-id, the id which is used in TCN for this measure

The following table gives a concise overview of the measure types:

Measure Description	**Decision parameters**	**Check value/parameters**	**Setpoint value/parameter**	**Measure Type No.**
Type A	1	none	Interpolation Table	2
Type B1	n	n values	1 value	5
Type B2	n	n values	1 parameter	6
Type B3	n	n parameters	1 value	7
Type B4	n	n parameters	1 parameter	8
Type B5	1	none	1 parameter	10
Type C1	none	none	from MATLAB	9
Type C2	none	none	from TCN	12

For measure type 5-8, the value of n may also be equal to 1.

It is possible to define multiple measures at one Flow controller. These measures can be linked to the same decision parameter as well, but that is not required. If one of the defined measures is active, the set point of local controller controlling the Flow structure will be adjusted to that measure.

In defining measures of type 5 to type 8, the multiple condition check is performed using ‘AND’ logic: only of all conditions hold, the set point will be set according to the decision rule.

An example file:

```
FLM2.0

FLCM id ’Sobek_cntrlid1’ nm ’Comment’ na 0 iv 0.0
SBMS pr 2 ty 2 bp ’BesComment blispar1’ nv 3 cv 3.0 3.54 4.0 sp 3.5
3.52 3.99 sbms
SBMS pr 1 ty 5 bp ’Decispar2’ ’Decispar5’ nv 2 cv 3.0 5.0 ch ’<’ ’>’ sp 5.0 sbms
SBMS pr 1 ty 6 bp ’Decispar2’ ’Decispar5’ nv 2 cv 3.0 5.0 ch ’<’ ’>’ psp ’Decispar1’ sbms
SBMS pr 1 ty 7 bp ’Decispar2’ ’Decispar5’ nv 2 cp ’Decispar4’ ’Decispar3’ ch ’>’ ’<’
sp 5.0 sbms
SBMS pr 1 ty 8 bp ’Decispar2’ ’Decispar5’ nv 2 cp ’Decispar3’ ’Decispar4’ ch ’<’ ’>’
psp ’Decispar1’ sbms flcm

FLCM id ’0-68’ nm ’Comment’ na 0 iv 18.0
SBMS pr 2 ty 9 sbms flcm

FLCM id ’10-68’ nm ’Comment’ na 0 iv 5.25
SBMS pr 1 ty 12 sbms flcm

FLCM id ’Sobek1D2D_BottomLevel controller location X’ nm ’Comment’ na 0 iv 12.75
SBMS pr 1 ty 10 psp ’Bottom Level at location X’ sbms
flcm
```

With the exception of the MATLAB and TCN measures of types 9 and 12 respectively, all measure types are related to a decision parameter (‘bp’ field in the data record). For the MATLAB and TCN measure types, everything has to be defined in the MATLAB m-file or the TCN csv-files.

Note:
It is possible to define multiple measures operating on the same controller, by giving them the same id. If multiple measures with the same priority are defined and active on the same controller, the last active one will define the setting of the controller.

3bmeas.rtc-file (Measures layer)

The Rainfall-Runoff measures are limited to structures (pumps, weirs, orifices, friction, Q-h relations) in Rainfall-Runoff. A typical application is pump operation. In normal conditions the pump operation in Rainfall-Runoff is based on water levels of open waters in Rainfall-Runoff only. In a combined Water Flow - Rainfall-Runoff calculation, pumps in Rainfall-Runoff can be switched off, based on the water levels in Water Flow. This is done using the Real-time Control module. A typical application is for a combined ’polder-boezem’ network, using Rainfall-Runoff for ’polder’ and Water Flow for the ’boezem’ network: polder pumps are switched off if the ‘boezem’ water level is too high. This is called a pumpstop measure.

Pumpstop measures for Rainfall-Runoff are specified using two types of records. First, a measure is defined using a decision parameter and switch-on and off-levels. This allows to determine whether a measure is active or not. Second, a pump in Rainfall-Runoff can be linked to a measure. If the measure is active, Real-time Control will pass on to Rainfall-Runoff that the pump should be switched off. If the measure is not active, Real-time Control will pass on to Rainfall-Runoff that the pump may be switched on again. Whether the Rainfall-Runoff pump is indeed switched on, is determined by the pump operation rules in Rainfall-Runoff.

Several pumps can be linked to the same measure. In fact, this is the main reason to use two different data records: you only have to define the measure or decision rule once, and you can link as many Rainfall-Runoff pumps to that measure as you like.

Also it is possible that a Rainfall-Runoff pump is linked to several measures. The measure with the highest priority determines what happens. If there are several measures of the same priority, the following rules apply: if one of the measures is active, the Rainfall-Runoff pump will be switched off. It can only be switched on again if all measures related to that pump are inactive again.

This set-up of measures also allows to take measures on RR-weirs and RR-orifices. A pump stop is then interpreted as a forcing to set Q=0; whereas with no pumpstop the discharge over the weir or orifice is determined by the normal discharge formula. At the moment there is not yet an option to distinguish the different high and low capacities of the Rainfall-Runoff pump: a pumpstop applies to the full capacity.

Since August 2000 an extra option is available to operate the RR-pumps using MATLAB. With MATLAB the user can change the switch-on and -off levels for the RR-pump at both the low and high capacity.

The general format of data-records is as follows:

MLST id 'Measure2' nm 'Comment 2' bp 'trend_ow' on -1 of 0 cn '$<$' cf '$>$' mlst
RRST id '38' nm 'Comment 3' na 0
RRMS ty 9 pr 1 rrms
RRMS ty 10 ms 'Measure2' pr 2 rrms rrst

where:

MLST record:

Id	id of measure (MLST record)
nm	name of the measure, which can be used as additional comment
bp	id of decision parameter
on	switch-on level measure
of	switch-off level measure
cn	check on switch-on level
cf	check on switch-off level

and:

RRST record:

id	id of Rainfall-Runoff pump
nm	name of RR-Pump, which can be used as additional comment
na	option not active, 0 = active, $<>$ 0 means not active.

This option can be used to switch off RTC for this pump while keeping the definition

The RRST record contains one or more RRMS sub-records with the specification of the used measure[s].

RRMS sub-record:

ty	measure type, 9= MATLAB, 10= normal measure.
ms	measure id, only applicable for type 10
pr	priority

An example input file:

```
RRM2.0

MLST id 'Measure1' nm 'Comment 1' bp 'peil_ow' on -595 of -605 cn '<' cf '>' mlst

MLST id 'Measure2' nm 'Comment 2' bp 'trend_ow' on -1 of 0 cn '<' cf '>' mlst

MLST id 'Measure3' nm 'Comment 3' bp 'trend2' on 1 of 0 cn '>' cf '<' mlst

RRST id '111' nm 'Comment 111' na 0
RRMS ty 10 ms 'Measure1' pr 1 rrms
RRMS ty 10 ms 'Measure2' pr 1 rrms
rrst

RRST id '2' nm 'Comment 2' na 0
RRMS ty 10 ms 'Measure3' pr 1 rrms
rrst

RRST id '38' nm 'Comment 3' na 0
RRMS ty 9 pr 1 rrms
RRMS ty 10 ms 'Measure2' pr 2 rrms
rrst
```

E Error Messages

E.1 Error Messages on Startup

Run-time Error ’53’: File not found: wlauth40.dll
This error will occur when the DS_Flex license manager has not been installed. SOBEK can only be used when the license manager is installed, even the Free Trial mode. It is possible to install the DS_Flex license manager by opening the SOBEK setup and following the default (recommended) installation options. For more information on installing the license manager, see https://publicwiki.deltares.nl/display/LMADMIN/Deltares+License+Management

E.2 General error messages or unexpected results

Decimal and digit grouping symbols
SOBEK uses the comma (,) as digit grouping symbol, and the dot (.) as decimal symbol. The SOBEK user interface will ensure the correct symbols are written to the data files. When directly editing the SOBEK data files (not recommended) incorrect use of the digit grouping symbol as decimal symbol will result in unexpected behaviour that may cause error messages, software crashes or unexpected results.

E.3 Error Messages Model data editor

Run time Error 62, Input past end of file.
This error may occur when you open the data editor for a 2D grid. It means that your 2D grid file does **not** entirely comply to the ASCII standards. We have encountered 2D grid files made by customers where the lines did not end with the obligatory "carriage linefeed" (Hexadecimal code 0D 0A, ASCII character codes 13 and 10). These files had been made with MATLAB. Read here how to create a correct 2D grid file from MATLAB: ***How to write a correct 2D-Grid file from MATLAB:***

If you use the command "open" in MATLAB for Windows, to start writing the grid file, you should not use the *-w* option (write), but the *-wt* option (write text). Using the *-wt* option means that MATLAB will write the text file platform dependently. In the case of a Windows environment, it will write a "carriage linefeed" at the end of each line.

E.4 Error Messages SOBEK-Rural / Urban 1DFLOW

- Fatal
  Node x is connected to itself
  Network input of this node is wrong
  Delete node and add it in the right way
- Fatal
  Missing or corrupted definition and data file
  Network and data can not be read from file by flow module
  <sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
- Fatal
  Inquire next group failed
  Network and data can not be read from file by flow module
  <sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
- Fatal
  Unable to close definition file
  Network and data can not be read from file by flow module
  <sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged

◇ Fatal
Unable to close data file
Network and data can not be read from file by flow module
<sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
◇ Fatal
Inquire group failure
Network and data can not be read from file by flow module
<sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
◇ Fatal
Inquire cell failure
Network and data can not be read from file by flow module
<sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
◇ Fatal
Inquire element failure
Network and data can not be read from file by flow module<sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
◇ Fatal
Get element failure
Network and data can not be read from file by flow module
<sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
◇ Fatal
Unable to open definition file
Network and data can not be read from file by flow module
<sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
◇ Fatal
Unable to open data file
Network and data can not be read from file by flow module
<sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
◇ Fatal
Unknown element type
Network and data can not be read from file by flow module
<sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
◇ Fatal
Variable already declared
Network and data can not be read from file by flow module
<sobek.mda> or <sobek.mdf> files might be write-protected or disk might be damaged
◇ Fatal
Running out of data space
Network and data can not be read into memory by flow module
Disk might be full
◇ Fatal
Running out of name space
Network and data can not be read into memory by flow module
Disk might be full
◇ Fatal
Estimated time step too small
(Error message: plchdt) The time step that is necessary to compute the next solution is smaller than 0.001s (the default minimum time step in SOBEK). This is due to an extreme condition that occurred at a location in the network. To see where the extreme condition occurs and what the extreme condition is, check the SOBEK log files (for example: <sobek.log>) . For example, there might be an extreme large discharge due to over-dimensioned structures. The description 'Estimated time step too small' is usually replaced with the error message 'plchdt' in <sobek.log>).

In order to obtain more information on time step reductions in your model, these advanced options are available:

- In Results in Charts and Results in Maps, the output for "Simulation Info at the Branch Segments". This output is stored in the file <FLOWANAL.HIS>.
- In the <sobek.log> file. More information about time step reductions becomes available in this file after setting the option "Debug=-1" in the <Sobeksim.ini> and running a simulation.
- In the <timers.his>. This file only becomes available after setting the option "UseTimers=-1" in the <Sobeksim.ini> file, under the "SimulationOptions" category. In order to decrease the amount of additional output this setting generates, set the value for the setting TimersOutputFrequency=1 to a higher number, for example 10. This generates Timer output every 10 time steps instead of every time step.

For more information about time step reductions, see also Time step reductions during the simulation.

Note: Starting with SOBEK 2.14, SOBEK will automatically reduce the calculation time step for locations where culverts are located below the cross-section bed level. This may result in reduced simulation performance or the plchdt error message for models that contain such inconsistencies. Besides manually ensuring culverts are not placed below the cross-section bed level in the network, SOBEK offers an option to automatically adjust the cross-section bed level when encountering this issue. For more information, search for the setting Maximum Lowering of Cross-section Bed Level at Culvert in this manual.

- Fatal
  Restart time step not found
  Restart data can not be read from fileDon not use this restart file
- Fatal
  Nefis error
  Network and data can not be read into memory by flow module
- Fatal
  Error while reading SobekB file. Maximum number of SobekB connections exceeded
  Module is not dimensioned for this large number of communication points
  Use fewer points
- Fatal
  Error while reading node identifiers
  <node.tbl> file is incorrect
- Warning
  No lateral discharges in file
  Sobekb has run but no lateral discharge is written to <runoff.out> file
  Run flow module stand alone
- Fatal
  Error while reading SobekB file. Number of identifiers on lateral discharge file exceeds maximum. Module is not dimensioned for this large number of communication points
  Use fewer points
- Fatal
  Error while reading SobekB file
  <runoff.out> file is not right
  <runoff.out> might be write-protected or disk might be damaged
- Warning
  Lateral discharge from file not found. Lateral discharge set to zero
  <qlat.tbl> and <runoff.out> file are inconsistent
  Check runoff
- Warning

Extreme differences in bottom level in branch segment x
Difference in bed level over branch segment is more than meter
Check data

◇ Fatal
Number of grid points from file and calculated inconsistent
There is an error in the network data
Check if all branches have a start and end node

◇ Warning
Lateral outflow higher than half the volume in node/calculation point x
Withdrawals can cause negative water depths
Check (negative) lateral discharge near this location

◇ Warning
Restart data at grid points inconsistent with present network. Continued with initial state
Network has changed since writing of restart file
Make new restart file

◇ Warning
Restart data at structures inconsistent with present network. Continued with initial state
Network has changed since writing of restart file
Make new restart file

◇ Warning
Restart data at controllers inconsistent with present network. Continued with initial state
Network has changed since writing of restart file
Make new restart file

◇ Warning
Bed level higher than bottom connected branch in well x. Bed level moved to bottomconnected branch Wrong (too high) bed level at this node
check data at this node

◇ Warning
Extreme differences in levels of connected branches in node x
Connected branches differ more than meter in level at this node
Check data

◇ Warning
Well surface table with one row in node x
This one value is used as constant surface
Use constant surface

◇ Warning
Street surface table with one row in node x
This one value is used as constant surface
Use constant surface

◇ Warning
Mass balance not closed in branch x at calculation point x
Netto inflowing water is not equal to storage. High discharges or velocities might be computed
Check data at this location

◇ Warning
Mass balance not closed in node x
Netto inflowing water is not equal to storage. High discharges or velocities might be computed
Check data at this location

◇ Warning
Steady state not branched in steps. Water level stop criterion = . m. Error = x
System is not suitable for a steady state calculation
Turn of discrete action like pumps switching on

◇ Warning

minutes not dividable by chosen time step
Results according to RIONED can not be given
Change time step

- Warning
  Street level lower than well depth in node x. Street level set one meter above well depth
  Street level at this node is wrong
  Change street level at this node
- Warning
  Missing tbl-files with identifiers
  Might be an old <convert.exe>.
  Use new <convert.exe>
- Fatal
  Error while reading branch identifiers
  <branch.tbl> file is incorrect
- Fatal
  Error while reading structure identifiers
  <struct.tbl> file is incorrect
- Fatal
  Error while reading lateral discharge identifiers
  <qlat.tbl> file is incorrect
- Warning
  Structure x might generate numerical oscillations due to broad crest and few storage area around the structure.
  Storage surface over one meter above crest level next to structure is less than . times the crest width. See if oscillations occur.
  If yes add storage surface
- Fatal
  Error while reading calculation point identifiers
  <grid.tbl> file is incorrect
- Fatal
  Error while reading branch-segment identifiers
  <rchsegdw.tbl> file is incorrect
- Warning
  Lateral discharge set to zero
  Withdrawal is to large to be used in a computation with a time step higher than.
  Change large withdrawal at this location
- Fatal
  Two Q-boundaries on branch segment x
  No calculation point on branch on which a solution can be found.
  Numerical scheme can not deal with this. Change network

E.5 Error Messages SOBEK-Rural / Urban RR (Rainfall-Runoff)

66. error
    unknown meteo-id or name of node
67. error
    node-id found, but inconsistent node names
71. fatal error
    fatal error <explanatory text>
902. fatal error
    error in sub <subroutine-identifier> file is corrupt
911. fatal error
    unexpected end of file
914. error

incorrect number of meteo stations

923. error

crop factor below zero or above 2.5

925. error

inconsistency in data

926. error

basin storage (min.) should be given as a percentage

932. error

time coefficient not in range 0–1

902. error

error reading file name <*name*>

F The SOBEK OpenMI interface

F.1 Introduction

The OpenMI standard defines an interface that allows time dependent models to exchange data at runtime (Moore *et al.*, 2005). Model components that comply with the OpenMI standard can, without any programming, be coupled to OpenMI modelling systems (Moore and Tindall, 2005; Gregersen *et al.*, 2007).

The intention of this chapter is to help modellers to setup OpenMI compositions with SOBEK. The OpenMI interface of SOBEK has been applied several times for model coupling (see e. g. Becker *et al.*, 2012c; Becker and Talsma, 2014; Becker, 2013; Becker and Gao, 2012; Becker *et al.*, 2012a; Schellekens *et al.*, 2012; Becker *et al.*, 2012b).

F.2 Installation

To use SOBEK within an OpenMI composition requires some installation steps. These steps are described in Becker (2013), Schwanenberg *et al.* (2011) and Becker and Gao (2012).

F.3 The `omi` file

The `omi` file populates the OpenMI compliant component. The main information is the location of the assembly (the computational core with OpenMI interface). Below an example for an `omi`-file is given.

```
<Arguments>
  <Argument Key="Model" ReadOnly="true" Value="CF" />
  <Argument Key="ID" ReadOnly = "true" Value = "WF" />
  <Argument Key="Directory" ReadOnly="true" Value=".\WF\CMTWORK" />
  <Argument Key="Schematization" ReadOnly="true" Value="sobeksim.fnm" />
  <Argument Key="ExchangeItemGroup" ReadOnly="true" Value="rtc" />
  <Argument Key="SplitSpecificElementSets" ReadOnly="true"
            Value="CalcPoints;Laterals;Pumps;Measurements;
            Structures;HBoundaries;QBoundaries" />
  <Argument Key="LogFile" ReadOnly="true" Value="D:\sh.txt" />
</Arguments>
```

Different argument keys are used to let the user to specify the component more in detail:

Model specifies the name of the model, appears in the yellow box in the OpenMI editor.

ID specifies an ID for the component, appears in the yellow box in the OpenMI editor.

Directory specifies the location of the schematization file (argument key **Schematization**) `Sobeksim.fnm` relative from the location of the `omi` file or absolute.

SplitSpecificElementSets defines element sets to split. By default all groups appear as one exchange item that groups all elements (for example all calculation points) as one element set. The element set appears in the list of available exchange items. Should elements of this element set be addressed separately, the element list should appear under this argument key.

ExchangeItemGroup specifies a group of exchange items that are used for typical tasks. The following groups are available (not case sensitive):

◇ *rtc* can be used for coupling of SOBEK models with real-time control models based on based on RTC-Tools (Deltares, 2013) as shown by Becker *et al.* (2012b) and Becker (2013).

◇ *nhi* has been set up for the Dutch National Hydraulic Instrument.
◇ *sobek* for coupling of multiple SOBEK models, examples given by (Becker and Gao, 2012).

Photo's by: BeeldbankVenW.nl, Rijkswaterstaat / Joop van Houdt.

Deltares systems

PO Box 177
2600 MH Delft
Rotterdamseweg 185
2629 HD Delft
The Netherlands

T +31 (0)88 335 81 88
F +31 (0)88 335 81 11

sales@deltaressystems.nl
www.deltaressystems.nl