



MIKE SHE

Fully Integrated Exercise

Step-by-step training guide



DHI headquarters Agern Allé 5 DK-2970 Hørsholm Denmark

+45 4516 9200 Telephone +45 4516 9333 Support +45 4516 9292 Telefax

mike@dhigroup.com www.mikepoweredbydhi.com



CONTENTS

MIKE SHE Fully Integrated Exercise Step-by-step training guide

2	Getting Started Exercise	5
2.1	Step 1 – Start MIKE SHE	5
2.1.1	Launch MIKE SHE	5
2.2	Step 2 – The MIKE Zero project	6
2.2.1	The MIKE Zero project window	6
2.2.2	Install the MIKE Zero examples	7
2.2.3	Open the default examples project	8
2.2.4	Getting help	9

3	Develop a Fully Integrated MIKE SHE Model	11
3.1	Step 1 – Create a new MIKE SHE setup	11
3.1.1	Create a new MIKE SHE document	11
3.1.2	Save the document file	12
3.2	Step 2 – Set up the map overlays	13
3.2.1	Set the display area	13
3.2.2	Add an image map overlay	14
3.2.3	Define the file and geo-reference the bitmap	15
3.3	Step 3 – Set up the simulation	16
3.3.1	Select simulation modules	16
3.3.2	Set simulation title	17
3.3.3	Specify simulation period	17
3.4	Step 4 – Set up the model domain	18
3.4.1	Define the model domain	18
3.4.2	Add the .shp polygon to the overlays	19
3.4.3	Adjust the display parameters for the .shp file	20
3.5	Step 5 – Set up the topography	21
3.5.1	Define surface topography	21
3.6	Step 6 – Define groundwater recharge	24
3.6.1	Define precipitation rate and temporal distribution	24
3.6.2	Define net rainfall fraction	25
3.7	Step 7 – Geological model	26
3.7.1	Saturated zone model specification	26
3.7.2	Define the number of geologic layers	27
3.7.3	Define layer bottom (lower level)	28
3.7.4	Assign hydraulic properties	30
3.7.5	Assign drainage option	30



3.7.6	Define drain levels	31
3.7.7	Define drain time constants	31
3.8	Step 8 – Define Attributes of Computational Layers	32
3.8.1	Define computational layers	32
3.8.2	Initial conditions	33
3.8.3	Edit the Lay1L.dfs2 grid	34
3.8.4	Outer boundary conditions	35
3.9	Step 9 – Pre-process the data	37
3.9.1	Run the preprocessor	37
3.9.2	Viewing the preprocessed data	38
3.10	Step 10 – Redefine the horizontal grid resolution	39
3.10.1	Change horizontal grid resolution	39
3.10.2	Run the preprocessor	39
3.10.3	Change the cell size to 400 meter cells	40
3.10.4	Change the cell size back to 500 meter cells	41
3.10.5	Save vour MIKE SHE document	41
3.11	Step 11 – Specifying Outputs and Calibration Targets.	42
3.11.1	Storing of results	42
3 11 2	Define detailed time series output items	43
3 11 3	Grid outputs	44
3 12	Step 12 – Run the Simulation	45
3 12 1	Specify the time step parameters	45
3 12 2	Simulation control	46
3 12 3	Run the simulation	46
3 13	Step 13 – View the Results	47
3 13 1	Locate the detailed time series output results	47
3 13 2	Display the gridded results	48
3 13 3	View the results in the results viewer	49
3 14	Step 14 – Add Geological Lenses	50
3 14 1	Add the geological lenses	50
3 14 2	Define horizontal extent of the lens	51
3 14 3	Define the top of the lens	52
3 14 4	Define the bottom of the lens	53
3 14 5	Define hydraulic properties for the lens	54
3 14 6	Evaluate the pre-processed data	54
3 15	Step 15 – Advanced use of the Results Viewer	55
3 15 1	Locate the Results	55
3 15 2	View the results in the results viewer	55
3 15 3	Time series plot at a point	56
3 15 /	Extract a profile of water levels	57
3 15 5	Display the computational grid on profile	58
3 15 6	Modify visible overlays and display the finite difference mesh in result viewer	50
3.16	Step 16 Create a Water Balance	60
3 16 1	Create a new water balance document	60
3.10.1	Extract the water balance data	61
3.10.2	Create water balance items	62
3.10.5	Define a Chart water balance	62
2 16 5	View the water balance output	64
2.10.0	Define a time verving water balance	65
2 16 7	View the water belance output	66
3.10.7	Stop 17 Add the River Module	67
2 17 1	Select simulation modules	67
3 17 0	Add the Diver Module	67
3 17 2	Finders the MIKE 11 simulation document	69
3 17 /	Change the results directory	00
5.17.4	טומוועה נוה ובסמונס מוובטנטוץ	03



3.17.5	Explore the branch network	70
3.17.6	Routing branches	71
3.17.7	Routing settings	72
3.17.8	MIKE SHE – MIKE 11 coupling	73
3.17.9	View cross-sections	73
3.17.10	View the boundary conditions	75
3.18	Step 18 – Add the Overland Flow Module	76
3.18.1	Select simulation modules	76
3.18.2	Define infiltration fraction	77
3.18.3	Specify the overland flow parameters	77
3.19	Step 19 – Run the Model	79
3.19.1	Define time steps	79
3.19.2	Overland flow parameters	80
3.19.3	Run the Preprocessor	81
3.19.4	View the river link setup	81
3.20	Step 20 – View the Results	82
3.20.1	View groundwater level results	82
3.21	Step 21 – Input Distributed Rainfall	83
3.21.1	Define rainfall stations	83
3.21.2	Associate the station areas with time-series data	84
3.22	Step 22 – Modify the Subsurface Drains	85
3.22.1	Modify the subsurface drain reference system	85
3.23	Step 23 – Add Land Use	86
3.23.1	Add the evapotranspiration and unsaturated flow modules	86
3.23.2	Define vegetation map (field map)	87
3.23.3	Define vegetation types for each vegetation area	88
3.24	Step 24 – Add Reference Evapotranspiration	89
3.24.1	Define evapotranspiration data	89
3.25	Step 25 – Build the Unsaturated Zone Model	90
3.25.1	Calculation column classification	90
3.25.2	Define the soil profile distribution map	91
3.25.3	Define soil profiles (horizons)	91
3.25.4	DEMO MODE only: Define groundwater table for UZ classification	92
3.26	Step 26 – Run the Simulation	93
3.26.1	Add field data to the observations	93
3.26.2	Run the simulation	94
3.27	Step 27 – Evaluate the Results	94
3.27.1	Locate the detailed time series output results	94
3.27.2	Gridded data results	95
3.27.3	Locate the detailed time series for MIKE 11	96
3.27.4	Water balance	96



Notes:

- The step-by-step exercises in this book are designed to illustrate the basic functionality available in the user interface and the basic steps required for integrated groundwater-surface water modelling. The exercises are not intended to illustrate best practices in integrated modelling.
- Although the MIKE SHE model used in these exercises is based on a real case study, it has been significantly simplified to make it run quickly. Thus, the parameter values may not be representative of real world values.
- Although the entire exercises will not run in the demo mode, the exercises can be easily modified so that they will run in the demo mode by reducing the number of nodes or the simulation length. The necessary changes are noted in the exercises.
- As the exercises are performed it is recommended that the reader initially spends some time familiarising themselves and understanding the figures on each page before reading the text on that page as this greatly aids in the understanding of the exercise instructions.



1 Introduction

In the hydrological cycle, water evaporates from the oceans, lakes and rivers, from the soil and is transpired by plants. This water vapour is transported in the atmosphere and falls back to the earth as rain and snow. It infiltrates to the groundwater and discharges to streams and rivers as base flow. It also runs off directly to streams and rivers that flow back to the ocean. The hydrologic cycle is a closed loop and our interventions do not remove water; rather they affect the movement and transfer of water within the hydrologic cycle.

MIKE SHE is an advanced, flexible framework for hydrologic modelling. It includes a full suite of pre- and post-processing tools, plus a flexible mix of advanced and simple solution techniques for each of the hydrologic processes. MIKE SHE covers the major processes in the hydrologic cycle and includes process models for evapotranspiration, overland flow, unsaturated flow, groundwater flow, and channel flow and their interactions. Each of these processes can be represented at different levels of spatial distribution and complexity, according to the goals of the modelling study, the availability of field data and the modeller's choices. The MIKE SHE user interface allows the user to intuitively build the model description based on the user's conceptual model of the watershed. The model data is specified in a variety of formats independent of the model domain and grid, including native GIS formats. At run time, the spatial data is mapped onto the numerical grid, which makes it easy to change the spatial discretisation.

1.1 Hydrologic processes simulated by MIKE SHE

MIKE SHE uses MIKE 11 to simulate channel flow. MIKE 11 includes comprehensive facilities for modelling complex channel networks, lakes and reservoirs, and river structures, such as gates, sluices, and weirs. In many highly managed river systems, accurate representation of the river structures and their operation rules is essential. In a similar manner, MIKE SHE is also linked to the MOUSE sewer model, which can be used to simulate the interaction between urban storm water and sanitary sewer net-works and groundwater. MIKE SHE is applicable at spatial scales ranging from a single soil profile, for evaluating crop water requirements, to large regions including several river catchments, such as the 80,000 km2 Senegal Basin. MIKE SHE has proven valuable in hundreds of research and consultancy projects covering a wide range of climatological and hydrological regimes.

The need for fully integrated surface and groundwater models, like MIKE SHE, has been highlighted by several recent independent studies that can be downloaded from the MIKE SHE web site. These studies compare and contrast available integrated groundwater/surface water codes. They also show that few codes exist that have been designed and developed to fully integrate surface water and groundwater. Further, few of these have been applied outside of the academic community.

1.2 Applications around the world

MIKE SHE has been used in a broad range of applications. It is being used operationally in many countries around the world by organisations ranging from universities and research centres to consulting engineers companies. MIKE SHE has been used for the



analysis, planning and management of a wide range of water resources and environmental and ecological problems related to surface water and groundwater, such as:

- River basin management and planning
- Water supply design, management and optimisation
- Irrigation and drainage
- Soil and water management
- Surface water impact from groundwater withdrawal
- Conjunctive use of groundwater and surface water
- Wetland management and restoration
- Ecological evaluations
- Groundwater management
- Environmental impact assessments
- Aquifer vulnerability mapping
- Contamination from waste disposal
- Surface water and groundwater quality remediation
- Floodplain studies
- Impact of land use and climate change
- Impact of agriculture (irrigation, drainage, nutrients and pesticides, etc.)

1.3 The MIKE SHE user interface

MIKE SHE's user interface can be characterised by the need to

- Develop a GUI that promotes a logical and intuitive workflow, which is why it includes
 - A dynamic navigation tree that depends on simple and logical choices
 - A conceptual model approach that is translated at run-time into the mathematical model
 - Object oriented "thinking" (geo-objects with attached properties)
 - Full, context-sensitive, on-line help
 - Customised input/output units to support local needs
- Strengthen the calibration and result analysis processes, which is why it includes
 - Default HTML outputs (calibration hydrographs, goodness of fit, water balances, etc.)
 - User-defined HTML outputs
 - A Result Viewer that integrates 1D, 2D and 3D data for viewing and animation
 - Water balance, auto-calibration and parameter estimation tools.
- Develop a flexible, unstructured GUI suitable for different modelling approaches, which is why it includes
 - Flexible data format (gridded data, .shp files, etc.) that is easy to update for new data formats
 - Flexible time series module for manipulating time-varying data
 - Flexible engine structure that can be easily updated with new numerical engines

The result is a GUI that is flexible enough for the most complex applications imaginable, yet remains easy-to-use for simple applications.





In addition to the MIKE ZERO Project Explorer, the MIKE SHE document consists of 4 parts:

- Along the top the Tool bar and drop-down Menus
- On the left the dynamic Data tree and tab control
- On the right the context sensitive **Dialogue area**
- Along the bottom the Validation area and Mouse-over data area

Tool bar - contains icon short cuts for many MIKE SHE operations that can be accessed via the Menus. Also, it changes depending on the tools that are currently in use.

Data tree - displays the data items required to run the model as it is currently defined. If you add or subtract hydrologic processes or change numeric engines, the make-up of the data tree will change.

Dialogue area - is different for each item in the data tree.

Validation area - displays information on missing data or invalid data items. Any items displayed here are hot linked to the dialogue in which the error has occurred.

Mouse-over area - displays dynamic coordinate and value information related to the mouse position in the map area of any of the spatial dialogues.





2 Getting Started Exercise

Overall objective

Learn how to create a MIKE Zero Project and the associated project documents. Use the **Examples Project** and save it locally for use

The Program Files menu looks different depending on your operating system and your

settings. The picture below is the default display for Windows 7 64 bit.

Important learning objectives

After completing this exercise you should be able to

- Start MIKE SHE
- Install the example file

2.1 Step 1 – Start MIKE SHE

2.1.1 Launch MIKE SHE



from the Program menu select MIKE Zero:

MIKE by DHI 2014 🚠 DHI License Administration OHI License Updater < DHI Software Updater < Model Warehouse **DHI Remote Simulation** DHI WaterData MIKE Animator Plus MIKE View MIKE Zero 📸 MIKE Zero Documentation Release Notes Tools 😻 Image Rectifier Caunch Simulation Engine 煮 MIKE 21 C Grid Generator MIKE to Google Earth

MIKE SHE is part of the MIKE Zero suite of modelling tools, which is a global DHI user interface for managing and manipulating data files and projects for many of the MIKE Powered by DHI software products. Thus, when you launch MIKE SHE, you are really launching MIKE Zero. MIKE Zero then provides the framework from which you can run MIKE SHE or any other product in the MIKE Zero family.



Also to notice in this menu is the

DHI License Administration icon – this launches a tool for managing your software licenses

MIKE View - This is a tool for evaluating the results of MIKE 11

Documentation - This links to a directory of all the installed documentation files

Release notes – Here you can find the latest information on the service packs that have been installed

MIKE to Google Earth – This tool allows you to map your results onto Google Earth

2.2 Step 2 – The MIKE Zero project

MIKE Zero is more than a set of modelling tools. MIKE Zero is a project management interface, with a full range of tools for helping you with your modelling project.

2.2.1 The MIKE Zero project window

The MIKE Zero Start Page includes two parts:

- 1. **The Project Overview table** lists the projects that you have recently accessed, along with their creation and last modified dates.
- 2. **The Project Explorer** is a view into the directory with all the files that are associated with the project.

The MIKE Zero project structure is designed to help you keep control of your project. By default it appears on the right hand side of the MIKE ZERO interface.

MOKE Zero - (Start Pi	age)				
Eile Yew Wind	ow Help				. # X
0.000	B 8 7 10				Close
SOFT WAT	WARE FOR ER ENVIRONN	MENTS			MIKE by DHI
Open an Existing Proj	ect	- 20			
Name	Created	Modified	Location		
Instal Examples					(few Project.) (Spee Project.) (Selete Project.)
est.				No Tracking	Mole



2.2.2 Install the MIKE Zero examples



The versions in the Program Files directory are read-only and provide you with an uncorrupted version of the example files. The **Install Examples** button copies all of the example files from the Program Files directory into your chosen directory location for use in these exercises. If you accidentally corrupt the files, then you can simply re-install the examples.

- Click on the Install Examples... button and then select to install all of the MIKE SHE example files
- Select your Destination Directory to be the Documents/MIKE Zero Projects directory

Now you have a local duplicate copy of the **Examples Project** for use in these exercises.

For all of these exercises, please use the versions found in the Documents directory just created.

	Install Example Files	×
Open an Existing Project	Examples	
MIKE SHE Examples	Destination Directory	
	C:\Data\MIKE Zero Projects	
∢ Install Examples	Add project files to recent project list	cel



2.2.3 Open the default examples project

After you have started MIKE SHE you will navigate to the default location of the installed MIKE SHE Examples Project.

- Select the Open Project button below the project list.
- Browse to the folder: \Documents\MIKE Zero Projects\MIKE_SHE
- Then open the existing project: MIKE SHE Examples.mzp
- The files contained within the MIKE SHE Examples project will now appear in the Project Explorer

If you have opened this project before, the project will be displayed in the list box. In this case, open by clicking on the project name.

Note: The project opens with all the directories expanded out. So, you might like to close up some of the directories in the Project Explorer by clicking on the little icons.





2.2.4 Getting help

You are welcome to ask your instructor at any time for help.

However, if you are working on your own, there is an extensive on-line help. The full Help can be accessed from the top **Help...** menu. From within the dialogues, context sensitive help is available by pressing the F1 key.

MIKE Zero	
Hor Tech Pint Options	
Contents Index. Search Favorites	Start Page
Working with MIKE Zere. Working with MIKE Zere. MiKE Zere Project Framework Working with a MIKE Zere Project Start Page MiKE Zere Project MiKE Zere Project Project View	INEE Zero will leep tack of the projects previously used in INEE Zero. When opening INEE Zero, a start page appears from which you can open old projects or create new projects. Furthermore you have the option to install one or more of the examples applied with the installation in a directory of your choice.
🗉 💑 Version Control Interface	MIKE
* *	

End of Getting Started Exercise.





3 Develop a Fully Integrated MIKE SHE Model

Overall objective

Starting with a simple groundwater model, develop a fully integrated MIKE SHE model.

Important learning objectives

After completing this exercise you should be able to:

- Use point data to interpolated gridded surfaces in MIKE SHE
- Use GIS polygon data and grid data to distribute model parameters spatially
- Add graphical overlays (bitmaps and shape files) to your model display
- Understand the concepts of *Geological Model* and *Computational Layers*
- Understand the properties and attributes that are attached to a Geological Model
- Master the input specifications required for a simple Geological Model
- Understand the properties and attributes that are attached to Computational Layers
- Master the input specifications required for Computational Layers
- Understand the concepts of Setup Data and Processed Data
- Run the simulation engine and view results using the Result Viewer
- Add a river to the model setup
- Add an unsaturated zone to the groundwater model setup
- Add an surface water component to the model setup
- Evapotranspiration, Vegetation and Land use?
- Complete a water balance of the fully integrated model

3.1 Step 1 – Create a new MIKE SHE setup

The models used in the following exercises are based on the Karup River watershed in western Denmark. In the following steps you will be creating and adding a new MIKE SHE model to the Karup River example project thus enabling use of already created files.

Ensure that you have installed the Example files in the previous Getting Started exercise, and have opened the **MIKE SHE examples.mzp** project file.

3.1.1 Create a new MIKE SHE document

A MIKE Zero project is a collection of documents. All of the MIKE Powered by DHI documents can be created from the **File\New\File...** menu in the top pull-down menu or

by clicking on the New File icon, \Box , in the toolbar.

When the New File dialogue appears,

- select MIKE SHE on the left hand side,
- select Flow Model (.she) on the right hand side,
- click OK.

The default MIKE SHE Setup dialogue will now appear and the new .she file will be added to the project under the Model directory.





The **.SHE** file contains all the user-specified information required to run MIKE SHE. However, the file does not contain the actual time series and grid data. MIKE SHE only stores the path names to the other data files for time-series data and grid data. This greatly improves the flexibility of the user interface for keeping your model up-to-date with new data and for running calibration and prediction scenarios

Alternatively, you can select any of the listed file types on the right hand side, or even navigate to one of the other items on the left hand side. For example, you may need to create a water balance (.wbl) file during calibration, or a new dfs0 time series file, which you will find under the MIKE Zero list.

oduct Types:	Documents:						
MIKE VYDRO MIKE 11 MIKE 121 MIKE 21 MIKE 21/3 Integrated Models LITPACK MIKE FLOOD MIKE SHE	Flow Model (.she)	Well Editor (.wel)	UZ Soil Propert	ET Vegetation Properties	Water Balance Calculation	Simple Shape Editor (.shp)	MIKE SHE Toolbox (.mst)
	l surface water						

3.1.2 Save the document file

The document file that you just created – in this case the .she document – is unnamed until you save it.

To save the file:

- **First**. Click on the *Model* directory in the Project Explorer (this becomes the default location for the file to be saved to and ensures that the .she file is located correctly within the project folder structure)
- Then, use the File\Save menu item, or the Save icon
- In the Save dialogue, type in a file name (e.g. SZ1 – this file will be used in the next exercise)
- Add a Note, if you want
- Click OK



Name:	SZ1	.she 👻
Location:	C:\data\MIKE Zero Projects\MIKE_SHE\Model	
otes:	E SHE model	5
ny miserina	E SHE IIIUUEI	

3.2 Step 2 – Set up the map overlays

The first thing to do in most projects is to define the maps and overlays that you are going to use in your project.

MIKE SHE allows you to add graphical overlays using bitmaps, ArcView .shp files, etc. These overlays will appear on all maps shown in the graphical view in the user interface.

3.2.1 Set the display area

The basic display area of the model map view is defined in the top item of the data tree dialogue under *Display*:

- Use your mouse pointer to select **Display**
- Leave the checkbox checked On



The Display item is located at the top of the data tree to make it easy to add and edit your background maps. In the Display item, you can add any number of images to your model setup, in a variety of formats. The images are carried over to the various editors, so you can keep a consistent display between the set up editor and, for example, the Grid Editor and the Results Viewer.

The option ' Default map display based on the Model Domain' means that the map view will be defined by the size of the model domain that you select in the next section of this exercise.

However, in some cases you may want the displayed map area should be much larger than the model domain, in which case you can define the map extents in this dialogue.

You can also import the extents from a shape or dfs2 file.





3.2.2 Add an image map overlay

In the Foreground dialogue under Display:

Click on the Add Item icon



Notice that a plus sign (+) appears beside the *Foreground* item in the data tree. This indicates that sub-items have been added in the data tree.

The Background and Foreground options refer to the way the overlays are displayed relative to the input data specified in the other dialogues.

Also, if you have multiple overlays, the order that they are listed defines the order in which they are displayed. Thus, you don't usually want a bitmap at the top of the list, since it would hide all of the lower overlays.



MIKE SHE Flow Model Descripti		
 ■ Topography ■ Precipitation 	Foreground	
		¥ + +
Extra Parameters	Type Display Overlay File 1 Image Image Image	

3.2.3 Define the file and geo-reference the bitmap

Now click on the plus sign beside the *Foreground* item in the data tree to expand the data tree. Then click on the sub-item to display the sub-dialogue. In the sub-dialogue:

- click on the browse button, ., and find the file
 .\Model Inputs\GIS Data/Karup0.bmp
- Define the Area Coordinates of the bitmap
 - Minimum X = 822

-

- Minimum Y = -1870
- Maximum X = 32845
- Maximum Y = 33270
- Set the *Display* style as *Blend Colors*, which will blend the map colours with any other displayed colours. This will prevent the bitmap from hiding the model data.
- Then navigate back to the *Display* dialogue, to ensure that the map is being displayed correctly.

MIKE SHE Flow Model Descripti	Image Ov	erlay				
Image: Karupu.bmp Model Domain and Grid	Display Image File:	Zero Project			Model Inn	uts\GIS Data\karun0 boo
Topography Precipitation Saturated Zone Storing of results Extra Parameters	Area Coordina Min coords:	ites x: 822	[m]	y: -1870	[m]	Import geo reference from file
	Image Styles Display style:	Blend	colors	33270	Tra	ansparent color:



Since a bitmap image does not contain any geographical information (it is simply a list of pixel locations and colours), the bitmap must be oriented in space. To do so, you have provided MIKE SHE with the coordinates of the lower left and upper right corners of the bitmap.

3.3 Step 3 – Set up the simulation

MIKE SHE includes several simulation modules. The navigation tree in the user interface depends on your choice of simulation modules.

3.3.1 Select simulation modules

In the Simulation Specification dialogue:

- Make sure that Saturated Flow is checked On
- Make sure that *Finite Difference* is selected as the saturated zone Numeric Engine.

	Simulation specification
	Numeric Engine
MIKE SHE Flow Model Description Display Model Domain and Grid Model Domain and Grid Model Domain and Grid Storagraphy Precipitation Storagraphy Storagrap	Water Movement (WM) Overland Flow (OL) Finite Difference Rivers and Lakes (OC) Unsaturated Flow (UZ) 2 Layer UZ Evapotranspiration (ET) Saturated Flow (SZ) Finite Difference Include Advection-Dispersion (AD) Water Quality
Setup Data Processed Results	



The Simulation Specification dialogue allows you to select, which flow components to include in your simulation. For example, if you want to include only MIKE 11 and the exchange to the saturated zone, then you only need to select *Rivers* and *Lakes* plus *Saturated Flow*.



This dialogue is where you also choose the numeric engine for the different hydrologic processes.

- There are three numeric engine options for the unsaturated zone and two each for both the Overland flow and Saturated Flow.
- The calculation method for the Evapotranspiration is automatically selected depending on the Unsaturated Flow option selected.
- The channel flow numerical method is selected in the MIKE 11 Setup.

3.3.2 Set simulation title

In the Simulation Title dialogue:

• Write any title and description you want

MIKE SHE Flow Model Descripti	
连 🗆 Display	
🖻 🗹 Simulation specification	
🚽 🖌 Simulation title	
🚽 🖌 Simulation period	
🖌 🖌 Time step control	
🚽 🖌 SZ Computational Contro	
🚽 🖌 Model Domain and Grid	
🚽 🖌 Topography	Simulation title
🕀 🖌 Precipitation	
🗄 🗙 Saturated Zone	Simulation Title:
🗄 🗉 Storing of results	
Extra Parameters	
_	
	Simulation Description:
↓	
Setup Proces Results	

3.3.3 Specify simulation period

In the Simulation Period dialogue:

- Specify the Start date and End date
 - Start Date: 1 January 1970
 - End Date: 31 December 1980



Demo Note: If you are using a Demo version, then the maximum length of simulation is 30 days. In this case you should use a shorter Simulation Period. For example, 1 June 1980 to 1 July 1980.

In this dialogue, you can either type in the dates or select the dates from a drop down calendar.

Specification of the simulation period at this early stage in the model development is not a requirement. However, it is convenient because the time-varying input data you specify later is validated against the simulation period. Thus, when you specify a rainfall time-series file, a check is made to make sure that the time-series covers the Simulation Period.



In most cases, MIKE SHE is run as a transient model. If the model is run in steady-state, then the steady-state solution also follows this simulation period, by generating a series of identical steady-state solutions for each specified time step.

The Hot-start section allows you to start the simulation from the end of a previous simulation.

If your simulation runs too slowly, then you can reduce the length of the simulation period without affecting the learning objectives.

3.4 Step 4 – Set up the model domain

The model domain and the surface topography are required for all MIKE SHE components.

The model domain defines the horizontal extent of the model area, as well as the horizontal discretisation used in the model for overland flow, unsaturated flow and saturated groundwater flow

3.4.1 Define the model domain

In the Model Domain and Grid dialogue:

- Choose Catchment defined by Shape File
- Using the Browse icon, select the file
 .\Model Inputs\GIS Data\catchment-meter.shp
- Set the Shape axis units to "meter".
- Set the grid dimensions to
 - Number of cells in the X direction, NX = 65
 - Number of cells in the Y direction, NY = 70
 - Cell size = 500m
 - Rotation = 0
 - Origin (X0, Y0) = 0, 0
 - Map Projection Type = NON-UTM

The MIKE SHE GUI automatically assigns values of 1 to the internal cells and values of 2 to the boundary cells.



When you preprocess the data, the preprocessor calculates the model domain based on the .shp file and defines the model domain as a dfs2-grid with integer values 1 (internal point), 2 (boundary point) and zero values for areas outside the model domain. The preprocessor assigns all of the model parameters based on the dfs2-grid that is calculated from the polygon that you have specified in this step.





If you wish to change the model domain, all you have to do is modify the .shp file and preprocess the data again. However, be aware that previously specified data must still cover the new polygon. If it does not, then either a warning will be issued (saying that some values were automatically interpolated) or an error will be issued (if it can't interpolate the data)

The Map Projection Type allows you to use any valid map projection. The only restriction is that whichever projection you chose, you must be consistent with respect to all other map inputs. You cannot mix, for example, maps from two different UTM zones. The NON-UTM option implies local coordinates.

3.4.2 Add the .shp polygon to the overlays

In the Foreground dialogue:

- Click on the Add Item icon,
- Change the *Type* to *Shape*



MIKE SHE Flow Model Descript.	Fore	ground			
Shape:Unknown					
- Background		Тупе	Dienlay	Overlay File	
🕀 🗇 🖸 Current Layer		Туре	Display		
😟 🗉 Simulation specification	1	Image		C:\8.Training\\Bitmaps\karupU.bmp	
- Model Domain and Grid	2	Shape		C:\8.Training\\GIS-Data\catchment_meter.shp	
Topography					
🗄 🗉 Precipitation					
🗄 🗉 Saturated Zone 📃					
🗄 🗉 Storing of results 📃 📃					
📃 🚽 Eutra Paramataro					
Setup Data Process Results					

- Now, go to the new Shape:Unknown data tree item, and
- Then click on the Browse icon,
- In the Open file dialogue, find the file
 .\Model Inputs\GIS Data\catchment-meter.shp

	Shape Overlay			
	Shape File:			Item:
	I			
	Parameters for Points			
	Point color:	Point style:	Transparent	7
	- Text Appotation	Point type:	Circle	_
	Color as point	Background:	Transparent	V
	C Individual color	_		
MIKE SHE Flow Model Descript	Parameters for Lines and Polyg	ons		
Erroreground	Line color:	Line style:	Solid	T
✓ Image: karup0.bmp		Polygon fill style:	Transparent	_
Background	T and formately a	Line thickness:	0.3	
Simulation specification Model Domain and Grid Topography	Color as line/polygon Individual color	Background:	Transparent	V
Precipitation Saturated Zone Storing of results Futra Parameters	Units Units of X- and Y-axises meter]		
Setup Data Process Results				

3.4.3 Adjust the display parameters for the .shp file

In the Shape: catchment_meter.shp dialogue:

- Choose line-colour
- Increase the line thickness (e.g. 0.5 or 1 mm)



Now in all of your map dialogues, such as the *Model Domain* and *Grid* dialogue, the model domain should now appear as a polyline on top of the map.

	Shape Overlay			
	🔽 Display			
	Shape File:			Item:
	C:\Program Files\DHI\MIKEZero\	Examples\MIKE_S	HE\Model\Mod	POLNUMBER
	Parameters for Points			
	Point color:	Point style:	Transparent	7
		Point type:	Circle	T
	Color as point	Background:	Transparent	_
	C Individual color	×		
MIKE SHE Flow Model Descripti	Parameters for Lines and Polygo	ns		
⊡… 🖌 Foreground 🖌 Image: karup0.bmp	Line color:	Line style:	Solid	•
Shape: catchment_met		Polygon fill style:	Transparent	•
		Line thickness:	0.3	
	Text Annotation Color as line/polygon	Background:	Transparent	
	C Individual color		Transparent	
🕀 🖬 Precipitation		1*		
🗄 🗉 🖬 Saturated Zone				
🗄 🗆 🖬 Storing of results	Units of X- and Y-avises			
Extra Parameters	meter			
	Imercen			
Setup Data Process Results				

3.5 Step 5 – Set up the topography

The surface topography is required for all MIKE SHE components.

The model topography defines the upper bound of the groundwater model as well as the upper surface of the unsaturated zone model. It is also used as the flow surface for overland flow.

3.5.1 Define surface topography

In the Topography dialogue:

- Choose Point/Line (.shp) for the Spatial Distribution
- Select the file
 .\Model Inputs\GIS Data\Topo-processed.shp
- If the map is covered by black dots, click off the 'Show shape raw data' checkbox. This will turn off the shape file data points.

The default interpolation method is Inverse Distance. Change the interpolation method to **Bilinear**



- Spatially distributed data, such as topography can be specified using:
 - a Uniform value
 - a Grid file (.dfs2),



- a Point/Line (.shp) ArcView or ArcGIS map, or
- an ASCII file with distributed xyz values (Point XYZ (.txt)).

If a .shp file or a xyz-file is used, MIKE SHE will interpolate the data to the mesh defined in the Model Domain and Grid menu.



Hint: you can always see the Z-value at the cursor position at the bottom of your Graphical View.

Interpolation method - You can choose between Bilinear, Triangular, Inverse Distance and Inverse Distance Squared interpolation methods by selecting from the Interpolation method combo box. The Inverse distance methods are good for scattered data. Bilinear Interpolation is a good method for interpolating from gridded data and the Triangular method is good for interpolating from digitised contour lines. You can use the Online Help to find out more information on the interpolation methods, by clicking F1. At the bottom of the Help page, are references to Related Items, which will take you to the detailed descriptions of the interpolation methods.

SURFER - If you want to use interpolation methods not available in MIKE SHE, then you can use a program such as SURFER by Golden Software. In SURFER, you can save the interpolated SURFER grid to an XYZ file, and use the bilinear method in MIKE SHE to reproduce the Surfer interpolation.

ArcGIS Grid – If you have gridded data from ArcGIS, then you can use the Grd2Mike Tool (New File\MIKE ZERO\MIKE Zero Toolbox\GIS) to convert an ArcGIS Grid to a dfs2 file.





Other tips:

- The search radius should be sufficiently large to ensure that each grid gets a value. For this exercise 1000 m is sufficient. However, the minimum search radius is two times the cell size. Search radii below this will have no effect. In this data set, the data density is quite high, and changing the search radius will have no effect.
- In some cases, such as when the grid is very large or the number of data points is large, then the interpolation can be time consuming. This can be a problem, since the grid is re-interpolated every time you enter the dialogue, as well as during the pre-processing step. To make the model more efficient, you can save the interpolation to a dfs2 file for use directly. To do this, right click on the map view and select save to *dfs2 file*.
- After you have saved the dfs2 file, you can then use it instead of the .shp file. Note though, that the link to the original .shp data is not lost, but simply hidden from view. This allows you to return to the original data if you want to change the discretisation of the dfs2 file, for example, if you change the size and shape of the Model Domain and Grid.



3.6 Step 6 – Define groundwater recharge

For an integrated hydrological model, you normally define the distribution of precipitation and reference evapotranspiration in both time and space. However, in this exercise, you are building only a groundwater model and thus the "precipitation" is actually "groundwater recharge".

3.6.1 Define precipitation rate and temporal distribution

The precipitation is the actual amount of rainfall.

In the Precipitation Rate dialogue:

- For now, choose *Uniform Spatial Distribution*, which means that the same precipitation rate will be used all over the model domain.
- Change the temporal distribution from constant to time varying.
- Click on the button to open the file browser
- Scroll through the list and pick the file Precipitation.dfs0 that has the green check mark beside it
- Keep the default Item "Precipitation station 1" and click OK

	DFS File & Item Selection		
	Files		
	Name	Path	Modified
MIKE SHE Flow Model Description ✓ Display ✓ Simulation specification ✓ Model Domain and Grid ✓ Topography Climate ✓ Precipitation Rate ✓ Saturated Zone ✓ Storing of results ✓ Extra Parameters	PotentialEvap.dfs0 flow.dfs0 Temperature_6894.d., exd.dfs0 Precipitation2.dfs0 karup.dfs0 Precipitation.dfs0 HeadObservations.dfs0	. Wodel Wodel Inputs \Time . Wodel Wodel Inputs \GIS D . \Wodel Wodel Inputs \Time . Wodel Model Inputs \Time . Wodel Wodel Inputs \GIS D . Wodel Wodel Inputs \Time . \Wodel Wodel Inputs \Time	16-12-2013 19:00:34 16-12-2013 19:00:32 16-12-2013 19:00:34 16-12-2013 19:00:34 16-12-2013 19:00:32 16-12-2013 19:00:32 16-12-2013 19:00:34 16-12-2013 19:00:32
	Items Name	Item	
	Any Item Type Precipitat	ion station 1	
			External File
Setup Data Process Results		Help	OK Cancel

This file browser lists of all of the files in your project that are the correct file type. The green check beside the file name indicates that the file contents match the data requirements – for example, the correct time period and units type. If you are in doubt, you can click on the button will show you the data requirements.

In this case, you are using spatially uniform recharge data. The precipitation file contains data from several rain gauges. The particular rain gauge to use is selected under the Item



in the browse dialogue. In this case, you are using only the data from Precipitation station 1.

Precipitation is specified the same way data is collected from rain gauges. It can be input as mean-step accumulated values (e.g. average rainfall per day in units of mm/day) or, as step accumulated values (e.g. measured rainfall in a tipping bucket rain gauge in units of mm since the last measurement).



3.6.2 Define net rainfall fraction

The Net Rainfall Fraction is the fraction of the precipitation that reaches the groundwater table.

In the Net Rainfall Fraction dialogue:

- Choose *Uniform*, which indicates that the same precipitation rate is used all over the model domain.
- Specify *Value* = 0.2, which indicates that only 20% of the rainfall reaches the groundwater table.



MIKE SHE Flow Model Description M Display Simulation specification M Model Domain and Grid M Topography M Climate M Precipitation Rate M Net Rainfall Fraction M Storing of results	
🥌 🗹 Extra Parameters	
	Net Rainfall Fraction
	Spatial Distribution:
	Value 0.2
Setup Data Process Results	



Normally, the amount of precipitation that reaches the groundwater table is only a small fraction of the total amount of precipitation – usually between 5 and 25%. The majority either runs off to surface water, or is returned to the atmosphere as evapotranspiration. Both of these processes are very dynamic and the actual fraction that reaches the groundwater table will vary both spatially and temporally.

When these processes are not simulated, then the Net Rainfall Fraction is used to reduce the actual precipitation to compensate for these processes. The Net Rainfall Fraction is a very important calibration parameter in groundwater modelling.

In the Karup catchment, the soils are very sandy, and well drained. So, there is little runoff as overland flow. In the Integrated exercise, we can use the water balance tool to find the calculated average net rainfall fraction and compare it to this value.

3.7 Step 7 – Geological model

Before defining your computational layers you must input a geological model, which is defined as part of the saturated zone.

A geological model may be defined as a combination of layers and lenses. Once the geological model is defined you may choose computational layers that are either identical to the geological layers or you may choose different computational layers. If the computational layers differ from the geological layers MIKE SHE's pre-processor will transfer the hydraulic properties of the geological model to the computational mesh.

3.7.1 Saturated zone model specification

In the Saturated Zone dialogue:

- Select *Include subsurface drainage*, which implies that water that flows to the drains is removed from the model. Water flows into the drains, whenever the groundwater table is above the drain level.
- Make sure that the *Include pumping* wells is not selected, which says that there are no groundwater pumping wells in the model.



Select Assign parameters via geological layers.





The geology data can be assigned using both Geological layers and Geological Units. If you chose the geological layers approach then the hydraulic properties are assigned as spatially distributed within the layer. If you use gridded data or point data, the hydraulic properties are smoothly interpolated to the model grid.

If you chose the geological units approach, then you must specify a distribution of geological units for each geological layer, for example, by a polygon file. In this case, all of the model cells in each polygon are assigned a value for each of the hydraulic properties. The geologic units method is often specified when you use the AUTOCAL program for parameter estimation and automatic calibration. In this case, the parameters for each polygon can be estimated automatically and the model sensitivity for each parameter calculated.

3.7.2 Define the number of geologic layers

In the Geological Layers dialogue:

• Rename the existing default layer name to Aquifer.



By default there is already one layer in the model. Additional layers can be added by clicking on the *Add Item* icon, , deleted by clicking on the *Delete Item* icon, , or moved up and down by clicking on one of the *Move Item* icons, .



MIKE SHE Flow Model Descrip	Geological Layers
Precipitation Saturated Zone Seturated Zone Geological Layers Aquifer Aquifer Aquifer Aquifer Active Level Aquifer Secological Hydra Secological Hydra Secological Hydra Secological Lenses Geological Lenses Secological Lense Secological Lenses Secological Lenses Secol	Geological layers Name 1 Aquifer

Increasing the number of saturated zone model layers allows you to better represent vertical flow and exchange between geologic layers. A fine vertical discretisation may also be required for solute transport simulations.

MIKE SHE allows you to have any number of layers in the saturated zone. Most models, however, have less than 10 model layers. The more model layers you have the higher the computational effort and the more memory that is needed. If you need more layers, it is always better to start with a few layers and do the initial calibration. Then add layers as necessary, while ensuring that the model remains stable and computational effort remains reasonable.

3.7.3 Define layer bottom (lower level)

In the Lower Level dialogue for Aquifer:

- Select Point/Line (.shp) in the Spatial Distribution combo box
- Using the file browse button, ., select the file
 .\Model Inputs\GIS Data\layer1.shp
- Use the Bilinear interpolation method, and
- Use a Search Radius of 1000m







Each of the Geological layers includes one item in the data tree for each of the properties. Additional layers will add additional sets of items to the data tree.



3.7.4 Assign hydraulic properties

The remaining hydraulic properties are listed under the Lower Level item. For each item:

- Select *Uniform* under the *Spatial distribution* combo box and specify the following values:
 - Horizontal hydraulic conductivity = 0.00052 [m/s]
 - Vertical Hydraulic Conductivity = 9.3e-5 [m/s]
 - Specific Yield = 0.2 [-]
 - Specific Storage = 0.0001 [1/m]

MIKE SHE Flow Model Descripti MIKE SHE Flow Model Descripti Model Domain and Grid Model Domain and Grid	
✓ Vertical Hydraulic C ✓ Specific Yield ✓ Storage Coeffici ✓ Geological Lenses ✓ Computational Layers ✓ Drainage ✓ Storing of results ✓ Extra Parameters ✓ Setup Data Process Results	Horizontal Hydraulic Conductivity Spatial Distribution: Uniform Value 0.00052 [m/s]

3.7.5 Assign drainage option

In the Drainage dialogue:

Select Drainage not routed, but removed from model.






In a fully integrated model the drainage water is part of the water balance and, normally you would want to route the drainage water to the river. However, in this example, you don't have a river specified so there is no place to send the water too. Thus, the water that flows to the drains will be removed from the model, but the amount of water flowing to the drains can still be evaluated in the water balance.

Water flows into the drains, whenever the groundwater table is above the drain level.

The method specified here is the same method as used in MODFLOW. That is, the drains act as head-dependent-flux boundaries but only as a water-sink.

3.7.6 Define drain levels

In the Drainage Level dialogue:

- Select Uniform under Spatial Distribution
- Use the value : -0.5 [m] (Note negative sign)
- Check on "Values relative to ground"

MIKE SHE Flow Model Descri	Level
MIRE SHE Flow Model Descri MIRE SHE Flow Model Descri Model Display Model Domain and Grid Model Domain and Gri	Spatial Distribution: Uniform Value Value 0.5

The Values relative to ground tells MIKE SHE that you have drains located 0.5 meter below ground surface in the entire model domain.



If the groundwater table is higher than the drainage level drain flow will be produced and removed from the model.

3.7.7 Define drain time constants

In the *Time Constant* dialogue:

- Select Uniform under Spatial Distribution
- Use the value = 5.6e-7 [1/s]



MIKE SHE Flow Model Deseri	Time Constant
E Displau	
	Spatial Distribution:
🚽 🖌 Model Domain and Grid	
🖌 Topography	Value
🗄 🛛 🗹 Precipitation	5.6e-007 [/sec]
🖻 🗹 Saturated Zone	,
🖃 🗹 Geological Layers	
🛨 🗠 🗹 Aquiter	
Geological Lenses	
✓ Viewel	
🖌 Time Constant	
🗄 🗹 Storing of results	
🔤 🖌 Extra Parameters	
Setup Proce Results	

The drain time constant can be thought of as an empirical factor that accounts for the time it takes for the water to drain. For example, in the case of agricultural drains, the time constant could be affected by drain spacing, drain diameter, clogging, etc.



In the case of natural drainage, the time constant is affected by the distribution of drainage ditches and channels.

A value closer to 1 implies that the water drains more quickly and the drain acts more like a constant head boundary, when the water table is above the drain level. Whereas, a value closer to 0 implies that the water drains more slowly. A value of 0 turns off the drainage.

An Extra Parameter option is available for time varying drains.

3.8 Step 8 – Define Attributes of Computational Layers

The computational layers are defined independently of the geological layers. While the *Geological Layers* have geological attributes, the *Computational Layers* include computational attributes such as initial conditions and boundary conditions. The properties of the geological layers are mapped and interpolated to the computational layers during the model preprocessing.

Although, MIKE SHE allows you to define *Computational Layers* independent of your *Geological Layers*, in this exercise computational and geological layers will be identical.

3.8.1 Define computational layers

In the Computational Layers dialogue:

- Select Defined by geological layers as the Type of Numerical Vertical Discretization
- Use Minimum layer thickness = 0.5 [m]







The Minimum layer thickness is used to keep layers from crossing or completely pinching out, which would cause numerical difficulties.

3.8.2 Initial conditions

In the Initial Potential Head dialogue:

- Select Grid file (.dfs2) under Spatial Distribution
- Using the file browse button, ., select the file
 .\Model Inputs\Maps\init-head-500.dfs2







You are using an initial head from a previous model run, which gives you a good starting point. In a real model, you might initially start from a value such as 3 m below the topography. However, such a value will require a "run in time", as the model may take several months to equilibrate.

An alternative is to run the model as a steady-state simulation and then use the steadystate solution as an initial condition. However, the steady state solution still might not be a reasonable starting point depending on how steady the groundwater table is.

3.8.3 Edit the Lay1L.dfs2 grid

- Click on the Edit button, Edit., next to the defined dfs2 file. This will open the dfs2 file in the Grid Editor, which allows you to view and edit the dfs2 file.
- Look at some of the editing options and close the grid editor without saving.



Demo Note: in Demo mode, you will not be able to save any changes to the dfs2 file

MIKE SHE includes sophisticated tools for managing gridded data as well as time series data. The grid editor can also manage a combination of both time series and gridded data - that is a time series of grids. This is very useful for time varying distributed parameters, such as precipitation that is interpolated from rain gauge data, or distributed recharge calculated by third party programs such as DaisyGIS.

The grid editor is a split window that can be dragged sideways to make the map smaller or larger. The table of values on the right side reflects highlighted grid shown on the left-hand map.

Ranges of values can be searched for, selected and changed using the select and

unselect tools,

沙 🖣 📰 🗭

, plus the Tools/Set Value... dialogue.



Important Note

The grid editor is a generic grid tool for all DHI Software, and was originally developed for the Marine programs MIKE 21 and MIKE 3. However, this often leads to confusion in the node and layer numbering because MIKE 21 and MIKE 3 use a different numbering system.

Node numbering: In the Grid Editor (and in MIKE 21 and MIKE 3) the nodes are numbered starting in the lower left from (0,0), whereas in MIKE SHE the nodes are numbered starting in the lower left from (1,1).

Layer numbering: In the Grid Editor (and in MIKE 21 and MIKE 3) the layers are numbered starting at the bottom from 0, whereas in MIKE SHE the layers are numbered starting at the top from 1.

The dfs2 file format includes geographic and cell-size information. The dfs2 file format is part of the larger dfs file format system developed by DHI. This includes:

- dfs0 files for time series data (e.g. rainfall data at a rain gauge),
- dfs1 data, which is not used in MIKE SHE, but used for time series of lines (e.g. the shape of a coast line)
- dfs2 data for time series of 2D grids (e.g. topography or gridded precipitation),
- dfs3 data for time series of 3D grids (e.g. MIKE SHE groundwater head results).

3.8.4 Outer boundary conditions

The *Outer Boundary* is defined by the row of cells on the outside of the model domain. The outer boundary is defined by the *Model domain and grid* file, as all points with a code value of 2 (boundary point).

In the Outer Boundary dialogue:

- Press the Add Item Icon, <a>Press
- Select Zero Flux as the boundary Type



This now gives you a no-flow boundary condition along the entire outside of the model. If the entire outer boundary is a no flow boundary, then this step is actually unnecessary because by default the outer boundary is no flow.

This is realistic in terms of a watershed boundary, but is not really very realistic in this case, because you are not going to specify a river boundary now. In this model, you are basically specifying the river outflow via drainage, in the sense that wherever the water table approaches the ground surface, water will be removed from the model.





The outside of the model can be divided into boundary sections, by clicking on the *Add Item* icon again.

Then click on the Add Point icon, where and click on the map to define the location of the boundary point. The boundaries are specified clockwise around the model starting at the first point in the table.

The outer boundary conditions are defined independent of the model grid. Thus, if you change your model grid the boundaries will be interpolated automatically to the new grid based on the nearest boundary cells to the point locations. If you change the shape or extent of the grid, you should check that your boundaries have not moved unexpectedly.

Internal boundary conditions Internal boundaries are used to specify such things as lakes and reservoirs that are not included in MIKE 11. In this case you could specify a constant head or a general head boundary for the lake.

Internal boundaries are distinguished from outer boundaries because on the outside of the model, you can specify flux and gradient boundaries.



3.9 Step 9 – Pre-process the data

You have now specified all the input required for the model, including

- Selecting the model components
- Specifying the model domain and grid
- Specifying the surface topography
- Specifying the net recharge (precipitation and net rainfall factor)
- Specifying the geological model (Geological layers), including the hydraulic properties.
- Specifying the computational layers, including initial conditions and boundary conditions.

Up to this point, all of the input data has been specified independent of the numeric model. You have specified only the characteristics of the numeric model that will be run. Before actually running the model you must run the pre-processor. The pre-processor extracts all of the spatial data that you have included and applies it to the numerical model that you have defined in the *Model Domain and Grid* and the *Computational Layers*. The pre-processor writes all input data to a binary Flow-Input-File (fif – file) which is then read by the simulation engine when running the simulation.

3.9.1 Run the preprocessor



The preprocessor will start the MIKE Zero Launch utility, which is allows you to set the process priority for the simulation.

Click OK

If you are starting a very long simulation, then you can also set up MIKE Zero to alert you when the simulation is finished.

Launch se	ettings					×
Launch	Purpose	1				
_ Simul	ation					
Notif	ication on	End of Simulation				
	/ia E-mail				_	
	/ia SMS	·				
CDU	Duizuihu					
CPU	Priority	Normal mode		_		
Refr	esh rate	200		milliseconds		
				1	1	-
			ОК	Cancel	Help	





3.9.2 Viewing the preprocessed data

After successfully running the Preprocessor,

- Click on the Processed Data Tab located at the bottom of the Navigation Tree.
- Then, click on the items in the data tree and explore the processed data.





The pre-processor creates a .fif file that contains the cell values etc. that will be used in the simulation. The processed data has been interpolated to the computational mesh, exactly as the simulation engine will read it. However, the .fif file is a binary format optimised for use by the numerical engine.

The fif-file embeds all geometric data. Temporal data (time-series data) are not contained in the fif-file, but are read directly from the source data files during the simulation.

To view the pre-processed data, a parallel set of dfs2 and dfs3 files are also created so that the standard MIKE Zero tools can be used to view the data.

It is the dfs data that is shown in the data tree and listed in the file name text box. Clicking on the View button opens the Grid Editor with the current pre-processed data file loaded – with all the current overlays.

If you want to edit values in the Grid Editor and use the edited values in your simulation, then you need to save the file to a new name after making the changes. Then specify the modified dfs file in the Setup data tree.

Spend some time moving around in the *Processed Data* and make sure you understand the input data and the relation between *Setup Data* and *Processed Data*.



3.10 Step 10 – Redefine the horizontal grid resolution

A very powerful feature of MIKE SHE is that you can change the horizontal grid without redefining your model parameters. Thus, you can perform sensitivity analysis of your model grid and grid resolution in a way that is impossible with other modelling user interfaces.



Demo Note: the demo version is restricted to a maximum of 70x70 cells, so you will not be able to pre-process or run the model with the refined grid.

3.10.1 Change horizontal grid resolution

In the Model Domain and Grid dialogue of the Setup tab:

- Double the number of grid cells in the X and Y directions
- *NX* = 130
- NY = 140
- Change the Cell Size to 250 m

	Model Domain and Grid	
MIKE SHE Flow Model Descript ✓ Display ✓ Simulation specification ✓ Model Domain and Grid ✓ Topography ✓ Precipitation	Catchment defined by Dfs File Catchment defined by Shape File Catchment size and orientation NX: NY: 130 140	C:\Data\MIKE Zero Projects\MIKE_SHE\Model\Model Inputs\ Create Edit Cell Size: Rotation: 250([m] 0 [Deg, Counter clockwise]
Storing of results Setup Proces Results	Catchment origin and map projectio X0: Y0: 0 [m] 0	n Map Projection Type: [m] NON-UTM -

3.10.2 Run the preprocessor

Click on the PP icon to start the pre-processor.

SMIKE Z	Zero - [SH	EPar2]			
🦉 File E	dit View	Refresh	Run	Window	Help
] 🗅 🚅 日	% 🖻	8 8 ?	₩?	Ø PP	им иц

 Verify in the pre-processor tab that all of the input data has been converted to a 250m grid



3.10.3 Change the cell size to 400 meter cells

In the Model Domain and Grid dialogue:

- Change the number of grid cells in the X and Y directions
- *NX* = 100
- NY = 100
- Change the Cell Size to 400 m
- Run the preprocessor

	Model Dom	ain and Grid			
■ MIKE SHE Flow Model Descript ■ ✓ Display ■ ✓ Simulation specification	 Catchment def Catchment def 	ined by Dfs File ined by Shape File	C:\Data\MIKE Zero	Projects\MIKE_SH	HE \Model \Model Inputs \ Create Edit
	Catchment size	and orientation			
M Topography	NX:	NY:	Cell Size:	Rotation:	
🕀 🗹 Precipitation	100	100	400 [r	m] 0	[Deg, Counter clockwise]
🕀 🗹 Saturated Zone					
🗄 🗹 Storing of results					
🔤 🗹 Extra Parameters	- Catchment origin	n and map projection	1		
	×0:	Y0:		Map Projection Ty	ype:
	0	[m] 0	[m]	NON-UTM	•
Setup Proces Results					

There are some constraints when using dfs2 files with different grid spacing or orientations.



If the grids are **coincident** and

- the data is real data (e.g. topography) the finer grid will be interpolated to the coarser grid using the bilinear interpolation to the centre node.
- the data is **integer** data, the integer values will be <u>assigned</u> based on a global population distribution. For example, if 50% of your land area is forest, then 50% of your cells will be assigned the integer code for forest.

If the grids are not coincident and

- the data is **real** data, then the grids will be interpolated in the same way as distributed XYZ and .shp data.
- The data is **integer** data, then an error will be generated because the model cannot <u>interpolate</u> integer data.



3.10.4 Change the cell size back to 500 meter cells

In the Model Domain and Grid dialogue:

- Change the number of grid cells in the X and Y directions
- NX = 65
- NY = 70
- Change the Cell Size to 500 m

	Model Domain and Grid	l de la companya de l
MIKE SHE Flow Model Descript ✓ Display ✓ Simulation specification ✓ Model Domain and Grid ✓ Topography	Catchment defined by Dfs File Catchment defined by Shape File Catchment size and orientation NX' NY'	C:\Data\MIKE Zero Projects\MIKE_SHE\Model\Model Inputs\ Create Edit Cell Size: Botation:
recipitation	65 70	500 [m] 0 [Deg, Counter clockwise]
Setup Proces Results	Catchment origin and map projecti X0: Y0: 0 [m] 0	on Map Projection Type: [m] NON-UTM 🗸

3.10.5 Save your MIKE SHE document

DNG write some TEXT here!



3.11 Step 11 – Specifying Outputs and Calibration Targets

The Detailed Time Series Output allows time Series plots of simulated data at specific locations to be automatically generated during a simulation. It also allows simulated output to be compared to observed output and calculates a number of statistics for model error.

Output is generated in HTML format and can be incorporated into web pages to document calibration progress.

3.11.1 Storing of results

Click on Storing of Water balance data



Set the Storing interval for grid series output to a daily frequency (24 hours) for *Prec*(ipitation), *SZ-heads* and *SZ-fluxes*

Model Domain and Grid Model Domain and Grid Precipitation Saturated Zone Storing of results Detailed timeseries out Grid series output Extra Parameters	Storing of results Water Movement Dutput Storing of Water balar Storing of Hot start da Only store Hot Hot start.stor	nce ta start data at the end ing interval: 1 series output	l of simulat	ion (hrs)	oring of Stor Stor	input data for WQ re SZ flow data only re all flow data	simulation
	Overland (OL):	Prec		SZ-Heads:		SZ-Eluves:	
	120	rs) 24	(hrs)	24	(hrs)	24	(hrs)

The Water balance items are used during the post-processing of the results to calculate water exchanges between the various components

The Hot start data is used for starting transient simulations from the end of a previous simulation. This allows you to modify things such as land use over time, or to do scenario analysis from the same starting scenario.

Gridded output is stored as dfs2 (2D grid time-series files) or dfs3 (3D grid time-series files). If output data is stored too frequently, you may get very large and unwieldy result files. Thus, before running the simulation you should consider how often you need to save the data.



3.11.2 Define detailed time series output items

- Select the Detailed timeseries output
- Use the Add item icon, , to add 4 items to the list
- For each item,
 - Set the name
 - Select head elevation in saturated zone under Data Type
 - Define the X, Y coordinates from the table below:

Name	X-coordinate	Y-coordinate
Obs 5	24103	8419
Obs 35	13800	17607
Obs 37	17508	20887
Obs 65	8866	25549

- Define the Depth = 5
- Make sure Incl. Obs. Data is not selected





÷

The X and Y location can be selected on the map using the 🕸 button.

10000

20000

30000

[meter]

5000

0

0

Observation data will be added in a later exercise.

.

Results

Proces

Setup ...

You are also welcome to add any other points and data items you like.



3.11.3 Grid outputs

In the *Grid series output* dialogue many items will already be selected if you have turned on the storing of Water Balance data.

• Some gridded items of interest are not connected to the of water balance data. By clicking on the check box, turn on *depth to phreatic surface (negative)*.

Gridded output is stored as dfs2 (2D grid time-series files) or dfs3 (3D grid time-series files). If all output items were saved, you may get very large result files. Thus, before running the simulation you should consider which output items you need.

The groundwater flow direction output is required so that you can plot flow vectors on the plots.

MIKE SHE Flow Model Descript				
🗄 🗹 🖌 Display				
🕀 🖌 Simulation specification	Grid seri	es output		
Model Domain and Grid				
	Enable	Item	Required for	Filename
🔤 🗹 Topography	1 7	precipitation rate	vVater Balance	SZ1-DHI\SZ1-DHI_ET_AllCells.dfs2
🕂 🖌 🖌 Precipitation	2	depth of overland water	Water Balance	SZ1-DHI\SZ1-DHI_overland.dfs2
Columbad Zone	3	overland water elevation		SZ1-DHI\SZ1-DHI_overland.dfs2
	4	infiltration to UZ (negative)	Water Balance	SZ1-DHI\SZ1-DHI_2DUZ_AllCells.dfs2
🖻 🗹 Storing of results	5 🔽	exchange between UZ and SZ (pos.up)	Water Balance	SZ1-DHI\SZ1-DHI_2DUZ_AllCells.dfs2
🚽 🖌 Detailed timeseries out	6	depth to phreatic surface (negative)		SZ1-DHI\SZ1-DHI_2DSZ.dfs2
	7 🔽	head elevation in saturated zone	Vater Balance	SZ1-DHI\SZ1-DHI_3DSZ.dfs3
📔 👘 🗹 Detailed M11 timeseries 📗	8 🔽	seepage flow SZ -overland	Vater Balance	SZ1-DHI\SZ1-DHI_2DSZflow.dfs2
Grid series output	9 🔽	seepage flow overland - SZ (negative)	Vater Balance	SZ1-DHI\SZ1-DHI_2DSZflow.dfs2
M and selles output	10 🔽	3D UZ recharge to SZ (negative)	Vater Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
🖙 🖌 Extra Parameters	11 🔽	groundwater flow in x-direction	Vater Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
	12 🔽	groundwater flow in y-direction	Vater Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
	13 🔽	groundwater flow in z-direction	Vater Balance	SZ1-DHI/SZ1-DHI_3DSZflow.dfs3
	14	SZ head elevation stored with SZ flows		SZ1-DHI/SZ1-DHI_3DSZflow.dfs3
	15 🔽	groundwater extraction	Vater Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
	16 🔽	SZ drainage flow from point	Water Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
	17 🔽	SZ flow to general head boundary	Water Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
Setup Proces Results				



3.12 Step 12 – Run the Simulation

Before running the model, you need to go back to the Setup menu to make a few changes to the simulation specifications, including simulation period, time-steps and storing of results.

3.12.1 Specify the time step parameters

In the Time step control dialogue:

- Specify Initial time step = 24 hrs
- Specify *Max allow SZ time step* = 24 hrs
- Max infiltration amount per time step = 10

Leave the rest of the parameters at their default values

	Time step control		
	Time Steps		
	Initial time step	24	[hrs]
	Max allowed OL time step	2	[hrs]
	Max allowed UZ time step	2	[hrs]
	Max allowed SZ time step	24	[hrs]
 ✓ Simulation title ✓ Simulation period ✓ Time step control 	Parameters for Precipitation-dependent	time step control	
Time step control	Parameters for Precipitation-dependent	time step control	
Model Domain and Grid	Max precipitation depth per time step	10	[mm]
 Precipitation Cohested Zene 	Max infiltration amount per time step	10	[mm]
Grow Saturated ∠one Grow Storing of results Grow Storing Parameters	Input precipitation rate requiring its own time step	0.1	[mm/hr]

The max infiltration amount and the max precipitation depth per time step are used to improve the numeric stability of the solution when there is ponded water on the ground surface.



If the maximum values are exceeded, then the time step will be automatically decreased until this limit is reached.

If you encounter small time steps during rainfall events, you can sometimes speed up the simulation by increasing these values. Look in the log files (information text files, which are created during the simulation) to see if the time step is being automatically reduced because of these criteria.



3.12.2 Simulation control

In the SZ Computational Control Parameter dialogue:

Choose Preconditioned Conjugate Gradient, Transient (default)



The default values for the simulation convergence parameters are normally suitable and in most cases need not be changed.

	SZ Computational Control Parameters							
	Solver Type Preconditioned Conjugate Gradient, Preconditioned Conjugate Gradient, Successive Overrelaxation Package	Transient Steady State (SOR)						
	Iteration Control							
	Maximum no. of iterations	50						
	Maximum head change per iteration	0.005	[m]					
	Maximum residual error [m/d]	0.0005						
	Sink de-activation in drying cells							
	Saturated thickness threshold	0.05	[m]					
MIKE SHE Flow Model Descript	Advanced Settings Image: Constraint of the set of the							
	Horizontal Conductance averaging b Under-relaxation No under-relaxation (Recommender)	etween iterations (ed)	n ecommendedj					
Model Domain and Grid ✓ Topography ✓ Precipitation ✓ Saturated Zone	 Under-relaxation with dynamic calc Under-relaxation with constant fac 	culation of factor tor (0.01-0.99):	0.99					
🗄 🗠 🗹 Storing of results	Maximum exchange from river during one time step							
	Max. fraction of H-point volume	0.9						
Setup Process Results								

3.12.3 Run the simulation

Click on the **PP** icon to re-run the pre-processor and then on the Water Movement icon to run the simulation

8 M	IIKE	Zero	- [SH	EPar2]			
	File	Edit	View	Refresh	Run	Window	Help
ם∥	B	R X		8 6 ?	₩?	Ø PP	им иц



3.13 Step 13 – View the Results

Upon successful execution of the model you're now ready to view the results.

The Results Viewer allows you to view and animate results from all DHI dfs file types. This first exercise will guide you through only the most basic functionality of the Result Viewer.

3.13.1 Locate the detailed time series output results

After successfully running the model,

Click on the Results Tab located at the bottom of the Navigation Tree.





The *MIKE SHE Detailed Time Series* output dialogue, under *Simulation Results*, lists the Detailed Time Series results you have chosen to store during the MIKE SHE simulation.

If you have specified more than 5 items for detailed time series output, then you will only see a page of links on the main *Detailed time series* page. The links will direct you to separate .html files with 5 graphs in each.



3.13.2 Display the gridded results

- Now select the Gridded Data Results Viewer
- Check on the Add XY flow vectors for the head elevation in the saturated zone plot
- Finally, click on the *View Result…* button to view the simulation results in the Results viewer tool.

	Gri	idded Data Results Viewer			
MIKE SHE Row Model Description	Layer 1	no. for Groundwater items			
Gridded Data Results Viewer		Item	Add XY-flow		Filename
Mike 11 Detailed Time Series	1	precipitation rate		View result	C:\Program Files\DH\\MIKEZero\Examples\
Post-Processing	2	depth of overland water		View result	C: Program Files/DHI/MIKEZero/Examples/
Run Statistics	3	infiltration to UZ (negative)		View result	C:\Program Files\DHI\MIKEZero\Examples\
- I	4	exchange between UZ and SZ (pos.up)		View result	C:\Program Files\DHI\MIKEZero\Examples\
	5	depth to phreatic surface (negative)		View result	C:\Program Files\DHI\MIKEZero\Examples\
	6	head elevation in saturated zone	Ē	View result	C:\Program Files\DHI\MIKEZero\Examples\
	7	seepage flow SZ -overland	Ē	View result	C:\Program Files\DHI\MIKEZero\Examples\
	8	seepage flow overland - SZ (negative)		View result	C:\Program Files\DHI\MIKEZero\Examples\
	9	groundwater flow in x-direction		View result	C:\Program Files\DHI\MIKEZero\Examples\
	10	groundwater flow in y-direction		View result	C:\Program Files\DHI\MIKEZero\Examples\
	11	groundwater flow in z-direction		View result	C:\Program Files\DHI\MIKEZero\Examples\
< III +	12	SZ drainage flow from point		View result	C:\Program Files\DHI\MIKEZero\Examples\
Setup Data Processed Results	•				

The *Results Viewer* dialogue lists the results you have chosen to store in the MIKE SHE results files.



The XY flow vectors check box adds velocity flow vectors calculated for each cell.

The *Layer no for groundwater items* is used to select the numerical layer number in groundwater models with multiple numerical layers.

3.13.3 View the results in the results viewer

Note that the velocity vectors will not appear in the initial time step, but rather appear first in the second time step.





3.14 Step 14 – Add Geological Lenses

The geological model defined up to now is a 1-layer system of homogenous sand. In this step you will add a clay lens within the sandy aquifer.

3.14.1 Add the geological lenses

Go back to the Setup Data and in the Geological Lenses dialogue

- Click on the Add Item icon, , to add a lens.
- Change the Name of the lens to "Clay"

MIKE SHE Flow Model Description	Geological Lenses	
 ✓ Moder Domain and Grid ✓ Topography ✓ Precipitation 	Geological Layers	
Image: Second Secon		
Geological Lenses Geological Lenses	Name 1 Clay	
Setup Data Processed Data Results		



3.14.2 Define horizontal extent of the lens

In the Horizontal Extent dialogue

- Select Grid file (.dfs2) files for Spatial Distribution
- Use the Browse button to select the file .Wodel Inputs\MapsVense3.dfs2



8

All the lenses will be assigned the same hydraulic properties. If you want to assign different hydraulic properties to different lenses, then you need to add additional lenses (see the step above)

Lenses can also be assigned using polygon shp files.



3.14.3 Define the top of the lens

In the Upper Level dialogue

- Select Grid file (.dfs2) files for Spatial Distribution
- Use the Browse button to select the file
 .Wodel Inputs\Maps\lense3U.dfs2





3.14.4 Define the bottom of the lens

In the Lower Level dialogue

- Select Grid file (.dfs2) files for Spatial Distribution
- Use the Browse button to select the file .Wodel Inputs Waps Vense 3L.dfs2





3.14.5 Define hydraulic properties for the lens

For each hydraulic property of the lens use a *Uniform Spatial distribution* with the following values:

- Horizontal Hydraulic Conductivity : 1e-006 [m/s]
- Vertical Hydraulic Conductivity: 1e-007 [m/s]
- Specific Yield : 0.2 [-]
- Storage Coefficient : 0.0001 [1/m]



3.14.6 Evaluate the pre-processed data

Run the preprocessor and explore the preprocessed data.



Lenses are treated as separate geologic layers in the cells where they are located. Thus, the pre-processor reads the lens geometry (area, top, bottom) and attributes (hydraulic properties) and interpolates the lens information into the hydraulic properties of the numerical layer. You will see the effect of the lens by looking at the hydraulic properties of the layer in the Pre-processed data tab.



3.15 Step 15 – Advanced use of the Results Viewer

This step demonstrates some of the advanced features of the Result Viewer that have not been discussed previously.

3.15.1 Locate the Results

After looking at the Detailed Time Series Output,

• click on the Gridded Data Results Viewer item in the Results navigation tree.

	G Laye	ridded Data Results Viewer			
		Item	Add XY-flow vectors		Filename
	1	precipitation rate		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
	2	depth of overland water		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
	3	infiltration to UZ (negative)		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
	4	exchange between UZ and SZ (pos.up)		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
	5	depth to phreatic surface (negative)		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
MIKE SHE Flow Model Description	6	head elevation in saturated zone	V	View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
Simulation Results	7	seepage flow SZ -overland		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
Mike SHE Detailed Time Series	8	seepage flow overland - SZ (negative)		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
Gridded Data Results Viewer	9	groundwater flow in x-direction		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
Mike 11 Detailed Time Series	10	groundwater flow in y-direction		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
	11	groundwater flow in z-direction		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
	12	SZ drainage flow from point		View result	C:\Program Files\DHI\MIKEZero\Examples\MIKE
Setup Data Processed Data Results	•				

The *Results Viewer* dialogue lists the results you have chosen to store in the MIKE SHE results file. In this exercise, you chose to store the precipitation (groundwater recharge), depth to phreatic surface, groundwater head, and the groundwater flow in the x-, y- and z-directions.

3.15.2 View the results in the results viewer

• Click on the View Results, <u>View result.</u>, button for the item *head elevation in* saturated zone.





3.15.3 Time series plot at a point

- Press the Time Series button
 on the Results Viewer tool bar
- Click once inside the map area and then while holding down the Ctrl-key, click on each additional point where you want time series output. Each selected locations is marked with a yellow x.
- Double click on the last point. (if you only want one point then simple double click without holding down the Ctrl-key)
- × Items | Item Overview Displ Accu ltem name head elevation in saturated zone (31, 41, 1)
 C:Program Files/DHI
 head elevation in saturated zone (26, 49, 1)
 C:Program Files/DHI
 head elevation in saturated zone (31, 35, 1)
 C:Program Files/DHI
 head elevation in saturated zone (31, 35, 1)
 C:Program Files/DHI
 head elevation in saturated zone (31, 35, 1) বন 65 **N** head elevation in saturated zone (31, 35, 1) C:\Program Files\DH F ΟK Cancel Help 00:00 1980-06-03 00:00 00:00 06-13 00:00 06-18 00:00 06-23 00:00
- Check *Display* and press OK to display the time series.

Export to a dfs0 file

- Right-click in the time series plot.
- Select Export... in the pop-up window.
- Check Export and press OK.
- Enter an appropriate file name for the dfs0 file and press Save.

	Zoom In Zoom Out	
	Previous Zoom	
	Next Zoom	
	Pan	(Shift)
	Refresh	
~	Grid	
	Copy to Clipboard	
	Copy to Clipboard Save to Metafile	
	Copy to Clipboard Save to Metafile Save to Bitmap	
	Copy to Clipboard Save to Metafile Save to Bitmap Font	
	Copy to Clipboard Save to Metafile Save to Bitmap Font Properties	



The j,k coordinates listed at the bottom of the Results Viewer go from 0 to (nx-1) and 0 to (ny-1) in the j and k directions, respectively.

Note that the time series time steps are equal to the storing time steps. The *Detailed timeseries output* option within the *Setup Data* allows you to save the output at every time step.



3.15.4 Extract a profile of water levels

- Press the button and the Results Viewer tool bar to extract a profile from the simulate output item displayed
- Click in the area of interest to select the points along the profile line. Double click on the last point of the profile to terminate the profile line. The profile line is indicated with a thick green line.

Check 3D head elevation in saturated zone and press OK.



	Display	Item name	File name	Item type	Unit	Glbr
1		3D Vertical Conductivity	E:\Projects\Tr	MShe	m/s	0
2		3D Storage Coefficient	E:\Projects\Tr	MShe	1/m	0
3		3D Specific Yield	E:\Projects\Tr	MShe	0	0
4		3D Horizontal Conductivity, Y-dir.	E:\Projects\Tr	MShe	m/s	0
5		3D Horizontal Conductivity, X-dir.	E:\Projects\Tr	MShe	m/s	0
6		3D head elevation in saturated zone	E:\Projects\Tr	MShe	meter	18.8068
7		3D Calculation layers	E:\Projects\Tr	MShe	meter	0
-						



3.15.5 Display the computational grid on profile.

In the Projects pull-down menu choose

Active View Settings -> Profile...

On the Graphical Items tab of the dialogue that appears, under Calculation layers,

- check Draw Calculation layer and select Draw as grid to show the computational grid or Draw as lines to show the upper and lower surface.
- Press OK to view the selection





3.15.6 Modify visible overlays and display the finite difference mesh in result viewer

First, return to the horizontal view of *head elevation in saturated* zone, by either closing the cross-section view or switching via the *Window* pull-down menu

In the Projects pull-down menu choose

Active View Settings -> Horizontal ...



In the dialogue that appears,

• Single click on the *Grid file* item **SZA_3DSZ.dfs3** to view the available display options.

Under the Miscellaneous items,

- check Element Mesh.
- Press OK to execute changes.

 ✓ Result Data ✓ Grid file: C\Data\Doci ✓ Grid file: C\Data\Docur ✓ Grid file: C\Data\Docur ✓ Image file: C\Data\Docur ✓ Shape file: C\Data\Docur ✓ Vector file: C\Data\Docur ✓ Vector file: C\Data\Docur 	Styles Colors 2D Grid Styles Contour type Box contour	Draw box contour without divide	Apply to all Data Sets	
	Isolines Draw isolines Draw labels Properties	Miscellaneous F Element mesh F Color legend Outline grid Smooth edge		
	Item/Layer Item head elevation in saturat Slice type: Layer slicing Layer 1	ed zone		
>				



3.16 Step 16 Create a Water Balance

The water balance utility in MIKE SHE is a vital part of the results analysis.

3.16.1 Create a new water balance document

- Click on the New document icon, D, or the File/New/File menu
- Select MIKE SHE and then the Water Balance Calculation (.wbl) document
- Click OK
- In the Project Explorer select the directory where you want to save the document
- Then give the file a name (e.g. SZWaterbalance) and click on the Save icon,





In MIKE SHE, water balances are calculated by the water balance utility. This is run separately after the simulation. Creating a water balance involves 3 steps:

- 1. Extract the water balance data from the MIKE SHE output files
- 2. Create specific water balances from the extracted data
- 3. Evaluate the water balance output

Save		×
Save As	Courses along	
Name:	SZWaterbalance	
Location:	C: (Data(MIKE 2010 Projects(MIKE_DHE (Result	
Notes:		
Noces.		
		Ŧ
Add to Ver	sion Control	OK Cancel



3.16.2 Extract the water balance data

- Use the browse button and find the .sheres file from your simulation in your Result directory
- Run the water balance extraction by clicking on the Extraction icon, selecting the drop down window *Run* and clicking on *Extraction*. If the extraction icon does not appear at all, most likely the Run toolbar is not activated. Select the drop down window View, Toolbars and select the Run toolbar (last entry).
- Check to make sure that the Extraction ran successfully, by looking for "Normal termination" in the message window

SZWaterbalance.wbl	
🖌 Extraction	Extraction
	Water movement simulation Flow result catalogue file: C:\Data\MIKE Zero Projects\MIKE_SHE\Result\SZ1.sf
	Type of extraction Area Type: Catchment
	Resolution Type: Area 👻
	Sub-catchment grid codes Type of input file: Dfs2 Item: Dfs2 file:
	Gross files
	Pre-hame of gross files: [C:\Data\MIKE Zero Projects\MIKE_SHE\Result\SZ1.she - Result Files\SZ1_1_1] []
Extraction Post pr Result	
Normal termination	A T
100 %	45 seconds ■ ↓

MIKE SHE saves the results in many different files. The results are grouped by process (e.g. OL, UZ, SZ) and by output type (i.e. 2D or 3D). The extraction process reads all of the various output files and builds a set of water balance files. These files can be efficiently read by the water balance utility.

The .sheres file is an ASCII catalogue of all the output files associated with the simulation.

The type of extraction is important, as it defines what subsequent water balances can be calculated.

- Area Type Choosing subcatchment allows you to create water balances for subareas in the model domain. Otherwise, the water balance is for the entire model area.
- Resolution type By default the water balance is for the entire catchment or subcatchment. The Single-cell option allows you to create maps of some of the water balance items.



3.16.3 Create water balance items

- In the *Postprocessing* tab, add two post processing items to the table by clicking on the new item icon,
- Change the name of the two items, so that you can distinguish them from one another



You can set up any number of water balance calculations for a single water balance extraction. Since these are all stored in the water balance document (.wbl file), you can re-run the water balance extraction and processing after each simulation.

The water balances are defined in a water balance .config file, which is located in the program installation directory (C:\Program Files (x86)\DHI\2016\bin).

If you want to create customised water balances, then you can edit the configuration file. This process is described in the MIKE SHE User Guide.

SZWaterbalance.wbl - Modified		
Postprocessings X Total water balance grap	Postprocessings	
🔤 🗙 Time varying water balan	Alternative config file:	Vse default Config file
	C:\Program Files (x86)\DHI\2011\bin\MSh	≥_Wbl_Config.pfs
	Postprocessing name	Comment
	2 Time varying water balance	
<►		
Extraction Post proce Result	•	
Normal termination		•
Validation Simulation	/	



3.16.4 Define a Chart water balance

- Select *Chart output: Total water balance* from the list of available Water balance types.
- Define a file name for the output file
- Run the water balance by clicking on the water balance icon *run selected* postprocessing,

SZWaterbalance whi					
Total water balance grap	Total water balar	nce graphic			
Time varying water balan	Water balance				
	Water balance type:	hart output: Total	water balance		-
	Description: C	hart output: Gene ntegrated)	ral water balance of	the entire model (depth)+
	Output period	Start date:		End date:	
	🔽 Use default period	1970/02/0	01 00:00 🚔 💌	1970/07/31 00:00	÷.
	- Output Timeseries Speci	ifications Outo	ut time step (hrs);	Tupe:	
	Use default output	time step	240	Accumulated	-
	Layer Output Specification	ons			
	Layer: All layers		Layer no.:		0
	Sub-Catchment Selectio	n	Single-Cell Lo	cation	
	Grid code:	0	X-index:	0 Y-index:	0
	Output File				
		Txt file:	TotaWB.txt		
Extraction Post proce Result					
					-
Normal termination					-
100 %][][45 seconds	.
Validation Simulation					



The Chart output creates a graphic illustrating the water balance items (see next step). The output is sent to an ASCII .txt file, and the graphic is generated automatically by a utility that reads txt file.

By default, the water balance is summed for the entire simulation period. However, you can easily change the water balance period, for example, to create seasonal water balances, or to remove the initialisation period.

If you defined the extraction on a sub-catchment basis, then you can select from the available sub-catchments in this dialogue.



3.16.5 View the water balance output

- On the Result tab, select the sub-item for the Chart output
- Click on the Open button beside the file path to create and open the water balance graphic



Note: your numbers may be different

The water balance graphic includes all of the non-zero water balance items in the current simulation (except for storage changes, which are always plotted).

In the graphic to the right, we can see that there was 31 mm of net precipitation during the simulation. That is the *Total precipitation* times the *Net rainfall fraction*.

All of this infiltrated to the groundwater system, where 17mm was lost to drainage and the groundwater storage increased by 14mm.

Note that the water balance items are all normalised to units of mm, which allows for easy comparison between the different components. If you want to convert depths of mm to volumes, you need to multiply the depth by the internal model area. That is, the total model area minus the area of the outer boundary cells. If you load the pre-processed data into the Grid Editor, select the Model and Domain item. Then, you can use the statistics function in the Grid Editor to find out the number of internal versus external cells.

Note that the sum of the water balance terms in the graphic do not add to zero. This is because some of the signs have been removed (e.g. Precipitation and ET have the same sign, but opposite inflow/outflow directions). In the water balance calculation, Inflows are negative and Outflows and Storage increases are positive. Thus, in the graphic to the right, the water balance is:

Storage change [+14] + Drain flow [+17] – Infiltration [-31] = 0



3.16.6 Define a time varying water balance

- Return to the Post-processing tab and select the second water balance item
- Select Total water balance from the list of available water balance types
- Leave the Output type as Accumulated
- Select the Output file type as a Time series file
- Input a filename with the .dfs0 extension and click on the browse button to specify the location where the file should be saved



Run the water balance by clicking on the run water balance icon, igveete

SZWaterbalance.wbl - Modified	
Postprocessings	Time varying water balance
Time varying water balan	Water balance
	Water balance type: Total waterbalance
	Description: General water balance of the entire model setup
	Output period Start date: End date:
	✓ Use default period 1970/02/01 00:00 ↓ 1970/07/31 00:00 ↓
	Output Timeseries Specifications
	Use default output time step 240 Accumulated
	Layer Output Specifications
	Layer: All layers
	Sub-Catchment Selection Single-Cell Location
	Laria code: U X-index: U Y-index: U
	Uutput File Type: Time series file Dfs0 file: sult\SZ1.she - Result Files\accTotalWB.dfs0
Extraction Post proce Result	
Normal termination	^
100.9/	
Validation Simulation	

If you chose the Table format for the Output file, then the output will be in a tab-delimited ASCII file that you can open in Excel.

In this exercise we have chosen an Accumulated water balance. This sums the water balance over the simulation period.

The alternative is to define an incremental water balance, which outputs the water balance time series at each saved time step.



3.16.7 View the water balance output

- On the Result tab, select the sub-item for the time series output, *Time varying water* balance
- Click on the Open button beside the file path to create and open the water balance dfs0 file





The water balance time series file includes all of the available water balance items. In the current water balance almost all of the items are zero.

The cumulative water balance sums the items over all of the stored time steps. Thus, the total precipitation to the right is 31.104 mm, which is the same as the 31mm in the graphic total water balance above.

Note, though, that the sign of the precipitation is negative in the time series file. Generally, all the items in one time period will sum to zero. There are exceptions, though, that are described in detail in the Water Balance documentation.

One important thing to remember is that the water balance is only an output of the saved time steps. Thus, the dynamics between the saved time steps are lost.


3.17 Step 17 – Add the River Module

You will now add a river to the groundwater model that you created in the SZ1 exercise. In this exercise, you will use an existing MIKE 11 river setup.

3.17.1 Select simulation modules

In the Simulation specification dialogue

Check Rivers and Lakes (OC) On

Your model now has a Rivers and Lakes branch in the Data Tree.

	Simulation specification
	Numeric Engine
MIKE SHE Flow Model Descript Display Model Domain and Grid Model Domain and Grid Model Domain and Grid Model Domain and Grid Model Domain and Grid Saturated Zone Storing of results Kata Parameters Setup Proces Results	Water Movement (WM) Overland Flow (OL) Finite Difference Rivers and Lakes (OC) Unsaturated Flow (UZ) 2 Layer UZ Evapotranspiration (ET) Saturated Flow (SZ) Finite Difference Include Advection-Dispersion (AD) Water Quality

3.17.2 Add the River Module

In the Rivers and Lakes Dialogue

Use the browse button to select the existing sim11 file
 .\Model Inputs\MIKE 11\karup.sim11

• Press the Edit... button to enter the MIKE 11 simulation editor



MIKE SHE Flow Model Descript	Rivers and Lakes
Simulation specification Model Domain and Grid Topography Precipitation Rivers and Lakes Saturated Zone Storing of results Extra Parameters	River Simulation File (*.sim11) Zero\Examples\MIKE_SHE\Model\Model Inputs\MIKE11\kļarup.sim11 Edit Inundation Areas Flood Codes Bathymetry
Setup Proces Results	

3.17.3 Explore the MIKE 11 simulation document

The sim11 file contains a complete river model setup. You are not going to build a river model as part of this exercise.

The Simulation editor includes five tabs:



- 4. The <u>Models</u> tab allows you to select the model components.
- 5. The <u>Input</u> tab allows you define the links to the model input files. A typical MIKE 11 model for MIKE SHE requires the following documents:
 - river network (nwk11),
 - cross-sections (xns11),
 - boundary conditions (bnd11), and
 - hydrodynamic parameters (HD11).

🥌 karup.sim11				
Models Input Sim	ulation Results Start			
Input Files				
Network	rojects\MIKE_SHE\Model\Model Inputs\MIKE11\ka	rup.nwk11	Edit	
Cross-sections	rojects\MIKE_SHE\Model\Model Inputs\MIKE11\ka	arup.xns11	Edit	
Boundary data	rojects\MIKE_SHE\Model\Model Inputs\MIKE11\ka	rup.bnd11	Edit	
RR Parameters			Edit	
HD Parameters	rojects\MIKE_SHE\Model\Model Inputs\MIKE11\ka	arup.HD11	Edit	
AD Parameters			Edit	
ECOLab Param.			Edit	
ST Parameters			Edit	
FF Parameters			Edit	
DA Parameters			Edit	
Ice Parameters			Edit	
HD Results				
RR Results				
	,			
0 %	6			



- 6. The <u>Simulation</u> tab allows you to set the time step and simulation length. However, the simulation length is over-ridden by MIKE SHE.
- 7. The <u>Results</u> tab allows you to set the output file name and storing frequency for the output. In dynamic simulations the time step for MIKE 11 can be quite small, so it is often necessary to reduce the storing frequency, otherwise the files may become very large.
- 8. The <u>Start</u> tab allows you to see if there are any data input errors in your setup and start the simulation when you are ready.

3.17.4 Change the results directory

The current results directory should be changed to the Results directory.

.\Result\MIKE 11\Karup.res11

 Use the browse button on the Results tab to change the Result directory and add the MIKE 11 directory if necessary

karup.sim1	L - Modified	rt		
Models input Results — HD: AD: ST: RR:	Filename E\Result/KARUP.RES11	Storing Frequency 24 1 1 1 1	Unit Time step v Time step v Time step v	
	0%			

Save your changes.



3.17.5 Explore the branch network

On the Input tab of the Simulation editor





The warning message that appears, is simply telling you that any overlays in the MIKE 11 file will be overwritten by the overlays from MIKE SHE.

All information in MIKE11 can be accessed and modified from the *network editor*. You are now in the *Network Editor*, which is the core editor of MIKE 11. From here, you can build and modify your river model.



3.17.6 Routing branches

Switch to the Network Tabular View, by either

- Selecting View from the top menu and then Tabular View... or
- Using the Ctrl+t keyboard shortcut while being in the graphical view
- In the Network > Branches Dialogue, note that the Branch Type for all branches are set to Kinematic Routing





A <u>regular</u> branch will calculate the water level based on the Saint-Venant equation (Kinematic, Diffusion or Dynamic Wave formulations). For detailed simulations, you would generally use regular branches.

However, often it is convenient to use simpler routing formulations, where the water in the river is moved down the river – without actually calculating the water levels. Water levels are calculated afterwards knowing the flow rate in the cross-section and back-calculating a water level.

The main advantage of the routing methods is their speed. They require less data, they are numerically stable and you can use very long time steps if you want.

The downside is that the calculated water levels are less precise and hydrodynamic effects are ignored. Water levels are simply calculated based on the amount of water in the river in the current time step.

You may also combine different approaches, for instance by using simple routing techniques on the upstream, steep branches and the dynamic wave approaches on the downstream low-gradient branches.



3.17.7 Routing settings



In the *Routing > Kinematic Routing Method* dialogue you do not need to specify anything. If no specifications are made, the model will:

- Calculate water levels using the *Manning Equation* assuming that the energy slope equals the bed slope derived from cross-sections (Manning numbers are specified in the HD11 file default is M=30 [m^{1/3}/s], n=1/M).
- Use the *No-Discharge Transformation* option, which implies that inflows are summed along the branch and transferred to the downstream end of the branch in the same time-step. This is the same as a full pipe with a unit of water entering one end and pushing a unit out the far end.
- Another discharge computation option is the Muskingum methods. It dampens the peak flows as they travel downstream, which is necessary when the travel time in the river is much longer than the time step. For example, in large river networks, it may take several days or longer for a flood wave to travel from the source to the river mouth.

		tiver Nar Bording (ttributes Compute Manning	me creek elevation by resistance mel	Chainage 7190 Con hod V No	npute discharge discharge tran	ID model cros = by sformation	s		– User-D	efined	IQH-Re	ation
		uskingu) verview	m Routing Para	neters × 0 Chainage	ID		Compute di		K		*	
	l h	1	Bording cr	7190	model cross	Mapping r	No dischar	n	<u> </u>	0	<u>^</u>	-
		2	Karup	51990	model cross	Manning r	No dischar	0		0		-
		3	Haderup R	14190	model cross	Manning r	No dischar	0		0		
Dverview		4	Hauge Cre	12090	model cross	Manning r	No dischar	0		0		
🖂 Mahuark		5	Feldborg	8090	model cross	Manning r	No dischar	0		0		
H Herwork H												-

You may include additional routing points at different locations along a branch, but this will not be covered in this course.

In this exercise, you are using the simplest option, where no routing is used. Thus, whatever water flows into the branch network at the boundaries or as lateral inflow, will be transferred downstream in the same time-step.

MIKE SHE keeps track of and conserves mass and does not allow water to infiltrate from the river to the groundwater if the river is empty.



3.17.8 MIKE SHE - MIKE 11 coupling



In this dialogue, you specify which MIKE 11 branches will be coupled to MIKE SHE. You may choose to include some of the MIKE 11 branches or all of them, or even just part of a branch.

The coupling specification includes the definition of the river-aquifer exchange, including the river bed leakage coefficient. The river leakage can also depend on the hydraulic conductivity of the aquifer material when the MIKE SHE cell size is much bigger than the river width.

By default, exchange between MIKE SHE and MIKE 11 is only one-way. Overland surface water will flow into MIKE 11, but water will <u>not</u> flow back onto the ground surface if the river floods. However, there are two options for flooding.

- Cells can be defined by flood codes, where they will be flooded if the elevation of the cell is below the water level in the nearest H-point.
- Alternatively, you can allow overbank spilling. In this case, the river will discharge the water onto the ground surface based on a weir formula.

	Cocation - Branch m Upstream Downstre Conducta Leakage	ame n Chainage sam Chainage ance A Coef. <u>1E</u> Include all of MIKE SHE	Bording creek 0 7200 uifer + Bed -005 Branches Coupling Reac	× In F B hes	feir data for overland Select weir option in Allow overbank sp Minim undation options by lood Area Option Iood Code ed Topography d Leakage	Hiver exchange MIKE SHE) Ight above ban Jilling um flow area for Flood Code No flooding Jse Grid Data Jse Grid Data	e Weir coefficie Head expone ik for full weir wid ar overbank spillin	nt 1.838 nt 1.5 th 0.1	
		Branch Name	US. Chainage	DS. Chainage	Conductance	Leakage Coef.	Flood Area	Flood Code	Bed 1
	1	Bording cr	0	7200	Aquifer + Bed	1E-005	No flooding		Use Gri
	2	Karup	0	52000	Aquifer + Bed	1E-005	No flooding		Use Gri
	3	Haderup R	0	14200	Aquifer + Bed	1E-005	No flooding		Use Gri
- Overview	4	feldborg	0	8100	Aquifer + Bed	1E-005	No flooding		Use Gri
The Makwada	5	hauge cre	0	12100	Aquifer + Bed	1E-005	No flooding		Use Gri
Structures Structures Routing MKE SHE links (5) MKE SHE links (0) Grid points	-								Þ

3.17.9 View cross-sections

In the *Network Editor (Graphical View)* at each of the red markers on the river, a cross-section has been defined.

- Move the cursor to a red marker and right click
- Choose Edit from the pop-up menu
- Choose Cross Section at: ..., and
- Click OK.



3 M Val		Edit Object	? ×
A V //		Point152 (17968.9, 12552.1) at: 21768.9	
	Point Properties	Branch Karup River 0 - 52000	
	Insert •	Lross Section at: Karup Hiver, model cross, 21768 []	
	Edit		
	Delete		
	Zoom In	-	
	Zoom Out		
	Zoom Out		
	Previous Zoom		
	Next Zoom		
	Pan (Shift)		
	Refresh	OK Cancel Hel	p
C Paralum.	🖌 Grid		

This will take you to the Cross-section Editor. Look at a couple of cross-sections.

Alternatively, you can open the cross-section file directly from the Simulation Editor (.xns11).



Note:

Cross-sections are identified by a Branch ID and a TOPO ID

Cross-section data are relative to the specified *Datum* value.



 Cross-section points are editable in the table and graphical area by selecting the points in the plot and using the *Move Points* option (right click).



• The *Marker* values 1 and 3 in the table define the extent of the cross-section to be used in the simulation.

Close the Cross-Section Editor and return to the Network editor, without saving any changes.

3.17.10 View the boundary conditions

By default, the boundary nodes are shown on the network with blue markers at 90 degrees to the cross-section markers.

- Right click on the boundary marker
- Chose Edit in the pop-up menu, and select the Boundary item

		AMD A	9 2	Edit Object	2 1
3ording Kirkeby		Nede Julia		Point167 (24737.4, 1371.9) at: 0 Branch Bording creek 0 - 7200 Cross Section at: Bording creek, model cross, 0 () Boundary (HD) at: Bording creek 0	
ing	Jul	er ianer		OK Cancel He	łp

This will open the boundary editor with the current boundary point highlighted.

Alternatively, you can open the boundary file directly from the Simulation Editor (.bnd11).Now take a quick look at the boundary specifications in MIKE 11.

All the upstream ends (Chainage = 0) are defined as Inflow boundaries with zero inflow. The downstream mouth of the river is defined as a water level boundary.

- Close the Boundary Editor and return to the Network Editor, without saving any changes.
- Close the Network Editor, without saving any changes.
- Close the Simulation Editor.



🦲 karu	ıp.bnd11							_ 🗆 🗙
	Boundary Description	Boundary Type	Branch Name	Chainage	Chainage	Gate ID	Boundary ID	
1	Open	Inflow	Karup River	0	0			
2	Open	Water Level	Karup River	52000	0			
3	Open	Inflow	Bording creek	0	0			
4	Open	Inflow	Haderup River	0	0			
5	Open	Inflow	hauge creek	0	0			
6	Open	Inflow	feldborg creek	0	0			
Data Type TS Type File / Value TS Info								
1	Discharge: TS File	karup.dfs0 <u> Edi</u>	t q=0					_
								-

3.18 Step 18 – Add the Overland Flow Module

3.18.1 Select simulation modules

In the Simulation Specification dialogue

Check Overland Flow (OL) On

Your model now has an Overland Flow branch in the data tree.

		Simulation specification
MIKE SHE Flow Model Descript MIKE SHE Flow Model Descript MIKE SHE Image: Security Mike She I	MIKE SHE Flow Model Descript Display Model Domain and Grid Model	Numeric Engine MIKE SHE ✓ Øverland Flow (0L) Finite Difference ✓ Rivers and Lakes (0C) Unsaturated Flow (UZ) Ølayer UZ Evapotranspiration (ET) ✓ Finite Difference ✓ Include Advection-Dispersion (AD) Water Quality



3.18.2 Define infiltration fraction



Normally, the amount of infiltration is determined by the infiltration rate of the top layer of the soil. When you add the overland flow module without specifying the unsaturated zone module, then a new parameter *Infiltration Fraction* is added under Precipitation.

The *Infiltration Fraction* is used to specify the fraction of the rainfall that is infiltrated versus the fraction that is available for ponding.

The default value is 1 with a *Uniform Spatial Distribution*, which means that all of the rainfall is infiltrated and there will be no ponding.

MIKE SHE Flow Model Descript Display Model Domain and Grid Model Domain and Grid Topography Comparison	
Interpretation Precipitation Rate Precipitation Rate Net Rainfall Fraction Infiltration Fraction Infiltration Fraction Verland Use Verland Flow Sturated Zone Storing of results Extra Parameters	Infiltration Fraction Spatial Distribution: Uniform Value 1
Setup Proces Results	



3.18.3 Specify the overland flow parameters

In the Overland Flow dialogue (in the MIKE SHE GUI), specify

- Manning's M equal to 0 [m^{1/3}/s]
- Detention storage equal to 0 [mm], and
- Initial water depth equal to 0 [m].

MIKE SHE Flow Model Descript	
😟 🗹 Display	
😟 🖅 Simulation specification	
🚽 🚽 Model Domain and Grid	
🚽 🚽 Topography	
🗄 🗹 Precipitation	
🚽 🗹 Land Use	
🚽 🚽 🖌 Rivers and Lakes	Manning Number
🚊 🖌 🖌 Overland Flow	Constitut Distributions
- 🖌 Manning Number	Spatial Distribution:
- 🖌 Detention Storage	Uniform
🔤 🖌 Initial Water Depth	Value
🗄 🗹 Saturated Zone	
🗄 🗹 Storing of results	
🖳 🖌 Extra Parameters	
1	
•	
Setup Proces Results	





The Karup catchment is quite sandy, and in this simulation there are no serious storm events. Thus, in reality, overland flow is insignificant for this simulation.

However, the overland flow module is required whenever the channel flow component is included. You can run overland flow without the channel flow component but not the opposite.

So, if you do not want to include overland flow you can globally disable overland flow by specifying a Manning's $M = 0 [m^{1/3}/s]$. Setting the Manning's M to zero locally is sometimes a useful way of speeding up the simulation by turning off the overland flow calculation in areas of permanently ponded water, such as small reservoirs. However, a Manning's M of zero will prevent any overland flow from entering the cell.

In this simulation, if you keep a non-zero Manning's M, the overland flow component will require the model to run with smaller time-steps (5-60 minutes) which will slow down the computation significantly.



3.19 Step 19 – Run the Model

3.19.1 Define time steps

In the Time step control dialogue:

• Define time step control parameters as shown in the dialogue to the right.



MIKE SHE uses different time-steps for the different hydrologic processes. Normally, the Overland flow time-step is the smallest and the Groundwater time-step the largest. The MIKE 11 time step is controlled in MIKE11.

The overland flow time-step is large in this simulation because you have disabled the overland flow simulation by setting the Manning's M to zero. To simulate overland flow you would typically need a time step on the order of 5-60 minutes, and possibly less than this if you are using the explicit solver and overbank spilling to simulate flooding.

The *Max precipitation depth per time step* is used to reduce the time step during heavy rainfall events.

The *Max infiltration amount per time step* is used to reduce the time step when water is ponded on the surface, for example, during a flood.

These two parameters are only important if the unsaturated zone model is included. You should set these to high values, so that the simulation time step is not reduced.

	Time step control		
MIKE SHE Flow Model Descript	Time Steps		
🖃 🗹 Simulation specification	Initial time step	4	[hrs]
····· ☆ Simulation title ····· ☆ Simulation period	Max allowed OL time step	12	[hrs]
Time step control	Max allowed UZ time step	12	[hrs]
✓ UL Computational Contri ✓ SZ Computational Contri	Max allowed SZ time step	24	[hrs]
Model Domain and Grid ✓ Model Domain and Grid ✓ Topography ✓ Precipitation ✓ I Land Use ✓ Rivers and Lakes ✓ Overland Flow	Increment of reduced time step length Increment rate (0-1)	0.05	
	Parameters for Precipitation-dependent t	ime step control	
	Max precipitation depth per time step	50	[mm]
	Max infiltration amount per time step	50	[mm]
۲	Input precipitation rate requiring its own time step	0.1	[mm/hr]
Setup Proces Results			



3.19.2 Overland flow parameters

Make sure that you are using

- the Successive Overrelaxation (SOR) solver, and
- the Manning equation

for calculating overland flow.



If you want to calculate flooding, then the explicit solver is recommended. However, the time step constraints on the explicit solver are much more restrictive.

	OL Computational Control Parameters		
n	Solver Type and Solver-specific Parameters		
MIKE SHE Flow Model Descript	 Successive Overrelaxation (SOR) 		
⊡… ✔ Display	Maximum number of iterations:	200	
🦳 🗹 Simulation title	Maximum head change per iteration:	0.0001	[m]
Simulation period	Maximum residual error:	0.0001	[m/d]
OL Computational Control	Under-relaxation factor: [0.01 - 1.0]	0.9	
✓ SZ Computational Contri ✓ Model Domain and Grid	Explicit (Recommended when overbank spilling is a	llowed)	
🖌 Topography	Maximum courant number: [0.1 - 0.9]	0.8	(for adaptive time step)
i⊞∽ 🖌 Precipitation	Sub-divide overland flow cells for multi-cell O	. method	
✓ M Land Ose ✓ Rivers and Lakes	Coarse cell sub-division	2	
	Common stability parameters		
	Threshold water depth for overland flow:	0.0001	[m]
🔤 🗹 Extra Parameters	Threshold gradient for applying	0.0001	11
	low-gradient flow reduction:	0.0001	
	Overland-River exchange calculation		
	Manning equation (using OL flow Manning number)	5)	
	🔿 Weir formula: (Weir data specified in MIKE 11)		
	Threshold head difference for applying	0.1	
Setup Proces Results	row-gradient now reduction.		,



3.19.3 Run the Preprocessor

Click on the **PP** icon to start the pre-processor, which will start the MIKE Zero Launch utility.

MIKE Zero - [SHEPar2]	
🥚 File Edit View Refresh Run Window Hel] 🗅 😅 🔜 🖇 🗈 🖻 🖉 🤗 💦 🗍 🗲 РР ым	р ыц
Launch settings	×
Launch Purpose	
Simulation	
Notification on End of Simulation	
🗖 Via E-mail	_
🗖 Via SMS	
CPU Priority Normal mode	
Refresh rate 200	milliseconds
OK	Cancel Help

Click OK

3.19.4 View the river link setup

In the Processed Data Tab the most recent pre-processed file is automatically loaded

Click on the Load button read in the Processed Data

Go to the *River Links* Item in the data tree to view the pre-processed river links:



View the location of the river links in the computational mesh.

The MIKE 11 network is interpolated to the cell boundaries of the model grid. The interpolated river locations are called river links.





3.20 Step 20 – View the Results

You will now view some of the outputs of this exercise using the *Result Viewer*. It is always a good idea to run and evaluate your model as you are setting it up. Otherwise, there is a danger that you when you are finished setting up a big model, you will have a lot errors that are difficult to track down.

3.20.1 View groundwater level results

In the Result Viewer dialogue:

- Look at whatever outputs you may find interesting, such as the SZ Exchange with river and the SZ drainage flow from point?
- Is the model predicting any significant drain flow? Why or why not? What is happening to the drain water in this exercise?
- Can you view river results?
- Do the results make sense so far?



3.21 Step 21 – Input Distributed Rainfall

You will now make the simulation a bit more realistic by adding distributed rainfall data.

3.21.1 Define rainfall stations



The rainfall will be spatially distributed using a dfs2-grid file with integer codes that are linked to a specified rainfall time-series.

In the Precipitation Rate dialogue:

- Choose Spatial Distribution = Station based
- Choose Data Type = Grid Codes (.dfs2)





You are now using a dfs2-grid code file to distribute rainfall time-series data. Each code (area) now needs to be associated with a rainfall time-series. The grid file contains 9 different codes (1-9). The value of the code is not important and may be any number. Each area (defined by a code) just needs to be associated with a rainfall time-series.

Station areas may also be assigned using .shp file polygons.



3.21.2 Associate the station areas with time-series data

The navigation tree now allocates space for 9 rainfall stations. Initially the station name is derived from the Grid code value. You may change the station name if you want.

You now need to link a rainfall time-series to each station (code).

- Set the Temporal Distribution to Time varying (.dfs0).
- Click the browse button and select the precipitation file Precipitation.dfs0
- At the bottom of the file selection dialogue, under Items, select the appropriate precipitation station from the combo box list or type it in (e.g. Precipitation station 3).
- Press Ok.

The Item name appears on the top right of each graph.

	Station Data - Precipitation	Rate
	Precipitation Station ID: Grid code value Station 3 3 Time Series File: C:\Program Files\DHI\MIKEZero\Examples\MIKE	Temporal Distribution: Time varying (.dfs0)
	(mm/day) 50 0 Hithing (min/day) 1970 1971 1972 1973	itemname: Precipitation station 3
MIKE SHE Flow Model Descript. MIKE SHE Flow Model Descript. Model Domain and Grid Model Domain and Grid Grid code = 1 Model Code = 1 Model Code = 2 Model Code = 2 Model Code = 2 Model Code = 4 Model Code = 4 Model Code = 5 Model Code = 6 Model Code = 6 Model Code = 8 Model Code = 9 Model Code = 8 Model Code = 1 Model Code = 8 Model Code = 1 Model Code = 6 Model Code = 8 Model Code = 8 Model Code = 8 Model Code = 1 Model Code = 8 Model Code = 1 Model Code = 7 Model Code = 8 Model Code = 8 Model Code = 1 Model Code = 1	Look in: Ime Image: Angle of the second	x
Setup Proces Results	Precip	itation station 8 OK

Repeat this for each grid code (station).



3.22 Step 22 – Modify the Subsurface Drains

In the SZ1 exercise, subsurface drains were included 0.5 meter below ground surface throughout the model domain. Thus, whenever the water table goes above the drainage level, water will drain from the saturated zone. In the SZ1 exercise, this drainage was exported from the model domain. You will now change the drainage setup so that drainage water is routed to the river instead.

3.22.1 Modify the subsurface drain reference system

- Choose Drainage routed downhill based on adjacent drain levels
- Leave the Level and Time Constant the same as in the SZ1 exercise.

MIKE SHE Flow Model Descript. ✓ Display ✓ Display ✓ Simulation specification ✓ Model Domain and Grid ✓ Topography ✓ Topography ✓ Topography ✓ Topography ✓ Jescriptation ✓ ✓ And Use ✓ ✓ Rivers and Lakes ✓ ✓ Overland Flow ✓ ✓ Saturated Zone ✓ ✓ Geological Layers ✓ ✓ ✓ Geological Lenses	
 	Drainage Drainage Option Drainage routed downhill based on adjacent drain levels Drainage routing based on grid codes
Setup Proces Results	 Distributed drainage options Drainage not routed, but removed from model



Drainage routing can be a very complex process. Simply removing the water from the model is the simplest drainage option. However, in most situations, drainage actually discharges to the river system.

In fact this is a very powerful feature of MIKE SHE.

The simplest routing method for the drainage is to route it downhill based on the drain levels. The drainage will enter the river whenever it intersects a river link. However, this can lead to ponding in local depressions in flat terrains.

More complicated is the second option *Drainage routing based on grid codes*, where the drainage is routed downhill but only from certain cells.

The *Distributed drainage options* allow you to route drainage to specific MIKE 11 nodes or MIKE Urban sewer manholes.





3.23 Step 23 – Add Land Use

3.23.1 Add the evapotranspiration and unsaturated flow modules

- Check Unsaturated Flow (UZ) on (Richards Equation)
- Check Evapotranspiration (ET) on

The Land Use, Evapotranspiration, and Unsaturated Flow sections are added to the data tree.

	Simulation specification
MIKE SHE Flow Model Description Mike SHE Flow Model Description Mike She Display Model Domain and Grid Mike Topography Mike She She She She She She She She She Sh	Numeric Engine MIKE SHE ✓ Overland Flow (DL) Finite Difference ✓ Privers and Lakes (OC) ✓ ✓ Pichards equation ✓ ✓ Finite Difference ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ Finite Difference ✓ ✓ Finite Difference ✓ Include Advection-Dispersion (AD) Water Quality
Setup Data Processed Results	



3.23.2 Define vegetation map (field map)

In the Vegetation dialogue:

- Use Station based Spatial distribution
- Specify Grid Codes (dfs2) as the Data Type
- Use the browse button to select the file
 .\Model Inputs\MAPS\Landuse.dfs2

The map now displays your vegetation distribution. The map (dfs2 file) contains 4 different integer codes that define 4 different areas. You now need to define the vegetation types and growth function which are associated with the 4 different areas.





3.23.3 Define vegetation types for each vegetation area

Each of the 4 different vegetation areas in the vegetation map appears as a sub-item in the data tree.

- Select the *Grid code* = 1 item
- Change the ID to Grass Crops
- Set the Temporal Distribution to Vegetation property file
- Use the new items icon, , to add 13 lines in the Crop Development table.
- For each line in the Crop Development table
 - Use the browse button to select the vegetation property file (database file):
 .\Model Inputs\MIKE_SHE_vege.ETV
 - From the pull-down Veg type combo box pick Grass1
 - Set the Start date to 1. Jan. 1969
- Repeat the above steps for each line in the Crop Development table, with a start date of 1970, 1971, ..., 1981.
- For the remaining three vegetation types, set the Temporal Distribution to Constant
- Use the following values for the remaining items:
 - Grid code = 2: Forest, LAI (Leaf Area Index) = 6, RD (Root Depth) = 800 [mm]
 - Grid code = 3, *Shrubs*, *LAI* = 2, *RD* = 440 [*mm*]
 - Grid code = 4, Wetlands, *LAI* = 4, *RD* = 500 [*mm*]





3.24 Step 24 – Add Reference Evapotranspiration

MIKE SHE calculates actual evapotranspiration based on the evapotranspiration rate, the vegetation properties, and the soil moisture.

3.24.1 Define evapotranspiration data

In the Reference Evapotranspiration dialogue:

- Use Uniform spatial distribution
- Use Time-varying (.dfs0) Data Type
- Use the browse button to select the file
 .\Model Inputs\Time\PotentialEvap.dfs0





Reference evapotranspiration can be specified in the same manner as precipitation, this is as constant or time-varying values and as uniform or spatially distributed values. As you're using *Uniform* distribution type, this evapotranspiration rate will be used throughout the model domain.



3.25 Step 25 – Build the Unsaturated Zone Model

The unsaturated zone model calculates recharge to the groundwater model and is closely linked to the actual evapotranspiration, as the roots remove water from the unsaturated zone. Soil physical properties are stored in a soil-database. The link between soil-type distribution maps and soils in the soil-database are established similarly to the link between vegetation-distribution maps and vegetation types.

3.25.1 Calculation column classification

In the Unsaturated Flow dialogue:

Choose Calculated in all grid points for the column classification type

	Unsaturated Flow	
MIKE SHE Flow Model Description ⊕ □ Display	Calculation Column Classification Type 1: Automatic 2: Specified calculation points 3: Calculated in all grid points 4: Partial automatic (combination of 1 and 2)	Initial Conditions
Display Display Display Simulation specification Model Domain and Grid Display Displa	Macropore Flow Image: Mone Simple by-pass flow Full macropore flow (Richards and Gravity only)	
Insaturated Flow Saturated Zone Storing of results Extra Parameters	Time Step Control 1 Max MP Infiltration per time step 1 Max MP-Matrix Node exchange per time step 0 Max MP-Matrix Column exchange per time step 2	mm 5 mm mm
Setup Data Processed Data Results	Use Green and Ampt model for infiltration	



MIKE SHE allows you to lump calculation profiles together rather than calculating unsaturated zone flows in each single profile. This can be done to speed up the computational time, which was very useful in the past when computational resources were limited.

However, the lumped (Automatic) classification is usually not suitable. For example, it is not allowed if overland flow or flooding is an important feature or if irrigation or water quality is to be simulated.

In most models it is probably best to avoid UZ classification.

Demo Note: the demo version is restricted to a maximum of 155 UZ computational cells. So, if you are running this exercise without a valid license, then you should select the Automatic classification type.



3.25.2 Define the soil profile distribution map

In the Soil Profile Definitions dialogue:

- Use *Distributed* for the Spatial Distribution
- Use Grid codes (dfs2) distribution data type
- Use the browse button to select the dfs2 grid file .\Model Inputs\MAPS\soil.dfs2





The soil map contains three different code values, which now need to be associated with three soil profiles.

3.25.3 Define soil profiles (horizons)

Each of the 3 different soil areas in the soils map appears as a sub-item in the data tree.

- Select the Grid code = 2 item
- Change the ID to Fine Sand
- Define one soil horizon for the profile by specifying To depth = 37
- Use the browse button to select the UZ soil-database file
 .\Model Inputs\Karup.uzs



- Define the Vertical Discretization as shown in the dialogue to the right.
- Repeat the above steps for the other two soil types using
 - For Grid Code = 3 use Coarse Sand
 - For Grid Code = 4 use Coarse Sand in the for the upper 2 meters and use Fine Sand in the remaining 35 meters (To Depth 37)

MIKE SHE Flow Model Descript								
😥 🗹 Display								
🗄 🗹 Simulation specification								
- Model Domain and Grid	UZ So	oil Profile I	Definition					
🗹 Topography	020.		Bonnaon					
🗄 🗹 Climate	Profile ID: Fine Sand Grid code value: 2							
🗄 🗹 🖌 Land Use								
Rivers and Lakes	Soil Profile	e.						
🗄 🗹 🗹 Overland Flow		From denth	To depth	Soil nam	ie I	IIZ Soil property file		
🖃 🗹 Unsaturated Flow		0	37	Fine Sand	Cillsei	rs))Model Inputs)Karun II7 Edit		
🚊 🗹 Soil Profile Definitions	ons							
Grid code = 2								
Grid code = 3								
Grid code = 4								
Initial Conditions	1'							
🗄 🗹 🖌 Saturated Zone	Vertical D	iscretization:						
🗄 🗹 Storing of results		From depth	To depth	Cell height	No of cells			
🗹 Extra Parameters	1	0	0.5	0.1	5	1		
	2	0.5	1.5	0.25	4			
✓	3	1.5	4	0.5	5]		
Setup Process Results	4	4	30	1	26			

3.25.4 DEMO MODE only: Define groundwater table for UZ classification

Demo Note: If you are using the Demo mode, then you must use the Automatic classification method, which requires you to specify a water table

The depth to the groundwater table is one of the criteria that are used when the automatic (lumped) unsaturated zone classification is used.

In the Groundwater depths dialogue:

- Use initial water table for classification
- Define 6 depth classes 1, 2, 3, 4, 5 and 10 meters.

MIKE SHE Row Model Descript				
Model Domain and Grid				X ★ ¥
Topography	D	lepth		▲
E	1 1			
Rivers and Lakes	2 2			
	3 3			
Onsaturated Flow Soil Profile Definitions	4 4			
Groundwater depths used	5 5			-
Initial Conditions				
	Sector Table	(Cl:C	- ť	
Extra Parameters		for Classifica	ation	
	💿 Use initi	ial water tabl	le for classification	
۰ <u>۱۱۱</u> ۲	C Specifie	ed water tabl	e for classification	
Setup Process Results				



3.26 Step 26 – Run the Simulation

3.26.1 Add field data to the observations

In the SZ1 exercise, you added 4 observation points, but did not include the observation data.



To add the observation data, you need to

- Choose Data type head elevation in saturated zone
- Check on the Incl. Obs. Data checkbox
- For each observation point,
 - Use the browse button to select the file
 .\Model Inputs\GIS Data\HeadObservations.dfs0
- In the DFS File Selector dialogue, you need to select the appropriate Item from the combo box. (e.g. Obs well 5 for the entry named Obs 5)

Imp	ort									
	Name	Data type	New plot	x	Ŷ	Depth		inci. Obs. Data	Obs. Data Filename	
1	Obs 5	head elevation in s	.	24103	8419	5	•		WeadObservations.dfs0	Edi
2	Obs 35	head elevation in s	•	13800	17607	5	۲		C:\Program Files\\GIS	. Edi
3	Obs 37	head elevation in s	•	17508	20887	5	۲		C:\Program Files\\GIS	. Edi
4	Obs 65	head elevation in s	V	8866	25549	5			C:\Program Files\\GIS	Edi

Lookin	CIS Data		-] 🙆 🏚 📂 🕅 👻		
LOOK IN						
en 1	Name 🔺	-	Date	▼ Туре	▼ Size	_
	flow.dfs0		27/09/2012 11:03 PM	TimeSeries.Docu	418	8
	HeadObserva	ations.dfs0	27/09/2012 11:03 PM	TimeSeries.Docu	178	в
Desktop Libraries	Precipitation2	2.dfs0	27/09/2012 11:03 PM	TimeSeries.Docu	59	7
Computer						
Network	•					•
Network	▲	HeadObse	ervations.dfs0			·
Network	✓ File name: Files of type:	HeadObse Time Serie	ervations.dfs0 is files (*.dfs0)		<u>p</u>	<u>.</u>
Network	✓ File <u>n</u> ame: Files of type: Select Item Pe	HeadObse Time Serie riod Info. Item I	ervations.dfs0 is files (*.dfs0) Info. Constraints Info.	¥ ¥	<u>1</u>	1
Network	File name: Files of type: Select Item Per Title:	HeadObse Time Serie riod Info. Item I D:\Proje	ervations.dfs0 is files (*.dfs0) Info. Constraints Info. acts\TalentProject(Autoca	■ ■ alibration\Mshe\Karup_	<u>-</u>	
Network	File name: Files of type: Select Item Pe Title: File Type:	HeadObse Time Serie riod Info. Item I D:\Proje Non Eq	ervations.dfs0 is files (*.dfs0) Info. Constraints Info. acts\TalentProject(Autoca uidistant Time Axis	alibration\Mshe\Karup_	2	J
Network	▲ File name: Files of type: Select Item Pe Title: File Type: Any Item Type	HeadObse Time Serie riod Info. Item D.\Proje Non Eq Dos Well 6 Obs Well 6 Obs Well 6 Obs Well 6 Obs Well 7	arvations.dfs0 Info. [Constraints Info.] ests\TalentProject(Autoc uidistant Time Axis Item	alibration\Mshe\Karup_	ţ	Canc



3.26.2 Run the simulation

• Click on the **PP** icon to re-run the pre-processor and then on the **icon** to run the simulation.

🗟 МІК	E Zero	- [SH	IEPar2]			
File	Edit	View	Refresh	Run	Window	Help
] 🗅 🖨		; 🖻	C # ?	N?	Ø PP	ым ма

3.27 Step 27 – Evaluate the Results

You will now view some of the outputs of this exercise.

3.27.1 Locate the detailed time series output results

After successfully running the model:

• Click on the *Results* Tab located at the bottom of the Navigation Tree.





Try to notice any differences compared to just the groundwater model.

All of the observations should also now be visible.

You can also add additional detailed time series items now and re-run the model.



3.27.2 Gridded data results

In the output tab, under Gridded Data Results, the number of output items has expanded considerably compared to when only the groundwater component was simulated. Many of these items are very useful for evaluating the integrated groundwater surface water interaction.

You should explore some of these output items. For example:

- Actual evapotranspiration this is the sum of all the ET components. It is useful for evaluating the spatial distribution of actual ET.
- **Depth of overland water** –typically this is a useful output for finding areas with permanently ponded water, or areas with closed depressions.
- Seepage flow SZ overland this can be used for determining where springs are occurring in the model.
- SZ exchange flow with river this is useful to find losing and gaining river reaches.
- SZ drainage flow from point this output maps the locations where groundwater is contributing water to the SZ drainage network.

There are also additional outputs that can be turned on in the Setup tab. Of particular interest might be:

 Total Recharge to SZ – this is a summary output map of all the items that contribute inflow to the SZ.

MIKE SHE Flow Model Description														
Simulation Results	Gr	idded Data Results Viewer												
Mike SHE Detailed Time Series														
Gridded Data Results Viewer	Layer	Layer no. for Groundwater items												
Mike 11 Detailed Time Series	1													
Post-Processing														
		Add												
		Item	XY-flow		Filename									
			vectors											
	1	precipitation rate		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	2	rooting depth		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	3	leaf area index		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	4	actual evapotranspiration		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	5	actual transpiration		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	6	actual evaporation from interception		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	7	actual evaporation from ponded water		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	8	canopy interception storage		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	9	evapotranspiration from SZ		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	10	depth of overland water		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	11	overland flow in x-direction		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	12	overland flow in y-direction		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	13	infiltration to UZ (negative)		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	14	exchange between UZ and SZ (pos.up)		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	15	UZ deficit		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	16	depth to phreatic surface (negative)		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	17	head elevation in saturated zone		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	18	seepage flow SZ -overland		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	19	seepage flow overland - SZ (negative)		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	20	groundwater flow in x-direction		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	21	groundwater flow in y-direction		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	22	groundwater flow in z-direction		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	23	groundwater extraction		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	24	SZ exchange flow with river		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
	25	SZ drainage flow from point		View result	C:\Users\dng\Documents\MIKE Zero Projects\MIKE_SHE\Result\Karup									
L														
Setup Data Processed Data Results														

Explore some of these outputs.



3.27.3 Locate the detailed time series for MIKE 11

You have not yet specified detailed output for MIKE 11. So, you cannot look at this yet. But, after successfully running the model, you can go back and add detailed time series locations for MIKE 11 outputs.

To add these you need to know the chainages of the h and q points. These can be found in the .nwk11 file.

Use the q-point chainages for the Discharge and h-point chainages for the Water Level.

For example, you can add a detailed discharge time series to the end of the Karup River:

- Name: 20.05 (outlet)
- Data type: Discharge
- Branch name: Karup River
- Chainage: 52000
- Incl Obs: check
- File name: .\Model Inputs\GIS Data\flow.dfs0, and select the 20.05

Detailed M11 timeseries output											
Minin	num output time step:	0.5 ([hrs]								
	Name	Data type	Branch name	Chainag e	Incl. Obs. Data	Obs. Data Filename					
1	St. 20.05 (outlet)	Discharge	Karup river	52000	V	C:\Users\\GIS Data\flow.dfs0	[] Edit	New			

3.27.4 Water balance

Following the steps in Section 3.16 - Step 16 Create a Water Balance on page 60, re-run the water balances to evaluate the changes in the water balance with the new components.

End of Exercise.