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

1 Introduction 1
1.1 Hydrologic processes simulated by MIKE SHE 1
1.2 Applications around the world 1
1.3 The MIKE SHE user interface 2

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 your 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
3.15.4 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 59
3.16 Step 16 Create a Water Balance 60
3.16.1 Create a new water balance document 60
3.16.2 Extract the water balance data 61
3.16.3 Create water balance items 62
3.16.4 Define a Chart water balance 63
3.16.5 View the water balance output 64
3.16.6 Define a time varying water balance 65
3.16.7 View the water balance output 66
3.17 Step 17 – Add the River Module 67
3.17.1 Select simulation modules 67
3.17.2 Add the River Module 67
3.17.3 Explore the MIKE 11 simulation document 68
3.17.4 Change the results directory 69

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:

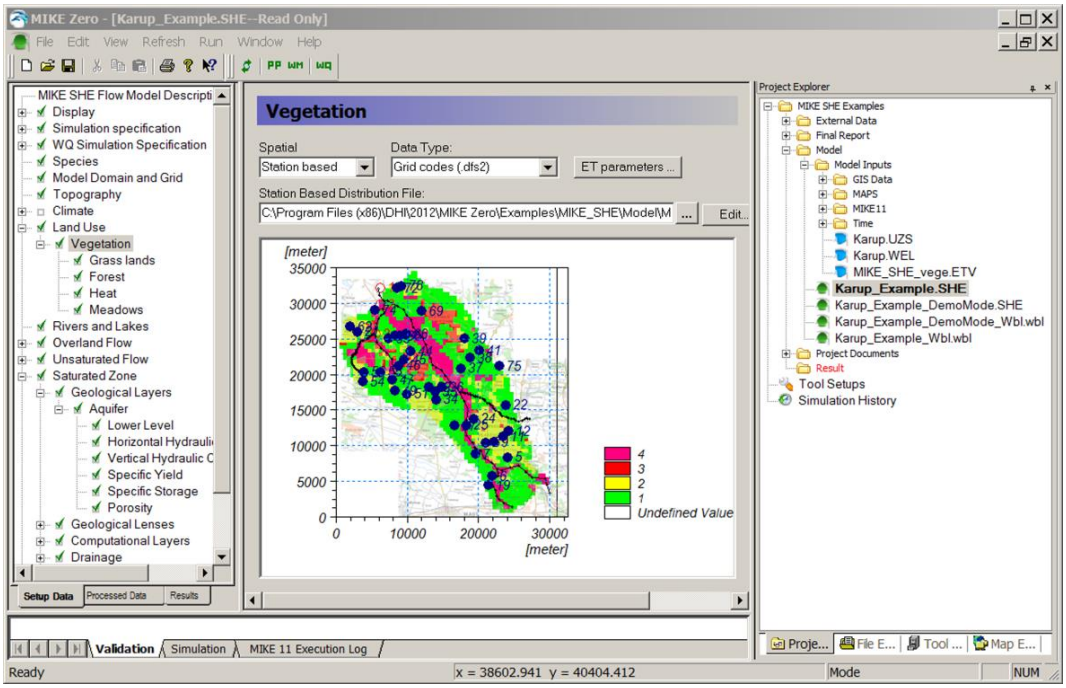
- River basin management and planning
- Water supply design, management and optimisation
- Irrigation and drainage
- Soil and water management
- Surface water impact from groundwater withdrawal
- Conjunctive use of groundwater and surface water
- Wetland management and restoration
- Ecological evaluations
- 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

Important learning objectives

After completing this exercise you should be able to

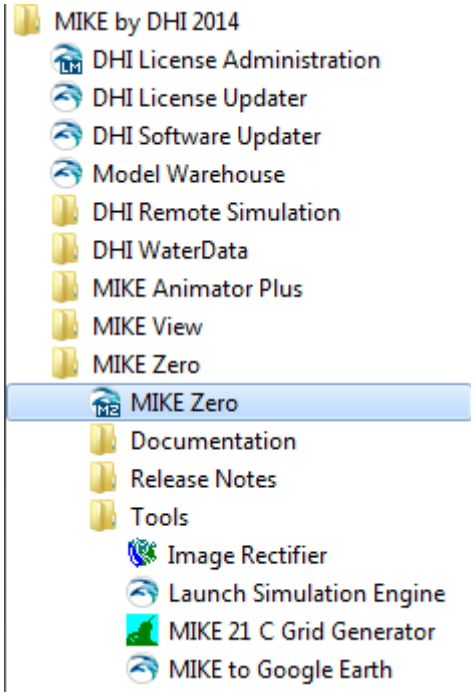
- Start MIKE SHE
- Install the example file

2.1 Step 1 – Start MIKE SHE

2.1.1 Launch MIKE SHE

- from the Program menu select MIKE Zero:

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.



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.

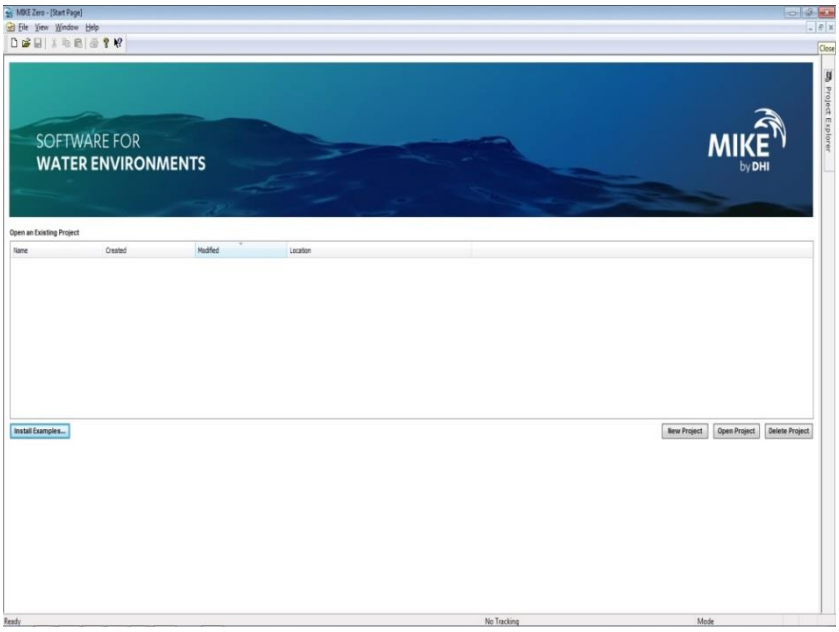
2.2.1 The MIKE Zero project window

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.



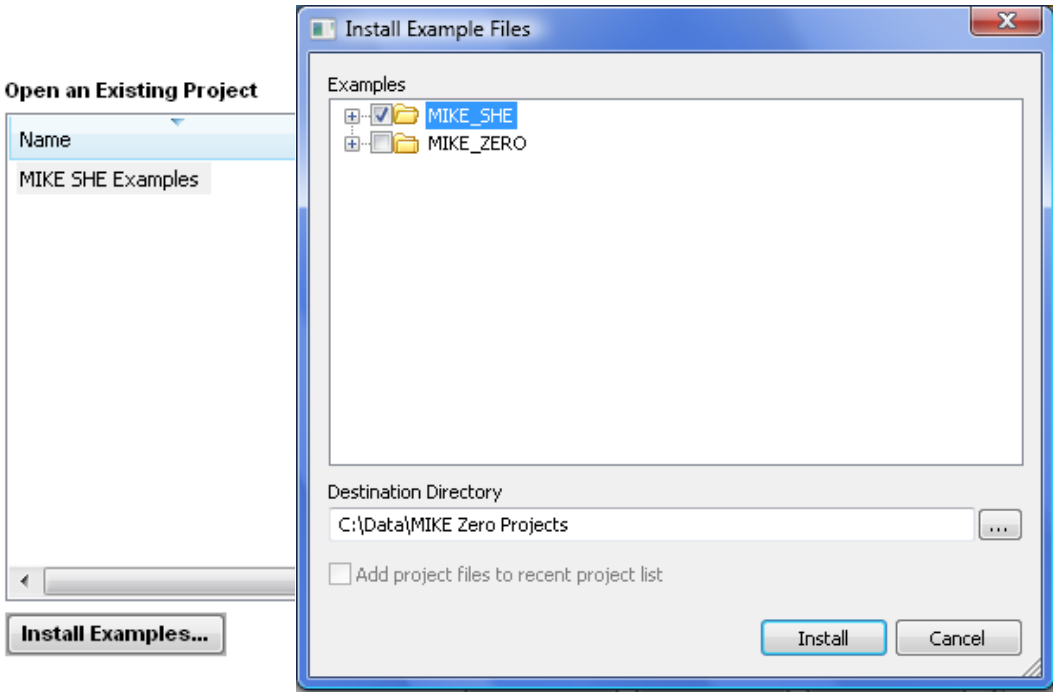
2.2.2 Install the MIKE Zero examples

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.



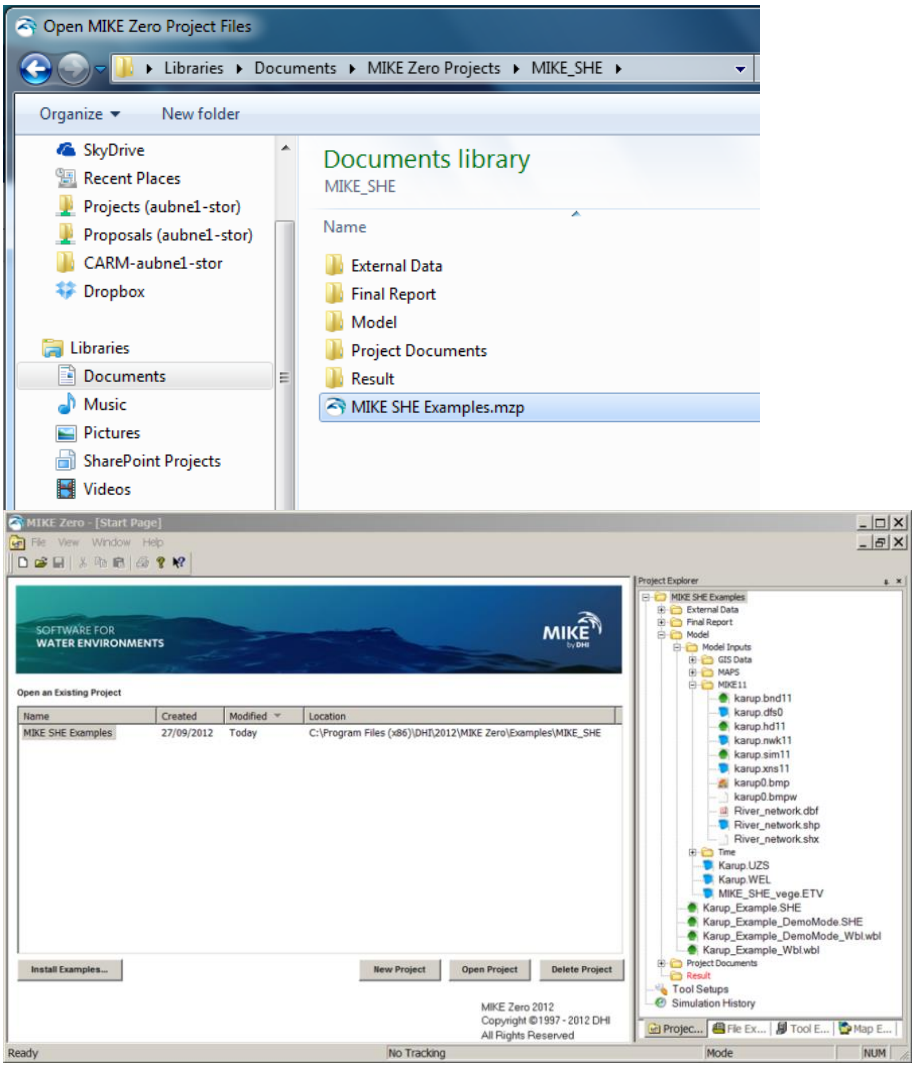
2.2.3 Open the default examples project

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.

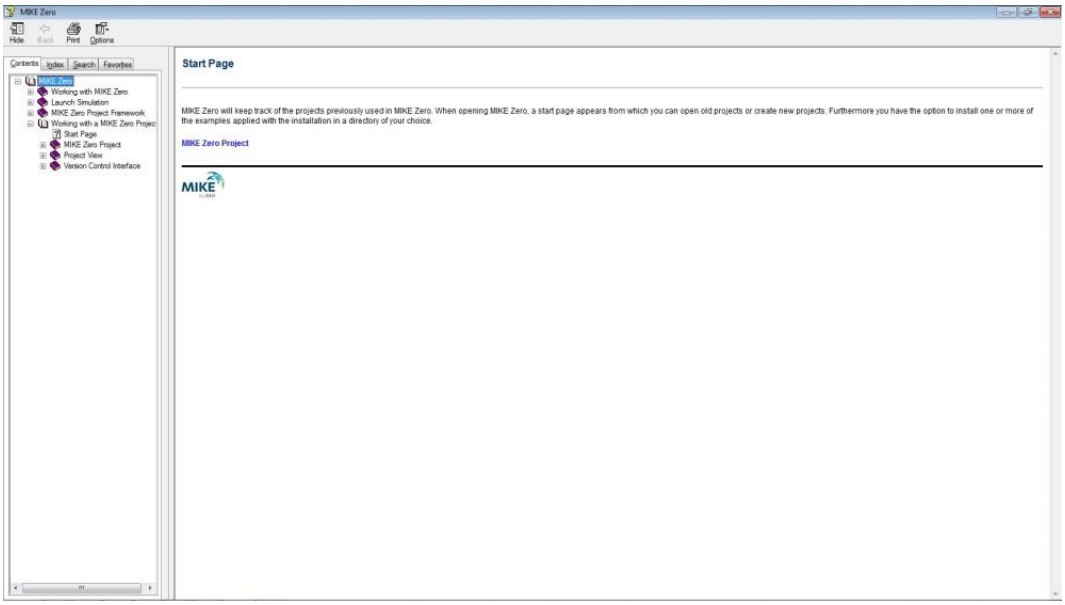
Note: The project opens with all the directories expanded out. So, you might like to close up some of the directories in the Project Explorer by clicking on the little icons.



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.



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

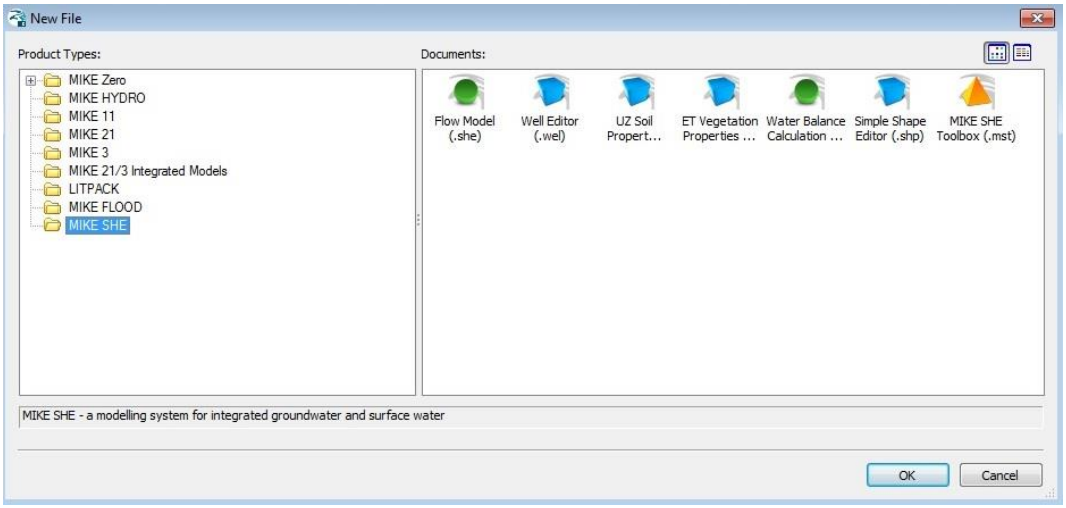
A MIKE Zero project is a collection of documents. All of the MIKE Powered by DHI documents can be created from the **File\New\File...** menu in the top pull-down menu or by clicking on the New File icon, , in the toolbar.

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.

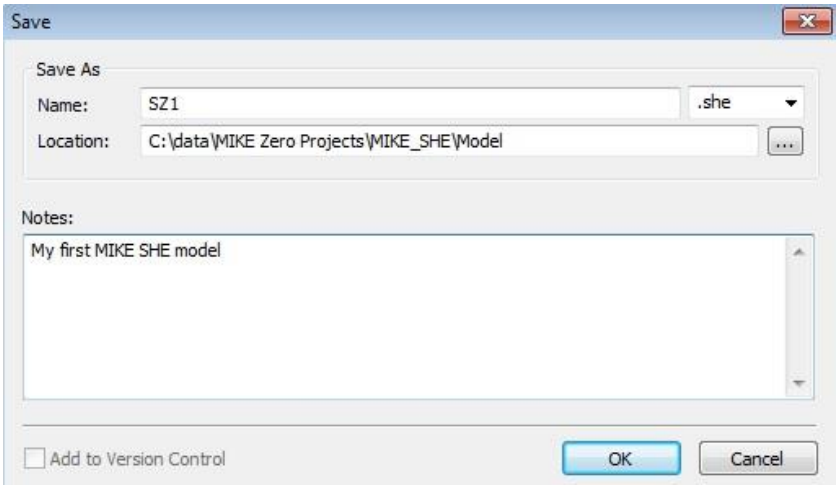


3.1.2 Save the document file

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



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*:

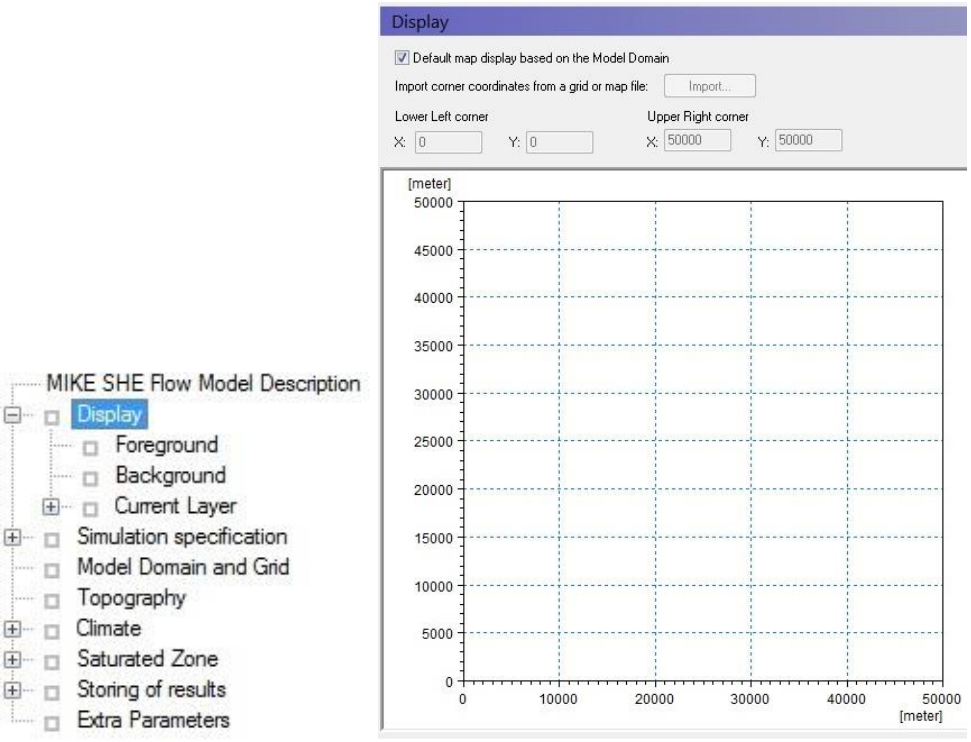
- Use your mouse pointer to select **Display**
- Leave the checkbox checked On

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

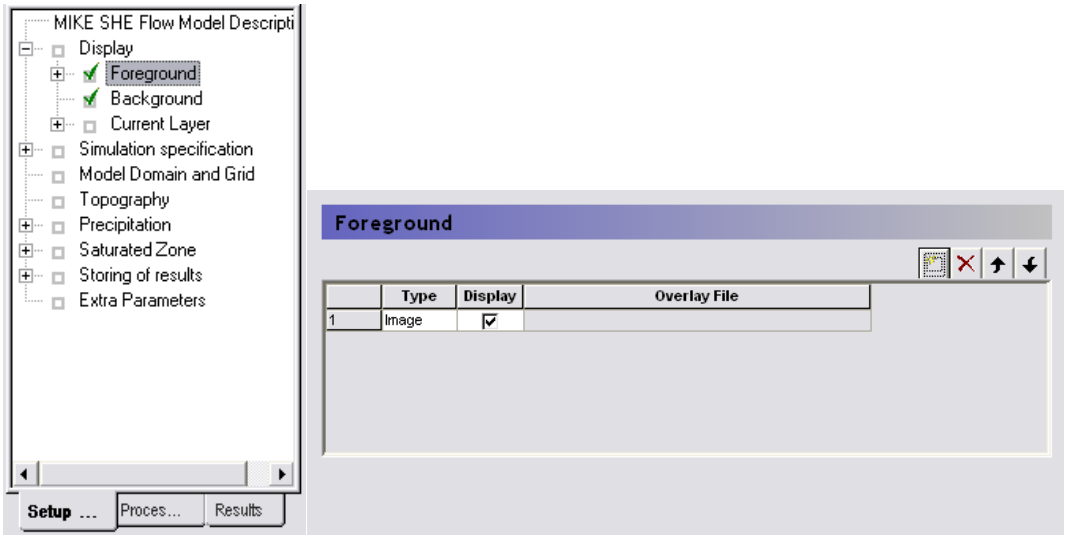
In the *Foreground* dialogue under *Display*:

- 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.

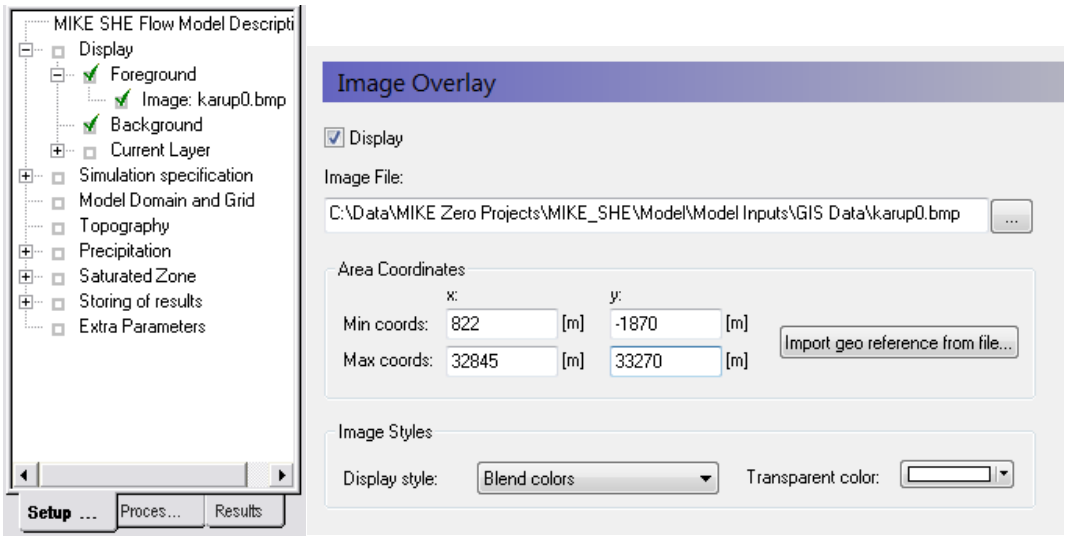
Also, if you have multiple overlays, the order that they are listed defines the order in which they are displayed. Thus, you don't usually want a bitmap at the top of the list, since it would hide all of the lower overlays.



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.



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.

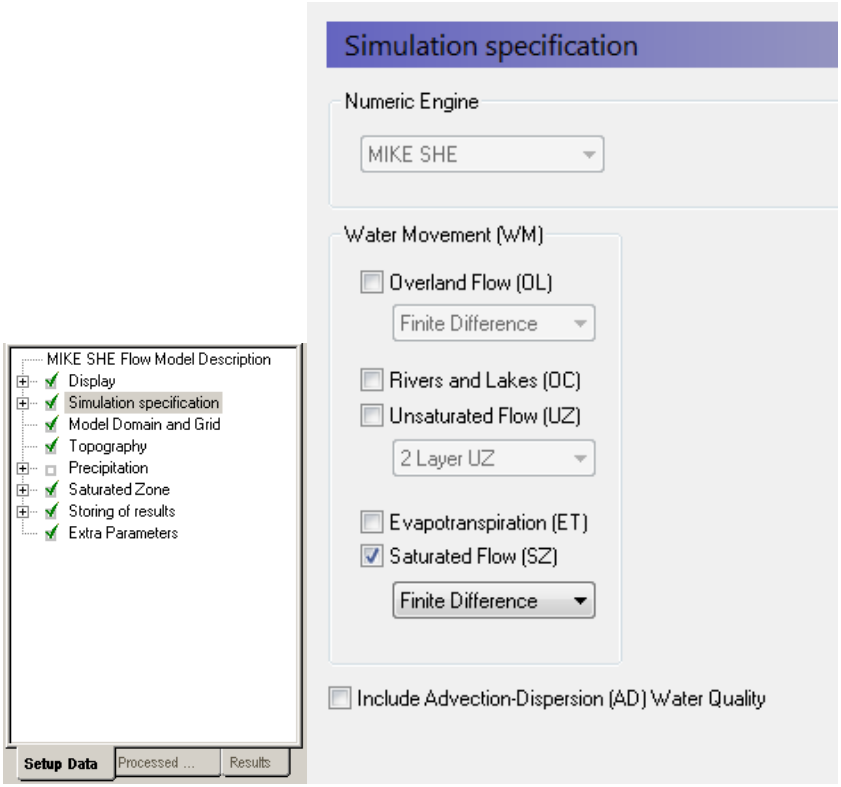
3.3 Step 3 – Set up the simulation

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.



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*.

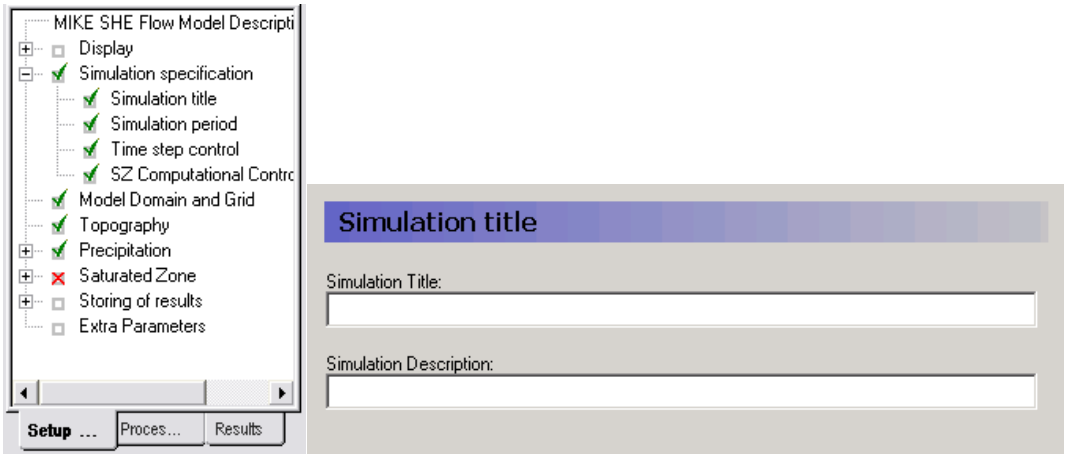
This dialogue is where you also choose the numeric engine for the different hydrologic processes.

- 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



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

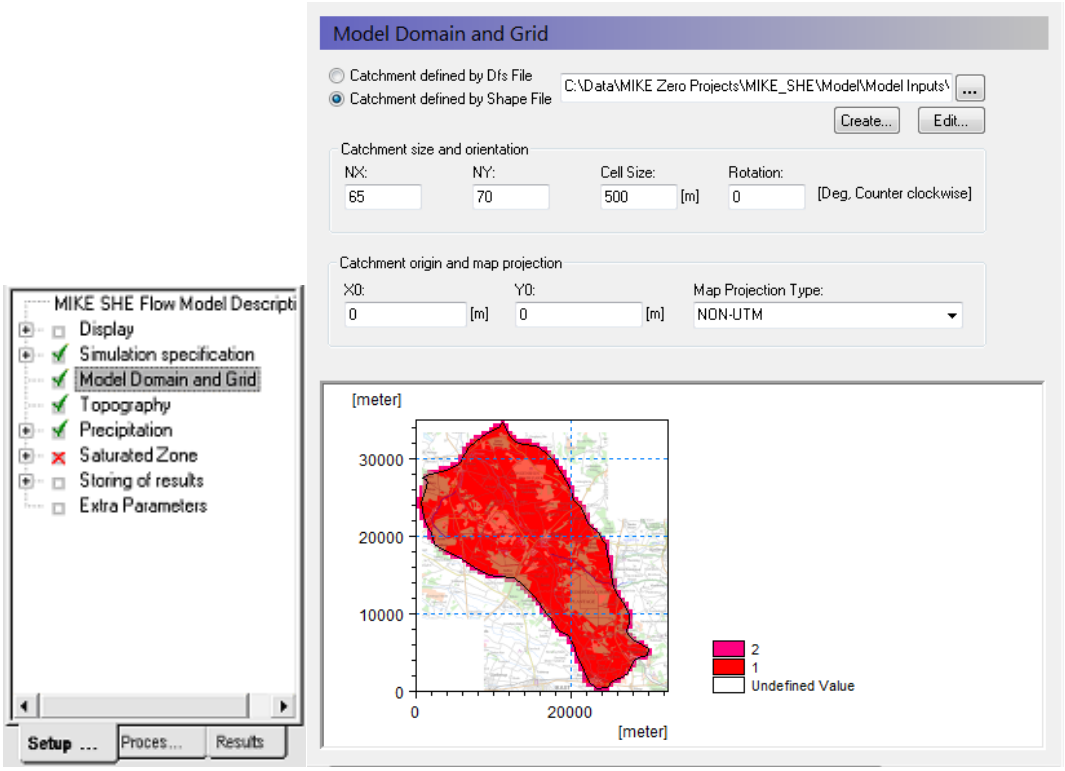
3.4.1 Define the model domain

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 pre-processor assigns all of the model parameters based on the dfs2-grid that is calculated from the polygon that you have specified in this step.



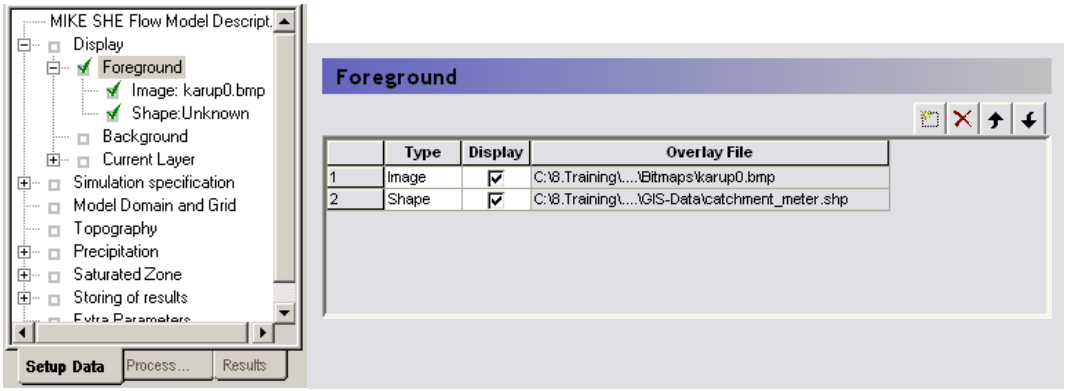
If you wish to change the model domain, all you have to do is modify the .shp file and preprocess the data again. However, be aware that previously specified data must still cover the new polygon. If it does not, then either a warning will be issued (saying that some values were automatically interpolated)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

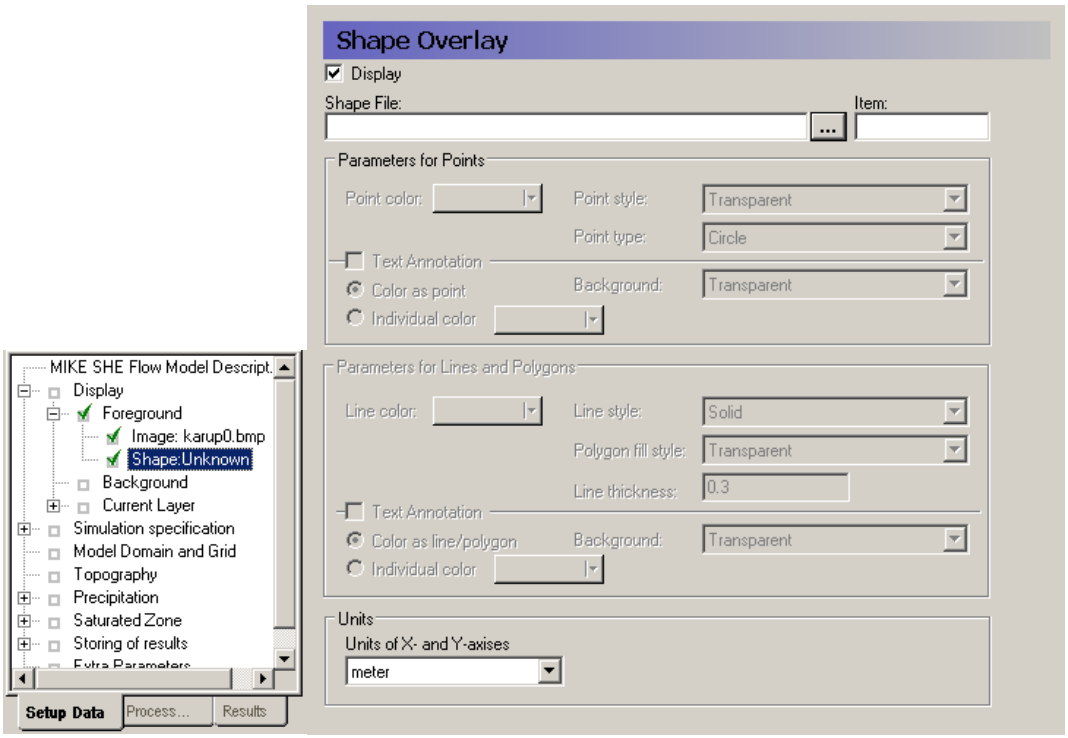
3.4.2 Add the .shp polygon to the overlays

In the *Foreground* dialogue:

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



- 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

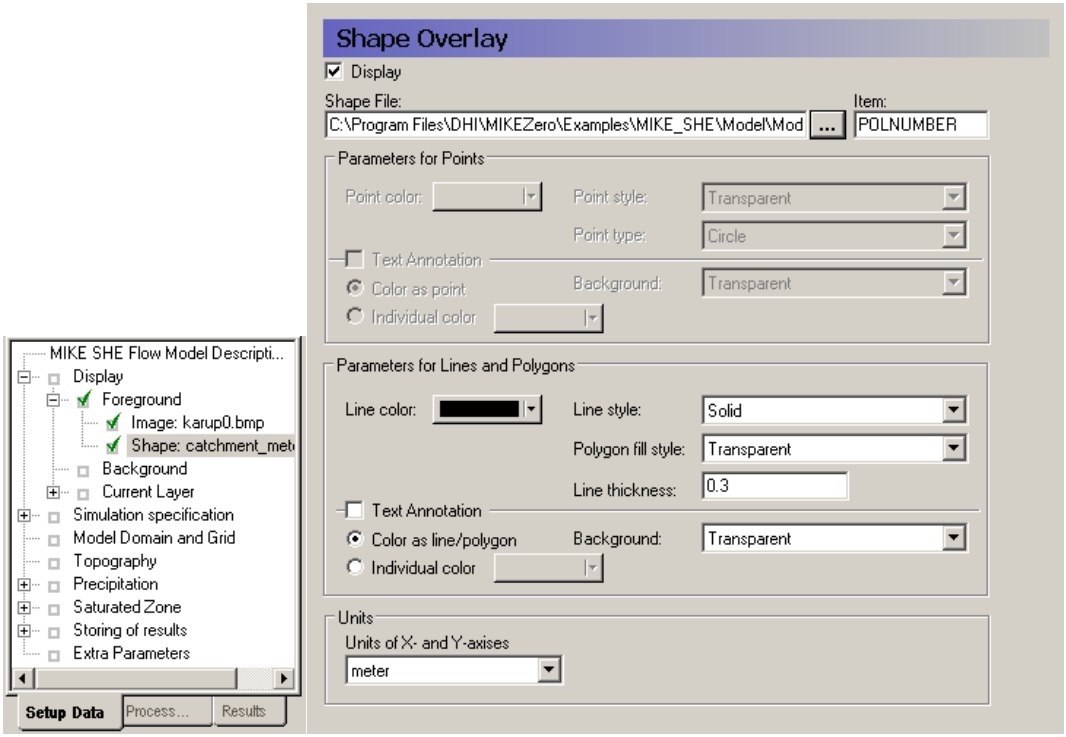


3.4.3 Adjust the display parameters for the .shp file

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.



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:

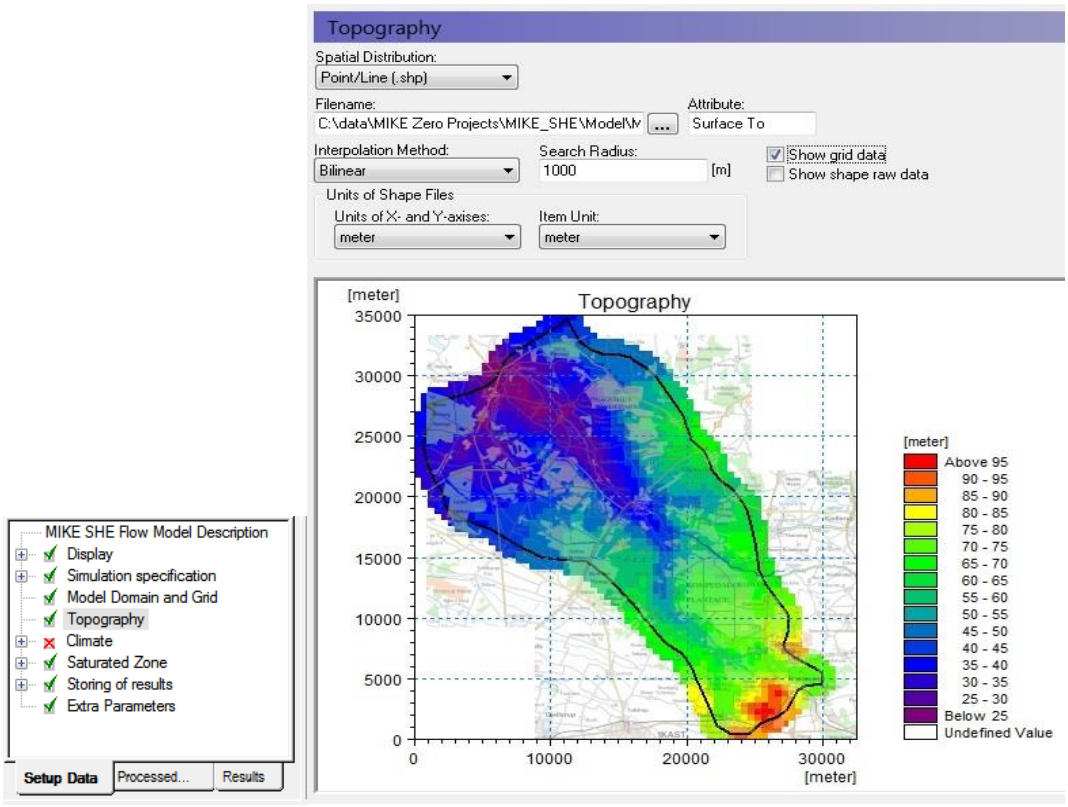
- Choose Point/Line (.shp) for the Spatial Distribution
- Select the file
  .\Model Inputs\GIS Data\Topo-processed.shp
- If the map is covered by black dots, click off the 'Show shape raw data' checkbox. This will turn off the shape file data points.

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.

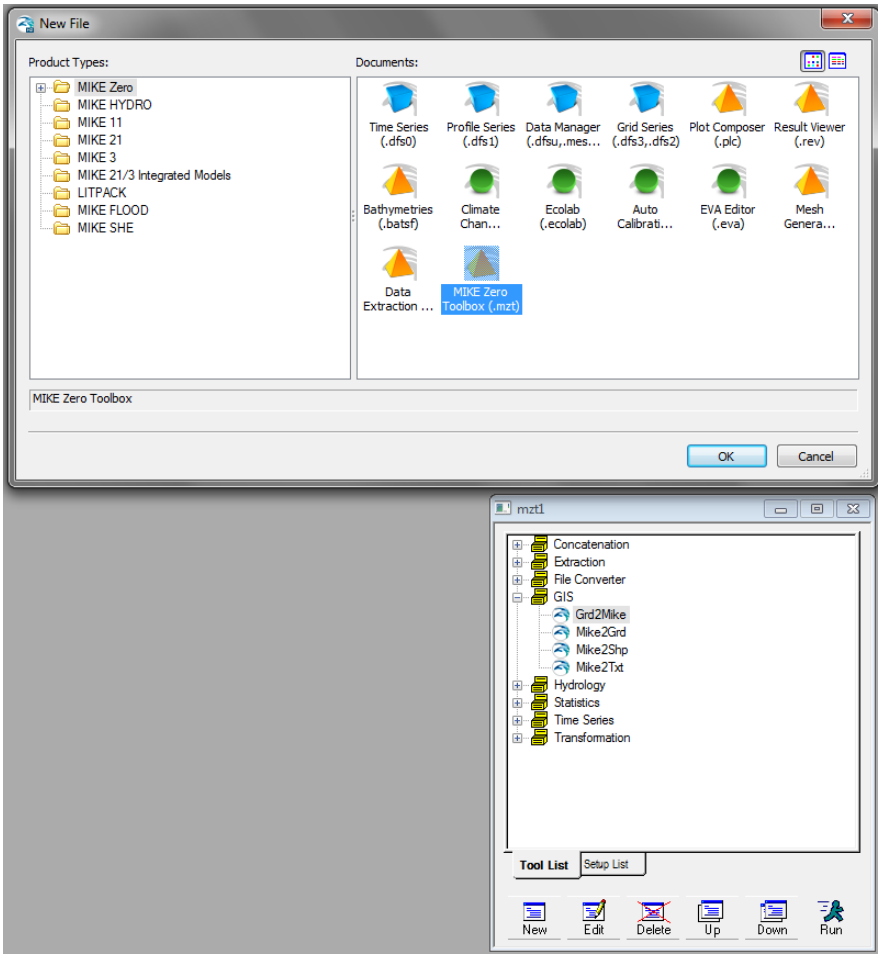


Hint: you can always see the Z-value at the cursor position at the bottom of your Graphical View.

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

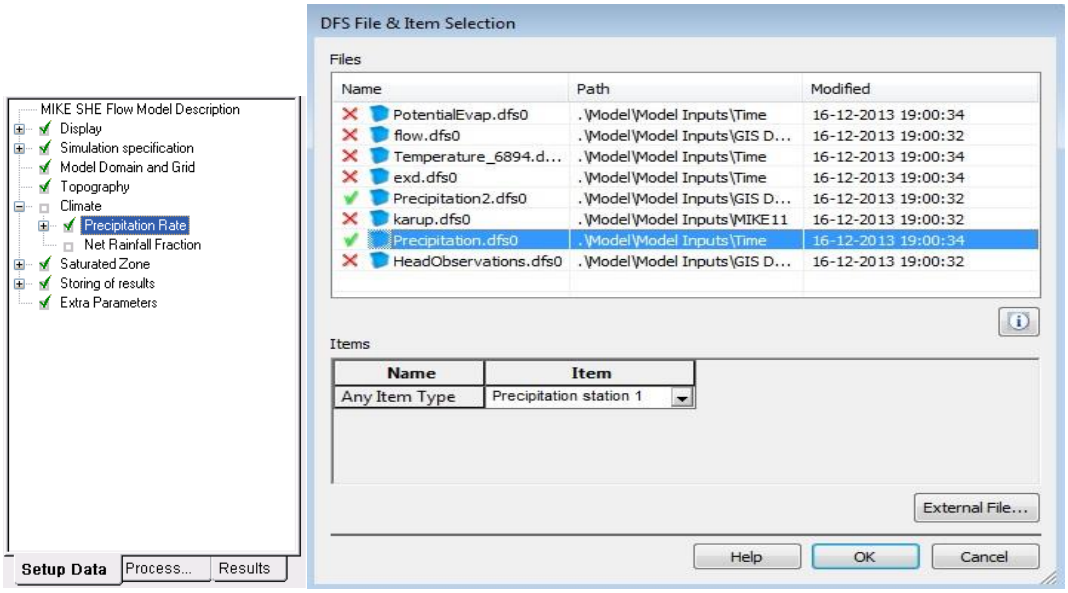
For an integrated hydrological model, you normally define the distribution of precipitation and reference evapotranspiration in both time and space. However, in this exercise, you are building only a groundwater model and thus the “precipitation” is actually “groundwater recharge”.

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

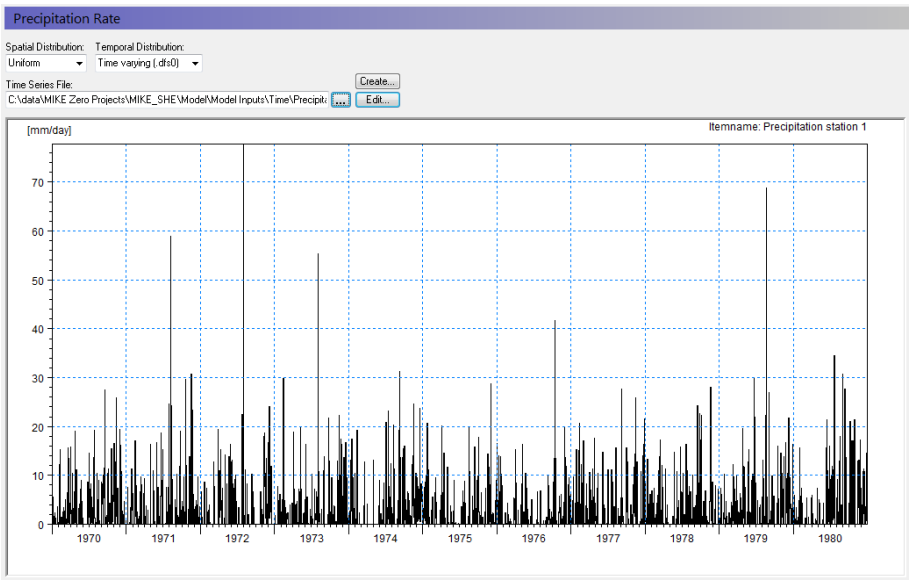


This file browser lists of all of the files in your project that are the correct file type. The green check beside the file name indicates that the file contents match the data requirements – for example, the correct time period and units type. If you are in doubt, you can click on the button will show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

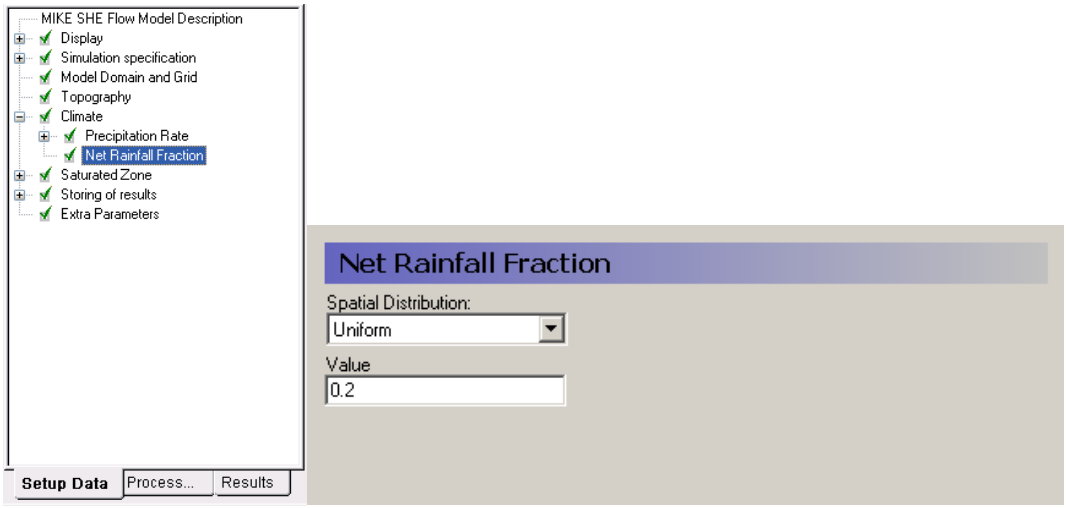


3.6.2 Define net rainfall fraction

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.



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.

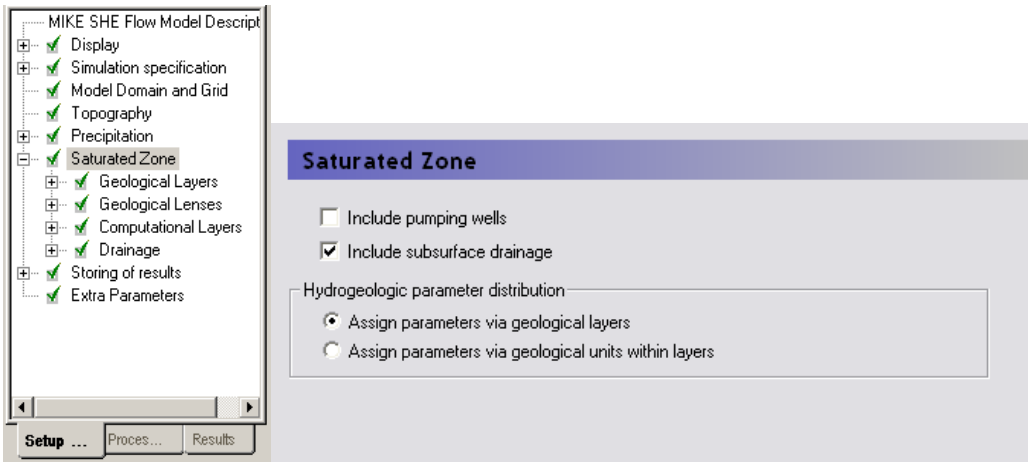
A geological model may be defined as a combination of layers and lenses. Once the geological model is defined you may choose computational layers that are either identical to the geological layers or you may choose different computational layers. If the computational layers differ from the geological 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.

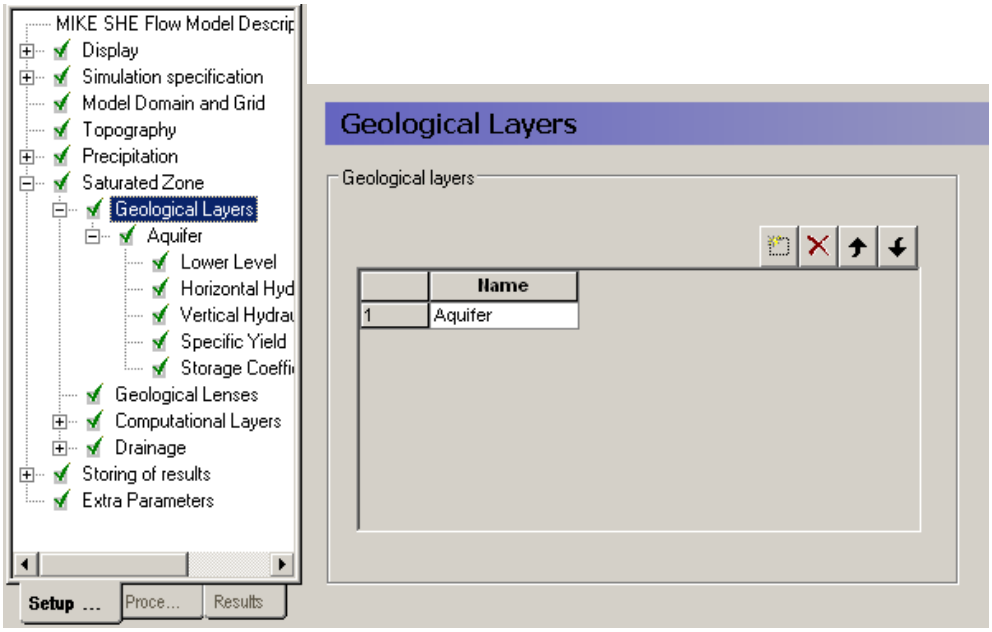
If you chose the geological units approach, then you must specify a distribution of geological units for each geological layer, for example, by a polygon file. In this case, all of the model cells in each polygon are assigned a value for each of the hydraulic properties. The geologic units method is 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, .



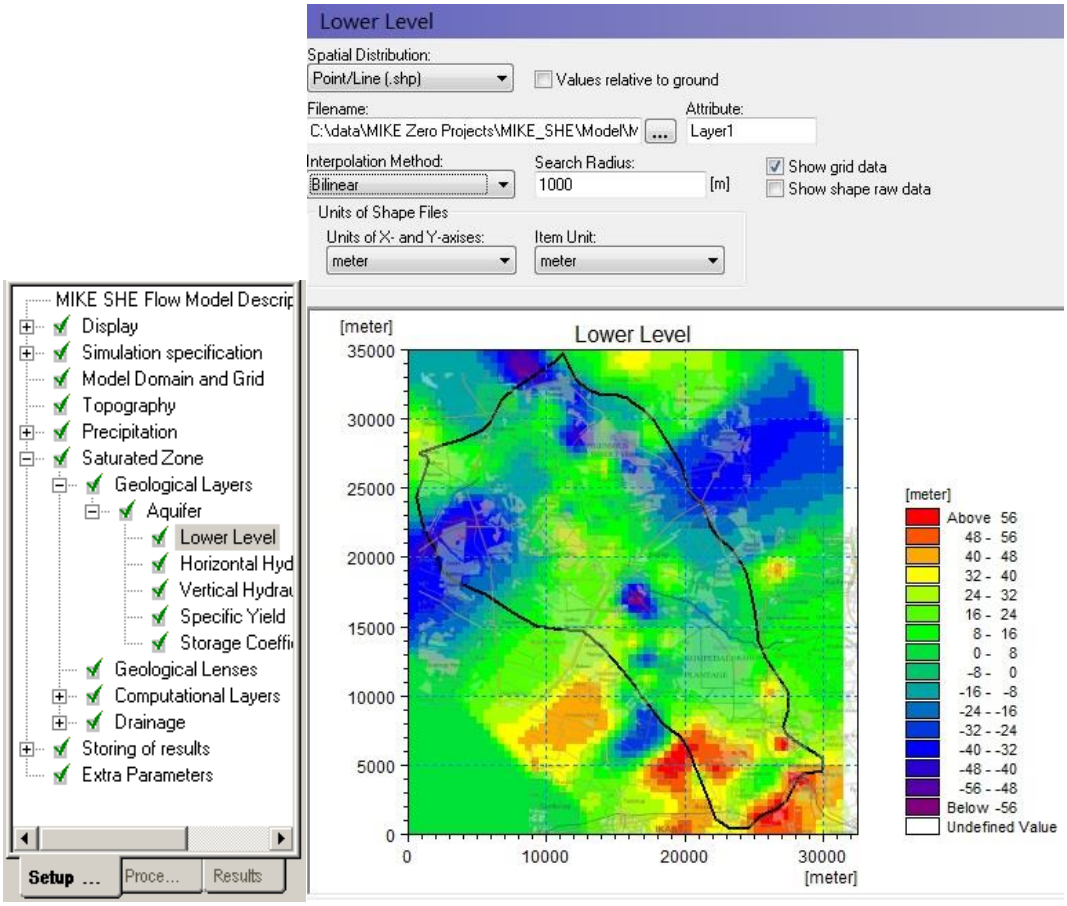
Increasing the number of saturated zone model layers allows you to better represent vertical flow and exchange between geologic layers. A fine vertical discretisation may also be required for solute transport simulations.

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

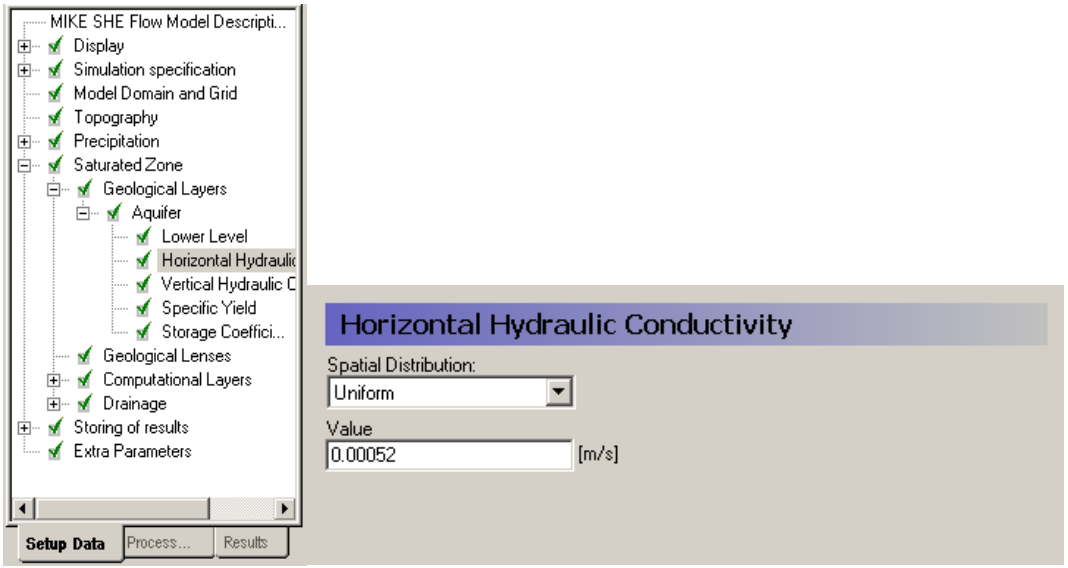


Each of the Geological layers includes one item in the data tree for each of the properties. Additional layers will add additional sets of items to the data tree.

3.7.4 Assign hydraulic properties

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

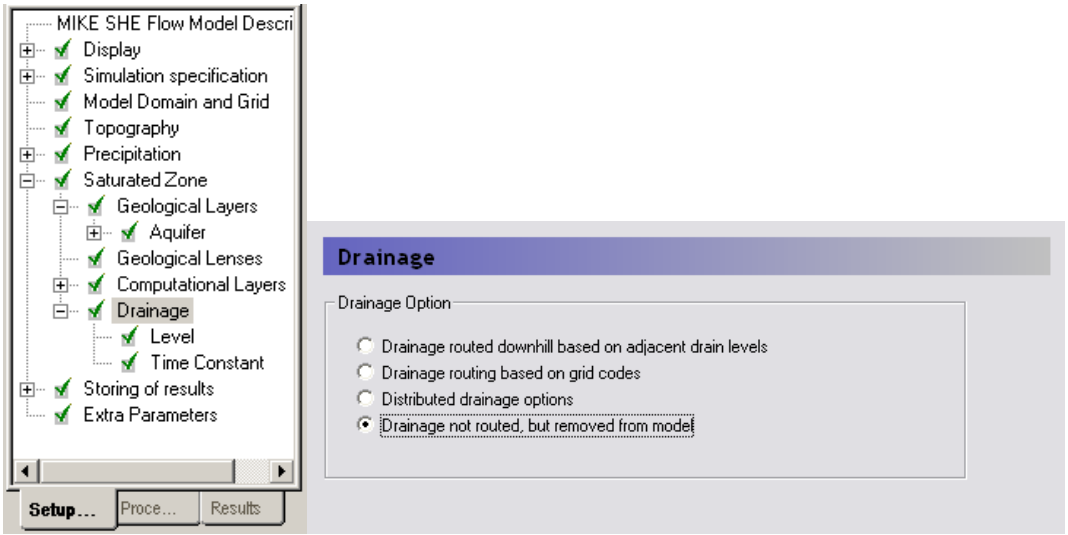
- Select *Uniform* under the *Spatial distribution* combo box and specify the following values:
  - Horizontal hydraulic conductivity = 0.00052 [m/s]
  - Vertical Hydraulic Conductivity = 9.3e-5 [m/s]
  - Specific Yield = 0.2 [-]
  - Specific Storage = 0.0001 [1/m]



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.

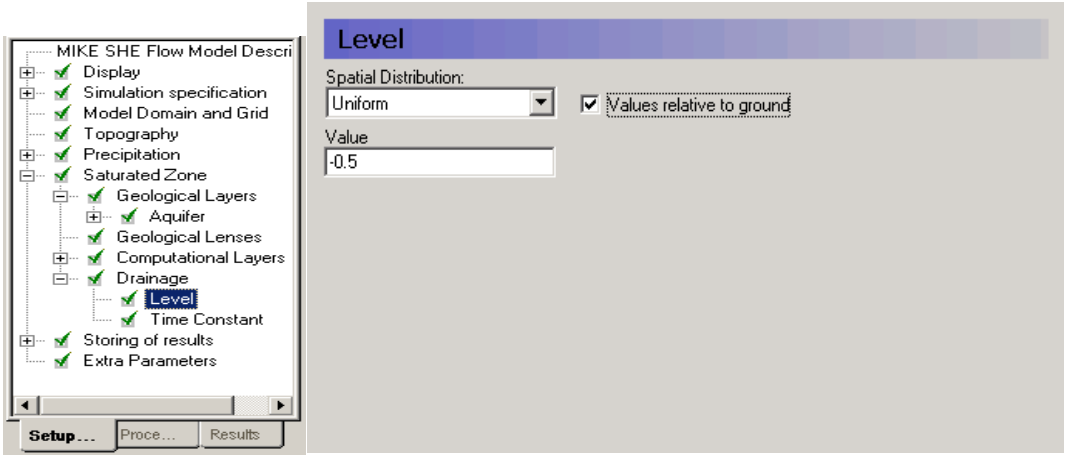
Water flows into the drains, whenever the groundwater table is above the drain level.

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*"



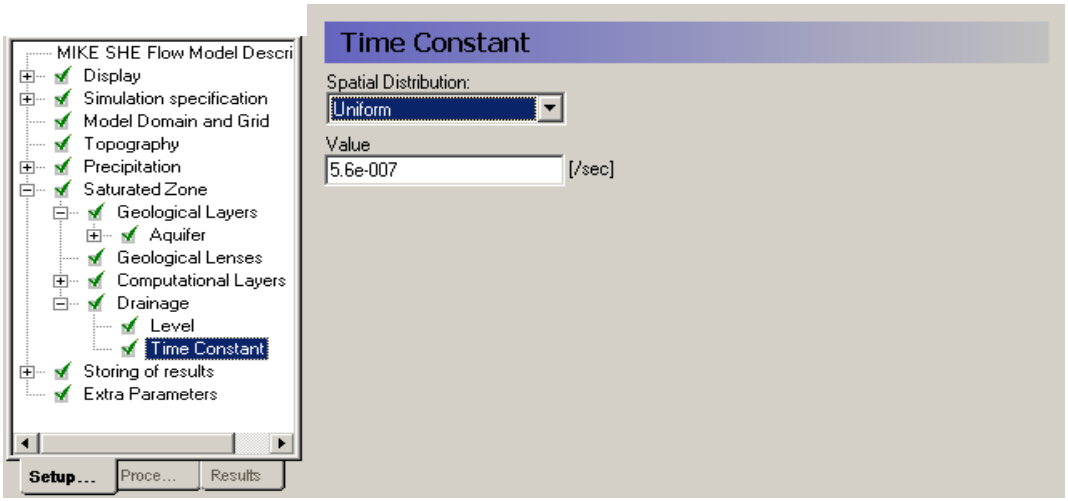
The *Values relative to ground* tells MIKE SHE that you have drains located 0.5 meter below ground surface in the entire model domain.

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]



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

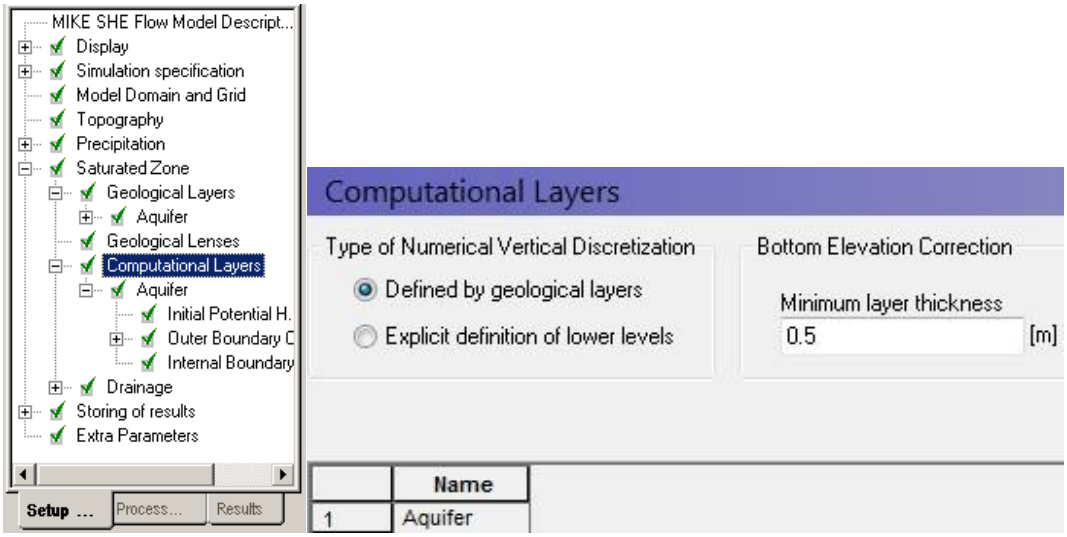
The computational layers are defined independently of the geological layers. While the *Geological Layers* have geological attributes, the *Computational Layers* include computational attributes such as initial conditions and boundary conditions. The properties of the geological layers are mapped and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

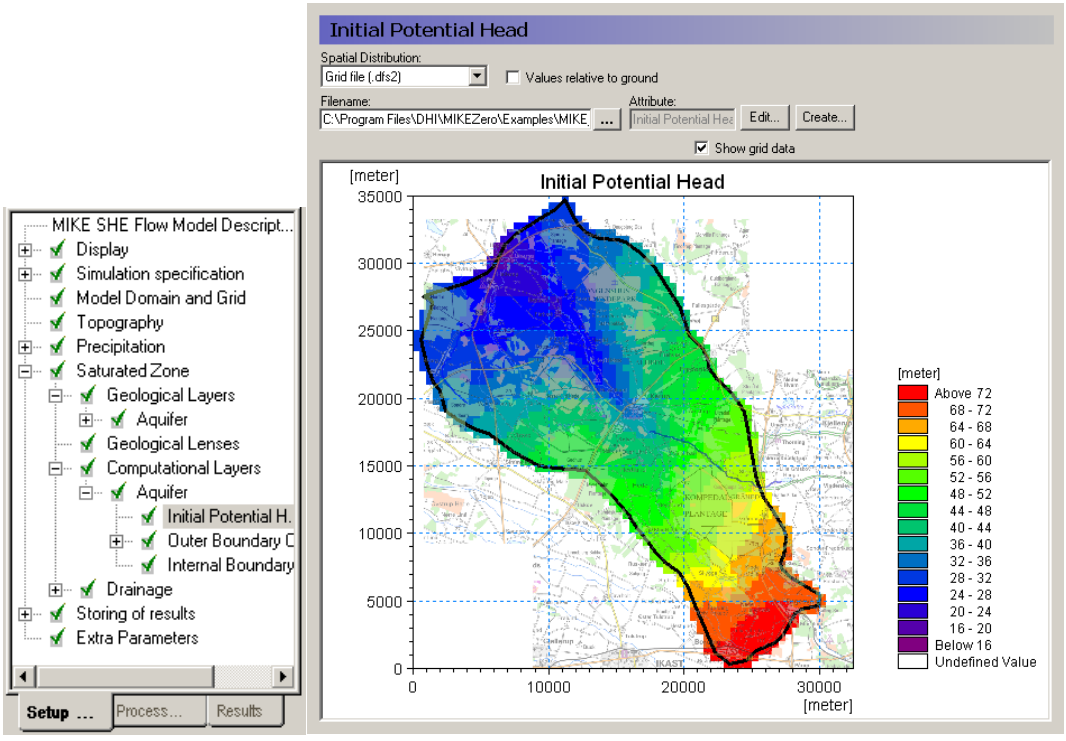


The Minimum layer thickness is used to keep layers from crossing or completely pinching out, which would cause numerical difficulties.

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**



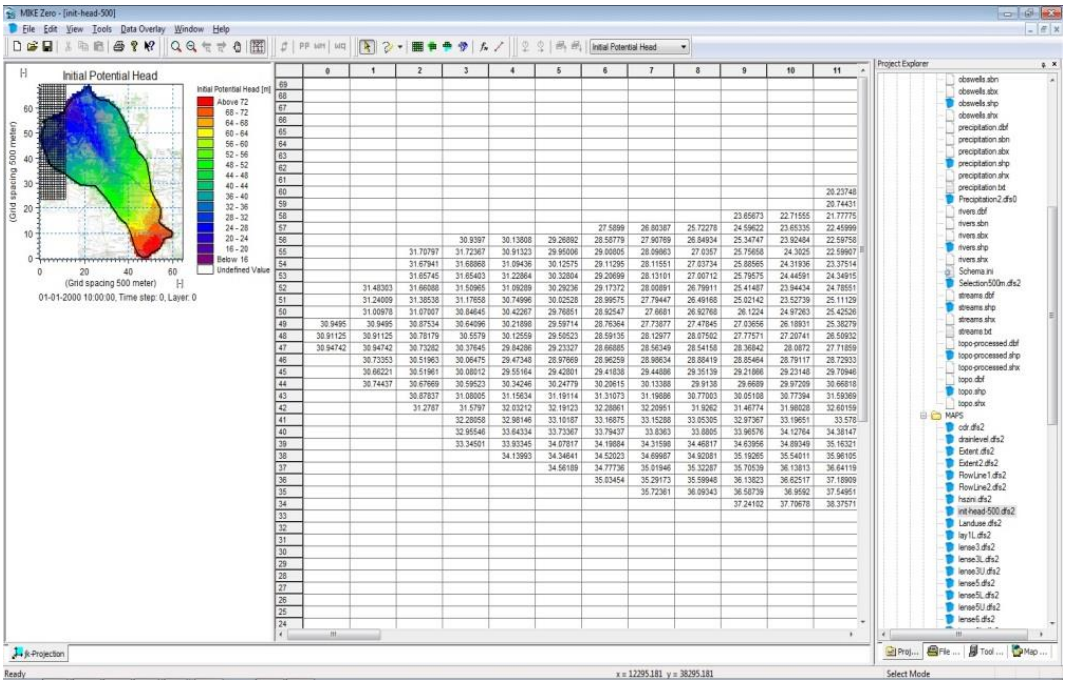
You are using an initial head from a previous model run, which gives you a good starting point. In a real model, you might initially start from a value such as 3 m below the topography. However, such a value will require a “run in time”, as the model may take several months to equilibrate.

An alternative is to run the model as a steady-state simulation and then use the steady-state 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,

, 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, , 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

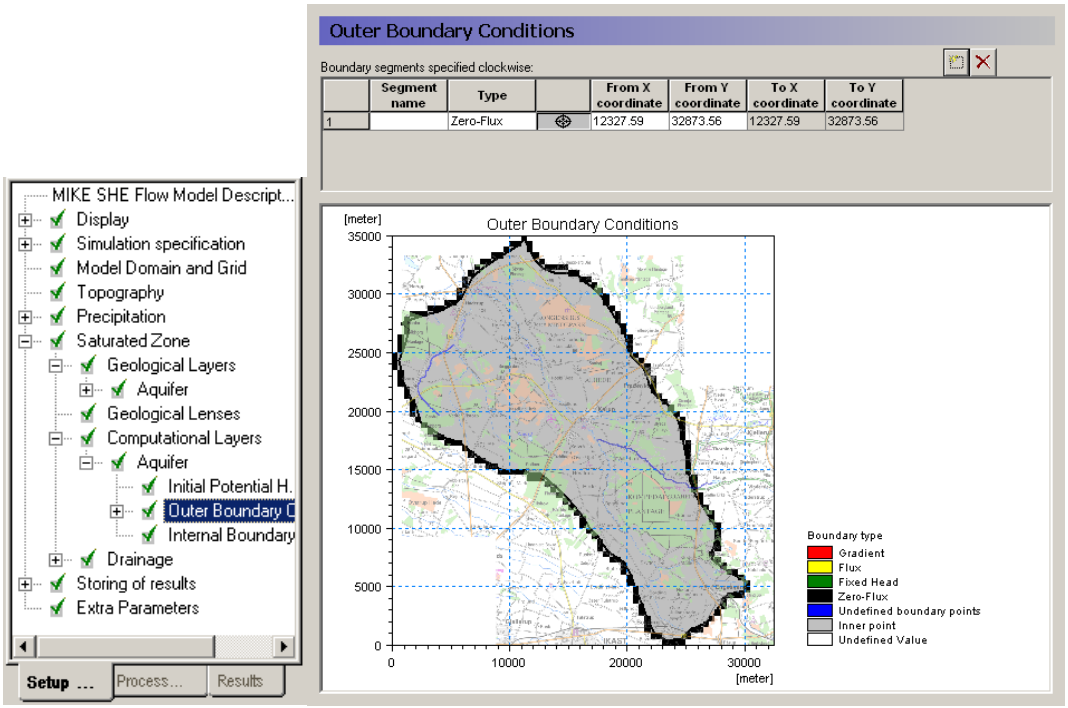
The *Outer Boundary* is defined by the row of cells on the outside of the model domain. The outer boundary is defined by the *Model domain and grid* file, as all points with a code value of 2 (boundary point).

In the *Outer Boundary* dialogue:

- Press the *Add Item* Icon,
- 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, , 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

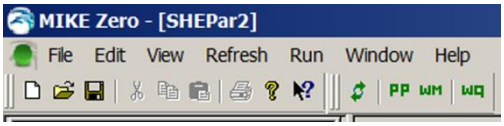
You have now specified all the input required for the model, including

- 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

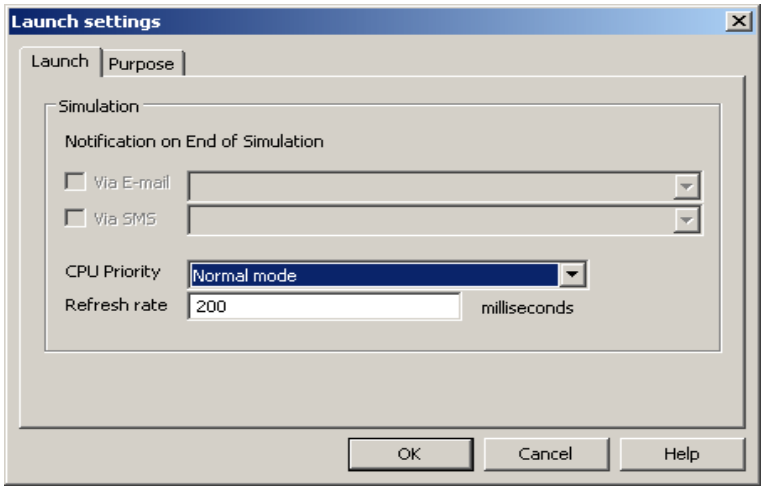
Click on the PP icon to start the pre-processor.



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

- Click OK

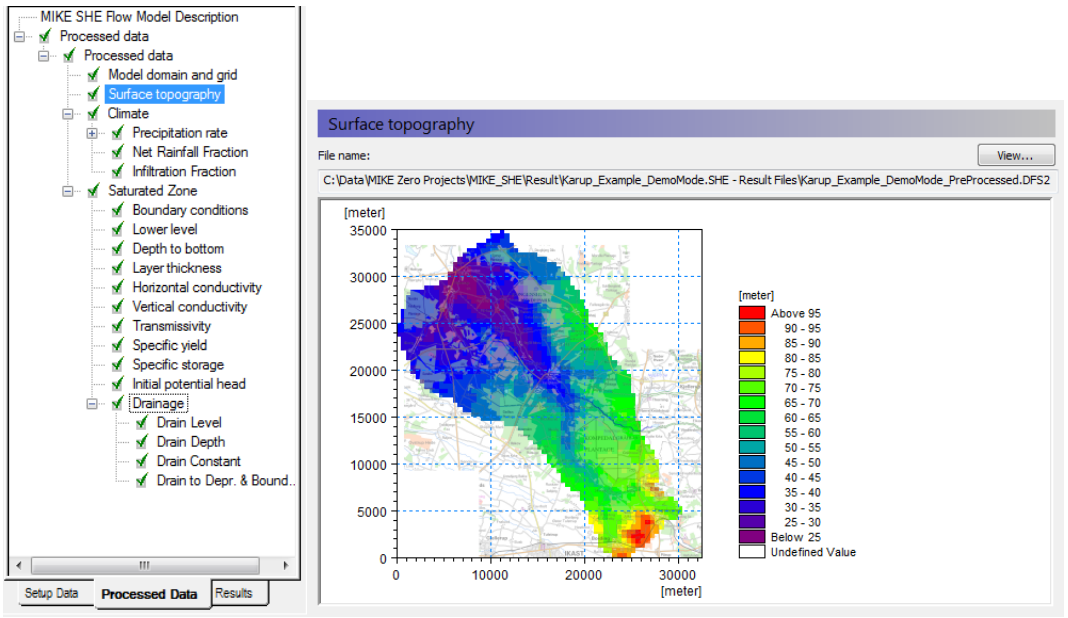
If you are starting a very long simulation, then you can also set up MIKE Zero to alert you when the simulation is finished.



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

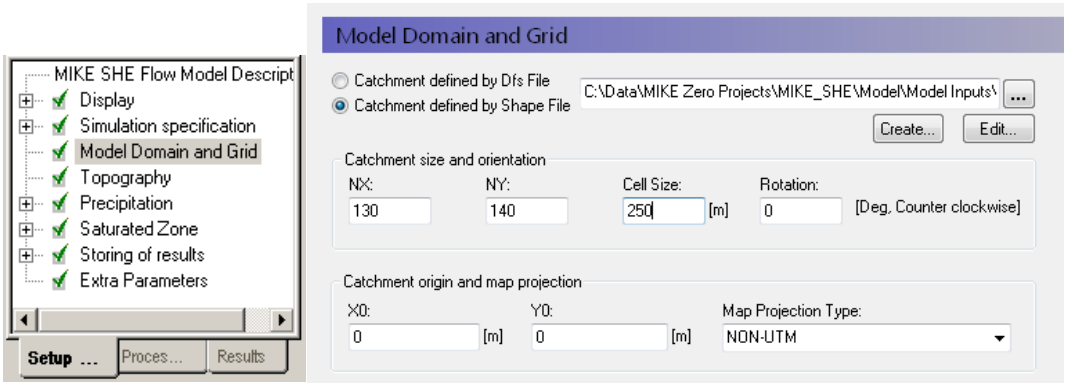
A very powerful feature of MIKE SHE is that you can change the horizontal grid without redefining your model parameters. Thus, you can perform sensitivity analysis of your model grid and grid resolution in a way that is impossible with other modelling user interfaces.

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

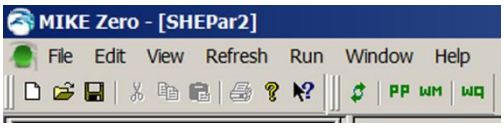
In the *Model Domain and Grid* dialogue of the Setup tab:

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



3.10.2 Run the preprocessor

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

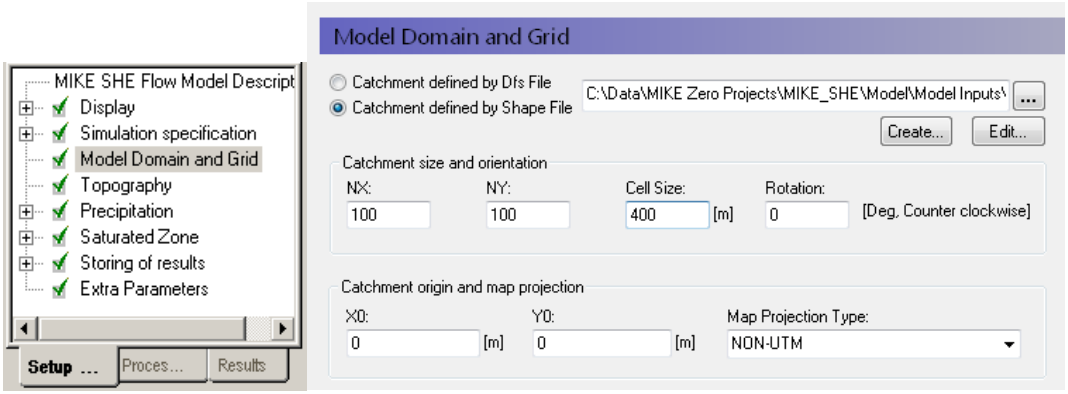


- Verify in the pre-processor tab that all of the input data has been converted to a 250m grid

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



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 assigned 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.

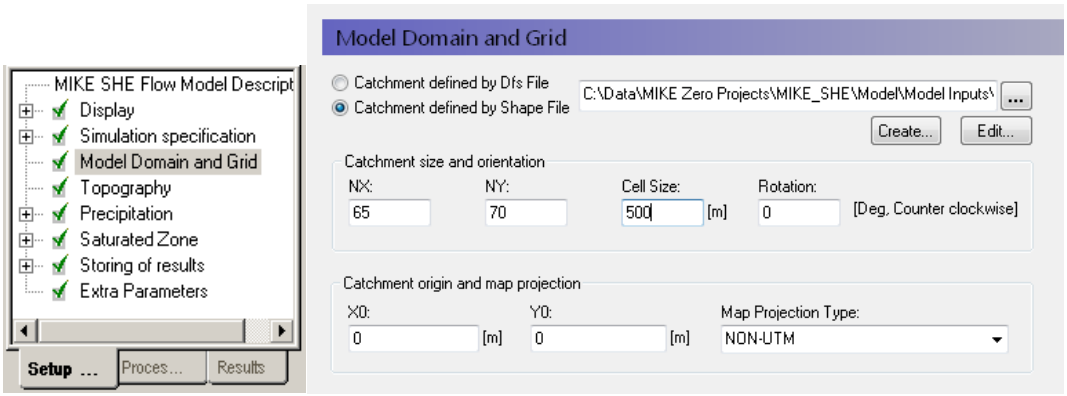
If the grids are **not coincident** and

- 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 interpolate 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



3.10.5 Save your MIKE SHE document

DNG write some TEXT here!

3.11 Step 11 – Specifying Outputs and Calibration Targets

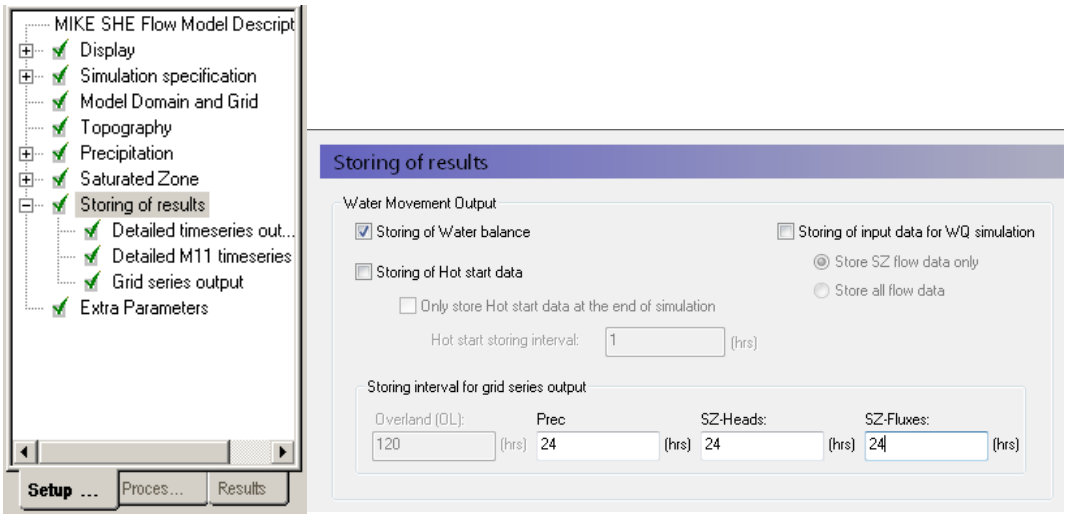
The Detailed Time Series Output allows time Series plots of simulated data at specific locations to be automatically generated during a simulation. It also allows simulated output to be compared to observed output and calculates a number of statistics for model error.

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*



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.

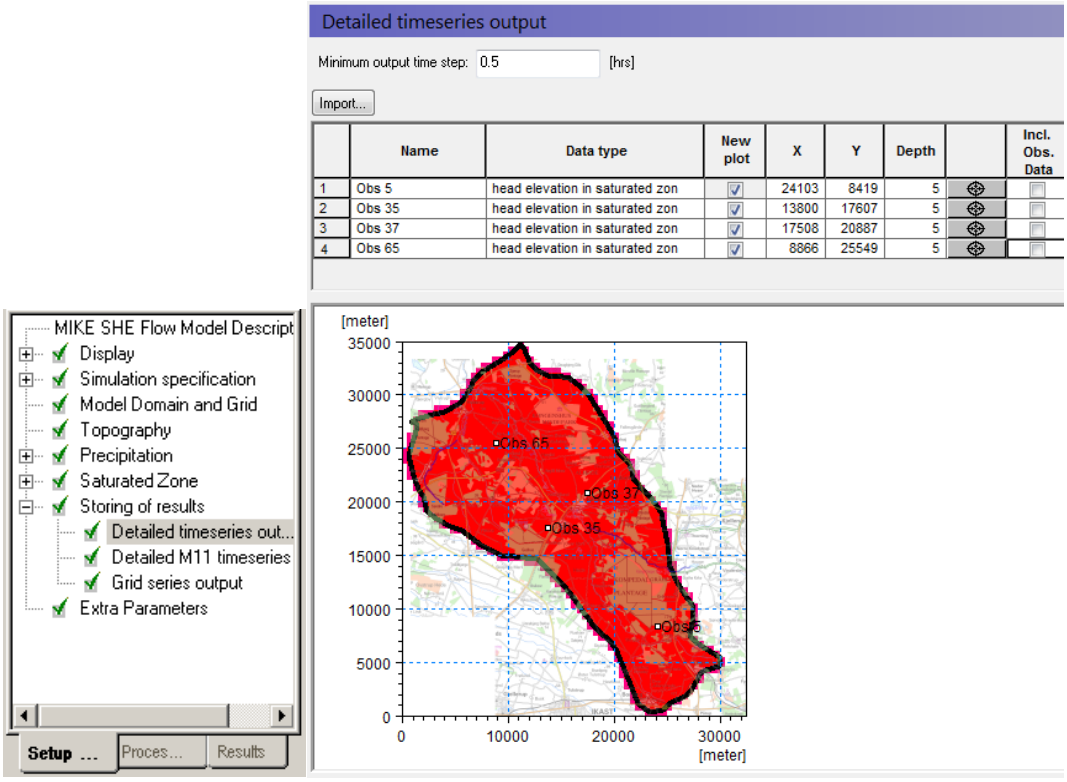
Gridded output is stored as dfs2 (2D grid time-series files) or dfs3 (3D grid time-series files). If output data is stored too frequently, you may get very large and unwieldy result files. Thus, before running the simulation you should consider how often you need to save the data.

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.

Observation data will be added in a later exercise.

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

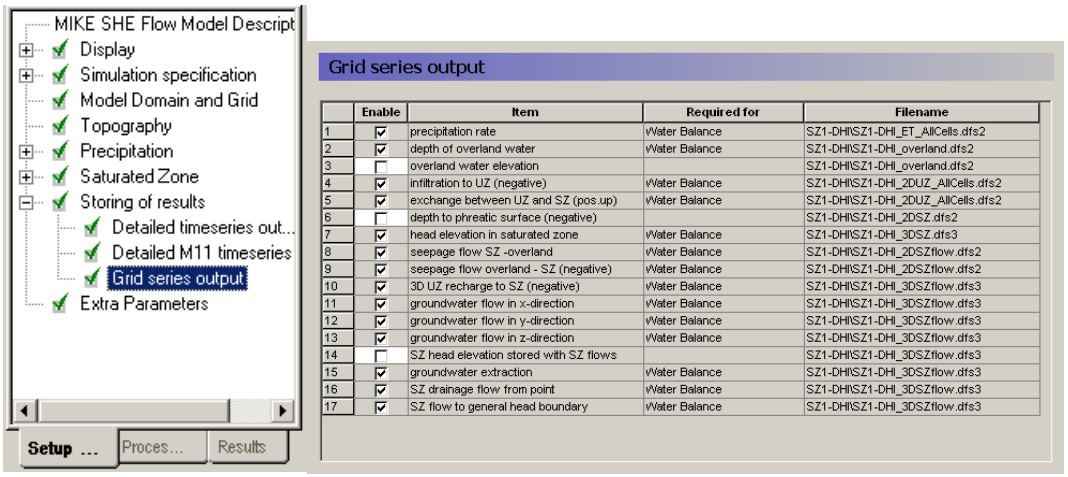
3.11.3 Grid outputs

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.



	Enable	Item	Required for	Filename
1	☑	precipitation rate	Water Balance	SZ1-DHI\SZ1-DHI_ET_AllCells.dfs2
2	☑	depth of overland water	Water Balance	SZ1-DHI\SZ1-DHI_overland.dfs2
3	☐	overland water elevation		SZ1-DHI\SZ1-DHI_overland.dfs2
4	☑	infiltration to UZ (negative)	Water Balance	SZ1-DHI\SZ1-DHI_2DUZ_AllCells.dfs2
5	☑	exchange between UZ and SZ (pos.up)	Water Balance	SZ1-DHI\SZ1-DHI_2DUZ_AllCells.dfs2
6	☐	depth to phreatic surface (negative)		SZ1-DHI\SZ1-DHI_2DSZ.dfs2
7	☑	head elevation in saturated zone	Water Balance	SZ1-DHI\SZ1-DHI_3DSZ.dfs3
8	☑	seepage flow SZ -overland	Water Balance	SZ1-DHI\SZ1-DHI_2DSZflow.dfs2
9	☑	seepage flow overland - SZ (negative)	Water Balance	SZ1-DHI\SZ1-DHI_2DSZflow.dfs2
10	☑	3D UZ recharge to SZ (negative)	Water Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
11	☑	groundwater flow in x-direction	Water Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
12	☑	groundwater flow in y-direction	Water Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
13	☑	groundwater flow in z-direction	Water Balance	SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
14	☐	SZ head elevation stored with SZ flows		SZ1-DHI\SZ1-DHI_3DSZflow.dfs3
15	☑	groundwater extraction	Water 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

3.12 Step 12 – Run the Simulation

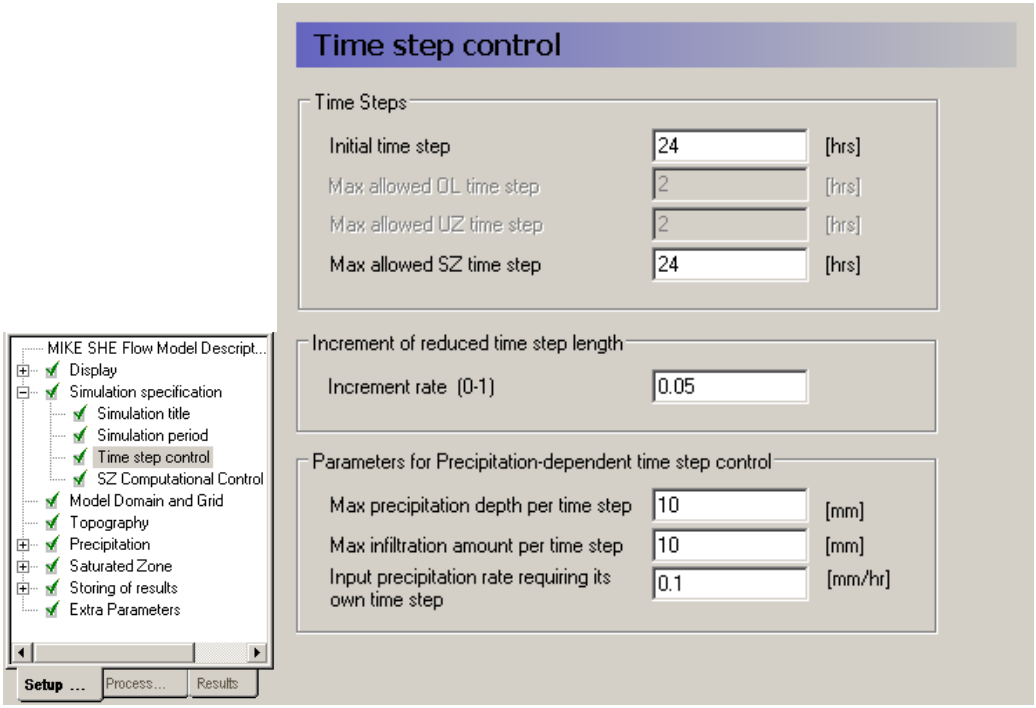
Before running the model, you need to go back to the Setup menu to make a few changes to the simulation specifications, including simulation period, time-steps and storing of results.

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



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.

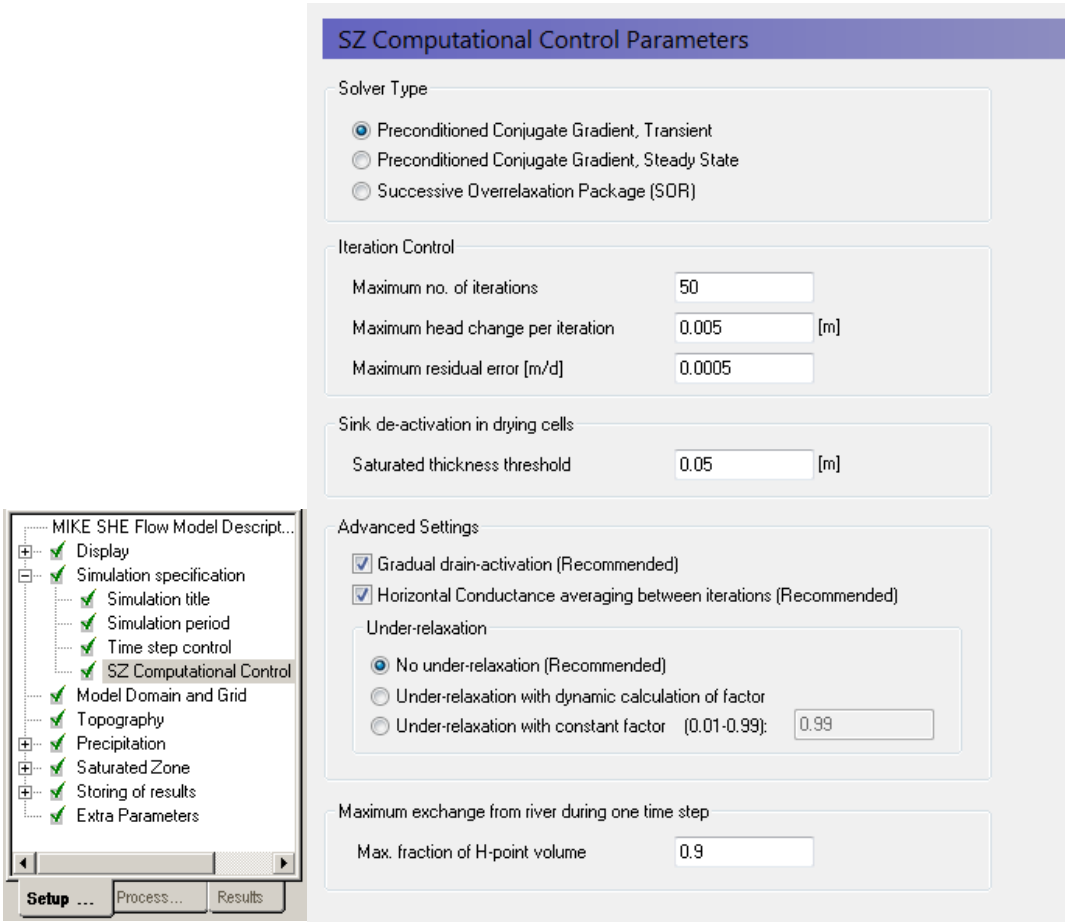
If you encounter small time steps during rainfall events, you can sometimes speed up the simulation by increasing these values. Look in the log files (information text files, which are created during the simulation) to see if the time step is being automatically reduced because of these criteria.

3.12.2 Simulation control

In the *SZ Computational Control Parameter* dialogue:

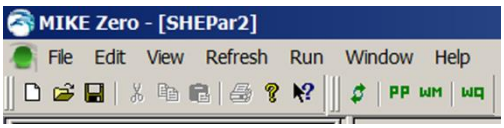
- Choose *Preconditioned Conjugate Gradient, Transient* (default)

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



3.12.3 Run the simulation

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



3.13 Step 13 – View the Results

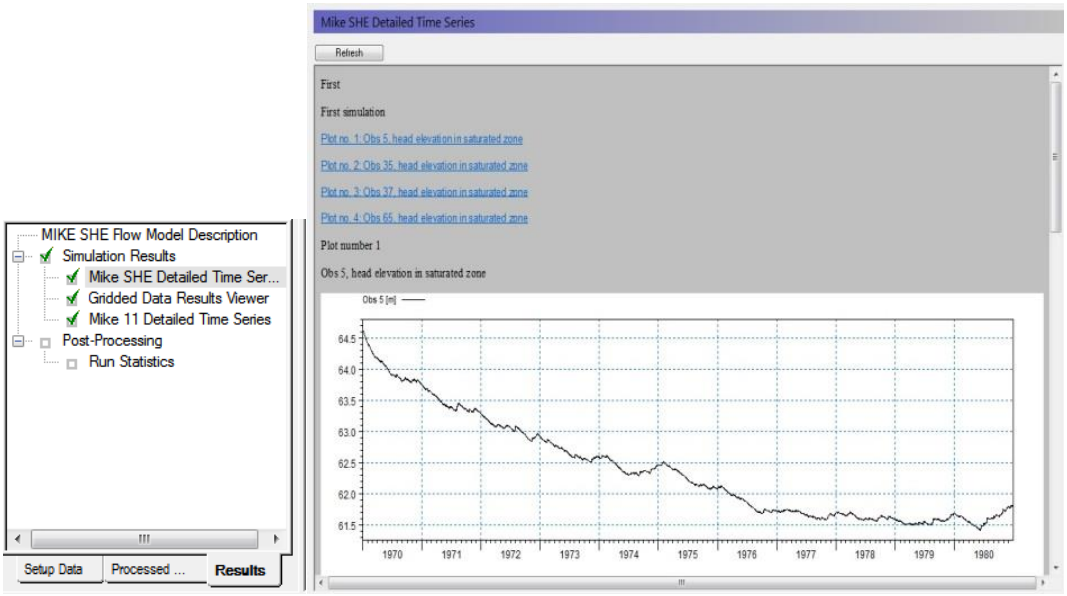
Upon successful execution of the model you're now ready to view the results.

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.

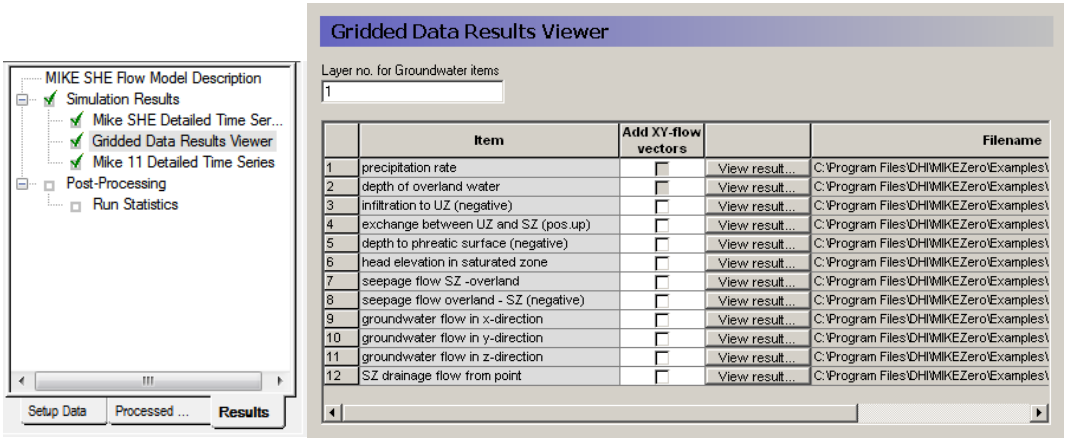


The *MIKE SHE Detailed Time Series* output dialogue, under *Simulation Results,* lists the Detailed Time Series results you have chosen to store during the MIKE SHE simulation.

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.



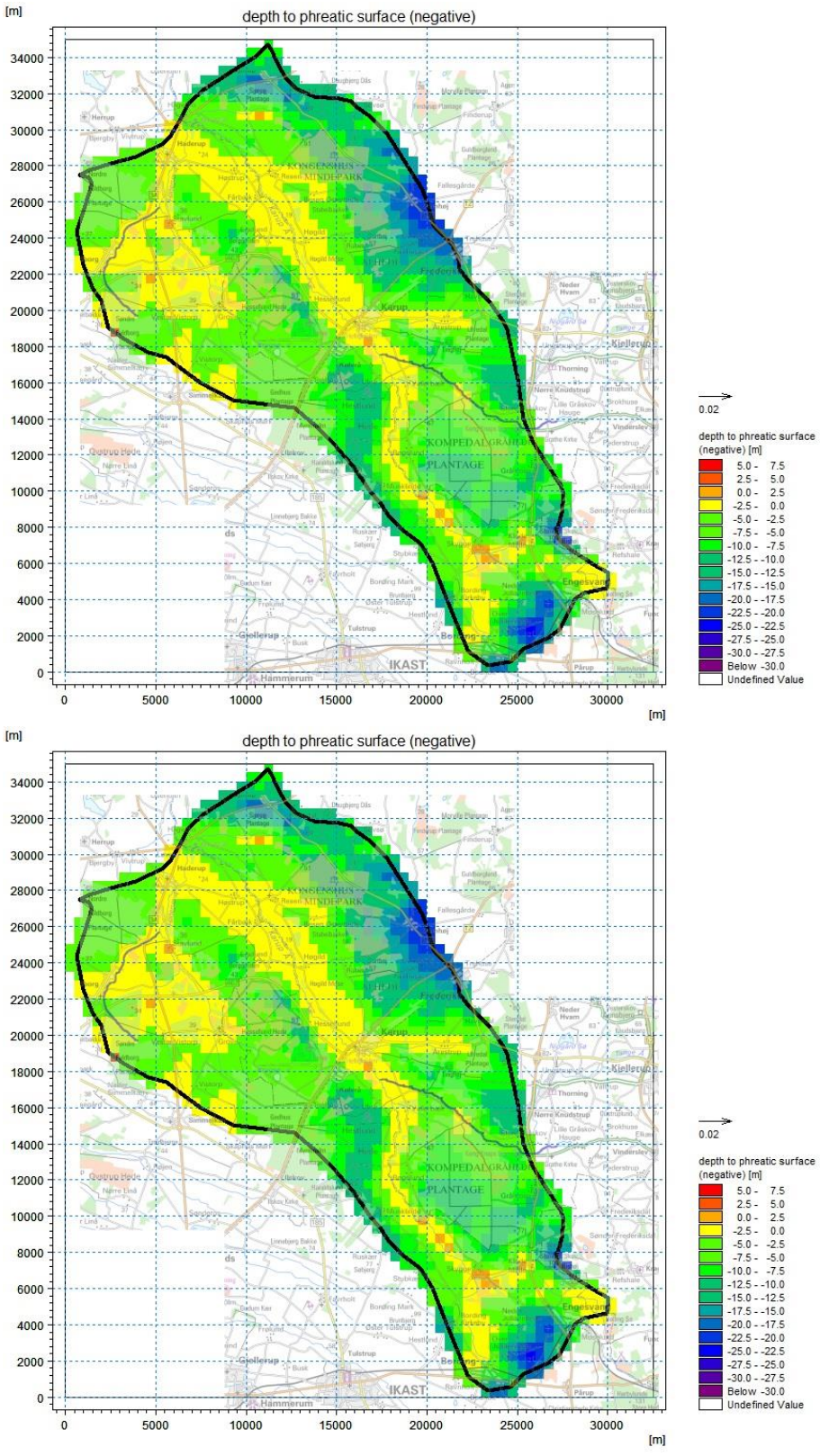
The *Results Viewer* dialogue lists the results you have chosen to store in the MIKE SHE results files.

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.



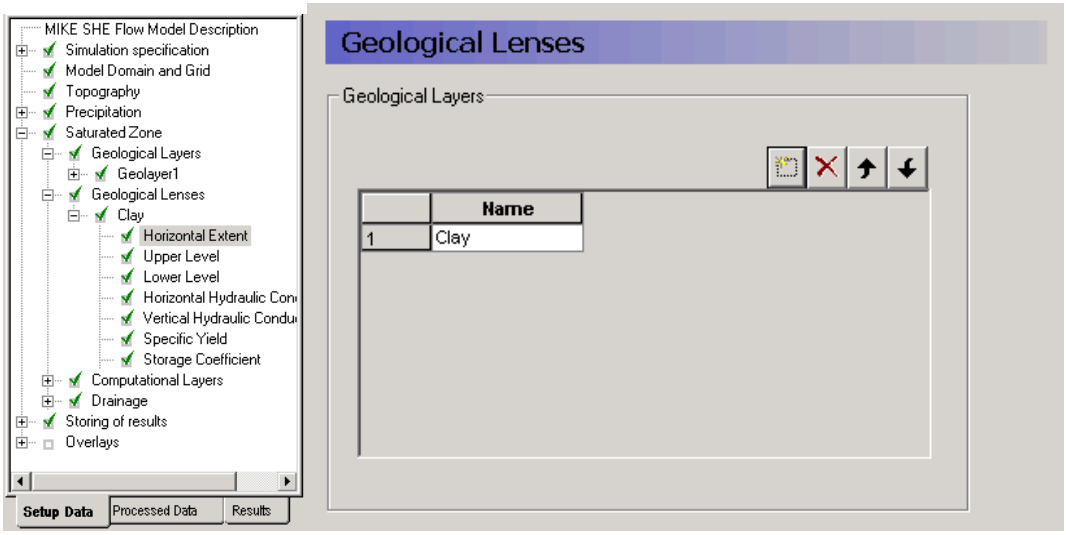
3.14 Step 14 – Add Geological Lenses

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

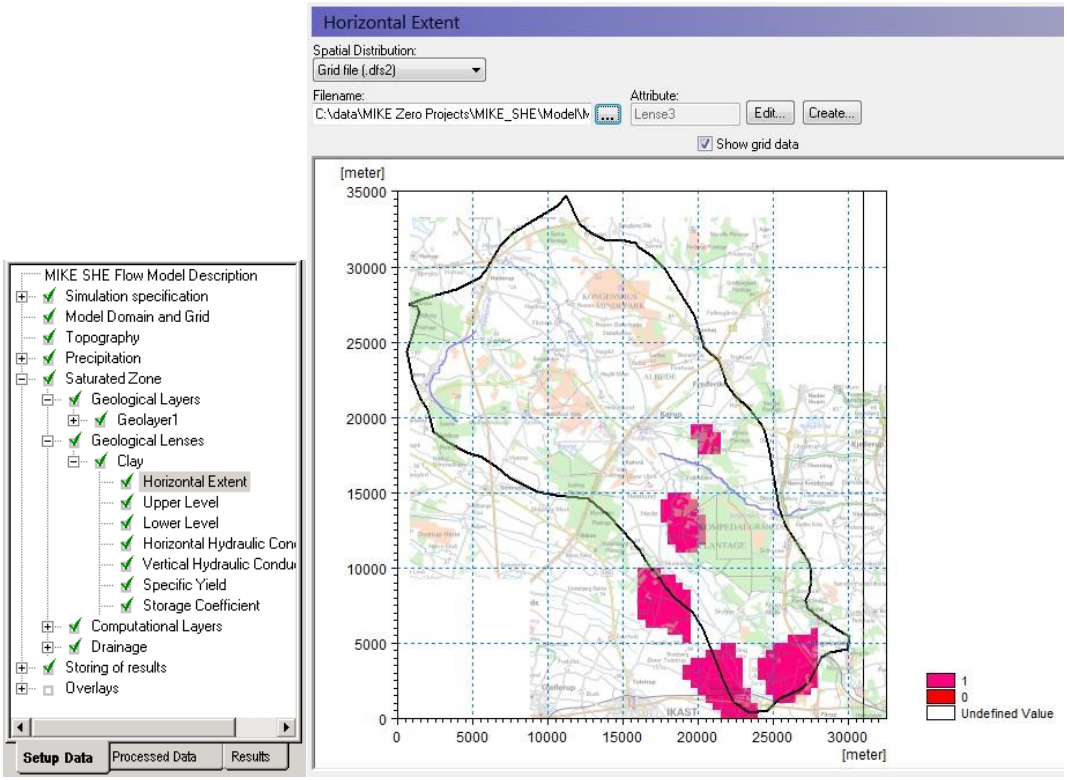
- Click on the Add Item icon, , to add a lens.
- Change the Name of the lens to “Clay”



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 ***.\Model Inputs\Maps\lense3.dfs2***



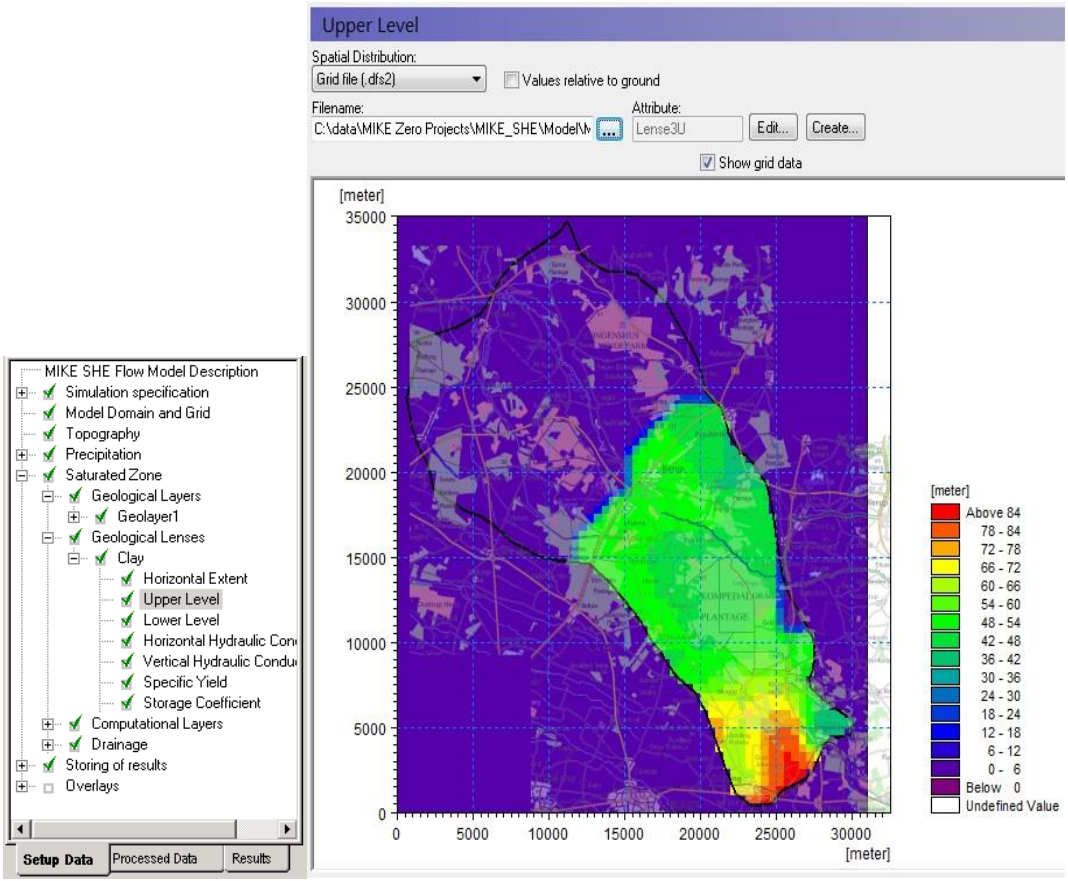
All the lenses will be assigned the same hydraulic properties. If you want to assign different hydraulic properties to different lenses, then you need to add additional lenses (see the step above)

Lenses can also be assigned using polygon shp files.

3.14.3 Define the top of the lens

In the *Upper Level* dialogue

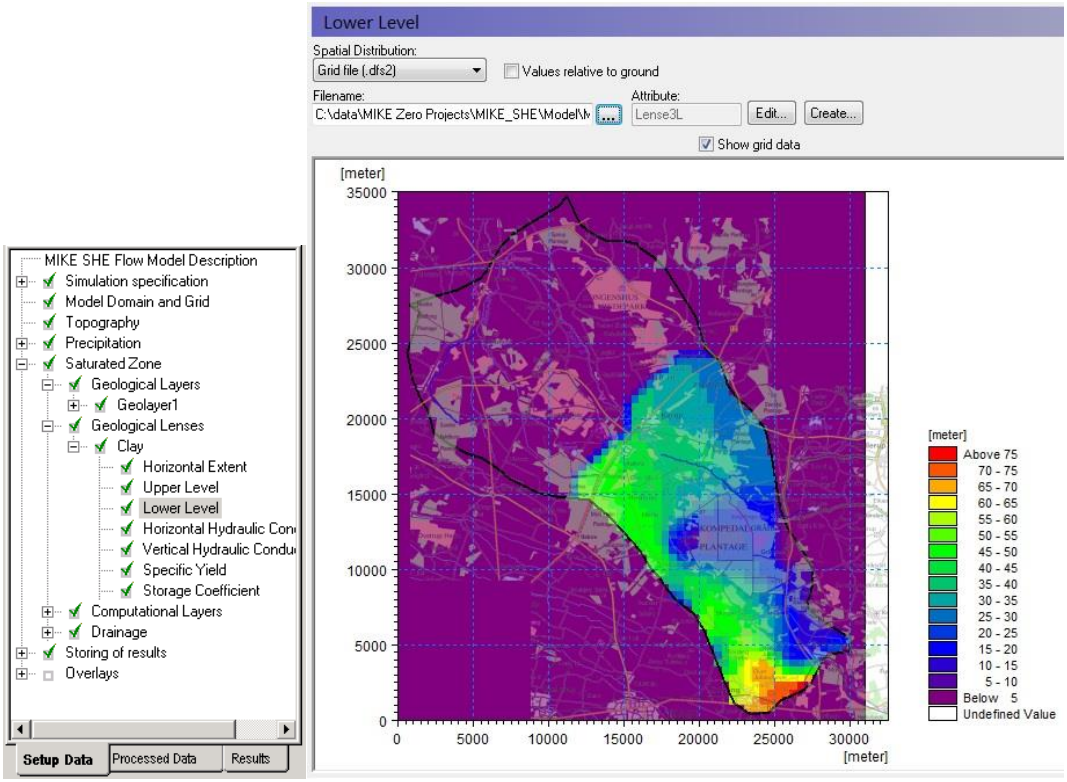
- Select *Grid file (.dfs2)* files for *Spatial Distribution*
- Use the Browse button to select the file ***.\Model Inputs\Maps\lense3U.dfs2***



3.14.4 Define the bottom of the lens

In the *Lower Level* dialogue

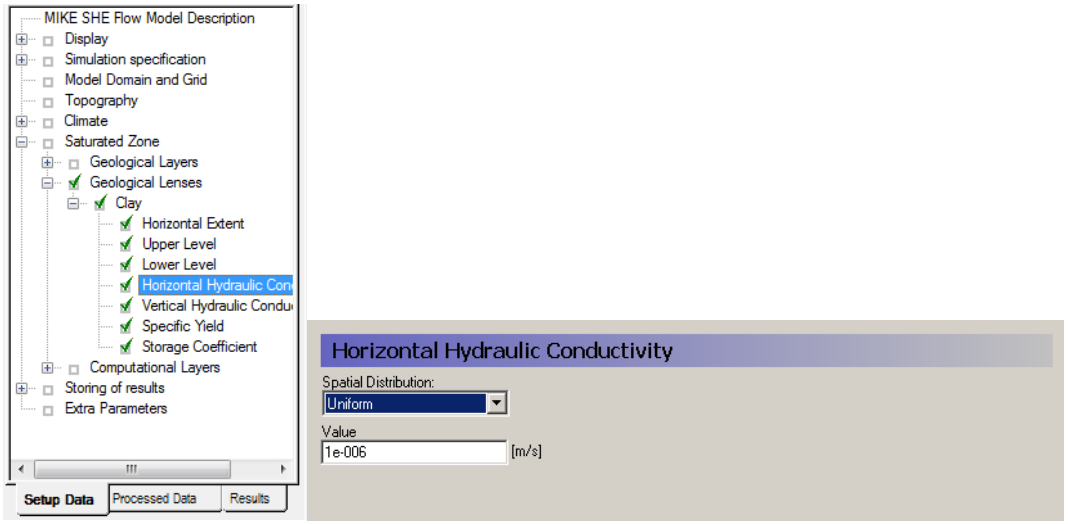
- Select *Grid file (.dfs2)* files for *Spatial Distribution*
- Use the Browse button to select the file ***.\Model Inputs\Maps\lense3L.dfs2***



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.

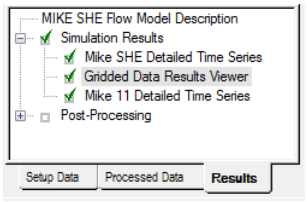
3.15 Step 15 – Advanced use of the Results Viewer

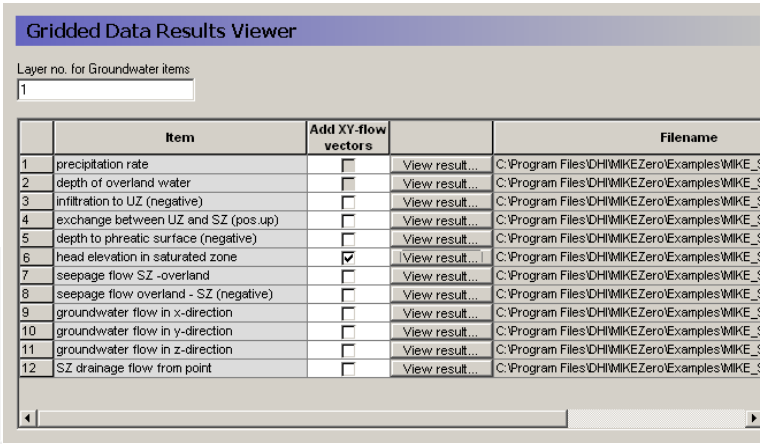
This step demonstrates some of the advanced features of the Result Viewer that have not been discussed previously.

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.



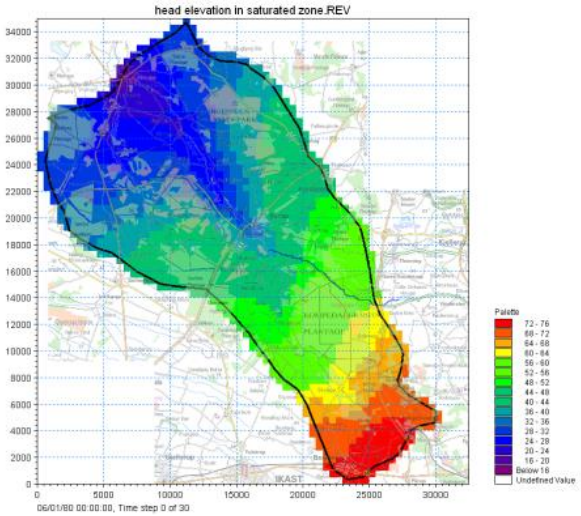


The *Results Viewer* dialogue lists the results you have chosen to store in the MIKE SHE results file. In this exercise, you chose to store the precipitation (groundwater recharge), depth to phreatic surface, groundwater head, and the groundwater flow in the x-, y- and z-directions.

3.15.2 View the results in the results viewer

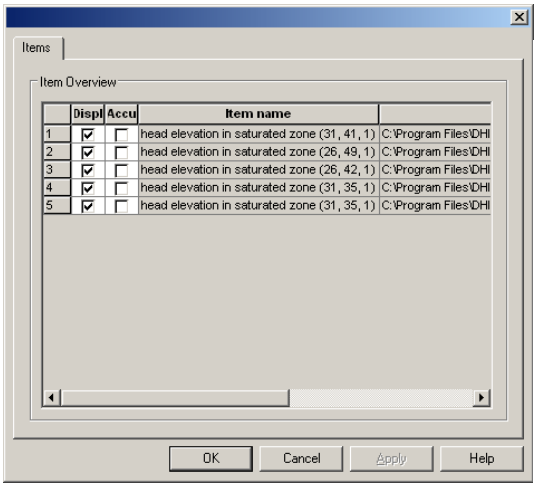
- Click on the View Results,

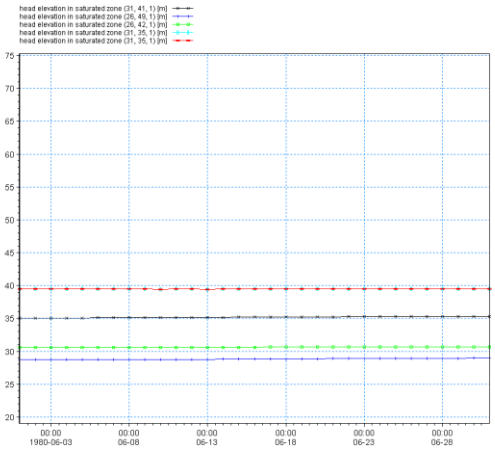
, button for the item *head elevation in saturated zone*.



3.15.3 Time series plot at a point

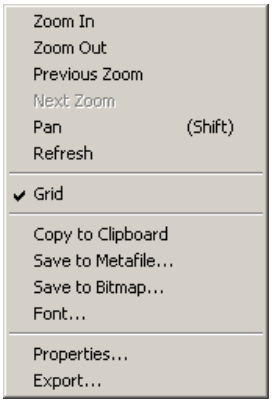
- Press the Time Series button on the Results Viewer tool bar
- Click once inside the map area and then while holding down the Ctrl-key, click on each additional point where you want time series output. Each selected locations is marked with a yellow x.
- Double click on the last point. (if you only want one point then simple double click without holding down the Ctrl-key)
- 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.



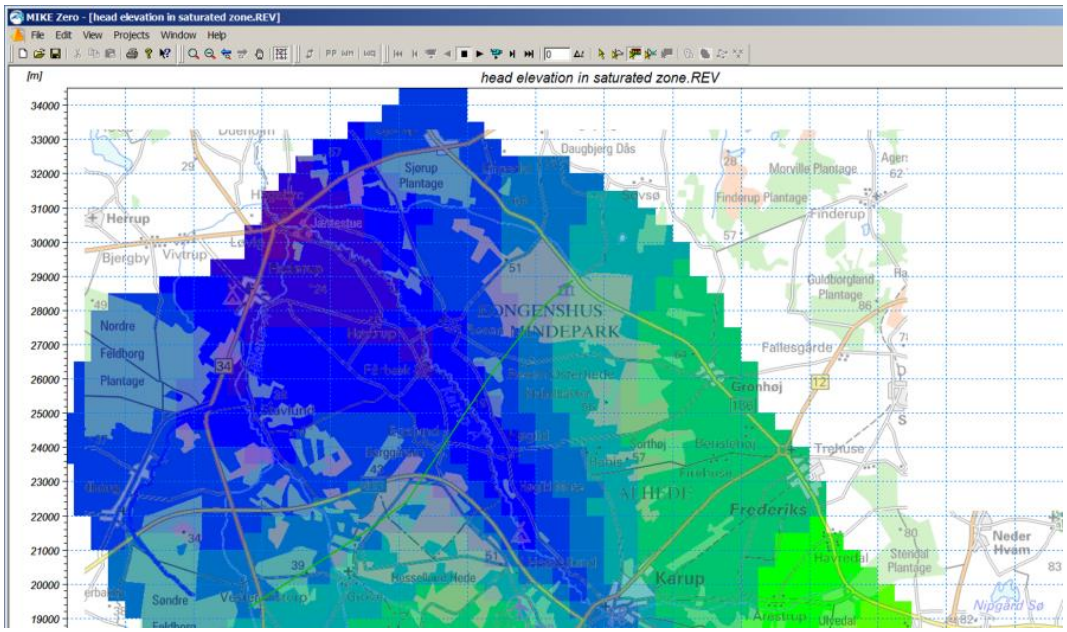
The j,k coordinates listed at the bottom of the Results Viewer go from 0 to (nx-1) and 0 to (ny-1) in the j and k directions, respectively.

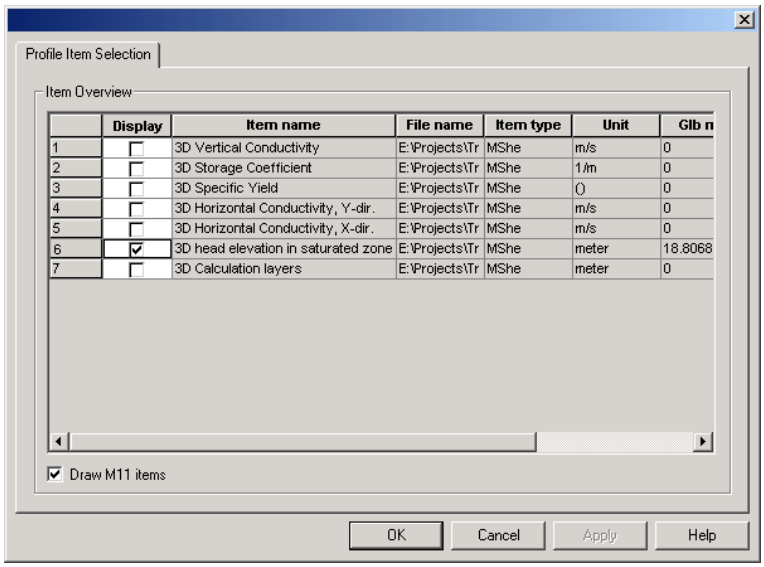
Note that the time series time steps are equal to the storing time steps. The *Detailed timeseries output* option within the *Setup Data* allows you to save the output at every time step.

3.15.4 Extract a profile of water levels

- Press the button on 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.





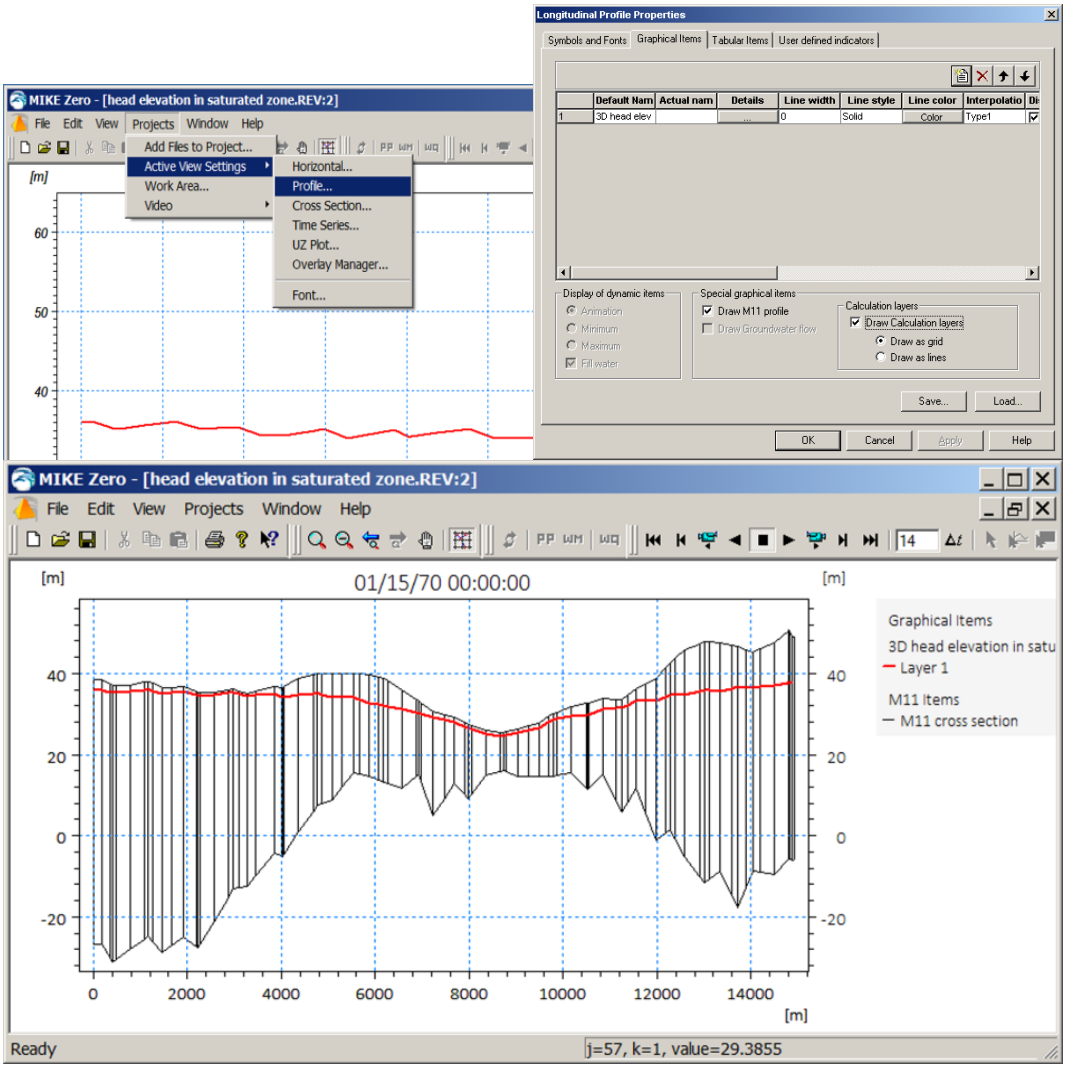
3.15.5 Display the computational grid on profile.

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

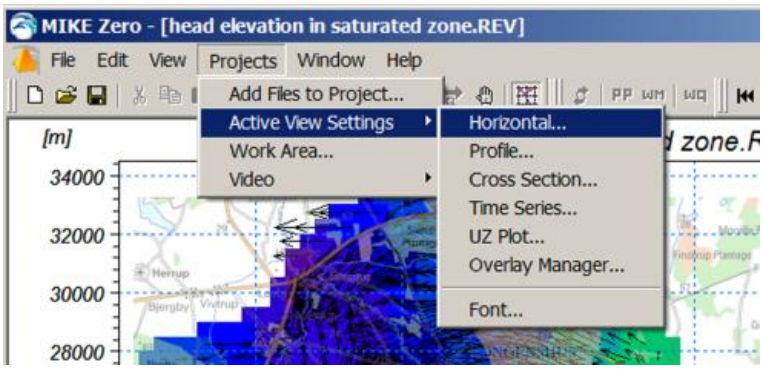


3.15.6 Modify visible overlays and display the finite difference mesh in result viewer

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 …*

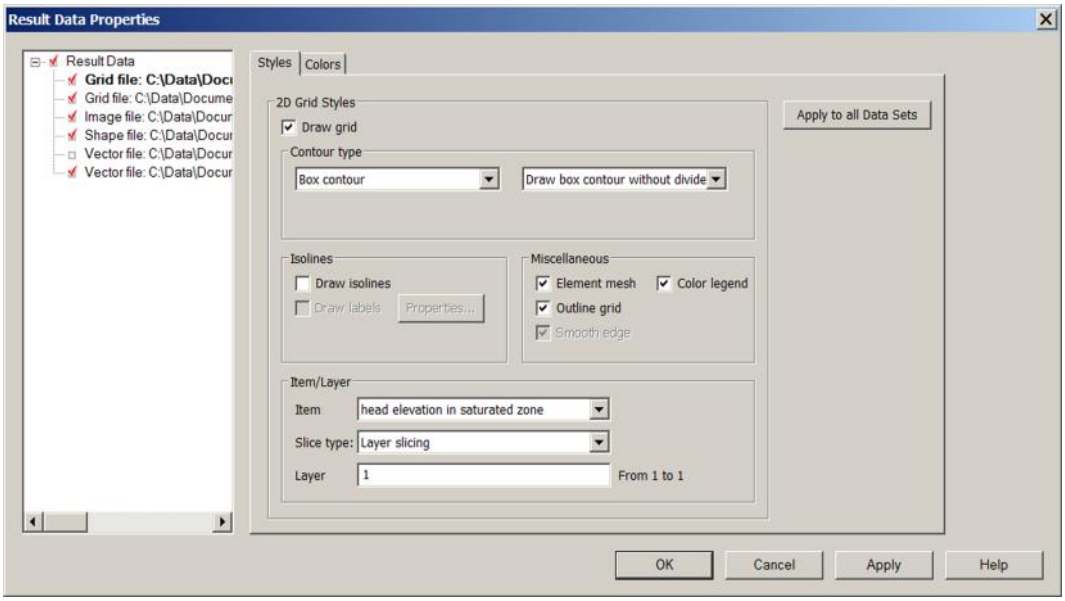


In the dialogue that appears,

- 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.

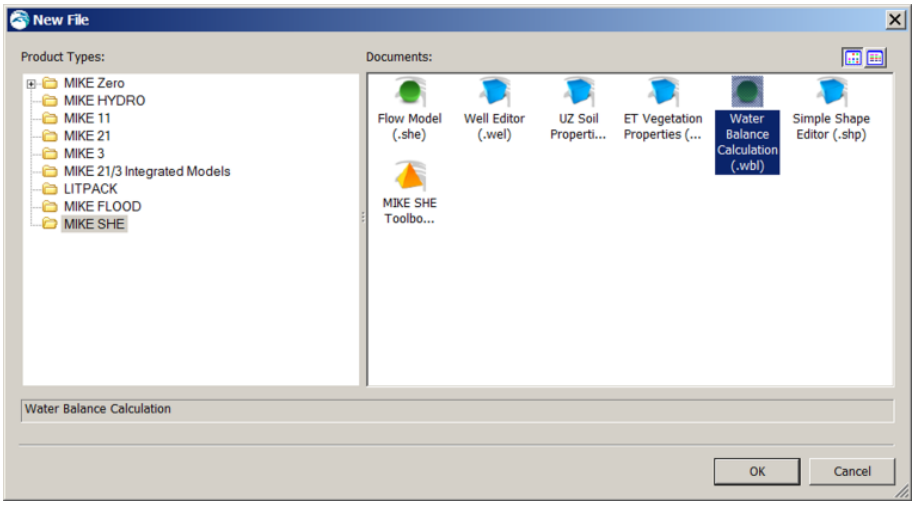


3.16 Step 16 Create a Water Balance

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

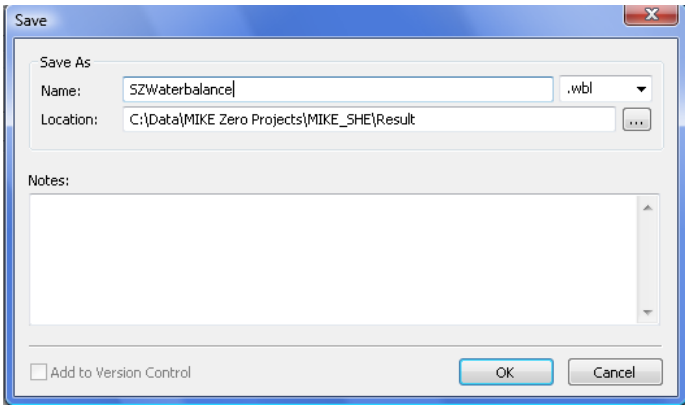
3.16.1 Create a new water balance document

- Click on the New document icon, , 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:

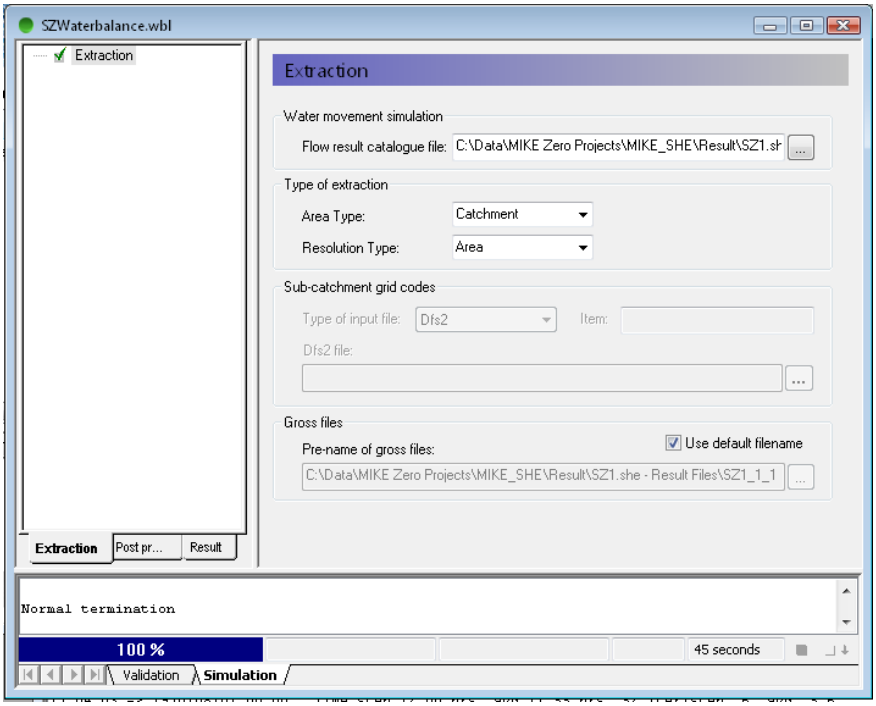
1. Extract the water balance data from the MIKE SHE output files
2. Create specific water balances from the extracted data
3. Evaluate the water balance output



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, or by 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



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 sub-areas 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 sub-catchment. The Single-cell option allows you to create maps of some of the water balance items.

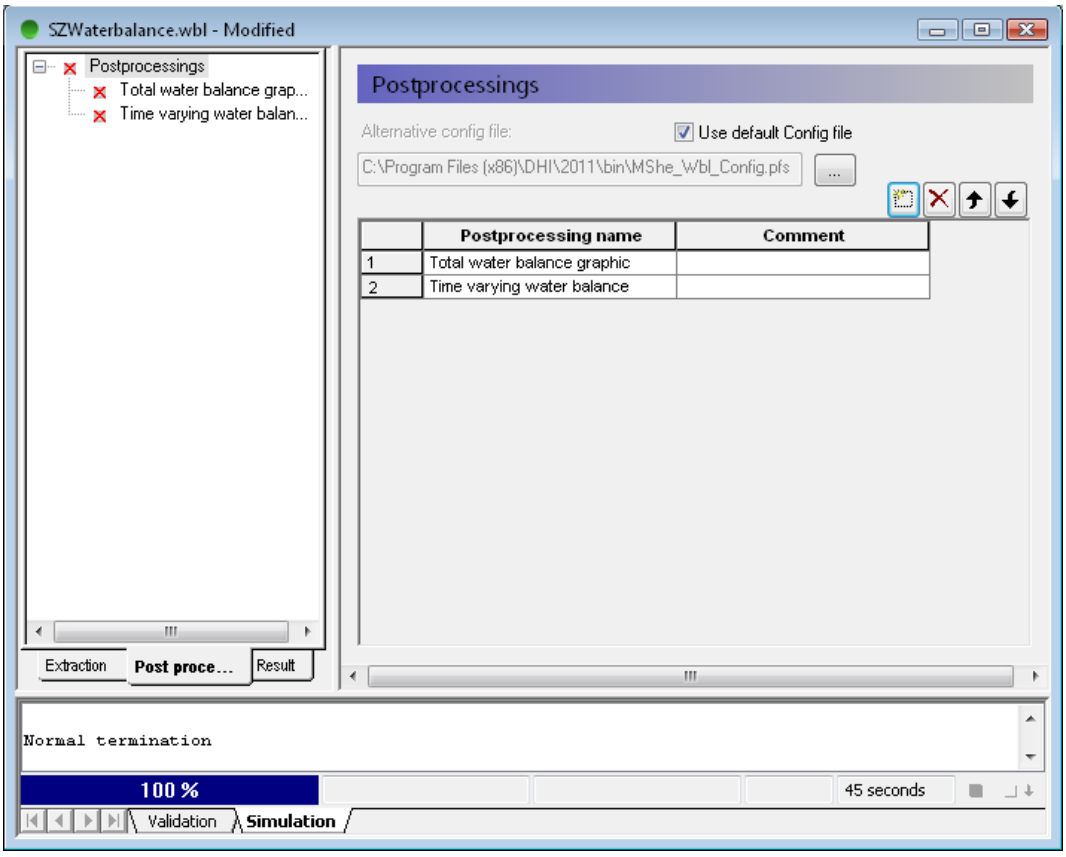
3.16.3 Create water balance items

- 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.

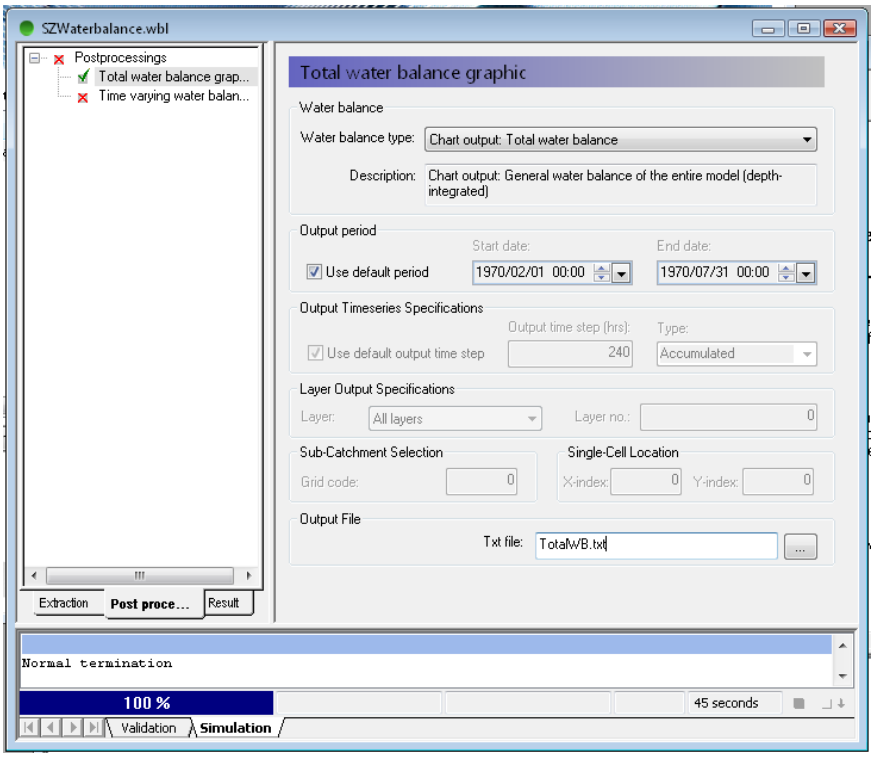
The water balances are defined in a water balance .config file, which is located in the program installation directory (C:\Program Files (x86)\DHI\2016\bin).

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.



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*,



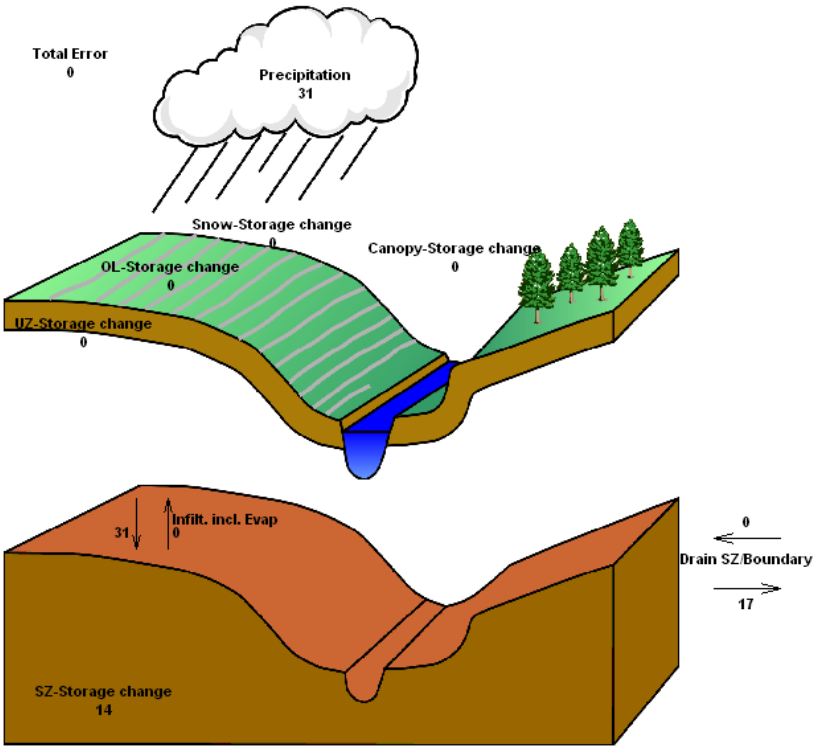
The Chart output creates a graphic illustrating the water balance items (see next step). The output is sent to an ASCII .txt file, and the graphic is generated automatically by a utility that reads txt file.

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.

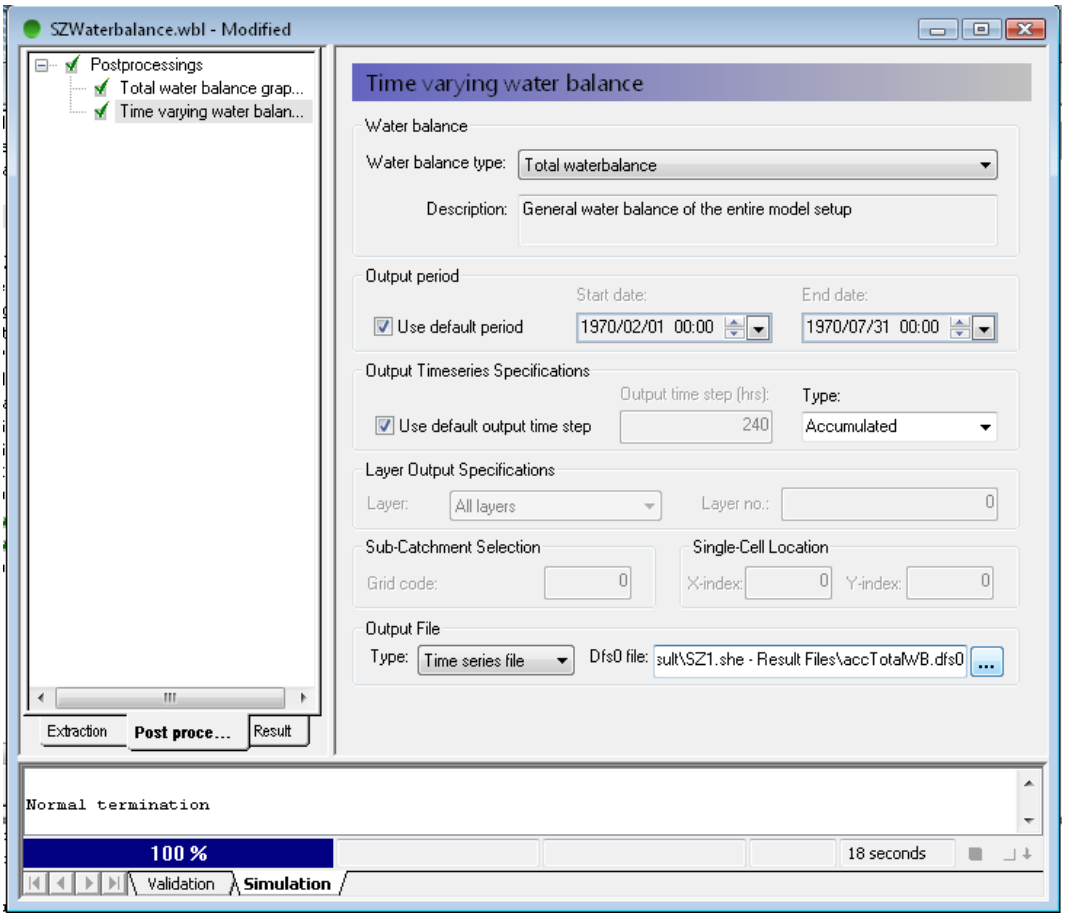
Note that the sum of the water balance terms in the graphic do not add to zero. This is because some of the signs have been removed (e.g. Precipitation and ET have the same sign, but opposite inflow/outflow directions). In the water balance calculation, Inflows are negative and Outflows and Storage increases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



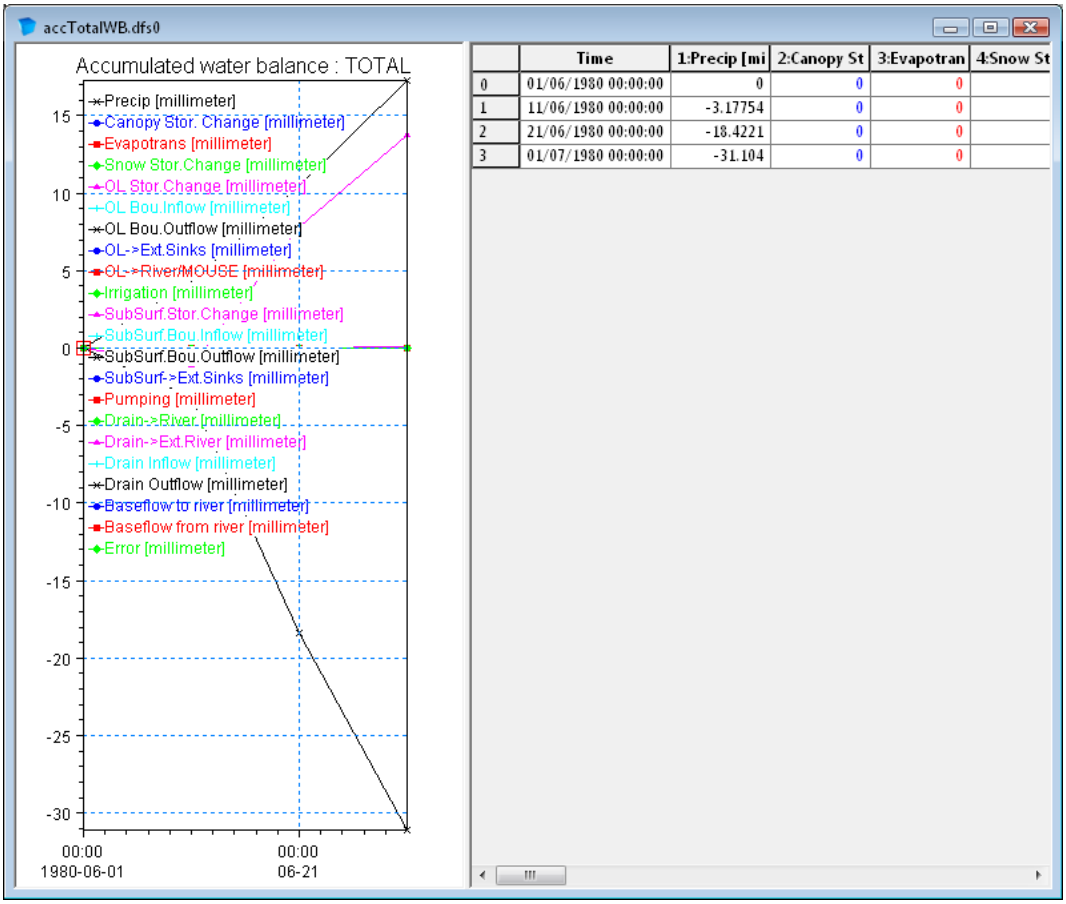
If you chose the Table format for the Output file, then the output will be in a tab-delimited ASCII file that you can open in Excel.

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

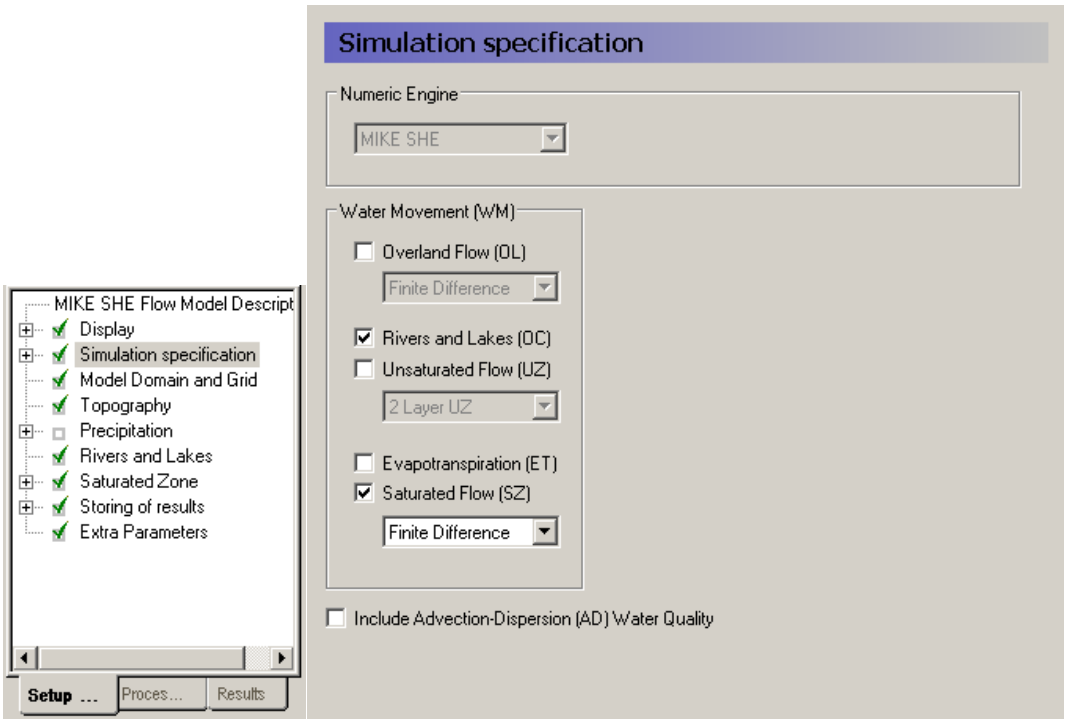
You will now add a river to the groundwater model that you created in the SZ1 exercise. In this exercise, you will use an existing MIKE 11 river setup.

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.

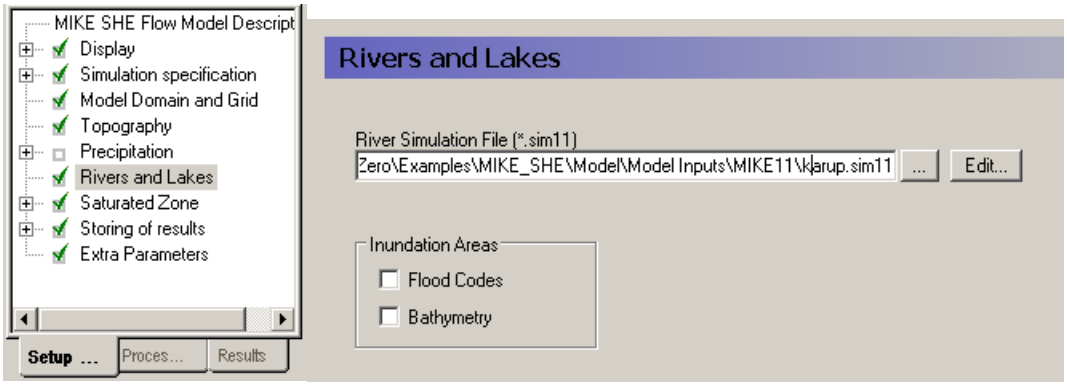


3.17.2 Add the River Module

In the *Rivers and Lakes* Dialogue

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

button to enter the MIKE 11 simulation editor



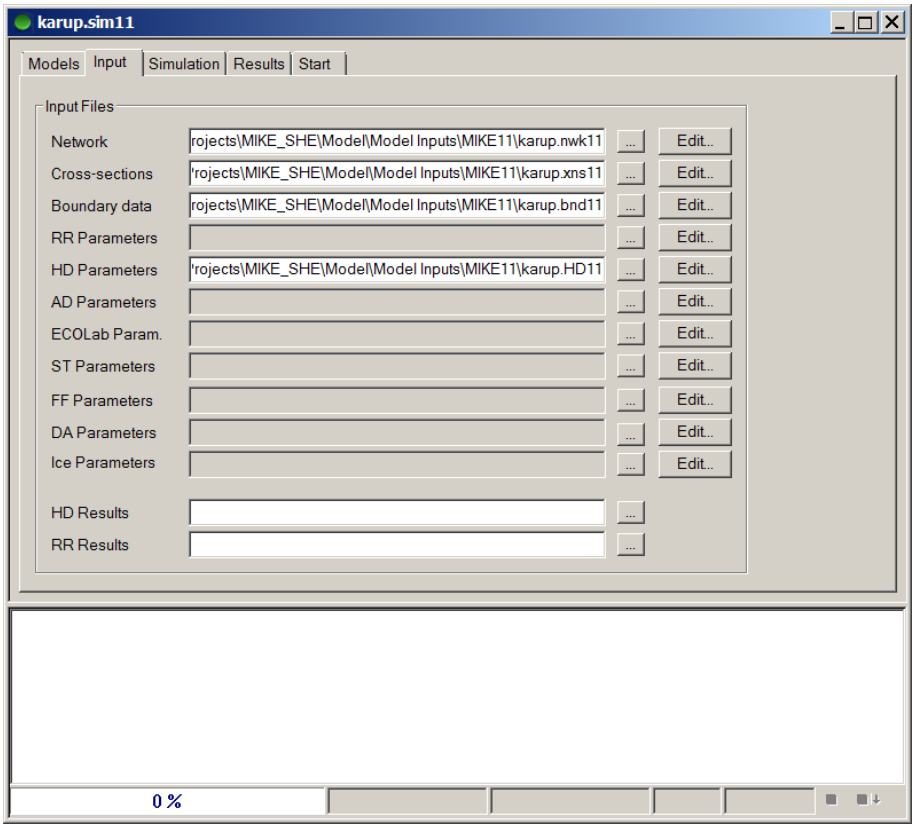
3.17.3 Explore the MIKE 11 simulation document

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 *Models* tab allows you to select the model components.

5. The *Input* 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).



6. The *Simulation* tab allows you to set the time step and simulation length. However, the simulation length is over-ridden by MIKE SHE.

7. The *Results* 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.

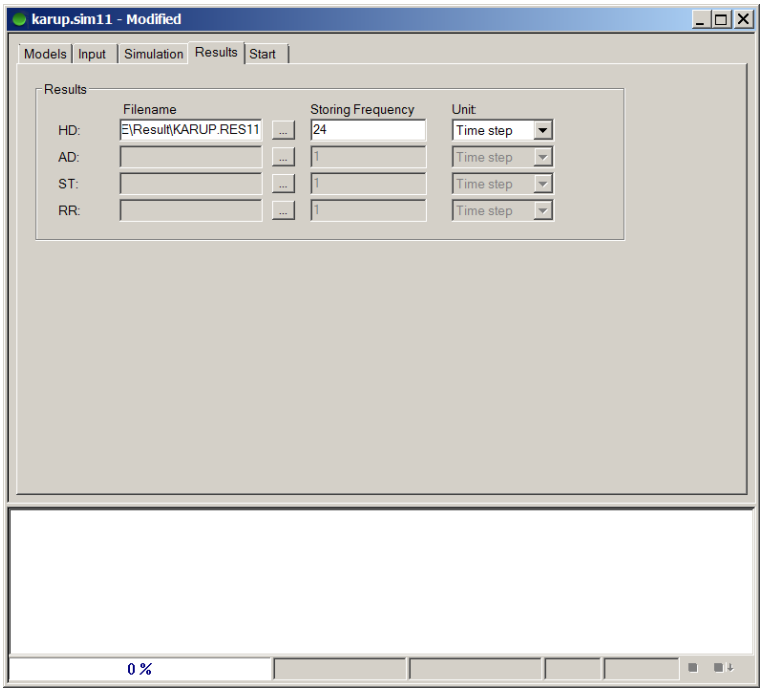
8. The *Start* tab allows you to see if there are any data input errors in your setup and start the simulation when you are ready.

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
- Save your changes.

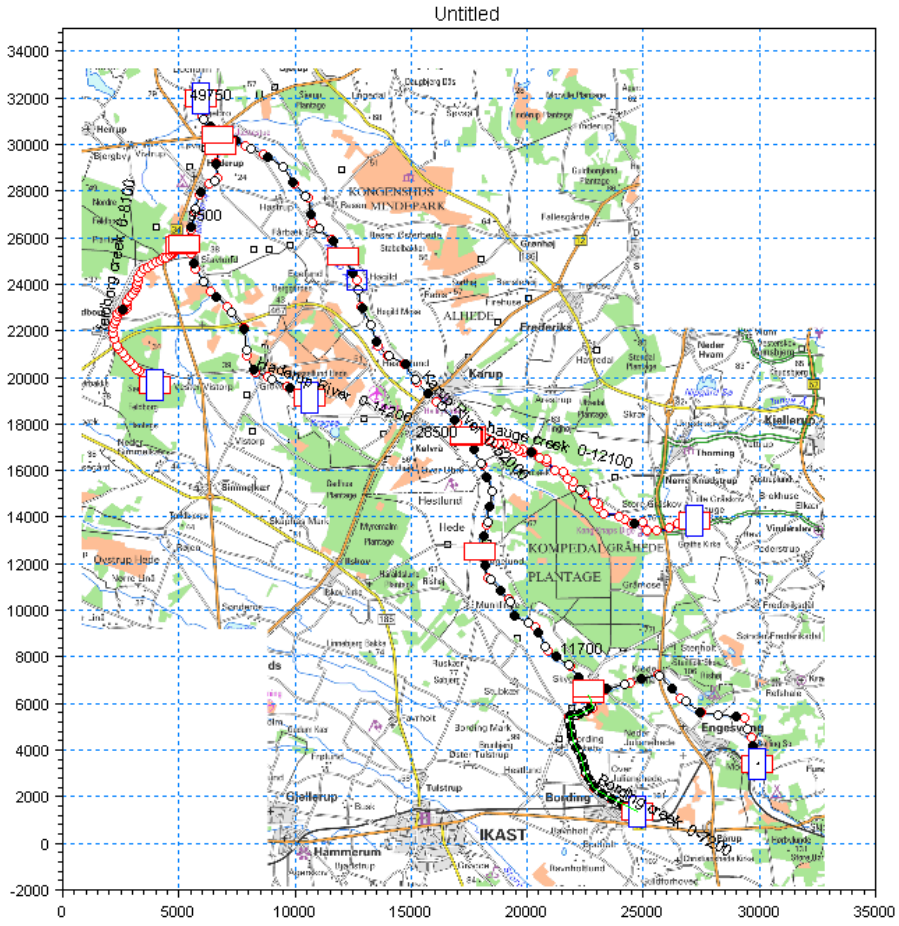


3.17.5 Explore the branch network

On the *Input* tab of the Simulation editor

- Press the

button on the Network line.



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.

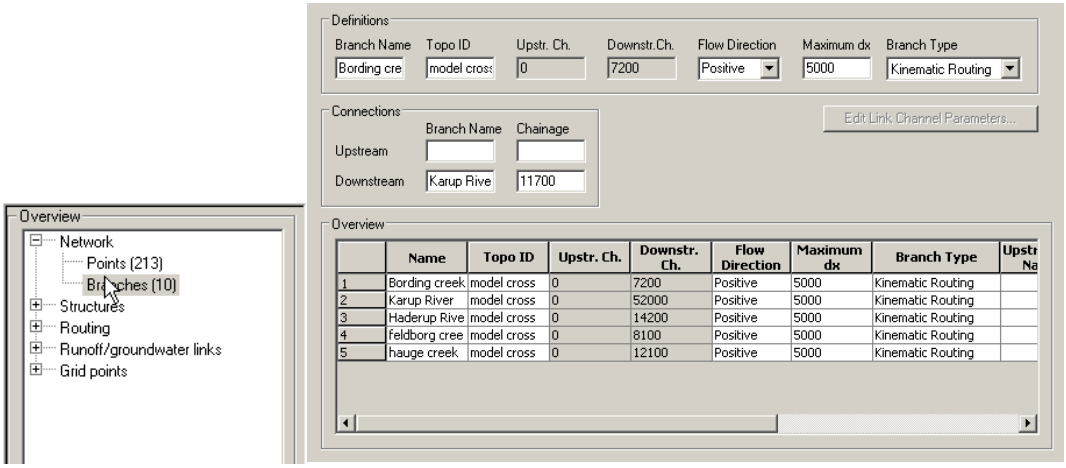
All information in MIKE11 can be accessed and modified from the *network editor*. You are now in the *Network Editor*, which is the core editor of MIKE 11. From here, you can build and modify your river model.

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 *regular* 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.

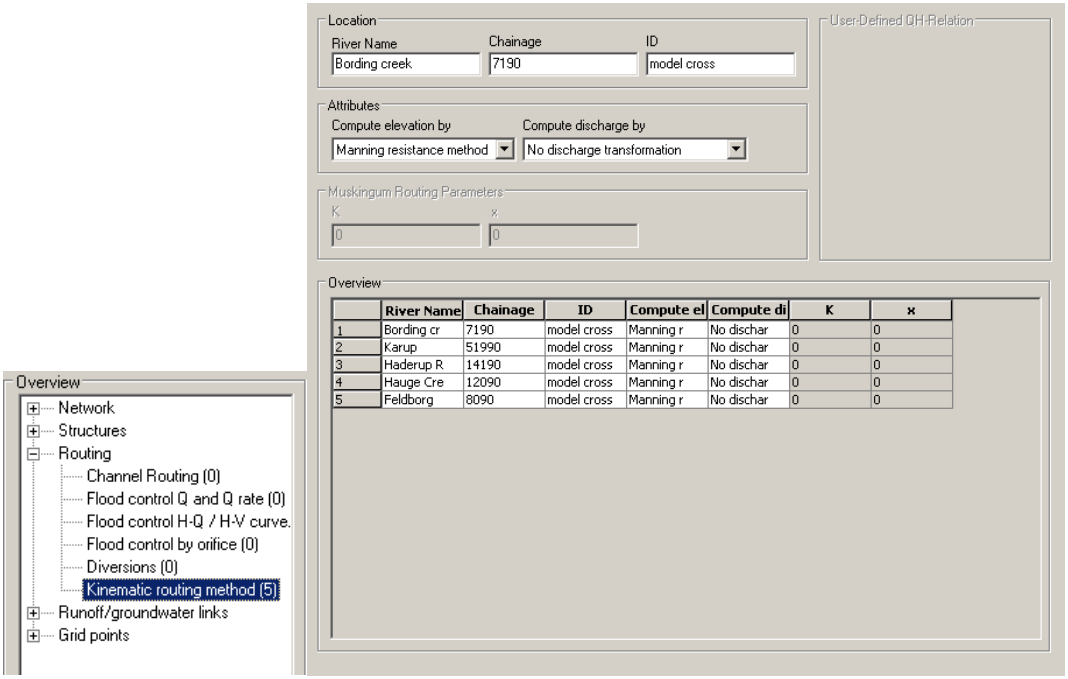
The downside is that the calculated water levels are less precise and hydrodynamic effects are ignored. Water levels are simply calculated based on the amount of water in the river in the current time step.

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.



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.

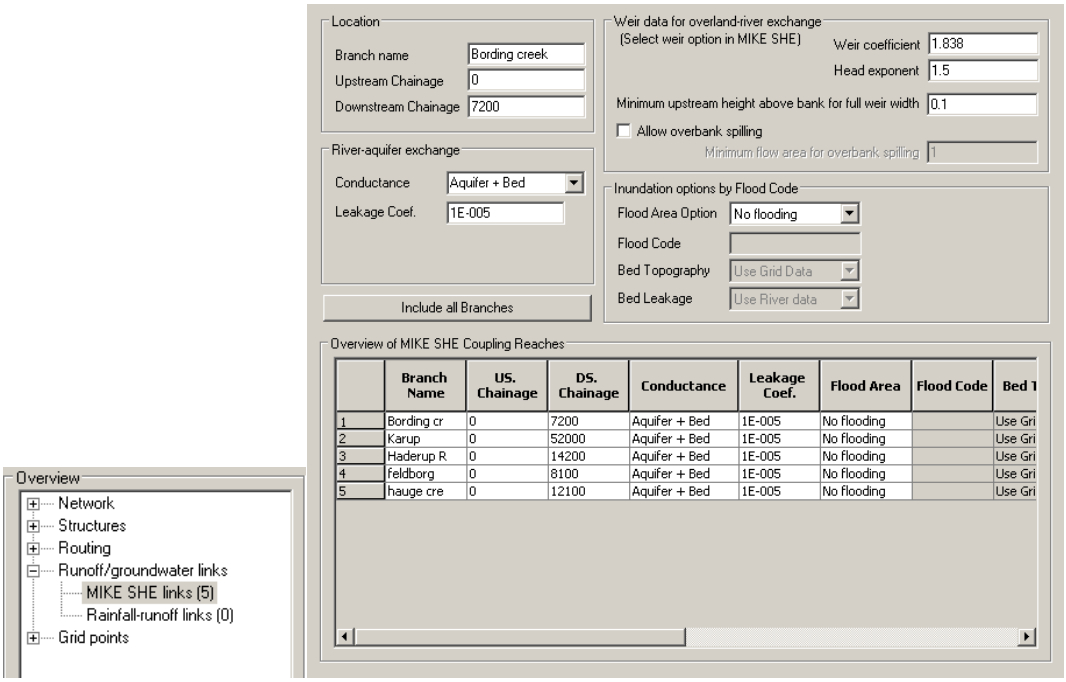
3.17.8 MIKE SHE – MIKE 11 coupling

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 not flow back onto the ground surface if the river floods. However, there are two options for flooding.

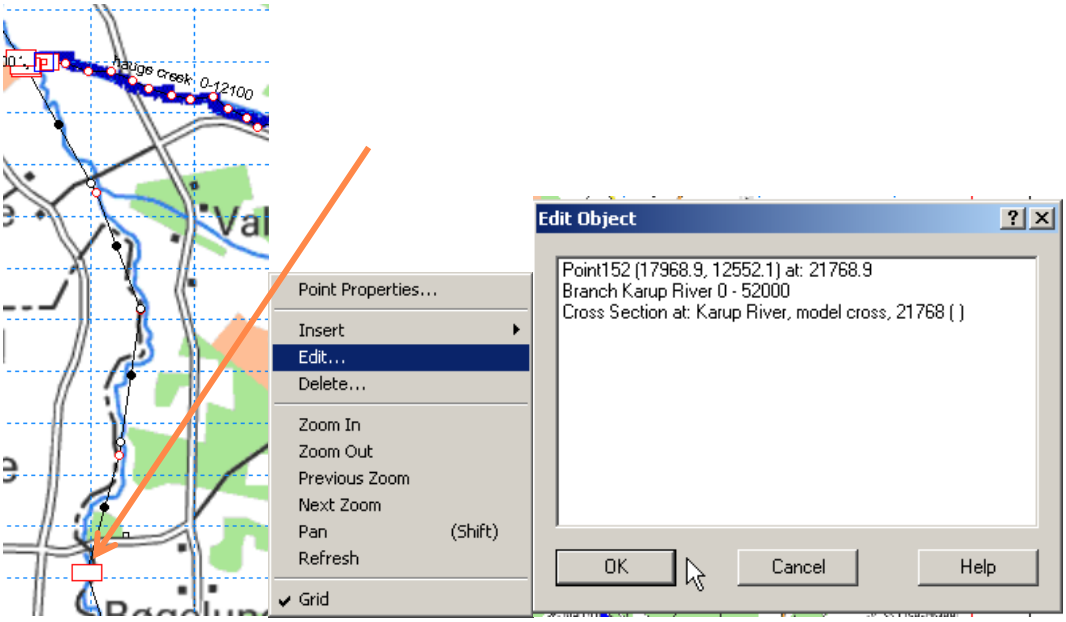
- Cells can be defined by flood codes, where they will be flooded if the elevation of the cell is below the water level in the nearest H-point.
- Alternatively, you can allow overbank spilling. In this case, the river will discharge the water onto the ground surface based on a weir formula.



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.

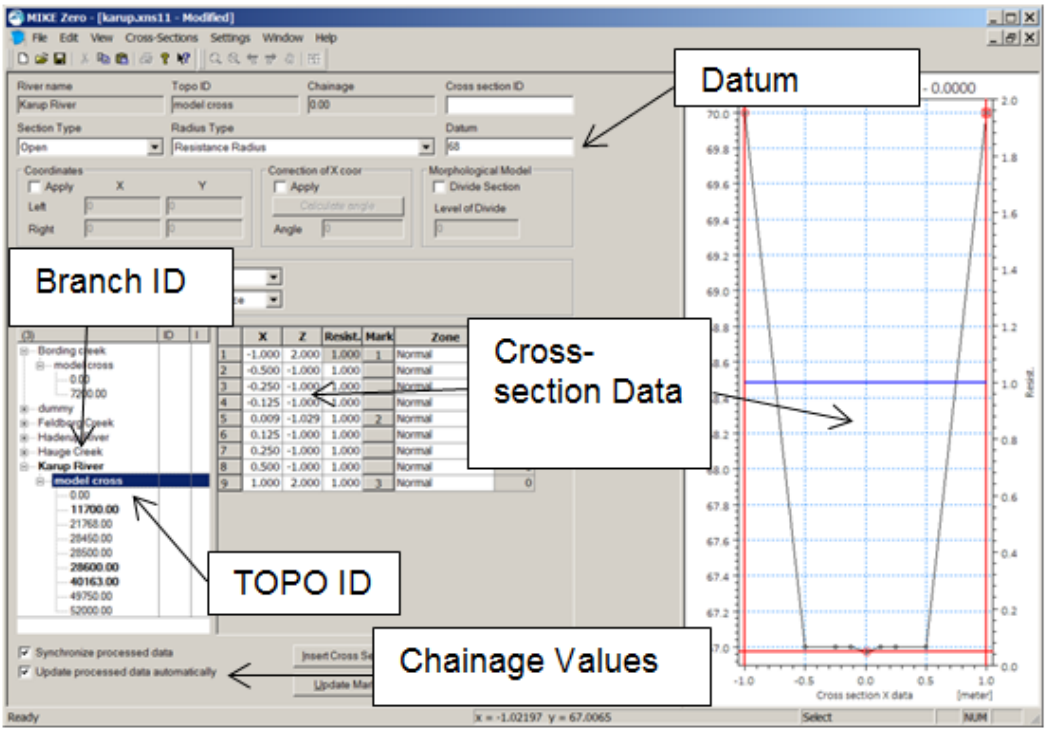


This will take you to the Cross-section Editor. Look at a couple of cross-sections.

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).

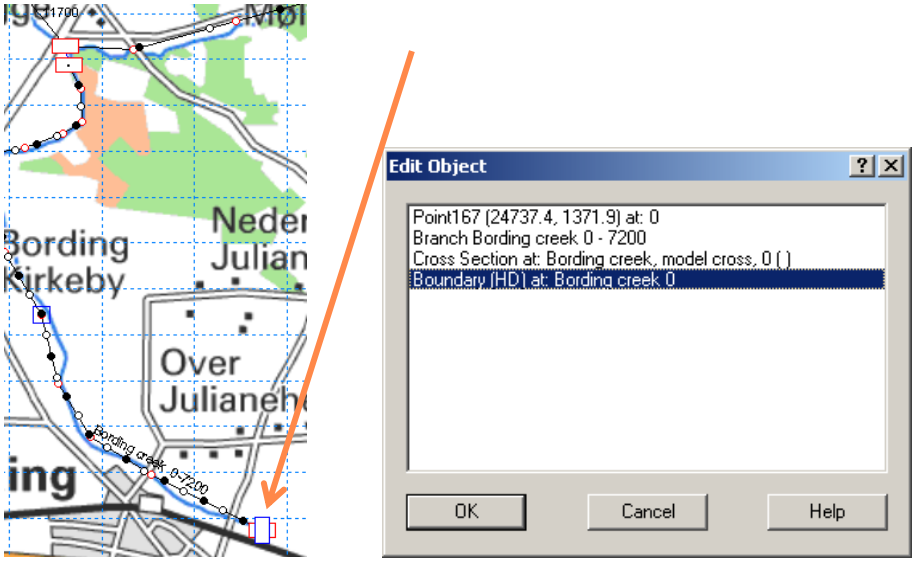
- The *Marker* values 1 and 3 in the table define the extent of the cross-section to be used in the simulation.

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

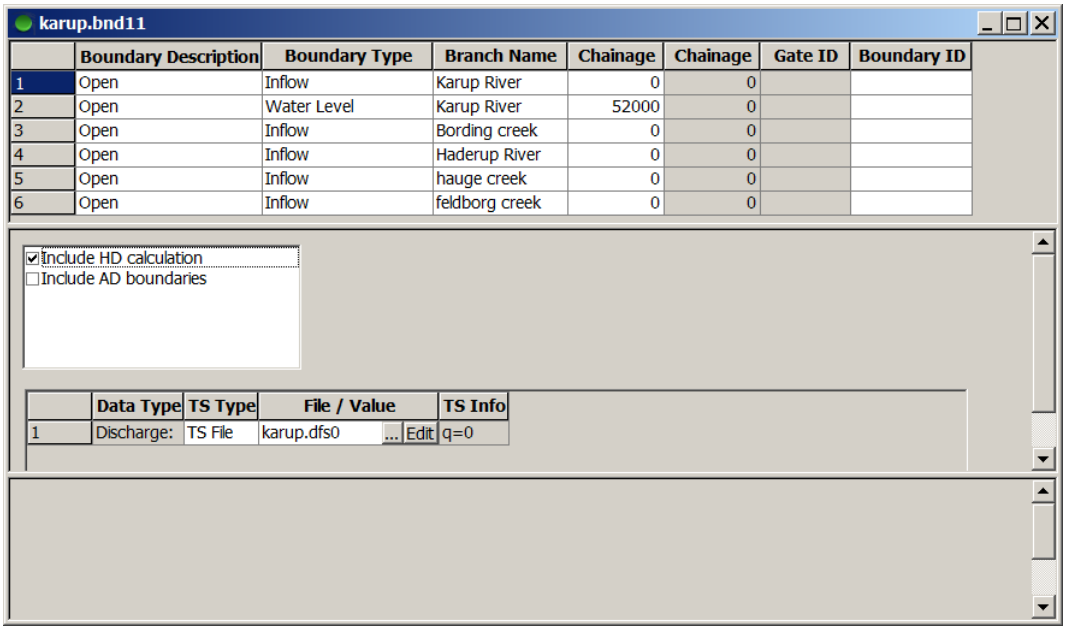


This will open the boundary editor with the current boundary point highlighted.

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.



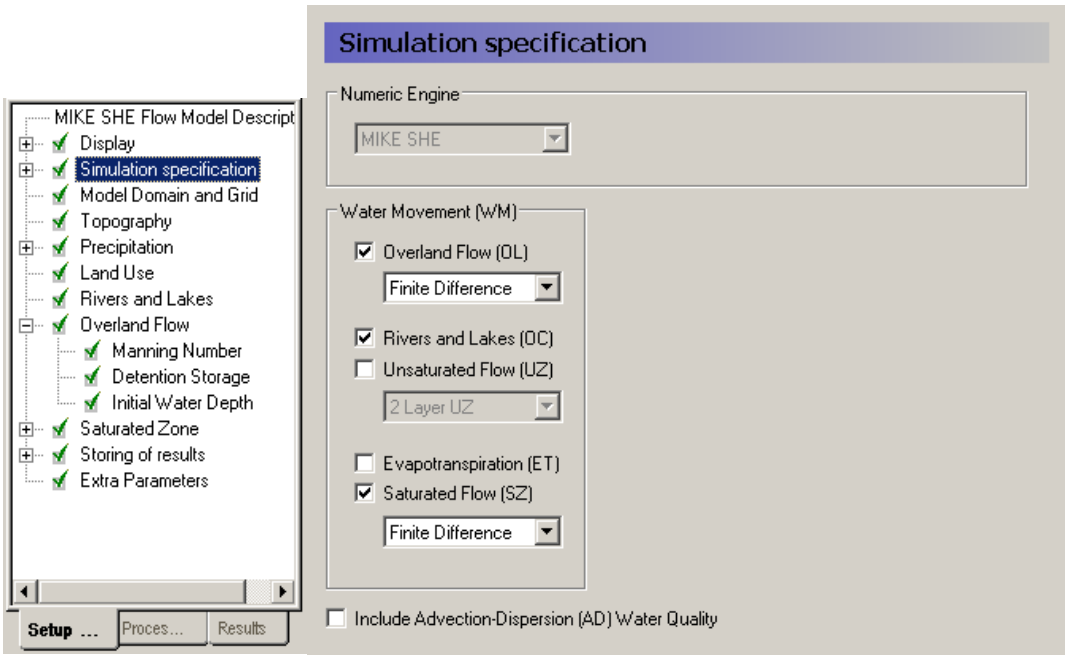
3.18 Step 18 – Add the Overland Flow Module

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.

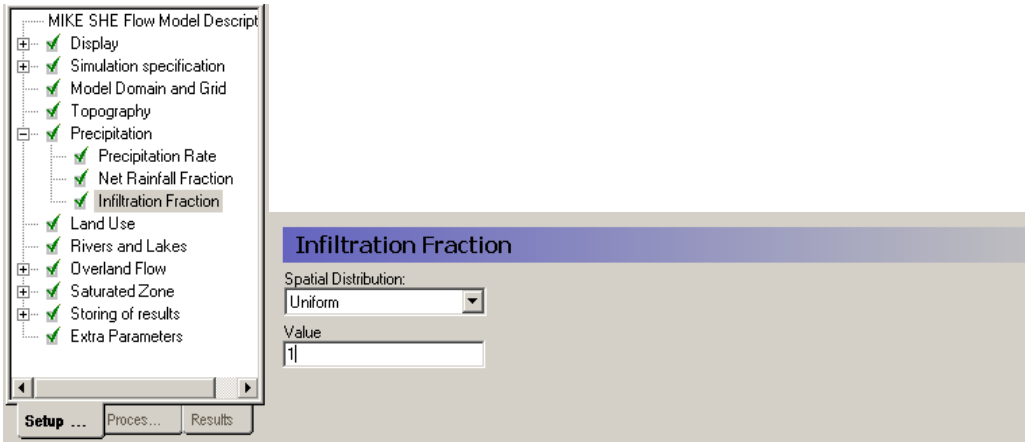


3.18.2 Define infiltration fraction

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.

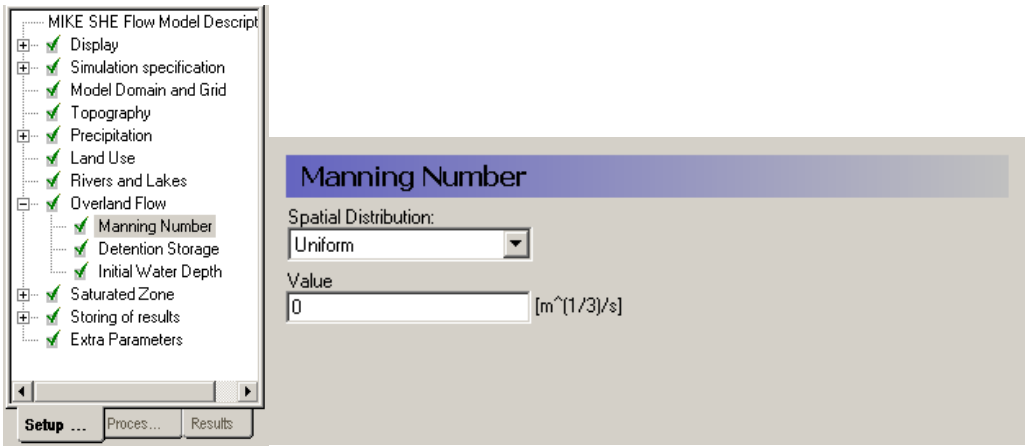


Note This item will disappear when you add the unsaturated zone later in this exercise.

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].



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.

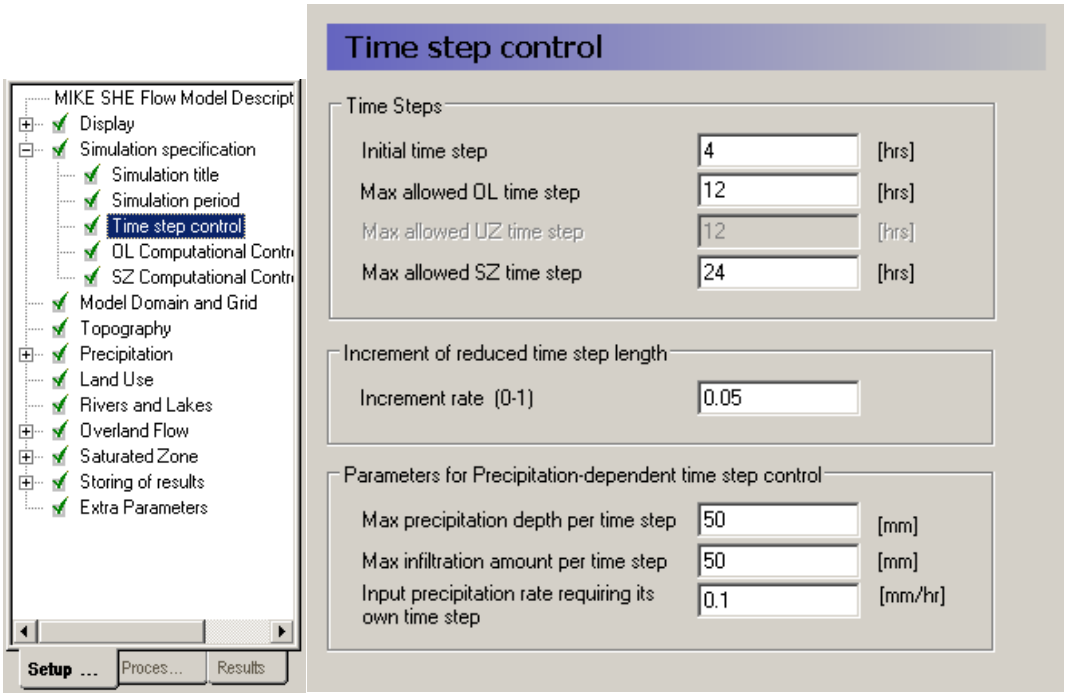
MIKE SHE uses different time-steps for the different hydrologic processes. Normally, the Overland flow time-step is the smallest and the Groundwater time-step the largest. The MIKE 11 time step is controlled in MIKE11.

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.



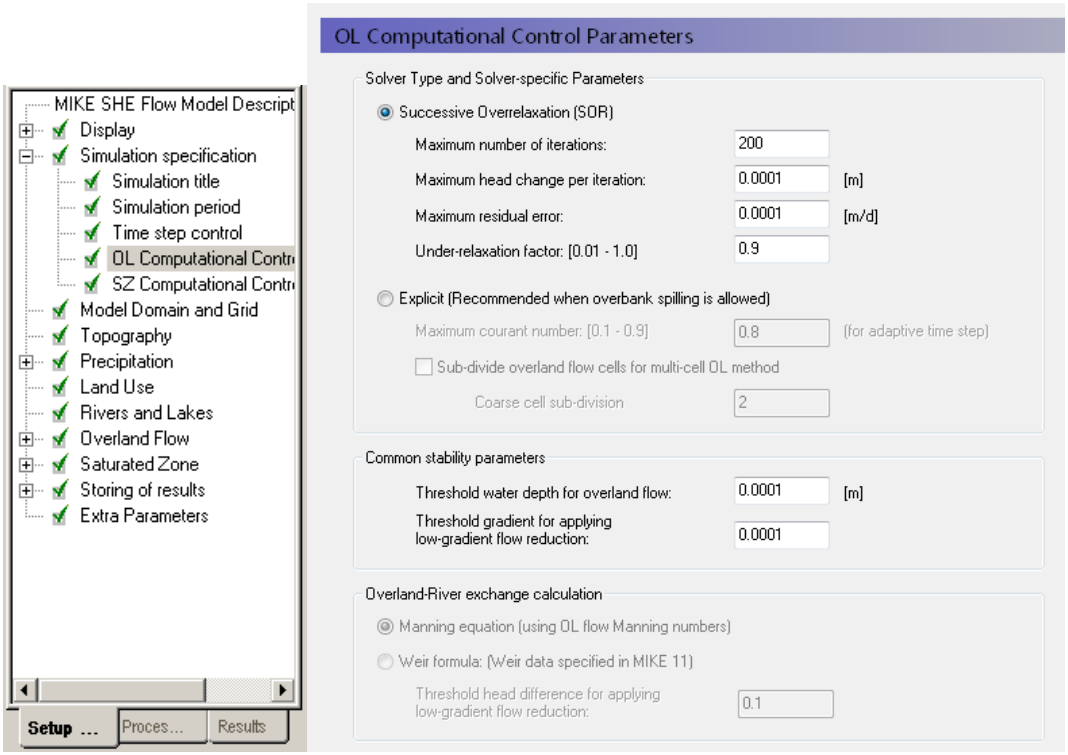
3.19.2 Overland flow parameters

Make sure that you are using

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

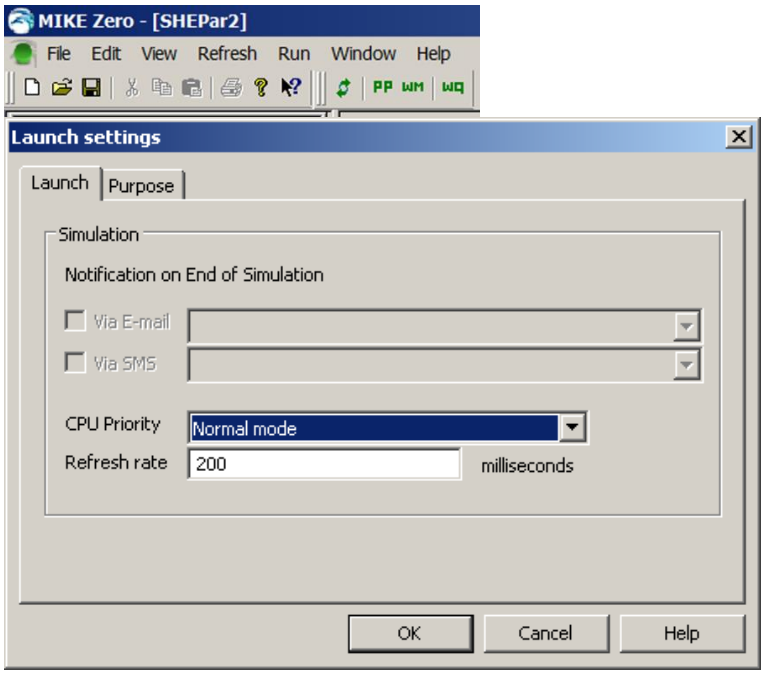
for calculating overland flow.

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.



3.19.3 Run the Preprocessor

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



- Click OK

3.19.4 View the river link setup

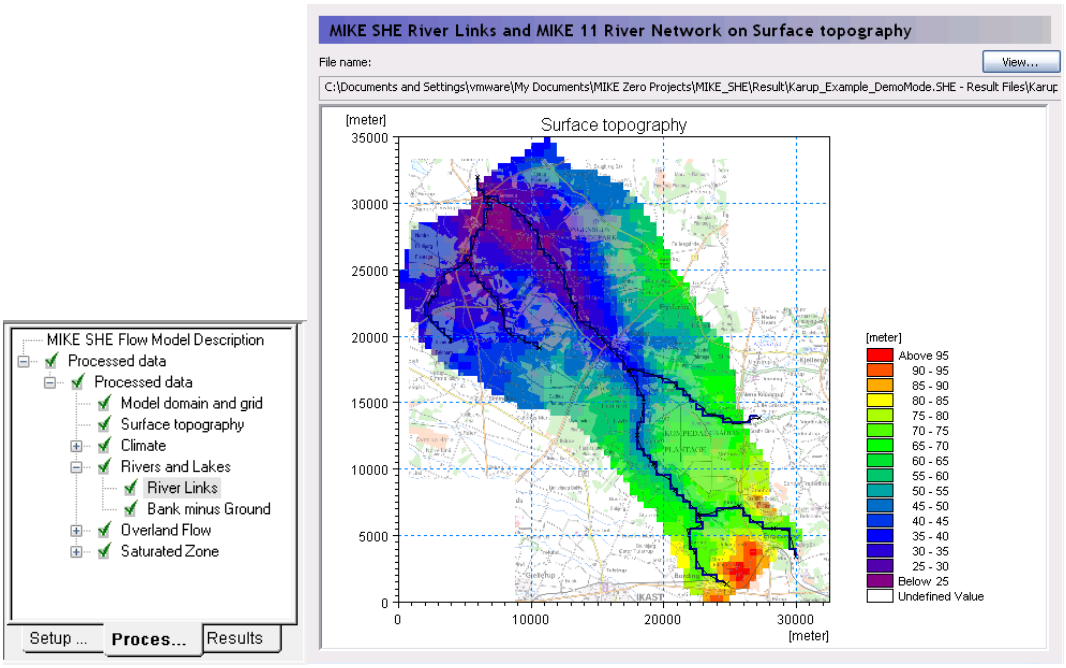
In the *Processed Data Tab* the most recent pre-processed file is automatically loaded

- 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

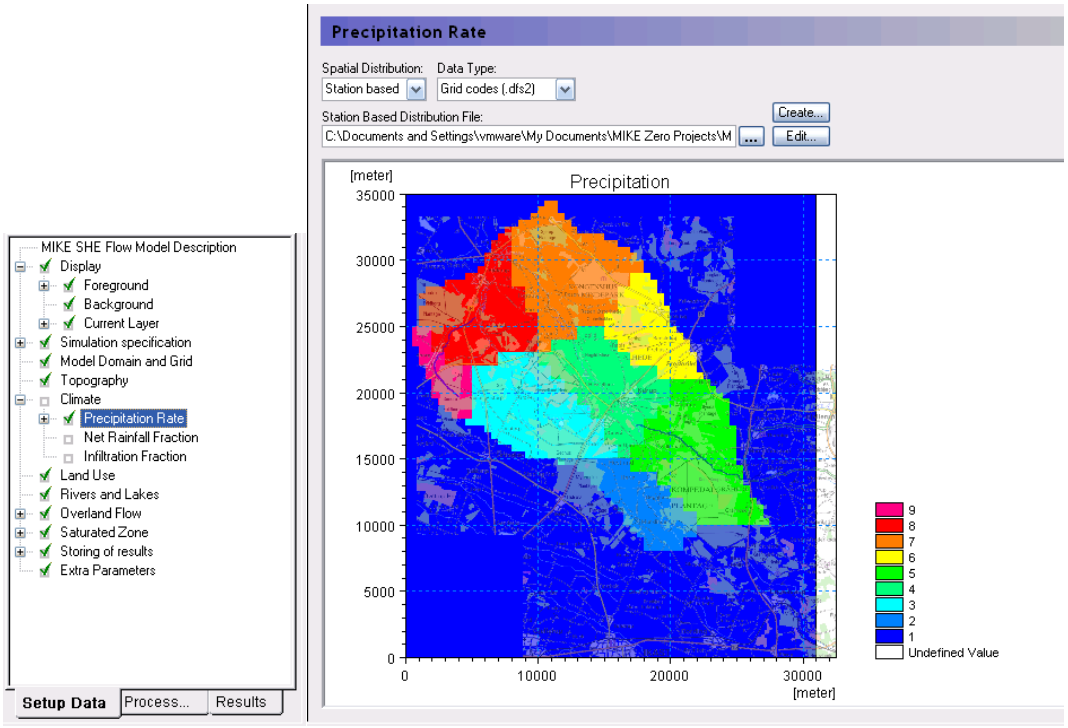
You will now make the simulation a bit more realistic by adding distributed rainfall data.

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)*
- Press and pick the file:
  .\Model Inputs\Maps\Precipitation.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.

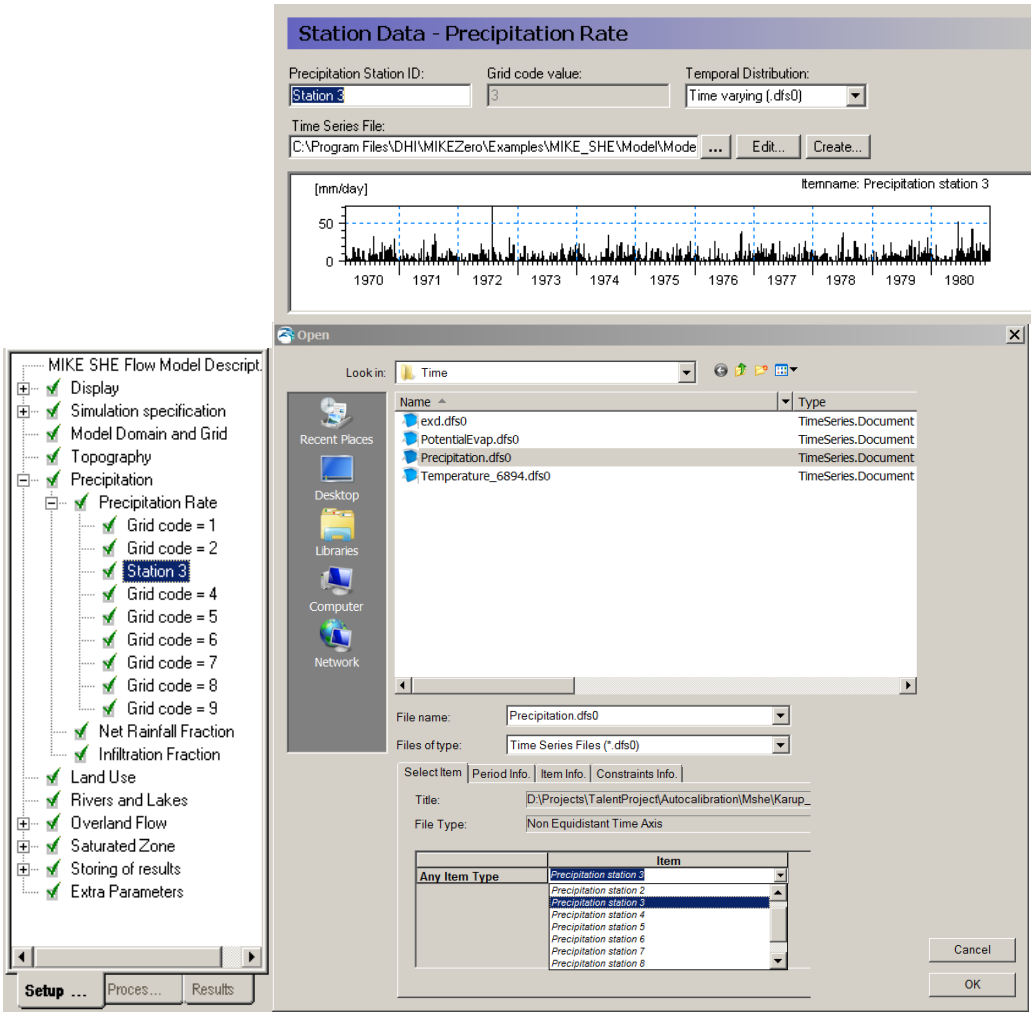
3.21.2 Associate the station areas with time-series data

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.



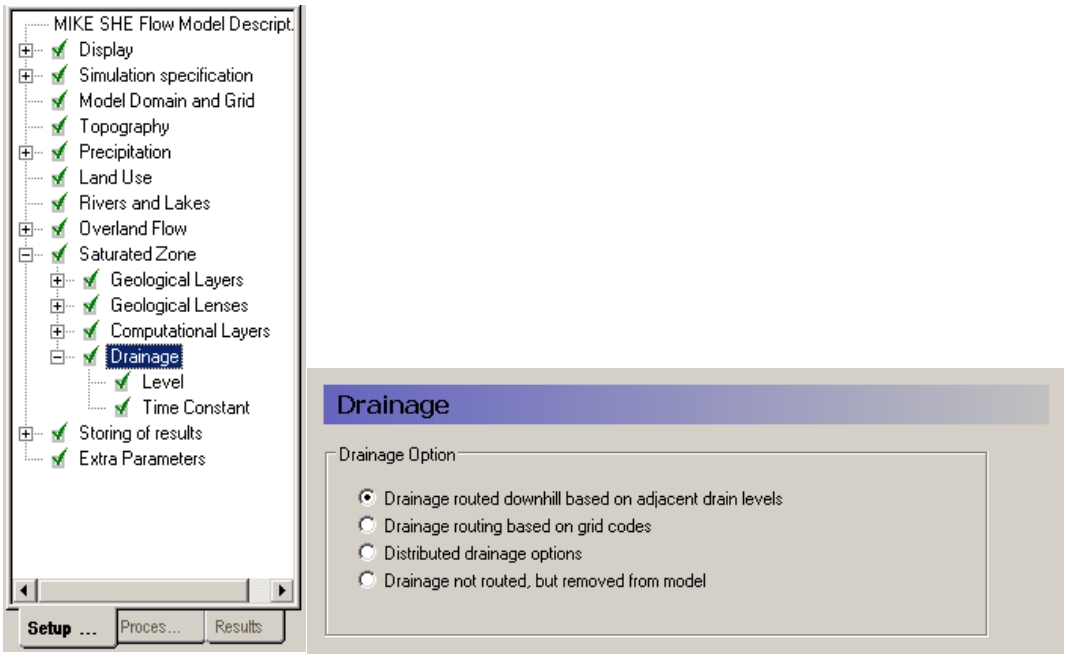
Repeat this for each grid code (station).

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.



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.

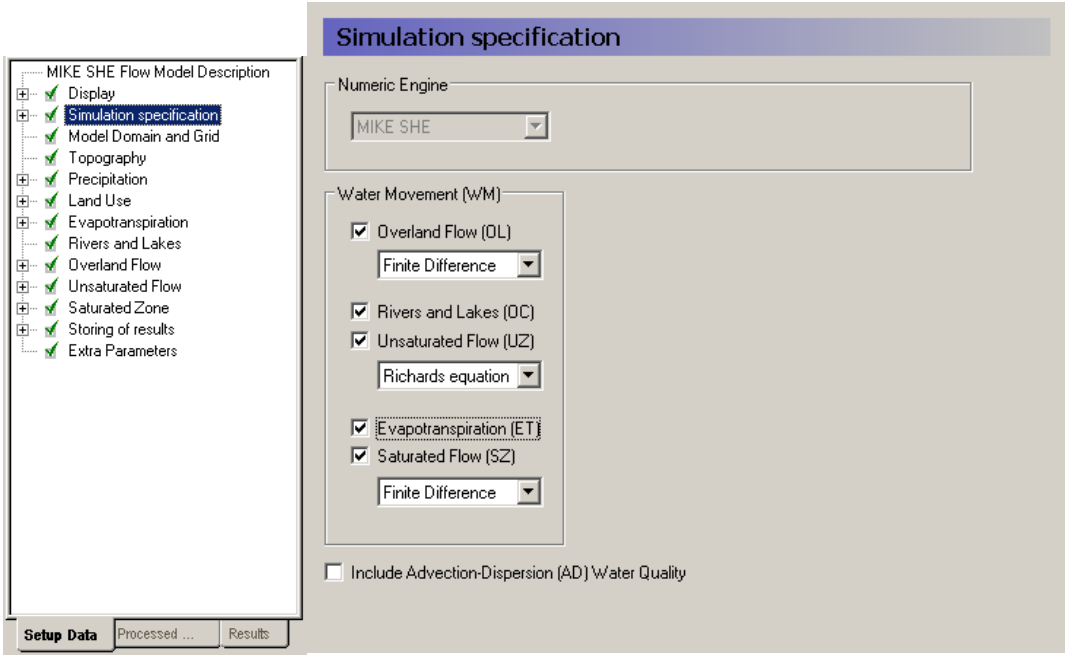
The *Distributed drainage options* allow you to route drainage to specific MIKE 11 nodes or MIKE Urban sewer manholes.

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.

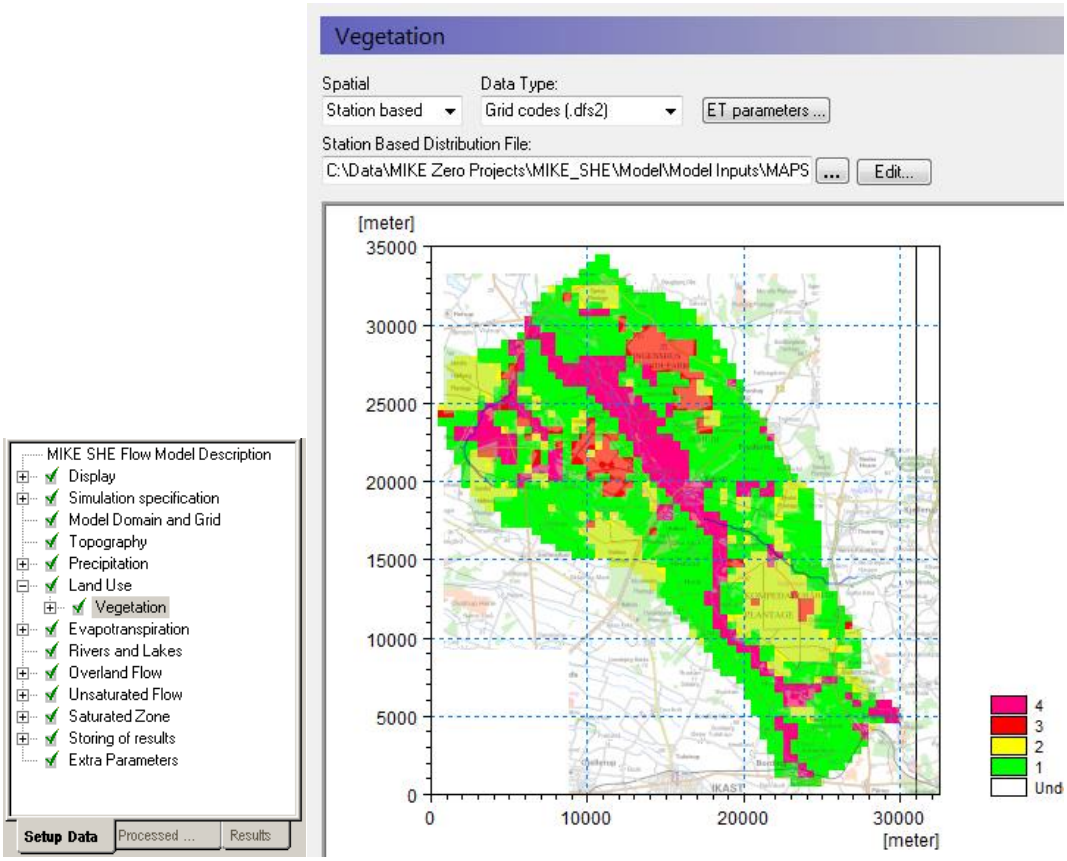


3.23.2 Define vegetation map (field map)

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**

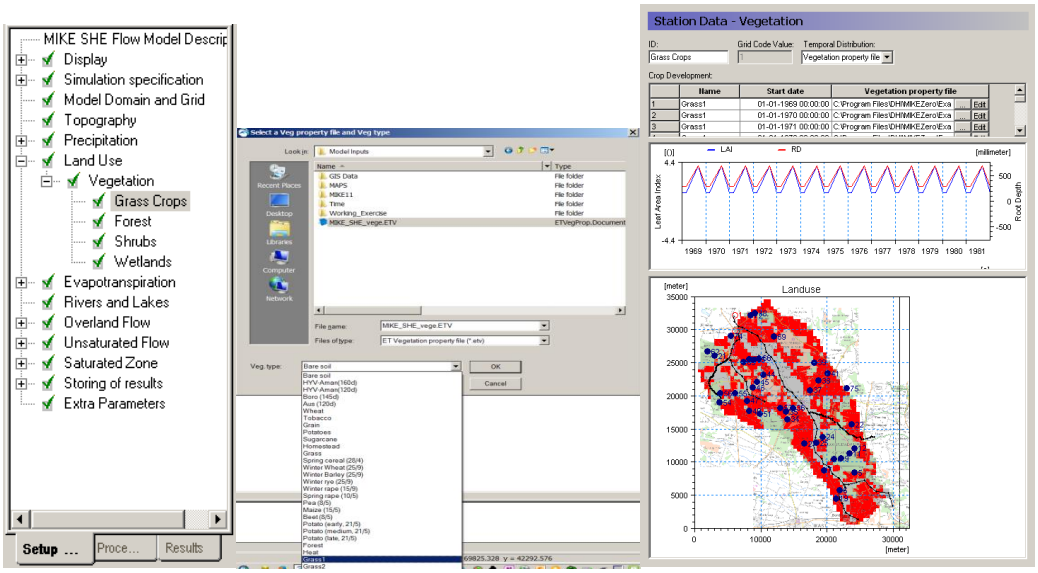
The map now displays your vegetation distribution. The map (dfs2 file) contains 4 different integer codes that define 4 different areas. You now need to define the vegetation types and growth function which are associated with the 4 different areas.



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]*



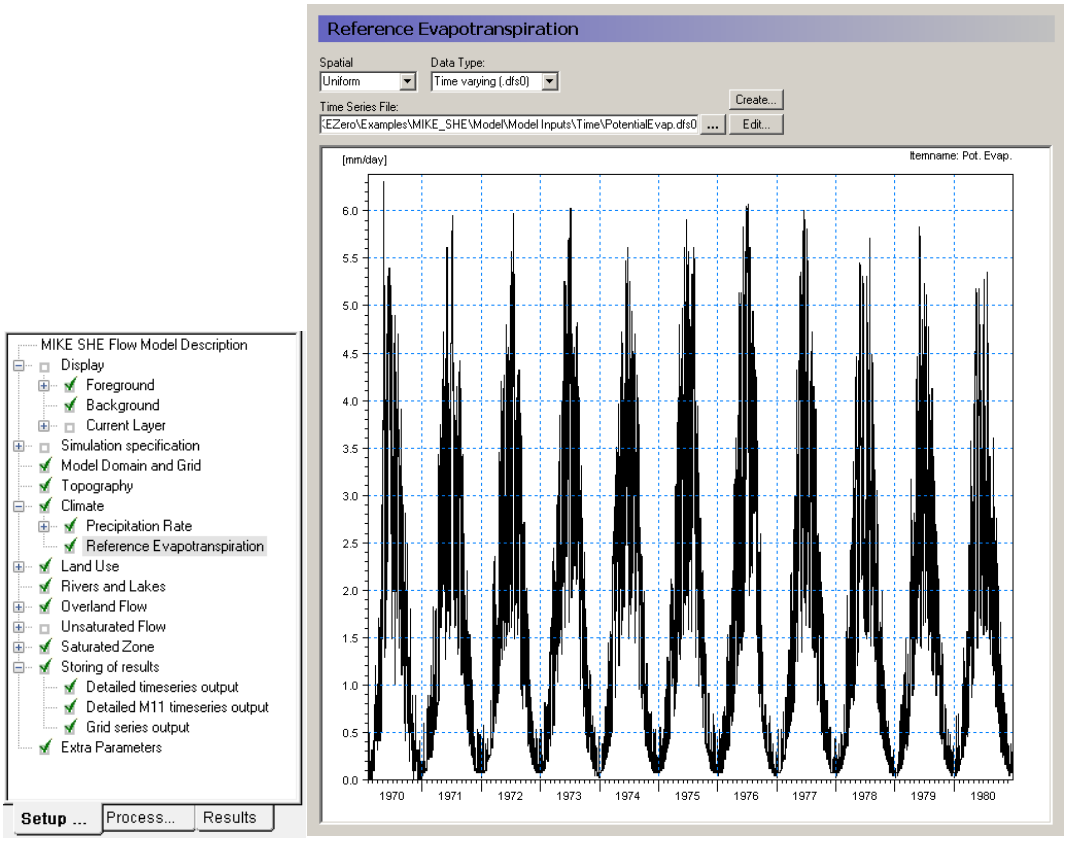
3.24 Step 24 – Add Reference Evapotranspiration

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.

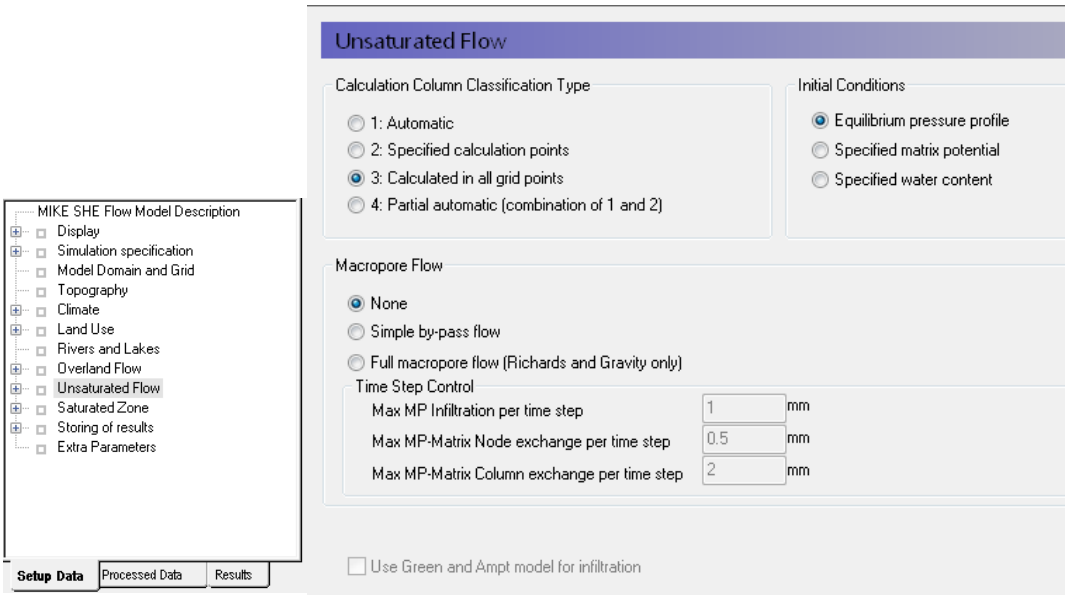
3.25 Step 25 – Build the Unsaturated Zone Model

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



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.

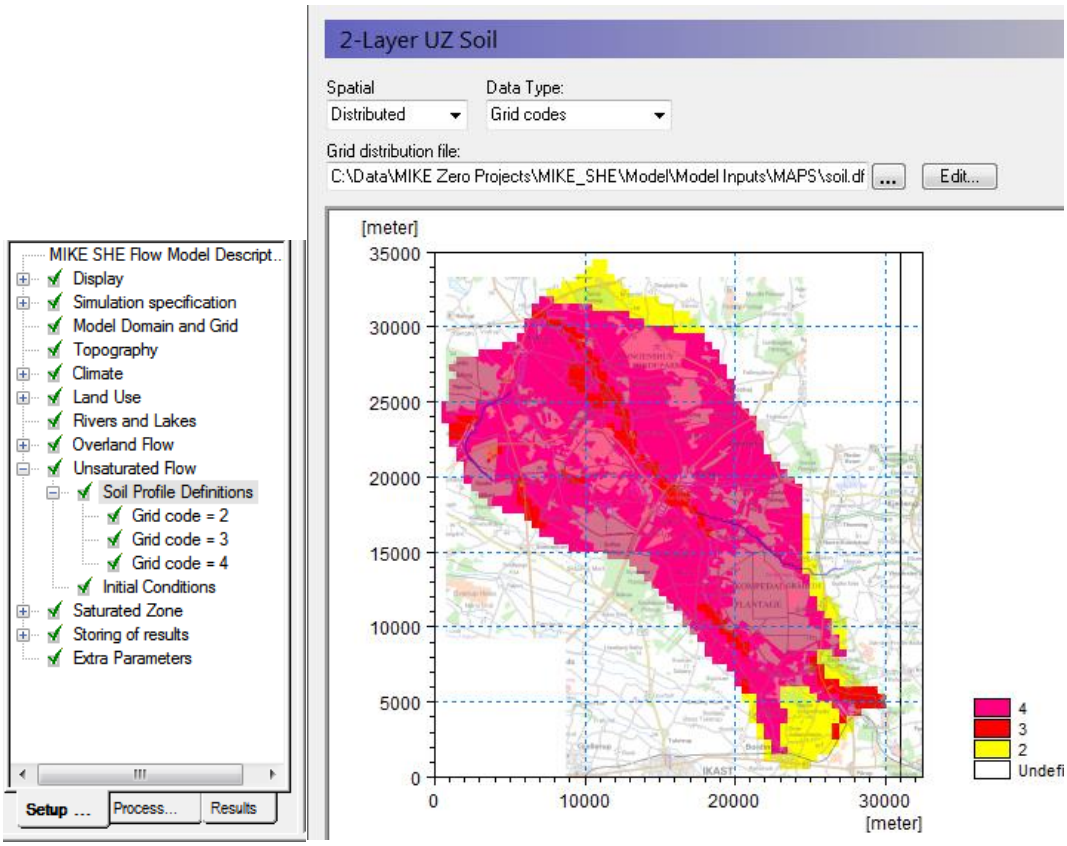
In most models it is probably best to avoid UZ classification.

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***



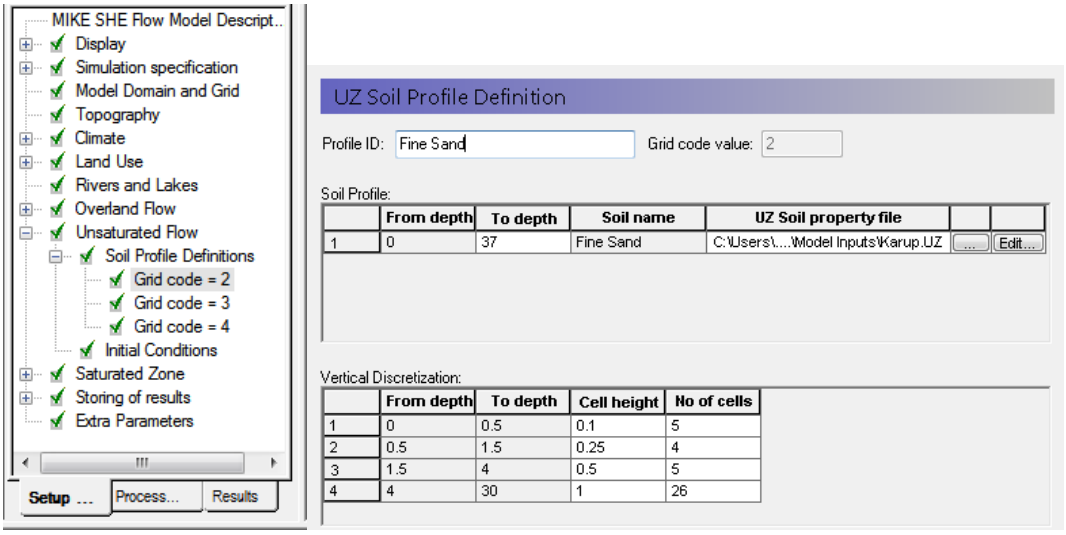
The soil map contains three different code values, which now need to be associated with three soil profiles.

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)



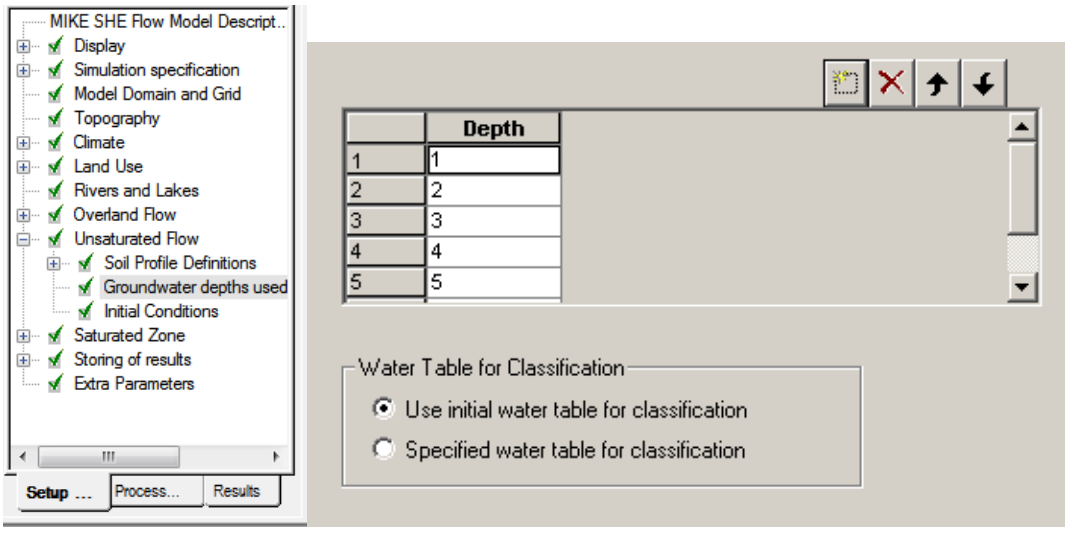
3.25.4 DEMO MODE only: Define groundwater table for UZ classification

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.



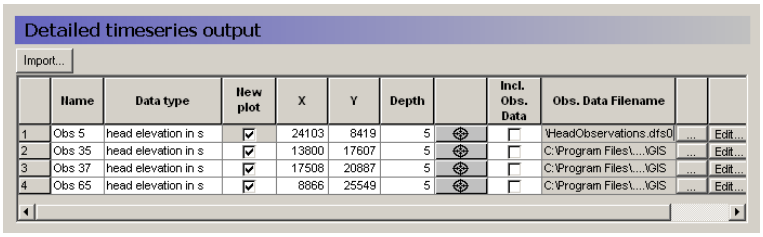
3.26 Step 26 – Run the Simulation

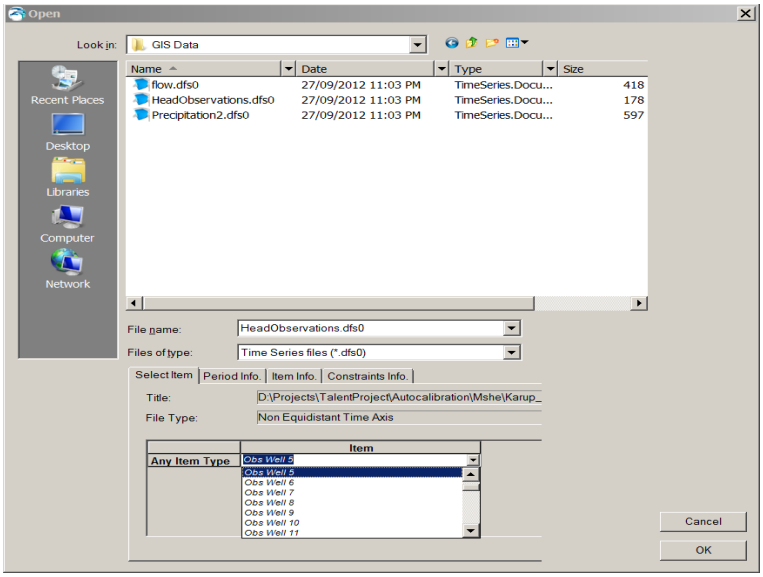
3.26.1 Add field data to the observations

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

To add the observation data, you need to

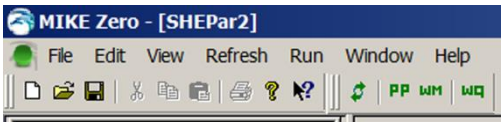
- Choose Data type *head elevation in saturated zone*
- Check on the *Incl. Obs. Data* checkbox
- For each observation point,
  - Use the browse button to select the file **.\Model Inputs\GIS Data\HeadObservations.dfs0**
- In the DFS File Selector dialogue, you need to select the appropriate Item from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





3.26.2 Run the simulation

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



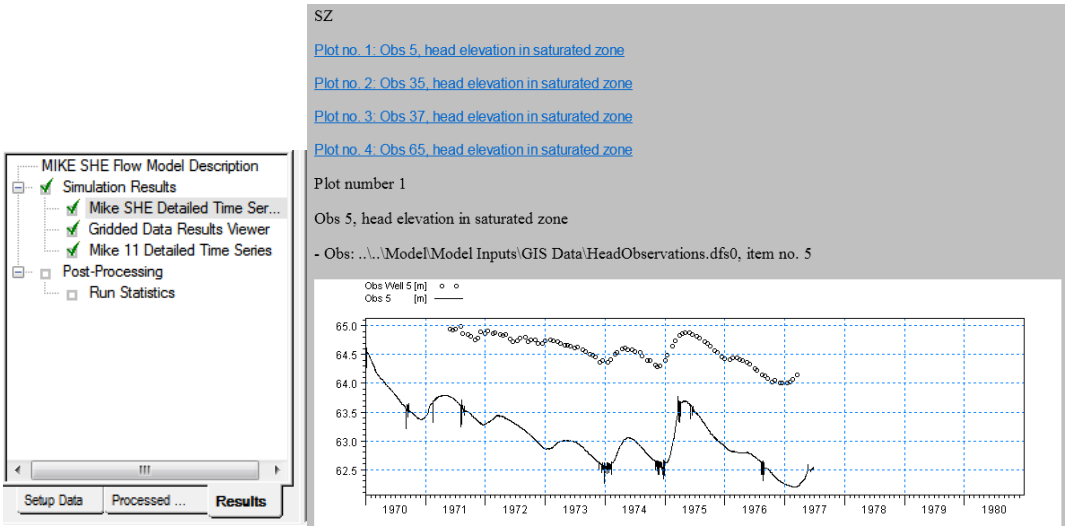
3.27 Step 27 – Evaluate the Results

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.

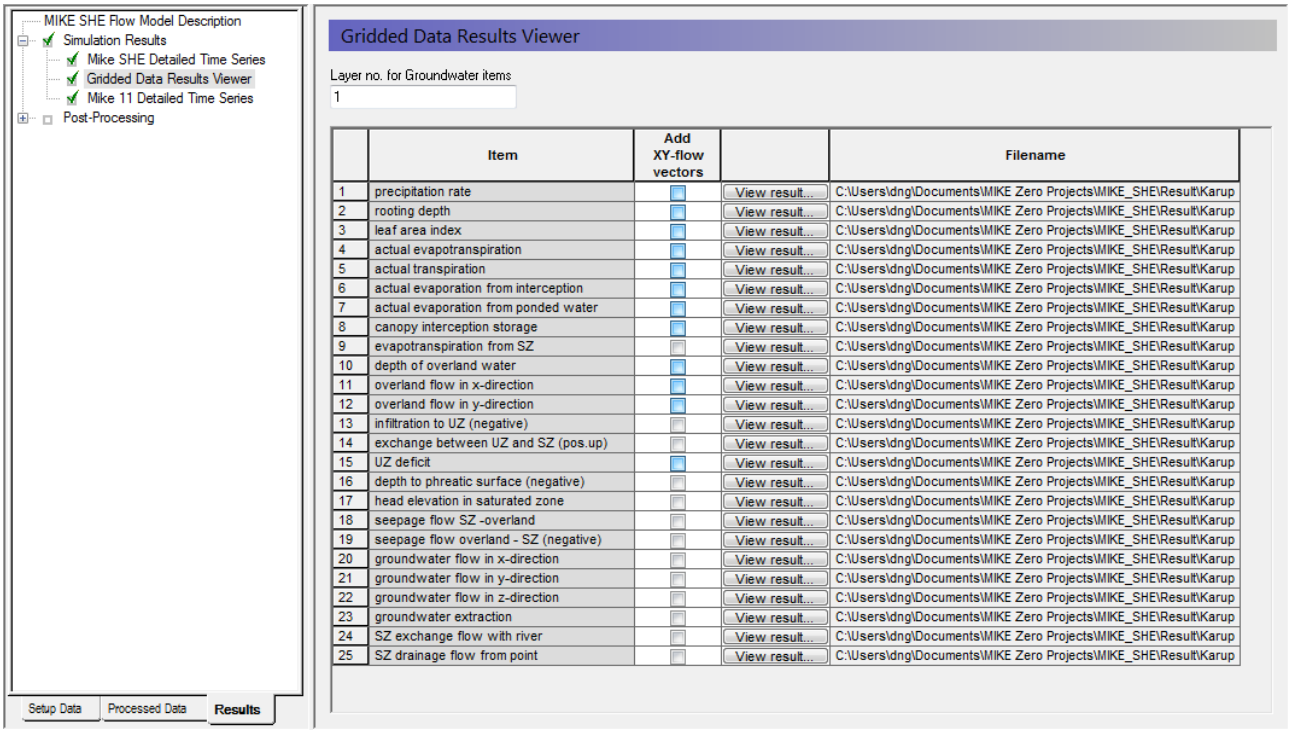
You should explore some of these output items. For example:

- **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.

Explore some of these outputs.



3.27.3 Locate the detailed time series for MIKE 11

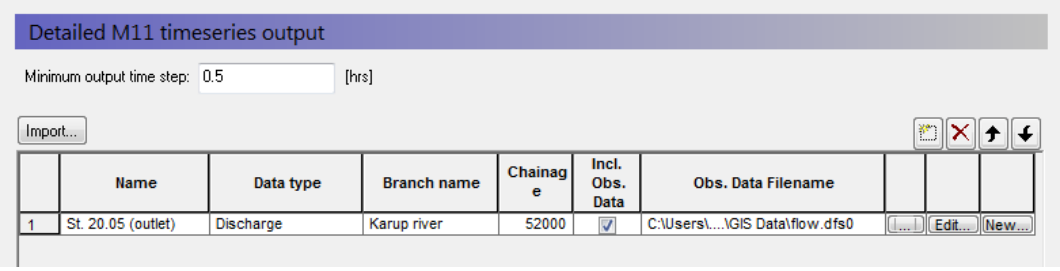
You have not yet specified detailed output for MIKE 11. So, you cannot look at this yet. But, after successfully running the model, you can go back and add detailed time series locations for MIKE 11 outputs.

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



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.