MIKE SHE USER MANUAL

VOLUME 1: USER GUIDE

Please Note

Copyright

Limited Liability

Printing History

December 2011
July 2011
September 2012

GETTING STARTED

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 baseflow. 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.

In 1969, Freeze and Harlan (Freeze and Harlan, 1969) proposed a blueprint for modelling the hydrologic cycle. In this original blueprint, different flow processes were described by their governing partial differential equations. The equations used in the blueprint were known to represent the physical processes at the appropriate scales in the different parts of the hydrological cycle.

From 1977 onwards, a consortium of three European organizations developed, and extensively applied, the Système Hydrologique Européen (SHE) based on the blueprint of Freeze and Harlan (Abbott et al., 1986a & b). The integrated hydrological modelling system, MIKE SHE, emerged from this work (see Figure 1.1)

Since the mid-1980's, MIKE SHE has been further developed and extended by DHI Water & Environment. Today, 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, (Butts et al. 2004). 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.

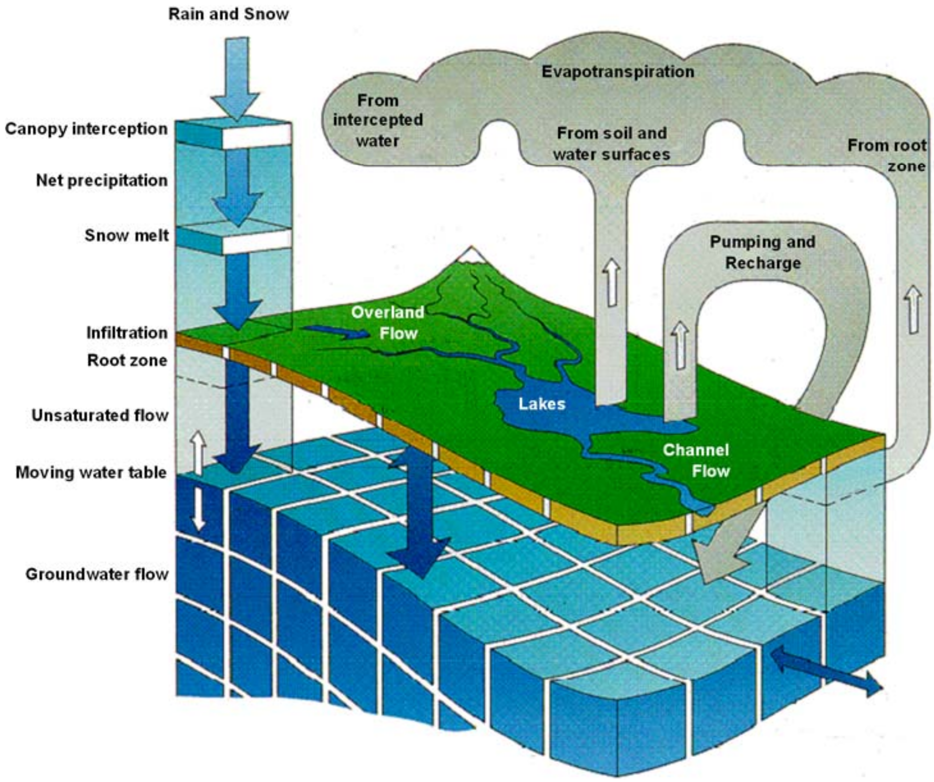


Figure 1.1 Hydrologic processes simulated by MIKE SHE

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 networks 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 km^2 Senegal Basin (e.g. Andersen et al., 2001). MIKE SHE has proven valuable in hundreds of research and consultancy projects covering a wide range of climatological and hydrological regimes, many of which are referenced in Graham and Butts (2006).

The need for fully integrated surface and groundwater models, like MIKE SHE, has been highlighted by several recent studies (e.g. Camp Dresser &

McKee Inc., 2001; Kaiser-Hill, 2001; West Consultants Inc. et al., 2001; Kimbley-Horn & Assoc. Inc. et al., 2002; Middlemis, 2004, which can all 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 (Kaiser-Hill, 2001).

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 organizations ranging from universities and research centres to consulting engineers companies (Refsgaard & Storm, 1995). 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 optimization
- 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.)

Graham and Butts (2006) contains a list of some easily accessible references for many of the application areas listed above.

User interface

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

1 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
   - Customized input/output units to support local needs

2 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.

3 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.

1.1 Process models

MIKE SHE, in its original formulation, could be characterized as a deterministic, physics-based, distributed model code. It was developed as a fully integrated alternative to the more traditional lumped, conceptual rainfall-runoff models. A physics-based code is one that solves the partial differential equations describing mass flow and momentum transfer. The parameters in these equations can be obtained from measurements and used in the model. For example, the St. Venant equations (open channel flow) and the Darcy equation (saturated flow in porous media) are physics-based equations.

There are, however, important limitations to the applicability of such physics-based models. For example,

- it is widely recognized that such models require a significant amount of data and the cost of data acquisition may be high;
- the relative complexity of the physics-based solution requires substantial execution time;
- the relative complexity may lead to over-parameterised descriptions for simple applications; and
- a physics-based model attempts to represent flow processes at the grid scale with mathematical descriptions that, at best, are valid for small-scale experimental conditions.

Therefore, it is often practical to use simplified process descriptions. Similarly, in most watershed problems one or two hydrologic processes dominate the watershed behaviour. For example, flood forecasting is dominated by river flows and surface runoff, while wetland restoration depends mostly on saturated groundwater flow and overland flow. Thus, a complete, physics-based flow description for all processes in one model is rarely necessary. A sensible way forward is to use physics-based flow descriptions for only the processes that are important, and simpler, faster, less data demanding methods for the less important processes. The downside is that the parameters in the simpler methods are usually no longer physics meaningful, but must be calibrated-based on experience.

The process-based, modular approach implemented in the original SHE code has made it possible to implement multiple descriptions for each of the hydrologic processes. In the simplest case, MIKE SHE can use fully distributed conceptual approaches to model the watershed processes. For advanced applications, MIKE SHE can simulate all the processes using physics-based methods. Alternatively, MIKE SHE can combine conceptual and physics-based methods-based on data availability and project needs. The flexibility in MIKE SHE's process-based framework allows each process to be solved at its own relevant spatial and temporal scale. For example, evapotranspiration varies over the day and surface flows respond quickly to rainfall events, whereas groundwater reacts much slower. In contrast, in many non-commercial, research-oriented integrated hydrologic codes (e.g. MODFLOW HMS, Panday et al., 1998; InHM, Sudicky et al., 2002), all the hydrologic processes are solved implicitly at a uniform time step, which can lead to intensive computational effort for watershed scale models.

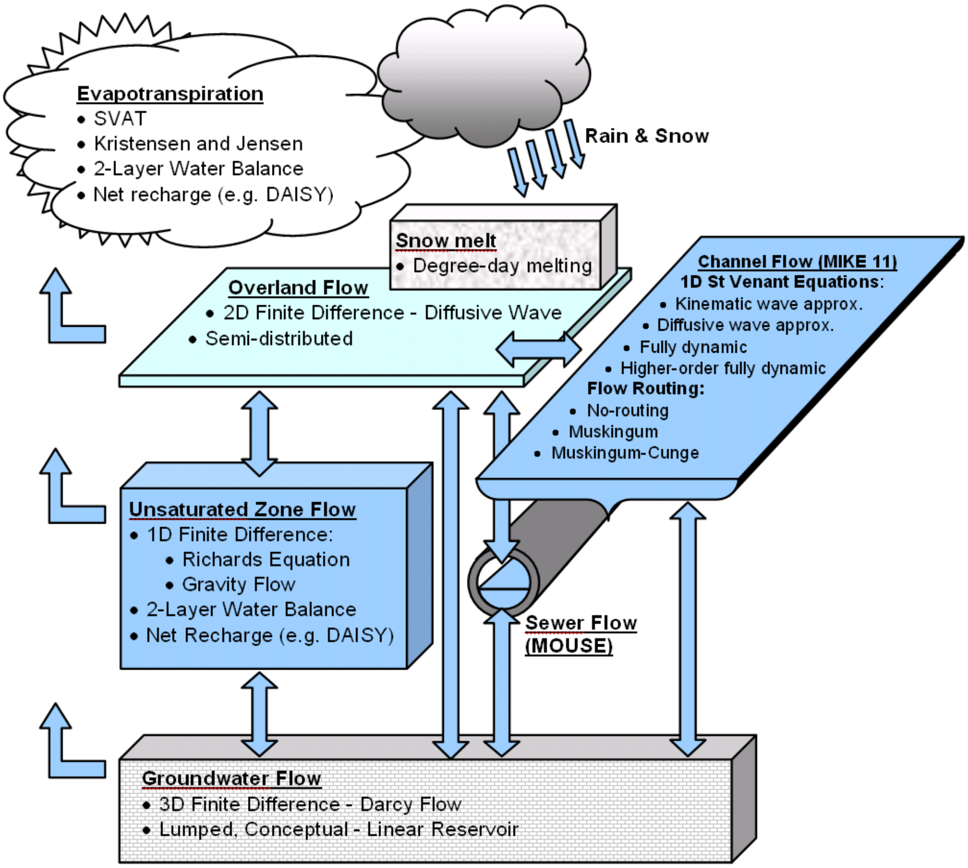


Figure 1.2 Schematic view of the process in MIKE SHE, including the available numeric engines for each process. The arrows show the available exchange pathways for water between the process models. Note: the SVAT evapotranspiration model is not yet available in the commercial version of MIKE SHE.

1.2 Requirements

The requirements to build and run a MIKE SHE model depend on the purpose of the model and the trade-offs that must be made between conceptualization and the practicality of simulation time.

1.2.1 Input requirements

The flexibility of MIKE SHE means that there is no predefined list of required input data. The required data depends on the hydrologic process included and the process model selected, which, in turn, depend on what problem you are trying to solve with MIKE SHE. However, the following basic model parameters are required for nearly every MIKE SHE model:

- Model extent - typically as a polygon,
- Topography - as point or gridded data, and
- Precipitation - as station data (rain gauge data).

Additional basic data is required depending on the hydrologic processes included, and their options:

- Reference evapotranspiration - as station data or calculated from meteorological data,
- Air Temperature - for calculating snowmelt (station data),
- Solar Radiation - for calculating snowmelt (station data)
- Sub-catchment delineation - for runoff distribution
- River morphology (geometry + cross-sections) - for river flow and water level calculations
- Land use distribution - for vegetation and paved runoff calculations
- Soil distribution - for distributing infiltration and calculating runoff
- Subsurface geology - for calculating groundwater flow

If you also want to calculate water quality then additional basic information includes:

- Species to be simulated, and
- Source locations

The data items listed above are the basic input data that define your problem. They are not usually part of the calibration. If we now look at each of the hydrologic processes, and the process models available for each, then we can separate out the principle calibration parameters.

Table 1.1 Principle parameters for MIKE SHE

	Principle calibration parameters	Other parameters
Overland flow (finite difference)	Surface roughness	Detention storage
Overland flow (subcatchment-based)	Surface roughness	Detention storage Slope parameters
River flow	River bed roughness River bed leakage coefficient	
Unsaturated flow (finite difference)	Saturated hydraulic conductivity	Soil water contents at saturation, field capacity, and wilting point Soil pedotransfer function parameters
Unsaturated flow (2-layer method)	Saturated hydraulic conductivity	Soil water contents at saturation, field capacity, and wilting point Capillary thickness
Actual Evapotranspiration	Leaf Area Index Root depth	Canopy Interception FAO Crop coefficient Kristensen and Jensen ET parameters
Groundwater flow (finite difference)	Hydraulic conductivity Specific yield Specific storage	Drain level Drain time constants

Table 1.1 Principle parameters for MIKE SHE

	Principle calibration parameters	Other parameters
Groundwater flow (linear reservoir)	Reservoir time constants Reservoir volumes (specific yield, depths)	Interbasin transfers (dead zone storage)
Water quality	Porosity Soil bulk density Dispersivities Sorption and degradation rate constants	Source strength

The parameter list in Table 1.1is not complete. There are many other parameters that can be modified if you are trying to simulate something specific, such as paved area discharge, or snowmelt, or if you eliminate one or more of the processes. If you do not simulate a process, then a place holder parameter is usually required that will need to be calibrated. For example, if you do not simulate the unsaturated zone and evapotranspiration, then precipitation must be converted to groundwater recharge using the Net Rainfall Fraction and Infiltration Fraction parameters to account for losses to evapotranspiration and runoff.

1.2.2 Model limits

Although, there are no physical limits to the size of your model, there are practical limits and hardware limits.

The practical limits are generally related to run time. We all want the model to be a little bit bigger or more detailed. However, that little extra detail or slightly smaller grid size can quickly lead to long run times.

The physical limits are generally related to memory size. If you model requires more memory than is physically installed on the computer, then the computer will start to swap data to the hard disk. This will vastly slow down your simulation. Also, if you are using a 32-bit computer, then the operating system will also put constraints on the maximum file size and memory.

If your model reaches the practical or physical limits of your computer, then critically evaluate your model to see if you really need such a large, complex model. For example, maybe you can reduce the number of UZ nodes or increase the grid size.

If the model is simply too slow, then you may be able do an initial rough calibration with a less complex model. For example, during the initial calibration, you could use Gravity flow instead of Richards equation, double the grid spacing, or shorten the calibration period. Afterwards, you can switch back to the original configuration for the final calibration. You might even be surprised that the rougher model is actually good enough.

1.2.3 MIKE SHE Demo model limits

If no dongle is installed, or if a current license file is not available, then MIKE SHE will run in demo mode. In this case, the model size is restricted. If you need a full size MIKE SHE to perform your evaluation, then you are welcome to contact your local DHI office to request a 30-day evaluation license.

The current demo restrictions are as follows:

- number of cells in x- and y-direction: 70
- number of computational cells per layer (incl. boundary cells): 2000
- number of computational saturated zone layers: 2
- number of river links: 250
- number of computational UZ columns (multi-layer UZ): 155
- number of nodes per UZ column (multi-layer UZ): 100
- simulation time: 4444 hours or 185 days
- number of UZ timesteps: 800
- number of SZ timesteps: 200
- no steady-state SZ
- no overbank spilling
- no ECO Lab linkage
- no irrigation

Further, there are some restrictions in the rest of the MIKE Zero tools in demo mode. The most critical of these is that the Grid and Time Series Editors do not allow you to save files in demo mode.

1.2.4 Hardware Requirements

The hardware requirements for MIKE SHE depend on the model that you are trying to simulate. As a rule of thumb, any good quality, new computer should be sufficient for an average MIKE SHE model. Thus, a typical machine for an average MIKE SHE model will have at least a 2GHz CPU, 2GB of RAM, and 100 GB of free disk space.

Note, however, these are minimum requirements. In particular, data storage is often a problem. A large model with a long simulation period and a short saved time step interval can easily generate very large output data sets. If you save multiple simulations (e.g. calibration runs or scenarios), then you can quickly have hundreds of Gigabytes of output data.

Note: MIKE SHE must run in a Windows environment and will not run on Linux workstations.

64-bit CPU

Most of the DHI numerical engines are compiled for a 64-bit processor, including MIKE SHE.

However, MIKE 11 is an older code that cannot be compiled for a 64-bit processor. Therefore, MIKE 11 will run as a 32-bit application in a 64-bit environment.

Multi-core/processor computers

The numerically intensive operations in the MIKE SHE engine have been optimized for multi-core computers. However, not all of the hydrologic processes scale equally well. Thus, the simulation speed improvements on multi-core computers depends on the model.

The AUTOCAL program for parameter optimization and sensitivity analysis has been updated to automatically spread out the simulation load to the available cores.

The standard MIKE Zero license is supports up to four cores/processors. If you want to take advantage of more than four cores, then you will need to contact your local DHI sales office to obtain additional run-time licenses.

RAM

MIKE SHE does not dynamically allocate RAM. That is, the amount of RAM required by the model is allocated at the beginning of the simulation based on the specified number of nodes. If you don’t have enough RAM, then MIKE SHE will swap to the hard disk, which can drastically slow down your simulation.

The amount of RAM may also be important when running multiple simulations at the same time, since each simulation will require a full memory space.

In a computer with a 32-bit operating system, each application is restricted to 2GB of RAM. If you have a 32-bit system, each MIKE SHE simulation can only use a maximum of 2GB of RAM - even if you have installed more than 2GB of RAM.

If your computer has a 64-bit operating system, then there is effectively no limit to the amount of RAM. In this case, MIKE SHE will use all available RAM.

CPU Speed

In general, the higher the CPU clock speed, the faster the calculations. However, simulation speed also depends on the chip design, which depends on the manufacturer (e.g. Intel vs AMD), the platform (e.g. laptop vs. desktop), etc. Given the huge range of chip designs and the rapid pace of development, it is difficult to give specific guidance on choice of CPU - other than "faster is usually better, all other things being equal".

1.3 Getting Help

If you click F1 in any MIKE SHE dialogue, you will land in one of the sections of The MIKE SHE Reference Guide. Likewise, if you click F1 in any MIKE 11 or other MIKE Zero dialogue, you will land in a relevant section of the on-line help.

This manual is a supplement to the basic on-line F1 help and provides you with additional information on how to use MIKE SHE to get the results that you want.

1.4 Service and Maintenance

As with any complex software package, the software is being continually improved and extended. Some of these improvements are fixes of problems that have slipped though our quality control. Others are fixes of known minor problems with the software. However, the vast majority of the changes in new releases and service packs are related to improvements to the functionality of the software.

Your initial purchase of the software is protected by a one-year subscription to our Service and Maintenance Agreement. Your Service and Maintenance Agreement entitles you free support for software problems via email or telephone and regular updates to the software.

We strongly recommend that you subscribe to the Service and Maintenance Agreement after the first year to further protect your investment. Improvements, extensions and fixes are continually being made, and we will make every effort to help you with any problems that you encounter, but we cannot provide fixes for any versions older than the current release.

1.4.1 Service Packs

As part of the Service and Maintenance, there is an auto update program installed with your software. This program automatically checks our website for Service Packs to the currently installed release and downloads the Service Pack if it is available. You will be asked before the installation begins, if you want the installation to proceed. We strongly recommend that you install the latest Service Pack as soon as they are released.

However, some clients prefer not to install the Service Pack during a project, or close to the end of a project. Occasionally, a fix in the numerical engine will slightly change your simulation results. This may require you to re-run previously finished simulations to obtain valid comparisons between simulations.

The Auto Updater overwrites your existing executable files. Therefore, if you are concerned about potential changes in your results, they you should backup all of the files in the MIKE SHE installation directory, before installing the Service Pack.

If you did not back up your installation directory, and you need to restore a previous version, DHI maintains an archive of all standard patch versions. Contact your local support centre and we will send you a copy of your previous executable.

2 BUILDING A MIKE SHE MODEL

The MIKE SHE user interface is organized around the workflow to build a model. Basically, your work flow follows the data tree. You typically start at the top of the data tree and work your way down. As you complete each of the items in the data tree, the red “x” will be replaced by a green check-mark. Thus, the basic work flow for a fully integrated MIKE SHE model is built around the following components:

1 The MIKE SHE User Interface *(V.1 p. 31)*

2 Background Maps *(V.1 p. 34)*

3 Initial Model Setup *(V.1 p. 34)*

4 Simulation parameters *(V.1 p. 36)*

5 Model domain and grid *(V.1 p. 37)*

6 Topography *(V.1 p. 39)*

7 Climate *(V.1 p. 40)*

8 Channel Flow *(V.1 p. 46)*

9 Overland Flow *(V.1 p. 47)*

10 Unsaturated Flow *(V.1 p. 52)*

11 Saturated Groundwater Flow *(V.1 p. 57)*

12 Storing of results *(V.1 p. 62)*

13 Preprocessing your model *(V.1 p. 149)*

14 Running your Model *(V.1 p. 149)*

2.1 MIKE Zero

MIKE SHE is part of the MIKE Zero suite of modelling tools. However, 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.

In any project, it is a challenge to maintain an overview of all of these files, not to mention keeping regular backups and archives of all of these files. As you progress through the calibration and validation phases, and then on to the scenario analysis and report writing phases, the number of model artifacts can become overwhelming. The MIKE Zero project structure is designed to include all of your modelling files; that is all of the raw data files, model input files, and model output files, as well as any reports, spread sheets, plots, etc.

The MIKE Zero project structure is designed to help you keep control of your project.

There is a separate introduction manual to help you get started working with MIKE Zero.

2.1.1 MIKE Zero Editors

The MIKE Zero also includes general tools for data editing, analysis and manipulation. Some of these have their own file types, or documents. The MIKE Zero documents include (with the tools commonly used for MIKE SHE in bold):

- **The Time Series Editor (.dfs0)**- for time series data
- The Profile Series Editor (.dfs1)- for time varying 1D data (profiles are not used in MIKE SHE)
- **The Grid Editor (.dfs2 and .dfs3)** - for time varying 2D and 3D data
- Data Manager - for finite element data
- **The Plot Composer (.plc)** - for creating standard report plots
- **Result Viewer (.rev)** - for results presentation
- Bathometry (.batsf) - for sea bed elevations
- Animator (.mza) - for 3D visualization of 2D surface water and waves
- **ECOLAB (.ecolab)** - for water quality in surface water, which can be used in MIKE 11 (but not yet in the rest of MIKE SHE)
- **AUTOCAL (.auc)** - for autocalibration, sensitivity analysis and scenario management
- EVA Editor (.eva) - for extreme value analysis of surface water flows
- Mesh Generator (.mdf) - for creating meshes for the finite element versions of MIKE 21 and MIKE 3
- Data Extraction FM (.dxfm) - for extracting data from finite element results files
- **MIKE Zero Toolbox (.mzt)** - various tools for data manipulation

The documentation for these tools is found in the printed MIKE Zero books and under MIKE Zero in the on-line help.

In addition to the MIKE Zero document-based tools, there are a number of other important MIKE Zero utilities that are accessed from the Start\Programs\MIKE by DHI menu, including:

- **MIKE View** - a results evaluation utility for MIKE 11 and MOUSE (sewers) 1D flow results.
- **Image Rectifer** - a simple tool for stretching and georeferencing image files
- **Launch Simulation Engin**e - a utility for launching and running MIKE Zero simulation engines independent of the Graphical User Interface.
- **GeoViewer** - a visualization tool for 3D layer data

2.2 The MIKE SHE User Interface

The MIKE SHE user interface is organized by task. In every model application you must

1 Set up the model,

2 Run the model, and

3 Assess the results.

The above three tasks are repeated until you obtain the results that you want from the model.

When you create or open a MIKE SHE model, you will find your self in the Setup Tab of the MIKE SHE user interface.

The following sections provide a quick overview of the main hydrologic processes in MIKE SHE. For more detailed information on the individual parameters, see Setup Data Tab *(V.2 p. 19)* chapter in the Reference Manual.

Alternatively, this manual also contains detailed user guidance and information in the sections:

- Surface Water *(V.1 p. 169)*
- Unsaturated Groundwater Flow *(V.1 p. 243)*
- Saturated Groundwater Flow *(V.1 p. 249)*
- Running MIKE SHE *(V.1 p. 147)*
- Results and Calibration *(V.1 p. 69)*

2.2.1 The Setup Editor

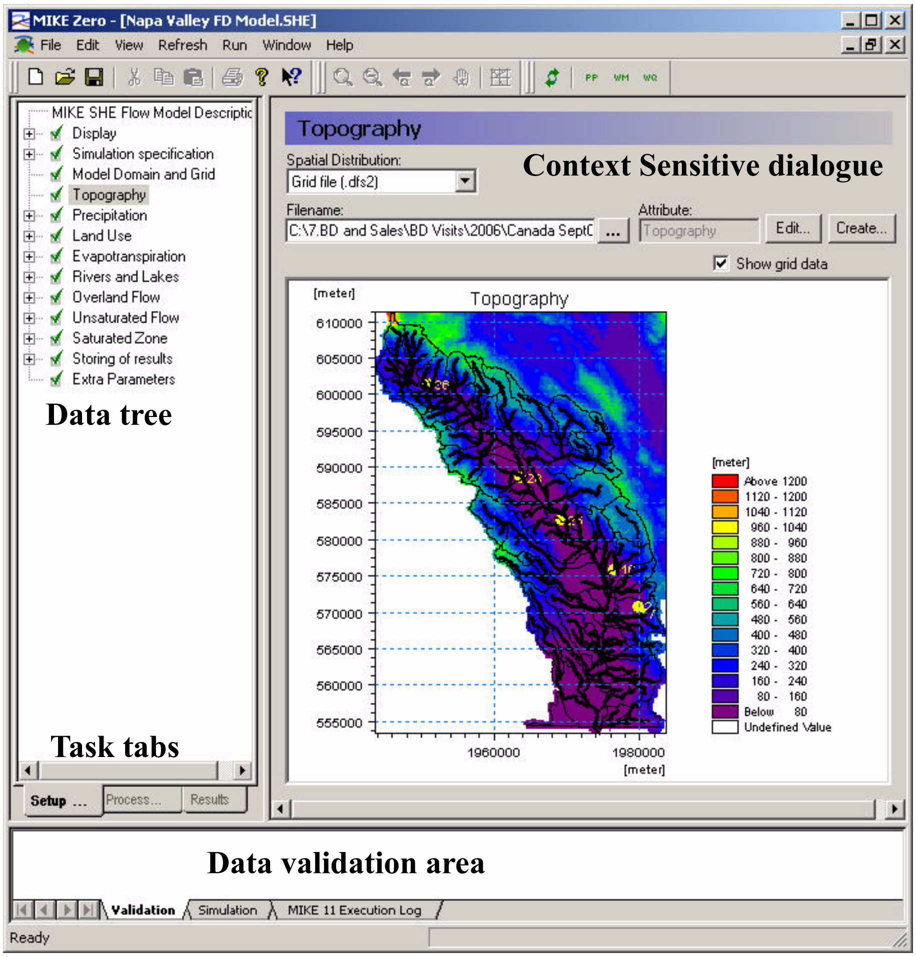


Figure 2.1 *Graphical overview of the in the MIKE SHE GUI, without the Project Explorer.*

The Setup editor is divided into three sections - the data tree, a context sensitive dialogue and a validation area.

The data tree is dynamic and changes with how you set up your model. It provides an overview of all of the relevant data in your model. The data tree is organized vertically, in the sense that if you work your way down the tree, by the time you come to the bottom you are ready to run your model.

The context sensitive dialogue on the right allows you to input the required data associated with your current location in the data tree. The dialogues vary with the type of data, which can be any combination of static and dynamic data, as well as spatial and non-spatial data. In the case of spatial and time varying data, the actual data is not input to the GUI. Rather, a file name must be specified and the link to the file is stored in the GUI. Furthermore, the distribution of the data in time and space need not correspond between the various entries. For example, rainfall data may be entered as hourly values and pumping rates as weekly values, while the model may be run with daily timesteps.

The validation area at the bottom of the dialogue provides you with immediate feedback on the validity of the data that you have input.

After you have set up your model, you must switch to the Processed Data tab and run the pre-processing engine on the model. This step reconciles all of the various spatial and time series data and creates the actual data set that will be run by MIKE SHE. Once the data has been pre-processed the simulation can be started. Using the Pre-processing tab at the bottom, you can view the pre-processed data.

After the simulation is finished, you can switch to the Results tab, where you can view the detailed time series output as in a report-ready HTML view. Alternatively, you can use the Results Viewer, which is one of the generic MIKE Zero tools, for more customized and detailed analysis of the gridded output.

2.2.2 The Setup Data Tree

Your MIKE SHE model is organized around the Setup Data Tree. The layout of the tree depends on the model components that are active in the current model, which are selected in the Simulation Specification dialogue. Opposite the data tree is the corresponding dialogue for the currently selected tree branch.

The data tree is designed to hide the components that are not needed for the current simulation. However, no data is ever lost if the branch is hidden. That is, all data is retained, even if the branch is not currently visible.

The design of the data tree is such that when you make selections in the current dialogue, the tree is automatically updated to reflect the selection. However, the layout of the data tree and the options available in the current dialogue are such that the data tree will only change along the current branch. That is, if you make a selection in the current dialogue, additional options or branches may become available further along the branch. However, no changes will occur in other branches of the data tree. For example, if you make a selection in the Precipitation dialogue, this will affect the Precipitation data branch. It will not affect the Evapotranspiration branch.

The only exception to the above rule is selections made in the Simulation Specification dialogue, which is used to set up the entire data tree. Thus, for example, if you unselect Evapotranspiration in the Simulation Specification dialogue, the entire Evapotranspiration branch will disappear.

2.2.3 Background Maps

Arguably, the first step in building your model is to define where your model is located. This generally involves defining a basic background map for your model area.

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.

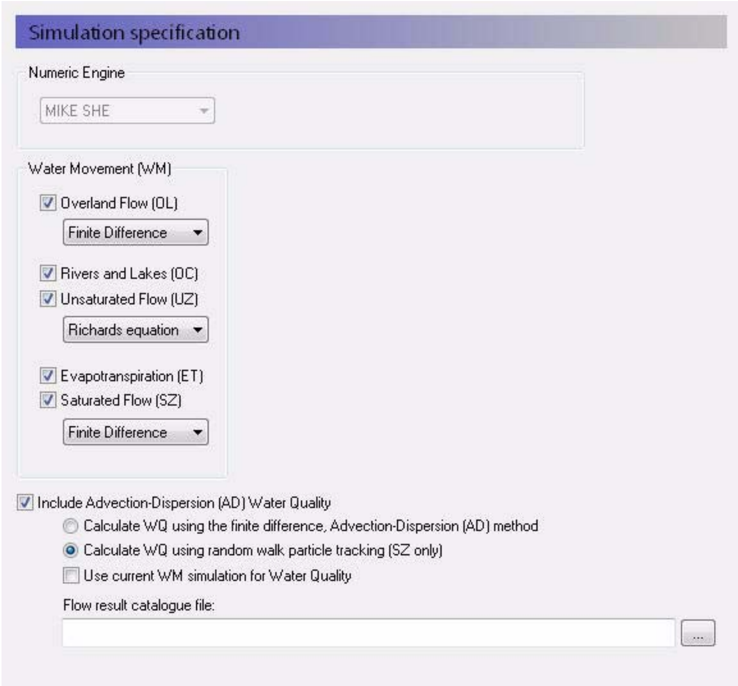
In the event that you are using scanned paper maps, if your maps are not rectilinear, or are not correctly georeferenced, then you can use the Image Rectifer (see on-line help under MIKE Zero) to align your image to the coordinate system you are using.

Note The display of the Mike 11 network is not carried over to the Results Viewer.

2.2.4 Initial Model Setup

MIKE SHE allows you to simulate all of the processes in the land phase of the hydrologic cycle. That is, all of the process involving water movement after the precipitation leaves the sky. Precipitation falls as rain or snow depending on air temperature - snow accumulates until the temperature increases to the melting point, whereas rain immediately enters the dynamic hydrologic cycle. Initially, rainfall is either intercepted by leaves (canopy storage) or falls through to the ground surface. Once at the ground surface, the water can now either evaporate, infiltrate or runoff as overland flow. If it evaporates, the water leaves the system. However, if it infiltrates then it will enter the unsaturated zone, where it will be either extracted by the plant roots and transpired, added to the unsaturated storage, or flow downwards to the water table. If the upper layer of the unsaturated zone is saturated, then additional water cannot infiltrate and overland flow will be formed. This overland flow will follow the topography downhill until it reaches an area where it can infiltrate or until it reaches a stream where it will join the other surface water. Groundwater will also add to the base-flow in the streams, or the flow in the stream can infiltrate back into the groundwater.

In the main simulation specification dialogue, you select the processes that you would like to include in your model. For the main water movement processes, you can also select the numerical solution method. In general, the simpler methods will require less data and run more quickly. Your choice here will be immediately be reflected in the data tree.



Water Quality

In this dialogue, you can also chose to simulate water quality. If you turn on the water quality, then several additional items will be added to the data tree. Also, you will be able to chose to simulate water quality using either the full advection-dispersion method for multiple species including sorption and decay. Or, you can chose to simulate water quality using the random walk particle tracking method.

You can also do water quality scenario analysis by using a common water movement simulation and defining only the water quality parameters. The common water movement simulation is defined by first unchecking the Use current WM simulation for Water Quality checkbox.

The Technical Reference contains detailed information on the numerical methods that can be selected from this dialogue:

- Overland Flow - Reference *(V.2 p. 265)*
- Channel Flow - Reference *(V.2 p. 287)*

- Evapotranspiration - Reference *(V.2 p. 295)*
- Unsaturated Flow - Reference *(V.2 p. 319)*
- Saturated Flow - Reference *(V.2 p. 355)*
- Particle Tracking-Reference *(V.2 p. 435)*
- Advection Dispersion - Reference *(V.1 p. 739)*

2.2.5 Simulation parameters

Once you have selected your processes, then there are several simulation parameters that need to be defined. None of these are initially critical and the default values are generally satisfactory initially. You can come back to all of these at any time.

However, we recommend that you set up you simulation period when you first create your model. The simulation period is used to verify all of you time series data to make sure that your time series cover your simulation period. You can still add your time series files, but if your simulation period is not correct, then you will get a warning message in the message field at the bottom of the page and the time series graphs will not display the proper portion of the time series.

In MIKE SHE, all of the simulation input and output is in terms of real dates, which makes it easy to coordinate the input data (e.g. pumping rates), the simulation results (e.g. calculated heads) and field observations (e.g. measured water levels).

Solver parameters

The default solver parameters for each of the processes are normally reasonable and there is usually no reason to change these unless you have a problem with convergence or if the simulation is taking too long to run. For more information on the solver parameters, you should see the individual help sections for the different solvers:

- OL Computational Control Parameters *(V.2 p. 36)*
- UZ Computational Control Parameters *(V.2 p. 41)*
- SZ Computational Control Parameters *(V.2 p. 42)*

Time step control

Likewise, the time step control is important, but the default values are usually reasonable to get your model up and running. Then, you should go back to the Time Step Control *(V.2 p. 31)* dialogue to optimize your simulation time stepping. For more information on time step control, you can go to the help section for the Time Step Control *(V.2 p. 31)* dialogue, or see the Controlling the Time Steps *(V.1 p. 161)* section.

Note: Although the different hydrologic processes can run on different time steps, the processes exchange water explicitly. Therefore, there are restrictions on the relationship between the time steps in the processes. In particular, the longer time steps must be even multiples of the shorter time steps. In other words, a 24 hour groundwater time step can included four 6-hour unsaturated flow timesteps, which can each include three 2-hour overland flow timesteps. See Time Step Control *(V.2 p. 31)* for more information.

2.2.6 Hot Starting from a previous simulation

Your MIKE SHE simulation can be started from a hot start file. A hot start file is useful for simulations requiring a long warm up period or for generating initial conditions for scenario analysis. Hot starting can also be an effective way to change parameters that are normally static (e.g. hydraulic conductivity) during the model process.

To start a model from a previous model run, you must first save the hot start data, in the Storing of Results *(V.2 p. 183)* dialogue. In this dialogue, you specify the storing interval for hot start data. Then in the Simulation Period *(V.2 p. 29)* dialogue, you can specify the hot start file and then select from the available stored hot start times.

Hot start limitations

There are a few limitations and caveats with the hot start process.

- The Water Quality simulations cannot be started from a hot start file.
- There is no append function for the hot start results, so your simulation will generate an independent set of results.
- The pre-processed data does not reflect the hot-start information. The pre-processed data is based on the specified input data, not the results file from which the simulation will be started. This primarily affects initial conditions.

2.3 Model domain and grid

Regardless of the components included in your model, the first real step in your model development is to define the model area. On a catchment scale, the model boundary is typically a topographic divide, a groundwater divide or some combination of the two. In general, there are no constraints on the definition of the model boundaries. However, the model boundaries should be chosen carefully, keeping in mind the boundary conditions that will be used for both the surface water and groundwater components.

All other spatial data defined in the data tree, such as topography, is interpolated during pre-processing to the Model Domain and Grid.

You can define your model domain and the grid using either a DHI grid file (dfs2 format) or a GIS shape file (.shp format).

Using a dfs2 file

If you define your model domain using a dfs2 grid file, then you must define the cell values as follows:

- Grid cells outside of the model domain must be assigned a delete value - by default -1.0e-35.
- Grid cells inside the model domain must be assigned a value of 1.
- Grid cells on the model boundary must be assigned a value of 2.

This distinction between interior grid cells and boundary cells is to facilitate the definition of boundary conditions. For example, drainage flow can be routed to external boundaries but not to internal boundaries.

Since the model domain is defined as part of the dfs2 file format, if you want to change the extent of your model domain, you must edit the .dfs2 file. However, if you want to change the grid spacing, then it is probably easier to create a new file.

The Model Domain and Grid does not have to have the same dimensions (size and spacing) as other specified dfs2 files (e.g. Topography). However, if the other dfs2 input files are coincident, that is if the rows and columns align with one another, then an average of the cell values is used. If the dfs2 files are not coincident, then the Bilinear Interpolation *(V.1 p. 355)* method is used to determine the cell value.

Note: The dfs2 files for **integer grid codes must be coincident with the model grid.** For more information on this see Integer Grid Codes *(V.1 p. 353)*.

Using an polygon shape (shp) file

It is much easier to define your Model Domain and Grid via a GIS polygon shape (.shp) file. In this case, the definition of integer code values is taken care of internally. Once you have defined the polygon file to use, then you specify the spatial extent and origin location of the model domain and grid.

An important advantage of using a polygon for the model domain, is that the number of rows and columns can be easily adjusted. See Using MIKE SHE with ArcGIS *(V.1 p. 345)* for more information.

Creating dfs2 or shp files

There is a Create button next to the Browse button that opens a dialogue where you can define a dfs2 grid file. This utility automatically creates the grid file with the appropriate Item Type.

In this dialogue, you can specify the overall dfs2 grid dimensions and origin. After you have created the file, then you can open and edit the file in the Grid Editor using the Edit button.

Geographic projections

MIKE SHE supports all available geographic projections. If you have defined the domain using a dfs2 file, then the geographic projection is defined in the dfs2 file. If you use polygon shape file, then you must defined the projection in the Model Domain and Grid *(V.2 p. 70)* dialogue. See Using MIKE SHE with ArcGIS *(V.1 p. 345)* for more information.

Note: All dfs2 and polygon shape files must use the same geographic projection. Any inconsistencies in the projections will results in an error during the pre-processing.

2.4 Topography

In MIKE SHE, the topography defines the upper boundary of the model. The topography is used as the top elevation of both the UZ model and the SZ model. The topography also defines the drainage surface for overland flow.

Many of the elevation parameters can be defined relative to the topography by means of a checkbox in the dialogue, including

- Lower Level *(V.2 p. 194)*,
- Upper Level *(V.2 p. 194)*,
- Initial Potential Head *(V.2 p. 165)*, and
- Drain Level *(V.2 p. 176)*.

Depth parameters, such as ET Surface Depth *(V.2 p. 141)*, are also measured from the topography.

File Formats

Topography is defined from a digital elevation model (DEM) using either a dfs2 grid file, a point theme shape (GIS) file, or an ASCII XYZ file.

Non-dfs2 files or dfs2 files that have a different grid definition than the model grid are all interpolated to the grid defined in the Model Domain and Grid.

The Bilinear Interpolation *(V.1 p. 355)* method is useful for interpolating previously gridded DEM data. Whereas, the Triangular Interpolation *(V.1 p. 359)* method is useful for contour data digitized from a DEM. Inverse Distance *(V.1 p. 361)* is usually used for sparse or irregularly spaced data.

ArcGIS Grid Files If you have an ArcGIS Grid DEM, this can be converted to a dfs2 file using the MIKE Zero Toolbox. For more information see the Using MIKE SHE with ArcGIS *(V.1 p. 345)* section. Alternatively, a dfs2 plug-in is available for ArcGIS, that allows you to read and write dfs2 files directly in ArcGIS.

Surfer Grid Files Surfer Grid files can be saved as an ASCII XYZ file and then interpolated in MIKE SHE.

Other DEM formats Most other DEM formats can be converted to either an ArcGIS Grid file or an ASCII XYZ file. If you have special requirements or difficulty, please contact your local support office.

2.5 Climate

Climate is the driving force for the hydrologic cycle. Spatial variation in solar radiation drives the weather resulting in evaporation, rainfall, and snow.

2.5.1 Precipitation

Precipitation is the measured rainfall. You can specify the precipitation as a rate, for example in [mm/hr], or as an amount, for example in [mm]. If you use the amount method, MIKE SHE will automatically convert this to a rate during the simulation.

If you use a rate, then the EUM Data Units *(V.1 p. 329)* must be “Precipitation” and the time series must be Mean Step Accumulated *(V.1 p. 343)*.

If you use an amount, then the EUM Data Units must be “Rainfall” and the time series must be Step Accumulated *(V.1 p. 343)*.

The Precipitation Rate item comprises both a distribution and a value. The distribution can be either uniform, station-based or fully distributed. If the data is station-based then for each station a sub-item will appear where you can enter the time series of values for the station.

2.5.2 Snow

If the Include snow melt *(V.2 p. 76)* checkbox is checked then rain accumulates as snow if the Air Temperature *(V.2 p. 84)* is below the Threshold Melting Temperature *(V.2 p. 88)* (the temperature at which the snow starts to melt - usually 0 C). If the air temperature is above the threshold, then the snow will melt at the rate specified by the Degree-day Melting or Freezing Coefficient *(V.2 p. 89)*.

Dry snow acts like a sponge and does not immediately release melting snow. Thus, melting snow is added to wet snow storage. When the amount of wet snow exceeds the Maximum Wet Snow Fraction in Snow Storage *(V.2 p. 91)*, the excess is added to ponded water, which is then free to infiltrate or runoff.

More detailed information on the snow melt process can be found in the on-line help for the individual dialogues and in the Snow Melt - Reference *(V.2 p. 289)* section.

2.5.3 Evapotranspiration

The calculation of evapotranspiration uses meteorological and vegetative data to predict the total evapotranspiration and net rainfall due to

- Interception of rainfall by the canopy,
- Drainage from the canopy to the soil surface,
- Evaporation from the canopy surface,
- Evaporation from the soil surface, and
- Uptake of water by plant roots and its transpiration, based on soil moisture in the unsaturated root zone.

The primary ET model is based on empirically derived equations that follow the work of Kristensen and Jensen (1975), which was carried out at the Royal Veterinary and Agricultural University (KVL) in Denmark. This model is used whenever the detailed Richards equation or Gravity flow methods are used in the Unsaturated zone.

In addition to the Kristensen and Jensen model, MIKE SHE also includes a simplified ET model that is used in the Two-Layer UZ/ET model. The Two-Layer UZ/ET model divides the unsaturated zone into a root zone, from which ET can occur and a zone below the root zone, where ET does not occur. The Two-Layer UZ/ET module is based on a formulation presented in Yan and Smith (1994). Its main purpose is to provide an estimate of the actual evapotranspiration and the amount of water that recharges the saturated zone. It is primarily suited for areas where the water table is shallow, such as in wetland areas.

The reference evapotranspiration (ET) is the rate of ET from a reference surface with an unlimited amount of water. Based on the FAO guidelines, the reference surface is a hypothetical grass surface with specific characteristics. The reference ET value is independent of everything but climate and can be calculated from weather data. The FAO Penman-Monteith method is recommended for determining the reference ET value.

The reference ET is multiplied by the Crop Coefficient to get the Crop Reference ET. The Crop Coefficient is found in the Vegetation development table in the Vegetation database. If the vegetation database is not used, then the Reference ET is the maximum ET rate.

The Reference Evapotranspiration item comprises both a distribution and a value. The distribution can be either uniform, station-based or fully distributed. If the data is station-based then for each station a sub-item will appear where you can enter the time series of values for the station.

2.5.4 Snow Melt

MIKE SHE includes a comprehensive snow melt module based on a modified degree-day method. Precipitation that occurs when the air temperature is below the freezing point accumulates as solid snow and does not infiltrate or contribute to runoff. The accumulated snow has a moisture content, and when the moisture content reaches a critical level, then additional melting contributes to runoff.

Air Temperature

For snow melt, the air temperature is critical. However, the air temperature changes significantly with elevation. In areas with significant elevation changes, snow will accumulate in upland areas - often were there is limited weather data available. The elevation correction for air temperature allows you to specify an elevation for the temperature stations and a temperature change rate with elevation. During the pre-processing, a temperature change factor is calculated for each cell and the actual temperature in the cell is calculated during the simulation using this factor.

In terms of snow melt, the air temperature along with the degree day melting coefficient determine the amount of melting that can occur. If you have daily temperature data it may be difficult to properly account for the diurnal melting and freezing cycles.

Air temperature can also be an important parameter during water quality simulations.

For more information on the snow melt parameters, see the specific snow melt dialogue information in the Climate *(V.2 p. 76)* section of the on-line help and User Interface manual

2.6 Land Use

The land surface plays a very important role in hydrology. In principle, the land use section is used to define the properties of the land surface. The most important of these is the distribution of vegetation, which is used by MIKE SHE to calculate a the spatial and temporal distribution of actual evapotranspiration.

However, the land surface comes into play in many ways and other sections of the data tree also include properties related to land use. Some of these properties are related to the vegetation distribution, and may even be spatially identical. For example:

Topography - The topography is a physical property of the land surface that defines the hydraulics of both the overland flow and the unsaturated flow. See Topography *(V.1 p. 39)*. Related to topography is the definition of Subcatchments *(V.2 p. 73)*, which is needed when you are using the Linear Reservoir method for groundwater or the simple, catchment based overland flow method.

Flood zones - In MIKE SHE, flood zones can be defined relative to the MIKE 11 branches using Flood codes. For details on how use Flood codes see the chapter on Surface Water *(V.1 p. 169)*.

Hydraulic properties - The properties related directly to overland sheet flow are found under Overland Flow *(V.1 p. 47)*. This includes the Manning number *(V.2 p. 118)* or surface roughness and the Detention Storage *(V.2 p. 119)*, both of which are influenced or even defined by the vegetation.

Hydraulic flow - Areas of the land surface can be hydraulically divided by man-made structures, such as road ways and embankments, which can be defined by Separated Flow Areas *(V.2 p. 123)*.

Infiltration properties - The infiltration rate is a property of the soil type, which may be modified by the land use. Related to the gross infiltration rate is the presence or absence of macropores and other soil features leading to rapid infiltration. Both of these properties are found in the Unsaturated Flow *(V.1 p. 52)* section. However, land surface sealing and compaction can be defined as a reduced contact between ponded water and the subsurface. This is defined in the Overland flow section as a Surface-Subsurface Leakage Coefficient *(V.2 p. 121)*.

Groundwater drainage - As the groundwater table rises, it intersects low lying topographic features, such as ditches, or other man-made drainage features, such as buried farm drains. These features are related to land use, but are specified as Groundwater Drainage *(V.1 p. 60)*

Paving - Paved areas are treated as a drainage feature for ponded water - not rainfall. The paved drainage function is part of the Land use section but requires Groundwater drainage to be defined and depends completely on the land use functions above that affect ponding of water.

2.6.1 Vegetation

By default, the only section under Land use is the vegetation distribution The vegetation properties are used to calculate the actual evapotranspiration from crop reference evapotranspiration defined under Climate.

The primary vegetation properties are Leaf Area Index (LAI) and Root Depth (RD). The LAI and Root Depth can be specified directly as a time series. Or, they can be defined as a crop rotation in the Vegetation Properties Editor *(V.2 p. 235)*.

A good source of local information on LAI and root depth is the agronomy department at your local university.

Leaf Area Index

The LAI is defined as the area of leaves per area of ground surface. The LAI values are characteristic of the plant type, season, and plant stress. LAI values are widely available in the literature for most major plant types.

The LAI is a lumped parameter for a cell that defines the average leaf area of the cell. In forests, it includes both the leaf area of the forest canopy and the understory. In more open areas, it is an average for all vegetation types, such as grass, brush and trees. In areas of largely open water the LAI is usually zero. If the LAI is zero, there will be no interception storage and no water will be removed from the unsaturated zone.

Root Depth

Root depth is defined as the depth below ground in **millimetres** to which roots extend. The root depth is not necessarily the average root depth. In some cases it may be the maximum root depth. The root depth defines the depth at which water can be extracted from the unsaturated zone. If the root depth is deeper than the depth of the capillary zone, then the roots will be able to extract water from the saturated zone. The thickness of the capillary zone is defined by the pedotransfer function in the soil properties for the Richards and Gravity flow methods. In the 2Layer UZ method, the thickness of the capillary zone is defined by the ET Surface Depth *(V.2 p. 141)*. If you are using the Richards or Gravity Flow UZ methods, then you will also be able to use the Root Shape factor (AROOT) for each vegetation type. This allows you, for example, to extract more water from the upper UZ cells than the lower cells, which is typical of grasses in semi-arid climate zones.

2.6.2 Paved areas

The paved area function allows you to drain rainfall directly to the MIKE 11 network. The paved area function is rather complex and restricted in several important ways. Most importantly, the paved area function

- requires that the SZ drainage function be turned on, which means that it only works when you are using the Finite Difference SZ method, and
- discharges only to river links - not internal depressions or boundaries.

If you turn on the paved area function, then you can enter the paved runoff coefficient, which is the fraction of the land surface that is paved.

There are two options for the paving function. There is an optional check on the water level in the discharge point. If the discharge water level is higher than or equal to the ponded water level, then no water will be discharged. The second option is that you can limit the discharge rate from paved areas.

For more details on the paved area function, see Paved Area Drainage *(V.1 p. 174)*.

2.6.3 Irrigation

The irrigation module allows you to simulate the transfer of irrigation water from multiple sources to multiple control areas.

The available sources include: shallow wells distributed across the cell, deep bores with defined screen intervals, river stretches defined by an upstream and downstream chainage, and external sources. The sources can be defined in a hierarchy, such that when one is unavailable, the water will be removed from the next.

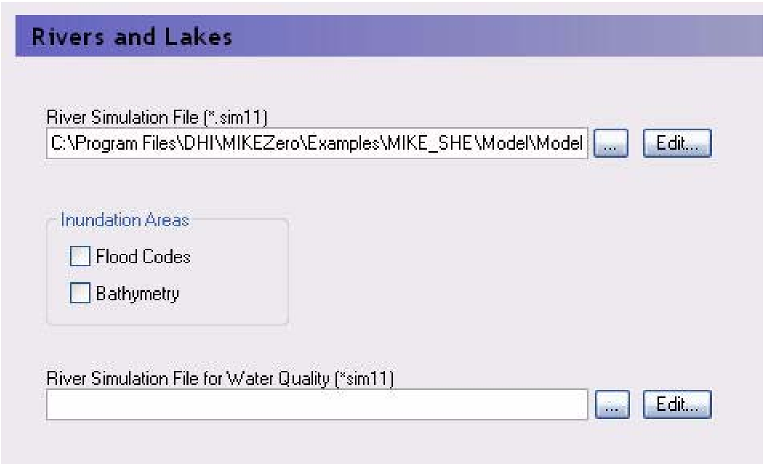
Irrigation is applied in control areas. Each control area is defined by an area and a control function that defines when and how much water will be applied.

For more information on Irrigation, see

- Irrigation Command Areas *(V.2 p. 103)*
- Irrigation Demand *(V.2 p. 111)*
- Irrigation Priorities *(V.2 p. 113)*

2.7 Channel Flow

In the Rivers and Lakes dialogue (below) you can link MIKE SHE to a MIKE 11 model.



The River Simulation File (.sim11) is the main MIKE 11 simulation file, which contains the file references to all the files used in the MIKE 11 model. For MIKE SHE, the primary MIKE 11 files are:

- the simulation control file (.sim11),
- the river network file (.nwk11),
- the cross-section database (.xns11),
- the boundary condition file (.bnd11) and
- the hydrodynamic setup file (.hd11).

In the Rivers and Lakes dialogue, there are two Inundation Areas options These options are always available for input, but are only used if you have selected specific options in the MIKE SHE Links dialogue in the MIKE 11 Network Editor. These options are

- Flood codes - a map used for the direct inundation of flooded areas in MIKE SHE based on water levels in MIKE 11, and
- Bathymetry - a detailed topography file that can be used to modify the defined topography with a more detailed flood plain topography in areas where Flood Codes have been defined.

Integrating a MIKE SHE and a MIKE 11 model is not very different from establishing a stand-alone MIKE 11 HD model and a stand-alone MIKE SHE model. In principle there are three basic set-up steps:

1 Establish a MIKE 11 HD hydraulic model as a stand-alone model and make a performance test and, if possible, a rough calibration using prescribed inflow and stage boundaries. You can also specify a default groundwater table (e.g. MIKE SHE's initial groundwater level) and leakage coefficients for any leakage calculations.

2 Establish a MIKE SHE model that includes the overland flow component and (optionally) the saturated zone and unsaturated zone components. An SZ drainage boundary can be used to prevent excessive surface flows in low lying areas and the river flood plain.

3 Couple MIKE SHE and MIKE 11 by defining branches (reaches) where MIKE 11 HD should interact with MIKE SHE. Modify your MIKE SHE and MIKE 11 models so that they work together properly. For example, by removing the specified groundwater table in MIKE 11 and adjusting your SZ drainage elevations if you used these in Step 2.

Detailed information on developing your surface water model, specifying flow on flood plains, and coupling to MIKE 11 is in the chapter Surface Water *(V.1 p. 169)*.

Additional documentation on MIKE 11 can be found in the MIKE 11 User Guide.

2.8 Overland Flow

Overland flow simulates the movement of ponded surface water across the topography. It can be used for calculating flow on a flood plain or runoff to streams.

You can run the Overland flow module separately, or you can combine it with any of the other modules. However, overland flow is required when you are using MIKE 11 in MIKE SHE, as the overland flow module provides lateral runoff to the rivers.

The Simplified Overland Flow Routing *(V.2 p. 279)* method can be used for regional applications when detailed flow is not required. This method assumes that ponded water in the upland areas of a subcatchment flows into the flood plain areas of the subcatchment, which in turn discharges uniformly into the stream network located in the subcatchment.

The Finite Difference Method *(V.2 p. 265)* uses the diffusive wave approximation and should be used when you are interested in calculating local overland flow and runoff. There are two solution methods available.

- Successive Over-Relaxation (SOR) Numerical Solution *(V.2 p. 270)*
- Explict Numerical Solution *(V.2 p. 271)*

The choice of method is a tradeoff between accuracy and solution time. The SOR solver is generally faster because it can run with larger time steps. The Explict method is generally more accurate than the SOR method, but is often constrained to smaller time steps. The time step constraint prevents flow from crossing a cell in a single time step. The time step constraint is determined by the cell with the highest velocity and applied to the entire model in the current time step.

The Explicit method is generally used when the river is allowed to spill from MIKE 11 onto the flood plain. Alternatively, you can use Flood codes *(V.2 p. 116)* to inundated flood plain areas based on the water level in MIKE 11.

The Multi-grid overland flow option allows you to take advantage of detailed DEM information if it is available. The multi-grid method, subdivides the overland flow cell into an even number of sub-cells. The gradients between the cells and the flow area between cells water surface elevation in the cell is then calculated based on the volume of water and the detailed topography information.

In MIKE SHE, the calculation of 2D overland flow can become a very time consuming part of the simulation. So, you need to be very careful when setting up your model to minimize the calculation of overland flow between cells when it is unnecessary.

Detailed information on Overland Flow, the coupling between MIKE 11 and MIKE SHE and the overbank spilling options, and ways to optimize the calculation of overland flow can be found in the chapter Surface Water *(V.1 p. 169)*.

Mannings M

The Manning M is equivalent to the Stickler roughness coefficient, the use of which is described in Overland Flow - Reference *(V.2 p. 265)*.

The Manning M is the inverse of the more conventional Mannings n. The value of n is typically in the range of 0.01 (smooth channels) to 0.10 (thickly vegetated channels). This corresponds to values of M between 100 and 10, respectively. Generally, lower values of Mannings M are used for overland flow compared to channel flow.

If you don't want to simulate overland flow in an area, a Mannings M of 0 will disable overland flow. However, this will also prevent overland flow from entering into the cell.

Detention Storage

Detention Storage is used to limit the amount of water that can flow over the ground surface. The depth of ponded water must exceed the detention storage before water will flow as sheet flow to the adjacent cell. For example, if the detention storage is set equal to 2mm, then the depth of water on the surface must exceed 2mm before it will be able to flow as overland flow. This is equivalent to the trapping of surface water in small ponds or depressions within a grid cell.

If you have static ponded water in an area and you do not want to calculate overland flow between adjacent cells (can be slow), then you can set the detention storage to a value greater than the depth of ponding.

Water trapped in detention storage continues to be available for infiltration to the unsaturated zone and to evapotranspiration.

Initial and Boundary Conditions

In most cases it is best to start your simulation with a dry surface and let the depressions fill up during a run in period. However, if you have significant wetlands or lakes this may not be feasible. However, be aware that stagnant ponded water in wetlands may be a significant source of numerical instabilities or long run times.

The outer boundary condition for overland flow is a specified head, based on the initial water depth in the outer cells of the model domain. Normally, the initial depth of water in a model is zero. During the simulation, the water depth on the boundary can increase and the flow will discharge across the boundary. However, if a non-zero value is used on the boundary, then water will flow into the model as long as the internal water level is lower than the boundary water depth. The boundary will act as an infinite source of water.

If you need to specify time varying overland flow boundary conditions, you can use the Extra Parameter option Time-varying Overland Flow Boundary Conditions *(V.1 p. 302)*.

Separated flow areas

The Separated Flow Areas *(V.2 p. 123)* are typically used to prevent overland flow from flowing between cells that are separated by topographic features, such as dikes, that cannot be resolved within a the grid cell.

If you define the separated flow areas along the intersection of the inner and outer boundary areas, MIKE SHE will keep all overland flow inside of the model - making the boundary a no-flow boundary for overland flow.

Multi-cell overland flow

The main idea behind the 2D, multi-cell solver is to make the choice of calculation grid independent of the topographical data resolution. The approach uses two grids:

- One describing the rectangular calculation grid, and
- The other representing the fine bathymetry.

The standard methods used for 2D grid based solvers do not make a distinction between the two. Thus, only one grid is applied and this is typically chosen based on a manageable calculation grid. The available topography is interpolated to the calculation grid, which typically does not do justice to the resolution of the available data. The 2D multi-grid solver in MIKE SHE can, in effect, use the two grids more or less independently.

In the Multi-cell overland flow method, high resolution topography data is used to modify the flow area used in the St Venant equation and the courant criteria. The method utilizes two grids - a fine-scale topography grid and a coarser scale overland flow calculation grid. However, both grids are calculated from the same reference data - that is the detailed topography digital elevation model.

In the Multi-cell method, the principle assumption is that the volume of water in the fine grid and the coarse grid is the same. Thus, given a volume of water, a depth and flooded area can be calculated for both the fine grid and the coarse grid.

In the case of detention storage, the volume of detention storage is calculated based on the user specified depth and OL cell area.

During the simulation, the cross-sectional area available for flow between the grid cells is an average of the available flow area in each direction across the cell. This adjusted cross-sectional area is factored into the diffusive wave approximation used in the 2D OL solver. For numerical details see Multi-cell Overland Flow Method *(V.2 p. 275)* in the Reference manual.

The multi-grid overland flow solver is typically used where an accurate bathymetric description is more important than the detailed flow patterns. This is typically the case for most inland flood studies. In other words, the distribution of flooding and the area of flooding in an area is more important than the rate and direction of ingress.

The multi-grid option is described in more detail in the chapter Multi-cell Overland Flow *(V.1 p. 181)*.

Overland Flow Performance

Calculation of overland flow can be a significant source of numerical instabilities in MIKE SHE. Depending on the model setup, the overland flow time step can become very short - making the simulation time very long.

The chapter Surface Water in MIKE SHE *(V.1 p. 171)* contains many more details on simulating overland flow and the coupling to MIKE 11. In particular the section Overland Flow Performance *(V.1 p. 178)* contains detailed information on improving the performance of the overland flow in your model.

MIKE FLOOD

MIKE SHE provides a useful means to simulate 2D flooding on a flood plain that includes the influence of infiltration and evapotranspiration. However, the detailed simulation of surface water flow paths and velocities on a flood plain can be very difficult. If you need to simulate more complex flood plain flow, for example the impact of flood plain structures and embankments, you may need to use MIKE FLOOD instead of MIKE SHE.

MIKE FLOOD is combination of the 2D MIKE 21 surface water model for detailed, accurate flow on the flood plain, and MIKE 11 for channel flow. MIKE FLOOD allows you to define flood plain structures such as embankments and culverts that can have very significant impacts on flow velocity and direction. MIKE FLOOD can also more accurately simulate flood wave propagation on a surface simply because of the higher order numerical method used.

2.9 Unsaturated Flow

Unsaturated flow is one of the central processes in most model applications. The unsaturated zone is usually heterogeneous and characterized by cyclic fluctuations in the soil moisture as water is replenished by rainfall and removed by evapotranspiration and exchange to the groundwater table.

Unsaturated flow is primarily vertical since gravity plays the major role during infiltration. Therefore, unsaturated flow in MIKE SHE is calculated only vertically in one-dimension, which is sufficient for most applications. However, this assumption may not be valid, for example, on steep hill slopes.

There are three options in MIKE SHE for calculating vertical flow in the unsaturated zone:

- the full Richards equation, which is the most computationally intensive, but also the most accurate when the unsaturated flow is dynamic;
- a simplified gravity flow procedure, which ignores capillary forces, and is suitable when you are primarily interested in the time varying recharge and not the dynamics in the unsaturated zone; and
- a simple two-layer water balance that is suitable when the water table is shallow and groundwater recharge is primarily influenced by evapotranspiration in the root zone.

More detailed information on the setup and calculation of unsaturated flow is found in the chapter Unsaturated Groundwater Flow *(V.1 p. 243)*.

The Technical Reference manual includes detailed information on the calculation methods - Unsaturated Flow - Reference *(V.2 p. 319)*.

2.9.1 Soil Profiles

The unsaturated zone usually includes several different soil types. For example, the soil profile could include a compacted upper zone or a loamy active layer with lots of humus and other organic matter. The lower layers could be alluvial zones with interbedded clay lenses, or less weathered bedrock layers.

The soil profile that you define can be as detailed as the available information. There is no restriction on the amount of detail that you can input. However, from a practical point of view, you are probably better off grouping similar soil types together and simplifying the soil profiles as much as possible.

The specified soil profile depth must be deeper than the vertical discretization.

In the 2-Layer UZ method, the soil profile is uniform with depth.

Soil properties database

The soil properties database is used to define the unsaturated flow properties and relationships for the different soil types, if you are using one of the finite difference UZ methods (i.e. the Richards Equation and Gravity methods). In the database, each soil type has a set of properties, and the profile is composed of different soil types.

Vertical Grid Discretisation

The vertical discretisation of the soil profile typically contains small cells near the ground surface and increasing cell thickness with depth. However, the soil properties are averaged if the cell boundaries and the soil property definitions do not align.

The discretisation should be tailored to the profile description and the required accuracy of the simulation. If the full Richards equation is used the vertical discretisation may vary from 1-5 cm in the uppermost grid points to 10-50 cm in the bottom of the profile. For the Gravity Flow module, a coarser discretisation may be used. For example, 10-25 cm in the upper part of the soil profile and up to 50-100 cm in the lower part of the profile. Note that at the boundary between two blocks with different cell heights, the two adjacent boundary cells are adjusted to give a smoother change in cell heights.

2.9.2 Initial Conditions

The default initial conditions for unsaturated flow are usually good, which means that initially there is no flow in the soil column. This means that the initial soil moisture content is based on the defined pressure-saturation relationship.

If the 2-Layer UZ method is chosen, then the initial conditions are automatically defined by the method.

2.9.3 Macropore flow

Macropores include vertical cracks, as well as worm and root holes in the soil profile. Macropores increase the rate of infiltration through the soil column.

Simple bypass flow - A simple empirical function is used to describe simple bypass flow in macropores. The infiltration water is divided into one part that flows through the soil matrix and another part, which is routed directly to the groundwater table, as bypass flow.

The bypass flow is calculated as a fraction of the net rainfall for each UZ time step. Typically, macropore flow is highest in wet conditions when water is flowing freely in the soil (e.g. moisture content above the field capacity, θ_{FC}) and zero when the soil is very dry (e.g. moisture content at the wilting point, θ_{WP}).

Simple bypass flow is commonly used to provide some rapid recharge to the groundwater table. In many applications, if all the rainfall is infiltrated normally, the actual evapotranspiration is too high and very little infiltration reaches the groundwater table. In reality some infiltration recharges the groundwater system due to macropores and sub-grid variability of the soil profile. In other words, there is usually sub-areas in a grid cell with much higher infiltration rates or where the unsaturated zone thickness is much less than that defined by the average topography in the cell.

Simple bypass flow is described in the Reference section under Simplified Macropore Flow (bypass flow) *(V.2 p. 334)*.

Full Macropore Flow - Macropores are defined as a secondary, additional continuous pore domain in the unsaturated zone. Full macropore flow is generally reserved for very detailed unsaturated root-zone models, especially in water quality models where solute transformations are occurring in the macropores. Full bypass flow is described in the Reference section under Full Macropore Flow *(V.2 p. 336)*.

2.9.4 Green and Ampt infiltration

The Green and Ampt algorithm is an analytical method to increase infiltration in dry soils due to capillarity. It is not applicable when using the Richards Equation method because capillarity is already included. However, when capillarity is not included (i.e. in the Gravity flow and 2-Layer methods), dry soils will absorb rainfall at a much higher rate than the defined infiltration rate (saturated hydraulic conductivity).

For more information on the Green and Ampt method, see the section Green and Ampt Infiltration *(V.2 p. 340)* in the Reference Guide.

2.9.5 UZ Column Classification

Calculating unsaturated flow in all grid squares for large-scale applications can be time consuming. To reduce the computational burden MIKE SHE enables you to compute the UZ flow in a reduced subset of grid squares. The subset classification is done automatically by the pre-processing program according to soil and, vegetation distribution, climatic zones, and depth to the groundwater table.

Column classification can decrease the computational burden considerably. However, the conditions when it can be used are limited. Column classification is either not recommended or not allowed when

- the water table is very dynamic and spatially variable because the classification is not dynamic,
- if the 2 layer UZ method is used because the method is fast and the benefit would be limited,
- if irrigation is used in the model because irrigation zones are not a classification parameter, and
- if flooding and flood codes are used, since the depth of ponded water is not a classification parameter

Thus, **the column classification should probably be avoided today** because the models have become more complex, MIKE SHE has become more efficient and computers have become faster.

If the classification method is used, then there are three options for the classification:

- **Automatic classification** With automatic option, the UZ columns are divided up based on the internal classification rules. The depth to the water table, Groundwater Depths used for UZ Classification *(V.2 p. 136)*, is the lower UZ boundary condition.
- **Specified classification** With the specified option, you must supply a list of grid codes, Specified classification *(V.2 p. 138)*, that defines the computational column and the columns to which the results will be applied.
- **Calculated in all Grid points (default)** In many models the classification system is not feasible or recommended. In this case, the UZ flow will be calculated in all soil columns.

- **Partial Automatic** Finally a combination of the Automatic classification and the Specified classification is available, where an. Integer Grid Code file must be provide (see Partial automatic classification *(V.2 p. 137)*) to define the different areas.

2.9.6 Coupling Between Unsaturated and Saturated Zone

A correct description of the recharge process is rather complicated because the water table rises as water enters the saturated zone and affects flow conditions in the unsaturated zone. The actual rise of the groundwater table depends on the moisture profile above the water table, which is a function of the available unsaturated storage and soil properties, plus the amount of net groundwater flow (horizontal and vertical flow and source/sink terms).

The main difficulty in describing the linkage between the two the saturated (SZ) and unsaturated (UZ) zones arises from the fact that the two components (UZ and SZ) are explicitly coupled (i.e. they run in parallel and exchange water only at specific times). Explicit coupling of the UZ and SZ modules is used in MIKE SHE to allow separate time steps that are representative of the UZ (minutes to hours) and the SZ (hours to days) domains.

Error in the mass balance originates from two sources:

- keeping the water table constant during a UZ time step, and
- using an incorrect estimate of the specific yield, S_y, in the SZ-calculations.

In the first case above, mass balance and convergence problems can be addressed by making the maximum UZ time step closer to the SZ time step.

In the second case above, the MIKE SHE forces the specific yield of the top SZ layer to be equal to the “specific yield” of the UZ zone as defined by the difference between the specified moisture contents at saturation, θ_s, and field capacity, θ_{fc}.This correction is calculated from the UZ values in the UZ cell in which the initial SZ water table is located. For more information see Specific Yield of the upper SZ numerical layer *(V.1 p. 252)*.

UZ - SZ limitations

The coupling between UZ and SZ is limited to the top calculation layer of the saturated zone. This implies that:

- As a rule of thumb, the UZ soil profiles should extend to just below the bottom of the top SZ layer.
- However, if you have a very thick top SZ layer, then the UZ profiles must extend at least to below the deepest depth of the water table.
- If the top layer of the SZ model dries out, then the UZ model usually assumes a lower pressure head boundary equal to the bottom of the uppermost SZ layer.
- All outflow from the UZ column is always added to the top node of the SZ model.
- UZ nodes below the water table and the bottom of the top SZ layer are ignored.

For more detailed information on the UZ-SZ coupling see Unsaturated Flow - Reference *(V.2 p. 319)*. The chapter, Unsaturated Groundwater Flow *(V.1 p. 243)*, also contains more detailed information on the setup and evaluation of the unsaturated model.

2.10 Saturated Groundwater Flow

The Saturated Zone (SZ) component of MIKE SHE calculates the saturated subsurface flow in the catchment. In MIKE SHE, the saturated zone is only one component of an integrated groundwater/surface water model. The saturated zone interacts with all of the other components - overland flow, unsaturated flow, channel flow, and evapotranspiration.

By comparison, MODFLOW only simulates saturated groundwater flow. All of the other components are either ignored (e.g. overland flow) or are simple boundary conditions for the saturated zone (e.g. evapotranspiration). On the other hand, there are very few difference between the MIKE SHE numerical engine and MODFLOW. The differences are limited to the discretisation and to some differences in the way some of the boundary conditions are defined.

Finite Difference Method

When the Finite Difference method has been selected, MIKE SHE allows for a fully three-dimensional flow in a heterogeneous aquifer with shifting conditions between unconfined and confined conditions. The spatial and temporal variations of the dependent variable (the hydraulic head) is described mathematically by the 3-dimensional Darcy equation and solved numerically by an iterative implicit finite difference technique. MIKE SHE includes two groundwater solvers - the SOR groundwater solver based on a successive over-relaxation solution technique and the

PCG groundwater solver based on a preconditioned conjugate gradient solution technique.

Linear Reservoir Method

The linear reservoir module for the saturated zone in MIKE SHE was developed to provide an alternative to the physically based, fully distributed model approach. In many cases, the complexity of a natural catchment area poses a problem with respect to data availability, parameter estimation and computational requirements. In developing countries, in particular, very limited information on catchment characteristics is available. Satellite data may increasingly provide surface data estimates for vegetation cover, soil moisture, snow cover and evaporation in a catchment. However, subsurface information is generally very sparse.

The linear reservoir method for the saturated zone may be viewed as a compromise between limitations on data availability, the complexity of hydrological response at the catchment scale, and the advantages of model simplicity.

For example, combining lumped parameter groundwater with physically distributed surface parameters and surface water often provides reliable, efficient

- Assessments of water balance and runoff for ungauged catchments,
- Predictions of hydrological effects of land use changes, and
- Flood prediction

2.10.1 Conceptual Geologic Model for the Finite Difference Approach

Before starting to develop a groundwater model, you should have developed a conceptual model of your system and have at your disposal digital maps of all of the important hydrologic parameters, such as layer elevations and hydraulic conductivities.

In MIKE SHE you can specify your subsurface geologic model independent of the numerical model. The parameters for the numerical grid are interpolated from the grid independent values during the preprocessing.

The geologic model can include both geologic layers and geologic lenses. The former cover the entire model domain and the later may exist in only parts of your model area.

You also have the option to set up your conceptual model

- by layers, where you specify the property distribution in the layer, or

- by units, where you specify the unit distribution in the layer.

Lenses

In building a geologic model, it is typical to find discontinuous layers and lenses within the geologic units. The MIKE SHE setup editor allows you to specify such units - again independent of the numerical model grid. Lenses are often useful when building up a geologic model where the units are discontinuous. For example, a coarse alluvial flood plain aquifer can be defined as a lense inside of a regional bedrock aquifer.

Lenses are specified by defining either a .dfs grid file or a polygon .shp file for the extents of the lenses. The .shp file can contain any number of polygons, but the user interface does not use the polygon names to distinguish the polygons. If you need to specify several lenses, you can use a single file with many polygons and specify distributed property values, or you can specify multiple individual polygon files, each with unique property values.

There are a number of special considerations when working with lenses in the geologic model.

- **Lenses override layers** - That is, if a lense has been specified then the lense properties take precedence over the layer properties and a new geologic layer is added in the vertical column.
- **Vertically overlapping lenses share the overlap** - If the bottom of lense is below the top of the lense beneath, then the lenses are assumed to meet in the middle of the overlapping area.
- **Small lenses override larger lenses** - If a small lense is completely contained within a larger lense the smaller lense dominates in the location where the small lense is present.
- **Negative or zero thicknesses are ignored** - If the bottom of the lense intersects the top of the lense, the thickness is zero or negative and the lense is assumed not to exist in this area.

2.10.2 Specific Yield of upper SZ layer

MIKE SHE forces the specific yield of the top SZ layer to be equal to the “specific yield” of the UZ zone as defined by the difference between the specified moisture contents at saturation, θ_s, and field capacity, θ_{fc}.This correction is calculated from the UZ values in the UZ cell in which the initial SZ water table is located. This is reflected in the pre-processed data.

For more information on the SZ-UZ specific yield see Specific Yield of the upper SZ numerical layer *(V.1 p. 252)*.

2.10.3 Numerical Layers

There is no restriction in MIKE SHE on the number of numerical layers in the SZ model. However, there may be practical limitations depending on your computer resources. As a rule of thumb, each additional SZ layer will significantly slow down your simulation.

The upper boundary of the top layer is always either the infiltration/exfiltration boundary, which in MIKE SHE is calculated by the unsaturated zone component or a specified fraction of the precipitation if the unsaturated zone component is excluded from the simulation.

The lower boundary of the bottom layer is always considered impermeable.

In MIKE SHE, the rest of the boundary conditions can be divided into two types: Internal and Outer. If the boundary is an outer boundary then it is defined on the boundary of the model domain. Internal boundaries, on the other hand, must be inside the model domain.

The UZ model only interacts and exchanges water with the top SZ layer. Therefore, the bottom of the top SZ layer is usually specified below the lowest water table level, so that the top SZ layer always includes the water table.

2.10.4 Groundwater Drainage

Saturated zone drainage is a special boundary condition in MIKE SHE used to defined natural and artificial drainage systems that cannot be defined in MIKE 11. It can also be used to simulate simple, lumped conceptual surface water drainage of groundwater.

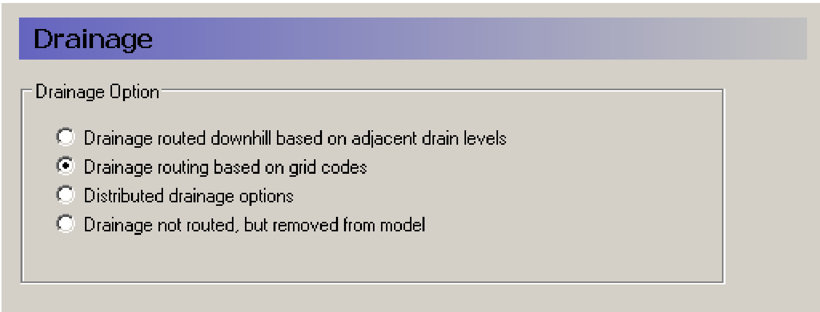
Saturated zone drainage is removed from the layer of the SZ layer containing the drain level. Water that is removed from the saturated zone by drains is routed to local surface water bodies, local topographic depressions, or out of the model. The amount of drainage is calculated based on the groundwater head and the drain level using a linear reservoir formulation.

When water is removed from a drain, it is immediately moved to the recipient. In other words, the drain module assumes that the time step is longer than the time required for the drainage water to move to the recipient. Conceptually, you can use a “full pipe” analogy. The drain is a pipe full of water. As groundwater is added to the pipe, an equivalent amount of water must be discharged immediately out of the opposite end of the pipe because the water is incompressible and there is no additional storage in the pipe.

Each cell requires a drain level and a time constant (which is the same as a leakage factor). Both drain levels and time constants can be spatially defined. A typical drainage level might be 1m below the ground surface and a typical time constant may be between 1e-6 and 1 e-7 1/s.

Drainage reference system

MIKE SHE requires a reference system for linking the drainage to a recipient node or cell. There are four different options for setting up the drainage source-recipient reference system



- **Drain Levels** The drainage recipient is calculated based on the drain levels in all the down gradient cells. That is, the location of the recipient cell is calculated as if the drain water was flowing downhill (based on the drain levels). This is the most common method of specifying drainage routing and the default setting.
- **Drain Codes** The drainage recipient is specified by the user based on a distribution map of integer code values.
- **Distributed option** With this option there are several different drainage possibilities, including a combination of Codes and Levels. The Distributed option can also be used to define a specific MIKE 11 H-point or MOUSE manhole as a recipient.
- **Removed** The fourth option is simply a head dependent boundary that removes the drainage water from the model. This method does not involve routing and is exactly the same as the MODFLOW Drain boundary.

2.10.5 Groundwater wells

Groundwater wells can be included in your SZ simulation. The groundwater well locations, filter depth, pumping rates etc. are stored in a .wel file that is edited using the Well editor *(V.2 p. 229)*.

2.10.6 Linear Reservoir Groundwater Method

In the linear reservoir method, the entire catchment is subdivided into a number of subcatchments and within each subcatchment the saturated zone is represented by a series of interdependent, shallow interflow reservoirs, plus a number of separate, deep groundwater reservoirs that contribute to stream baseflow.

The lateral flows to the river (i.e. interflow and baseflow) are by default routed to the river links that neighbour the model cells in the lowest topographical zone in each subcatchment.

Interflow will be added as lateral flow to river links located in the lowest interflow storage in each catchment. Similarly, baseflow is added to river links located within the baseflow storage area

Three Integer Grid Code maps are required for setting up the framework for the reservoirs,

- a map with the division of the model area into Subcatchments,
- a map of Interflow Reservoirs, and
- a map of Baseflow Reservoirs.

The division of the model area into subcatchments can be made arbitrarily. However, the Interflow Reservoirs must be numbered in a more restricted manner. Within each subcatchment, all water flows from the reservoir with the highest grid code number to the reservoir with the next lower grid code number, until the reservoir with the lowest grid code number within the subcatchment is reached. The reservoir with the lowest grid code number will then drain to the river links located in the reservoir.

For baseflow, the model area is subdivided into one or more Baseflow Reservoirs, which are not interconnected. However, each Baseflow Reservoir is further subdivided into two parallel reservoirs. The parallel reservoirs can be used to differentiate between fast and slow components of baseflow discharge and storage.

For more detailed information on the Linear Reservoir method, see the section Linear Reservoir Method *(V.2 p. 373)* in the Reference manual.

2.11 Storing of results

The integrated nature of MIKE SHE means that very large amounts of output can be generated during a simulation. Thus, the output specification is designed to allow you to save only the necessary information. However, the downside is that if you failed to save a specific output during the simulation run, then you will have to re-run the simulation to obtain this information.

The output in MIKE SHE can be divided into two types: Time series and Grid Series. From a practical viewpoint, time series output generated during the simulation is saved at every simulation time step, whereas grid series output is saved at a specified time interval. You can easily obtain missing time series from a grid series output file, but the time resolution will be the same as the specified saving interval.

Thus, at the locations where you want detailed results of a particular value, you define a point in the Detailed Time Series dialogue. If you are interested in the spatial and general temporal trends of a parameter, then it is usually sufficient to save only the Grid Series output.

Water balance output

The water balance is often a vital part of assessing the results of a MIKE SHE simulation. The water balance describes the flow of water within your catchment.

If the water balance checkbox is turned on, then all of the data necessary for calculating the water balance will be automatically saved. If you do not check on this box, then you will not be able to calculate a water balance for your simulation and you will have to re-run your simulation to generate the needed output data.

Water balances are calculated using a separate water balance utility, which is described in detail in the chapter Using the Water Balance Tool *(V.1 p. 105)*.

Hot start output

It is often very useful to be able to start a simulation from a consisten predefined starting point. For example, you may want to simulate the first five years and then start all of your scenarios from this starting point. This could save you considerable calculation time.

You can append individual simulation output files together using the Concatination tool in the MIKE Zero Toolbox. However, you will not be able to create a water balance of the entire period including the first five years.

Using the hot start involves:

- Turning on the hot start by checking the hot start checkbox,

- Then either storing the hot start data at the end of the simulation only (which will create only one possible hot start point), or
- Storing the hot start information at regular storing intervals. Frequent hot start storage can create very large files and may slow down the simulation as all of this data must be written to the hard disk.

Water quality output

If you want to run a water quality simulation after the water movement simulation, then you must turn on the storing of the water quality output. If the water quality is turned on the main Simulation Specification dialogue, then the water quality output is automatically stored during the water movement simulation. Manual activation is only required if the water movement simulation is being run separately.

Storing intervals

Storing intervals for both the water movement and the mass balance define the frequency at which grid data is stored. Grid data is the most space consuming output.

The grid output data is viewed in the Results Viewer and is used for calculating the water balance. Thus, you cannot calculate a water balance or spatial output maps at a finer temporal resolution than the storing intervals. If you want detailed output of a specific parameter at more frequent intervals, then you should use the Detailed Time Series Output function.

2.11.1 Detailed Time Series Output

The detailed time series output allows you to save any output parameter at every time step of the particular process. Since the different processes run at different time steps, you may get, for example, much more detailed output for the unsaturated zone than for the saturated zone.

Each item in the Detailed time series is displayed automatically in an HTML format graph on the Run tab while the simulation is running.

You can also add observation data to each of the detailed time series items

A full list of available output items, as well as more detail on the individual items is found in the section Output Items *(V.1 p. 72)*.

Importing ASCII data

Detailed MIKE SHE Time Series data can be imported directly into the Detailed MIKE SHE Time Series dialogue using the Import button. The data file must be a tab- delimited ASCII file without a header line. The file must contain the following fields and be in the format specified below.

```
Name>data typeCode>NewPlot>X >Y >Depth>UseObsdata>dfs0Filename>dfs0ItemNumber
```

where the > symbol denotes the Tab character and

Name - is the user specified name of the observation point. This is the name that will be used for the time series item in the Dfs0 file created during the simulation.

data typeCode - This is a numeric code used to identify the output data type. See the list of available Data Type Codes in Table 3.1 and Table 3.2 under Output Items *(V.1 p. 72)*.

NewPlot - This is a flag to specify whether a new detailed time series HTML-plot will be created on the Results Tab:

0 = the output will be added to the previous plot.

1 = Create a new plot

X, Y - This is the (X, Y) map coordinates of the point in the same EUM units (ft, m, etc.) as specified in the EUM Database for Item geometry 2-dimensional. (see EUM Data Units)

Depth -This is the depth of the observation point below land surface for subsurface observation points. The value is in same EUM units (ft, m, etc.) as specified in the EUM Database for Depth Below Ground (see EUM Data Units). A depth value must always be included, even if not needed.

UseObsData - This is a flag to specify whether or not an observation file needs to be input: 0 = No; 1 = Yes

dfs0FileName - This is the file name of the dfs0 time series file with observation data. The path to the dfs0 file must be relative to the directory containing the MIKE SHE *.she document. The .dfs0 extension is added to the file name automatically and should be not be included in the file name. For example, the following input line

```
.\Time\Calibration\GroundwaterObs
```

refers to the file `GroundwaterObs.dfs0` located in the subdirectory `Time\Calibration`, which is found in the same directory as the .she model document.

dfs0ItemNumber - This is the Item **number** of the observation data in the specified DFS0 file.

Import Example

The following is a simple example of a tab delimited ASCII file with two MIKE SHE observation points, where the file containing the observations is called `obsdata.dfs0`:

```
Obs_1    20   1234500. 456740.  0.   0    .\time\obsdata    1
Obs_2    15   1239700. 458900.  10.  1    .\time\obsdata    2
Obs_3    16   0241500. 459310.  20.  1    .\time\obsdata    3
```

2.11.2 Detailed MIKE 11 Time Series Output

MIKE 11 output is normally analysed using the MIKE View program. However, the default MIKE 11 output is only at specified time intervals. Every item in the Detailed MIKE 11 Time Series table is output at every MIKE 11 time step.

Like the Detailed Time Series Output (above), each item in this table is output automatically to an HTML graph in the Run Tab. You can also specify an observation file for each item, which is more convenient that using MIKE VIEW.

Importing ASCII data

Detailed MIKE 11 Time Series data can be imported directly into the Detailed MIKE 11 Time Series dialogue using the Import button. The data file must be a tab- delimited ASCII file without a header line. The file must contain the following seven fields:

Name - is the user specified name of the observation point. This is the name that will be used for the time series item in the Dfs0 file created during the simulation.

data typeCode - This is a numeric code used to identify the output data type (1=water level, 2=discharge).

Branch_name - The name of the MIKE 11 branch

Chainage - The location of the MIKE 11 h-point or q-point (the nearest one will be taken within a tolerance).

UseObsData - This is a flag to specify whether or not an observation file needs to be input: 0 = No; 1 = Yes

dfs0FileName - This is the file name of the dfs0 time series file with observation data. The path to the dfs0 file must be relative to the directory containing the MIKE SHE *.she document. The .dfs0 extension is added to the file name automatically and should be not be included in the file name. For example, the following input line

```
.\Time\Calibration\GroundwaterObs
```

refers to the file `GroundwaterObs.dfs0` located in the subdirectory `Time\Calibration`, which is found in the same directory as the .she model document.

dfs0ItemNumber - This is the Item **number** of the observation data in the specified DFS0 file.

2.11.3 Grid Series Output

The grid time series output allows you to save spatial output data at every saved time step of the particular process.

Each item in the Grid time series table is listed on the Run tab. You can open and plot each of these items while the simulation is running.

A full list of available output items, as well as more detail on the individual items is found in the section Output Items *(V.1 p. 72)*.

RESULTS AND CALIBRATION

3 MIKE SHE RESULTS

The available output from MIKE SHE depends on the processes selected in the Simulation Specification dialogue. Thus, for example, results for Overland Flow only appear when Overland flow is being calculated.

3.1 Output Files

The output from MIKE SHE is stored in a combination of files.

.sheres - this is an ASCII file that is a catalogue of all the output files associated with a simulation.

.frf - this is a binary output file containing all of the static information on the simulation, as well as all of the time series results that cannot be easily stored in a dfs format.

dfs files - The rest of the output is stored in a series of dfs0, dfs2 and dfs3 files.

The dfs file format is a binary time series format. Each file can contain multiple output items, but each of the items must be stored at the same time step interval. Thus, the output for each of the processes that has an independent storing time step is stored in separate output file (e.g. OL water depth is stored separately from SZ Recharge, even though each is a 2D output item).

Viewing Output Files

The primary means of viewing the dfs2 and dfs3 output is the Results Viewer. The gridded output files can be also viewed in the Grid Editor. The Grid Editor includes icons in the icon bar to step between layers and time steps, as well as to switch between output items.

Dfs0 output is viewed most easily in the Time Series Editor.

All three of these are MIKE Zero tools and are described in the MIKE Zero documentation. See the section on The Results viewer *(V.1 p. 81)* for more details on the Results Viewer.

3.1.1 Log files

There are three main log files (where the xx refers to your document file name). All three of these files are found in the default results directory along with the other result files.

xx_PP_Print.log - This is the main output file from the pre-processor.

xx_WM_Print.log - This is the main output from the water movement engine.

xx_WQ_Print.log - This is the main output from the water quality engine.

3.2 Multiple simulations

There are several things to consider when running multiple MIKE SHE simulations.

- If you run simulations one after another, the results files will be overwritten unless you move or copy them first.
- If you set up multiple simulations using the same MIKE 11 model, the MIKE 11 results files will be overwritten. To prevent this, you must create different .sim11 files and change the results file name in the .sim11 file.
- If you are starting from a Hot Start file, then you need to be careful that the Hot start file you are using is the one you want. The easiest way to ensure this is to change the name of the hotstart file.
- You can run a chain of models - hot starting from the end of the last simulation. This can be done using a batch command, for example. You can concatenate the results files using the Concatenate Tool in the MIKE Zero toolbox. This will allow you to build up a set of continuous results files that includes the entire simulation. However, you will not be able to create a continuous water balance because the .sheres file and the .frf files will not be correct.

3.3 Output Items

Some of the available output items are calculated as part of another process. For example, the depth of overland water is calculated based on seepage to and from the groundwater and as part of the MIKE 11 surface water calculations, even if the overland flow is not directly simulated.

Furthermore, some of the output items require that more than one process be simulated. For example, the leaf area index is only available if both evapotranspiration and unsaturated flow are calculated.

In the absence of an explicit remark, the sign convention for MIKE SHE's output is positive in the positive direction. In other words, all flows in the direction of increasing X, Y and Z coordinates are positive. Thus, vertical downward flows, such as infiltration are negative.

Flows that do not have a direction are positive if storage or outflow is increasing. Thus, all flows leaving the model are positive, and water balance errors are positive if the model is generating water.

Also important to remember is that the output items related to flow are accumulated over the storing time step. In many cases, these values are required for the Water Balance program described in the section Using the Water Balance Tool *(V.1 p. 105)*. The values that are part of the water balance are automatically turned on when the water balance option is selected.

However, the output items that are not flows, such as temperature, water depth and Courant number represent the current value at the end of the storing time step.

Finally, some of the output items are actually input items. For example, precipitation is usually input as a time series for several polygons or grid code areas. The output file is a fully distributed dfs2 version of the input time series files.

The available output items for gridded data and time series data are listed in Table 3.1 and Table 3.2. Table 3.2 lists a number of additional output items, such as the number of solver iterations, that can only be displayed as a time series.

Code - In Table 3.1 and Table 3.2, a **Data Type Code** is needed when importing time series items into the Detailed time series output *(V.2 p. 186)* dialogue.

The following are some additional notes on the gridded output items

3.3.1 Overland flow velocity

The overland flow velocity in the list of available output items is used for the water balance calculations. **It is not the cell velocity**.

The cell velocity cannot be directly calculated because the overland water depth is an instantaneous value output at the end of storing time step.

The overland flow in the x- and y- directions are mean-step accumulated over the storing time step. Thus, the overland flow is not the flow in the cell, but rather the accumulated flow across the cell face on the positive side of the cell.

You may be tempted to calculate a flow velocity from these values. But, you can easily have the situation where the accumulated flow across the boundary is non-zero, but at the end of the storing time step, the water depth is zero. Or, you could have a positive inflow and a zero outflow, which may be misleading when looking at a map of flow velocities.

The overland flow velocities are discussed in more detail in the section, Overland Flow Velocities *(V.1 p. 177)*.

3.3.2 Recharge

The data item Total recharge to SZ (positive for downwards flow) contains the following items:

- Exchange between UZ and SZ, calculated by the UZ solver
- Recharge from Bypass or Macropores if included
- Direct flow between SZ and overland (when groundwater table is above ground)
- Transpiration from SZ (when the roots reach the groundwater)

So neither baseflow (SZ-M11) nor drain flow is included. These items can be found in the two data items:

- - SZ exchange flow with river (positive when flow from SZ to M11, negative the other way)
- - SZ drainage flow from point

The Total recharge to UZ should correspond with the water balance items, but note the sign. The easiest way to check this is to look at a Saturated zone water balance, table type:

- Recharge: exchange between UZ and SZ + Bypass flow or Macropore recharge if included + direct flow between SZ and Overland + transpiration from SZ, all POSITIVE UPWARDS
- Drain: Drainage flow from point
- SZ->River: SZ exchange flow with river, positive for flow to the river

Note the various units. The total recharge result type is a flux (i.e. mm/d, mm/h, m/s, etc) depending on the chosen user unit for Recharge. Whereas, the SZ river exchange and Drainage are flows (i.e. m3/s or similar). The Water balance output is in units of Storage depth (mm). That is, it is normalized with the catchment area (using the area inside the outer boundary), or the subcatchment area if a sub-catchment water balance has been extracted.

3.3.3 Summary of all output items

The following table includes a summary of all output items for both the gridded data and the time series data.

Table 3.1 Available output items for gridded data and time series.
–Key to symbols
–ET - Evapotranspiration
–OL - Finite Difference Overland Flow
–SubOL - Sub-catchment based Overland Flow
–UZ - Richards or Gravity Unsaturated flow,
–2LUZ - 2-Layer Unsaturated Water Balance
–SZ - Finite Difference Saturated Zone flow,
– LR - Linear Reservoir groundwater
–AD - Advection Dispersion (Water Quality)
–PT - Particle Tracking
–SM - Snow melt

Code	Output Item	Appears with these processes	
10	precipitation rate This is the distributed actual precipitation in the model, accumulated per storing time step.	Always	
128	average water content in the root zone	UZ+ET	2LUZ+ET
11	rooting depth	UZ+ET	2LUZ+ET
12	leaf area index	UZ+ET	2LUZ+ET
182	crop coefficient	UZ+ET	2LUZ+ET
15	actual evapotranspiration	UZ+ET	2LUZ+ET
16	actual transpiration	UZ+ET	2LUZ+ET
13	actual soil evaporation	UZ+ET	
17	actual evaporation from interception	UZ+ET	2LUZ+ET
18	actual evaporation from ponded water	UZ+ET	2LUZ+ET
19	canopy interception storage	UZ+ET	2LUZ+ET
14	evapotranspiration from SZ	SZ+UZ+ET	SZ+2LUZ+ET
	total snow storage	ET+SM	
	Dry snow storage	ET+SM	
	Wet snow storage	ET+SM	
	Wet snow storage fraction	ET+SM	
	Fraction of cell area covered by snow	ET+SM	
	Precipitation and Irrigation added to snow	ET+SM	
	Total snow converted to overland flow	ET+SM	
	Freezing due to air temperature	ET+SM	
	Melting due to air temperature	ET+SM	

Table 3.1 *Available output items for gridded data and time series.*
–Key to symbols
–ET - Evapotranspiration
–OL - Finite Difference Overland Flow
–SubOL - Sub-catchment based Overland Flow
–UZ - Richards or Gravity Unsaturated flow,
–2LUZ - 2-Layer Unsaturated Water Balance
–SZ - Finite Difference Saturated Zone flow,
– LR - Linear Reservoir groundwater
–AD - Advection Dispersion (Water Quality)
–PT - Particle Tracking
–SM - Snow melt

Code	Output Item	Appears with these processes			
	Melting due to SW solar radiation	ET+SM			
	Melting due to energy in rain	ET+SM			
99	Snow evaporation	ET+SM			
61	depth of overland water This is the instantaneous depth of water at the end of the storing time step.	OL	SZ	SubOL	
58	overland flow in x-direction This is the flow across the boundary from $cell_i$ to $cell_{i+1}$ in volume/time e.g. m^3/s.	OL			
59	overland flow in y-direction (this is the flow across the boundary from $cell_i$ to $cell_{i+1}$ in volume/time e.g. m^3/s)	OL			
	flow from flooded areas to river				
	Overland flow to MOUSE				
	External sources to Overland (for OpenMI)				
62	paved area drainage to river or MOUSE	OL+M11/MOUSE +Drainage+ Paved			
	Overland water elevation				
	Mean OL wave courant number (explicit OL)				
	Max OL wave courant number (explicit OL)				
	Max output OL-OL per cell volume (explicit OL)				
141	Water content in root zone (2-layer UZ)	2LUZ			
142	Water content below root zone (2-layer UZ)	2LUZ			
143	Maximum water content (2-layer UZ)	2LUZ			
144	Minimum water content (2-layer UZ)	2LUZ			
121	infiltration to UZ (negative)	UZ	2LUZ	SZ	LR
122	exchange from UZ to SZ (negative)	UZ	2LUZ	SZ	LR
	bypass flow UZ (negative)				
57	UZ deficit	UZ	2LUZ		
	infiltration to macropores (negative)				
	macropore recharge to SZ (negative)				

Table 3.1 Available output items for gridded data and time series.
–Key to symbols
–ET - Evapotranspiration
–OL - Finite Difference Overland Flow
–SubOL - Sub-catchment based Overland Flow
–UZ - Richards or Gravity Unsaturated flow,
–2LUZ - 2-Layer Unsaturated Water Balance
–SZ - Finite Difference Saturated Zone flow,
– LR - Linear Reservoir groundwater
–AD - Advection Dispersion (Water Quality)
–PT - Particle Tracking
–SM - Snow melt

Code	Output Item	Appears with these processes			
37	average soil moisture content in top 5 compartments	LR+UZ			
	Total recharge to SZ (positive down) This is a sum of: + exchange from UZ to SZ + bypass flow to SZ (if active) + macropore flow to SZ (if active) + flow between OL and SZ (if water table at/above topography) + transpiration from SZ SZ drainage and exchange to MIKE 11 is NOT included in this term. The Recharge is positive downwards and has the EUM units of precipitation rate, so that it can be used directly as input to another MIKE SHE model in place of precipitation.				
119	rate of change in UZ storage	UZ			
123	epsilon calculated in UZ	UZ	2LUZ		
120	accumulated error in UZ (water balance in the UZ cells only)	UZ			
	column mean macropore water content above GW				
	column total net exchange matrix to macropores				
	column total exchange matrix to macropores				
	column total exchange macropores to matrix				
45	groundwater feedback to the unsaturated zone	LR+UZ		LR+2LUZ	
117	unsaturated zone flow	UZ			
118	water content in unsaturated zone	UZ			
159	pressure head in unsaturated zone	UZ			
129	root water uptake	UZ+ET			
	macropore water content				
	root water uptake				
20	irrigation: actual water content in root zone	UZ+ET+Irrigation			
135	irrigation: soil moisture deficit in root zone	UZ+ET+Irrigation			
21	total irrigation	UZ+ET+Irrigation			
26	irrigation from river	M11+UZ+ET+Irrigation			

Table 3.1 Available output items for gridded data and time series.
–Key to symbols
–ET - Evapotranspiration
–OL - Finite Difference Overland Flow
–SubOL - Sub-catchment based Overland Flow
–UZ - Richards or Gravity Unsaturated flow,
–2LUZ - 2-Layer Unsaturated Water Balance
–SZ - Finite Difference Saturated Zone flow,
– LR - Linear Reservoir groundwater
–AD - Advection Dispersion (Water Quality)
–PT - Particle Tracking
–SM - Snow melt

Code	Output Item	Appears with these processes
28	irrigation from wells	SZ+UZ+ET+Irrigation
22	irrigation from external source	UZ+ET+Irrigation
23	irrigation index	UZ+ET+Irrigation
24	irrigation shortage	UZ+ET+Irrigation
25	irrigation total demand	UZ+ET+Irrigation
153	sprinkler irrigation	UZ+ET+Irrigation
154	drip and sheet irrigation	UZ+ET+Irrigation
27	ground water extraction for irrigation	SZ+UZ+ET+Irrigation
106	depth to phreatic surface (negative)	SZ
101	head elevation in saturated zone	SZ
107	seepage flow SZ -overland (the flow up from SZ onto the topography)	SZ
108	seepage flow overland - SZ (negative) (the flow down into the saturated zone)	SZ
	External inflow to SZ drain (for OpenMI)	
113	3D UZ recharge to SZ (negative)	SZ+NegPrec
102	groundwater flow in x-direction (a flow rate, e.g. in [m^3/s]	SZ
103	groundwater flow in y-direction (a flow rate, e.g. in [m^3/s]	SZ
104	groundwater flow in z-direction (a vertical darcy flow rate, e.g. in [mm/day]	SZ
	SZ head elevation stored with SZ flows Use this if you wan to display heads in SZ cross-sections in the Results Viewer	SZ
109	groundwater extraction	SZ+Extraction
115	SZ exchange flow with river	SZ+River
112	SZ drainage flow from point	SZ+Drainage
105	SZ flow to general head boundary	SZ+GHB
	SZ flow to MOUSE	

Table 3.1 Available output items for gridded data and time series.
–Key to symbols
–ET - Evapotranspiration
–OL - Finite Difference Overland Flow
–SubOL - Sub-catchment based Overland Flow
–UZ - Richards or Gravity Unsaturated flow,
–2LUZ - 2-Layer Unsaturated Water Balance
–SZ - Finite Difference Saturated Zone flow,
– LR - Linear Reservoir groundwater
–AD - Advection Dispersion (Water Quality)
–PT - Particle Tracking
–SM - Snow melt

Code	Output Item	Appears with these processes			
	External sources to SZ (for OpenMI)				
216	Overland concentration	OC			
217	Overland sorbed concentration	OC			
218	Overland mass/area	OC			
219	Air temperature	OC			
220	UZ concentration (matrix phase)	UZ			
221	UZ sorbed concentration (matrix phase)	UZ			
222	UZ concentration (macropore phase)	UZ			
223	UZ sorbed concentration (macropore phase)	UZ			
224	UZ mass flux (matrix phase)	UZ			
225	UZ mass flux (macropore phase)	UZ			
226	UZ soil temperature	UZ			
227	SZ concentration (mobile phase)	SZ			
228	SZ sorbed concentration (mobile phase)	SZ			
229	SZ concentration (immobile phase)	SZ			
230	SZ sorbed concentration (immobile phase)	SZ			
231	SZ soil temperature	SZ			
232	SZ porosity	SZ			
233	Number of particles	SZ			
234	Number of registered particles	SZ			
235	Most recent registration zone code	SZ			
236	Average age	SZ			
237	Average transport time to nearest registration cell	SZ			

.

Table 3.2 Additional output items for time series.
–Key to symbols
–ET - Evapotranspiration
–OL - Finite Difference Overland Flow
–SubOL - Sub-catchment based Overland Flow
–UZ - Richards or Gravity Unsaturated flow,
–2LUZ - 2-Layer Unsaturated Water Balance
–SZ - Finite Difference Saturated Zone flow,
– LR - Linear Reservoir groundwater
–AD - Advection Dispersion
–PT - Particle Tracking

Code	Output Item	Appears with these processes	
145	SimStatus: Basic time step length	UZ	OL
146	SimStatus: SZ time step length	SZ	
147	SimStatus: No. of SZ iterations / time step	SZ	
148	SimStatus: Avg. no. UZ iterations / column / time step	UZ	
149	SimStatus: No. of Overland iterations per time step	OL	
29	recharge to interflow reservoirs	LR	
30	interflow from interflow reservoirs	LR	
31	percolation from interflow reservoirs	LR	
32	interflow reservoir storage	LR	
33	change in interflow reservoir storage	LR	
34	inflow to baseflow reservoir	LR	
211	dead zone inflow to baseflow reservoir	LR	
35	baseflow from baseflow reservoir	LR	
36	groundwater feedback from baseflow reservoir	LR+UZ	LR+2LUZ
44	pumping from baseflow reservoir	LR	
46	storage in baseflow reservoir	LR	
212	dead zone storage in baseflow reservoir	LR	
38	change in subcatchment storage in baseflow reservoir	LR	
213	change in dead zone storage in baseflow reservoir	LR	
155	simple overland water depth	SubOL	
156	simple overland exchange to lower zone or river	SubOL	
157	simple overland recharge	SubOL	

4 THE RESULTS VIEWER

4.1 Toolbars

Many of the functions in the Results Viewer are the same as those available in other DHI software tools (e.g., 2D Grid Editor). Additional tools available in the result viewer are summarized in Table 4.1.

Table 4.1 Description of Result Viewer tools

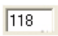
Button	Name	Description
	Rewind	Rewinds result files to first time step
	Previous Step	Rewinds result files to the previous time step.
	Video Reverse	Generates an avi file from the current time step to the first time step
	Play Reverse	Plays result files from the current time step to the first time step. Identical to Video Reverse except an avi file is not generated.
	Stop Animation	Stops forward and reverse playing of result files and creation of avi files
	Play Forward	Plays result files from the current time step to the last time step. Identical to Video Forward except an avi file is not generated.
	Video Forward	Generates an avi file from the current time step to the last time step
	Next Step	Advances result files from the current time step to the next time step
	Wind	Advances results files to the last time step
	Go to time step	Rewinds or advances result files to the specified time step

Table 4.1 Description of Result Viewer tools

Button	Name	Description
	Time step	Change the time step used by the result viewer. The time step can be less than or greater than the result file time step
	Default	Default extraction tool.
	Time Series extractor	Tool to extract time series data from result files. Multiple time series can be extracted by holding down the Ctrl key while left-clicking. A single extraction or the last multiple extraction is selected using a double left-click. See Displaying a time series at a point *(V.1 p. 92)*
	Profile extractor	Tool to extract vertical profiles (cross-sections) from 3D result files. Vertices of a profile line are specified with a single left click and the profile line is closed with a double left-click. See Saturated Zone Cross-section Plots *(V.1 p. 95)*
	Cross-section extractor	Tool to extract cross-sections of MIKE 11 results at H-points. Additional information is given below
	UZ Plot extractor	Tool to extract a UZ plot of the water content in the unsaturated zone. This tool generates a plot of water content versus depth with time. This tool can only be used on one cell at a time. A cell is selected by double left-click. Additional information is given below.
	UZ Scatter plot	Limits displays of results to unsaturated zone calculation cells. This button is only activated if unsaturated zone data is displayed in the result viewer. Additional information is given below.
	UZ Filled Plot	Displays interpolated unsaturated zone results in non-calculation cells and unsaturated zone results in calculation cells. This button is only activated if unsaturated zone data is displayed in the result viewer. Additional information is given below.

4.2 Modifying the plot

When the Results viewer is opened from MIKE SHE, a default plot is created. However, in many cases, you will want to edit these plot settings. Typical changes fall into four broad categories:

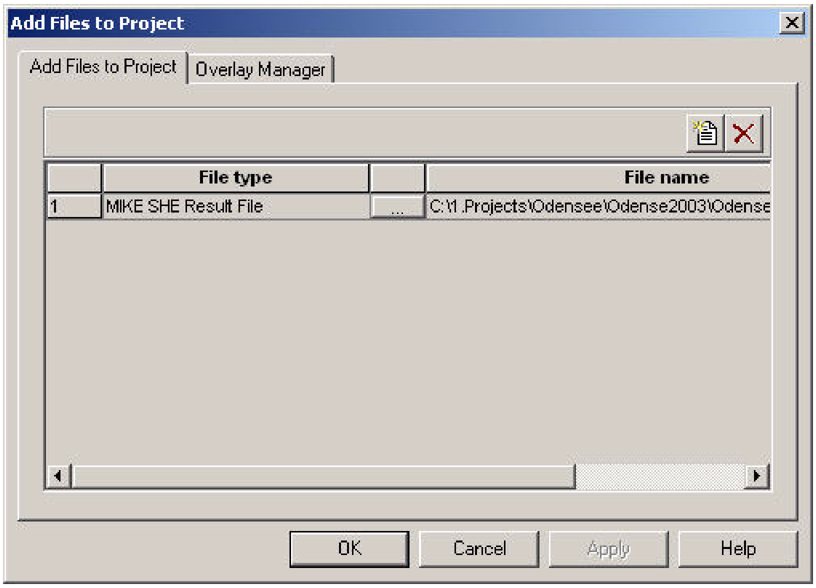
- Adding additional result files and overlays *(V.1 p. 83)*
- Adding or modifying vectors *(V.1 p. 85)*
- Changing the shading and contour settings of gridded data *(V.1 p. 87)*
- Changing the legend and colour scale *(V.1 p. 88)*

4.2.1 Adding additional result files and overlays

A project (or view or plot) in the Results Viewer is a collection of results files and overlays. You can add additional results files or overlays to your current plot by following these steps

1 Select Projects/Add Files to Project... from the top pull-down menu. This will open the dialogue below

:



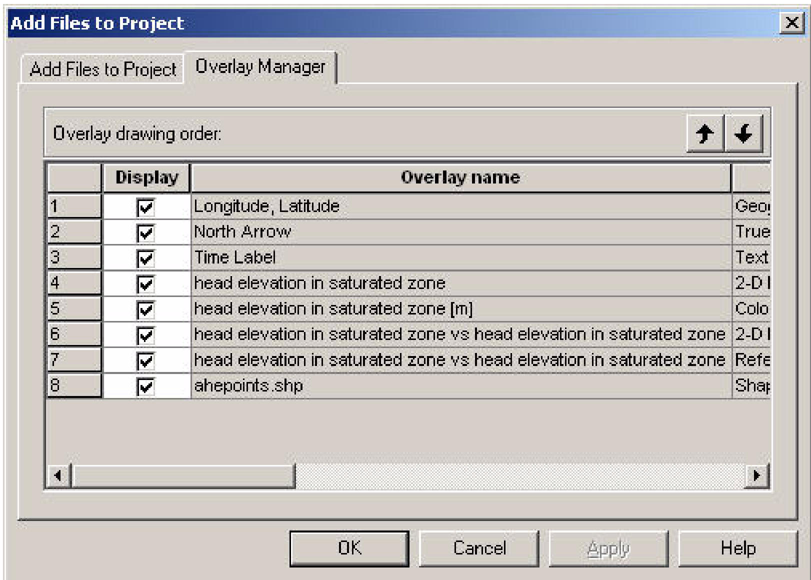
2 Click on the Add item button in this dialogue to add a line to the list of files attached to the current project.

3 In the left hand column select the type of file to add, including image files, additional results files, and MIKE 11 files.

4 Click on the browse button, to find the file that you want to add. All project files will be displayed that are the correct data type.

5 If you are adding shape files, you must remember to specify the coordinate axes or the file will not be displayed properly. To do this, you must scroll the dialogue to the right and change the units in the Units combobox.

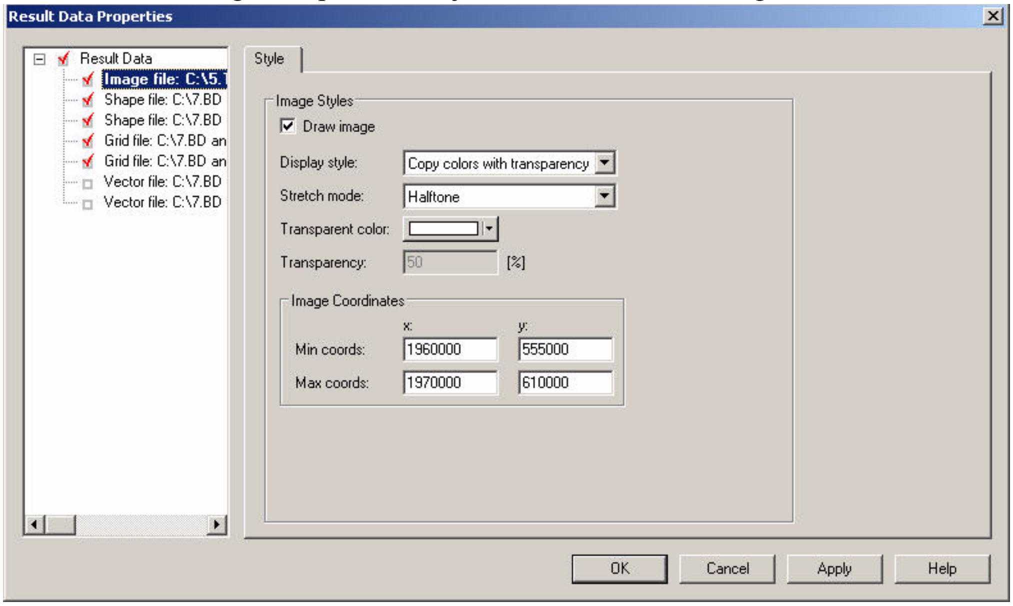
6 After adding the additional file or files, you can modify the drawing order from the Overlay Manager tab



The up arrow and down arrow buttons are used to move an item up or down in the drawing order. The Overlay Manager uses the convention that items are drawn from the lowest to highest item number (i.e., items on the bottom of the overlay list are drawn last and are on top of all other items). The Overlay Manger can also be used to turn overlays on and off by selecting or unselecting overlay items using the check box.

The Overlay Manager can also be accessed from the menu bar by selecting Project / Active View Setting / Overlay Manager.

7 After adding the file, open the Property dialogue by right-clicking in the results map and selecting Properties from the pop-up menu or by using the top menu Projects / Active View Settings / Horizontal.



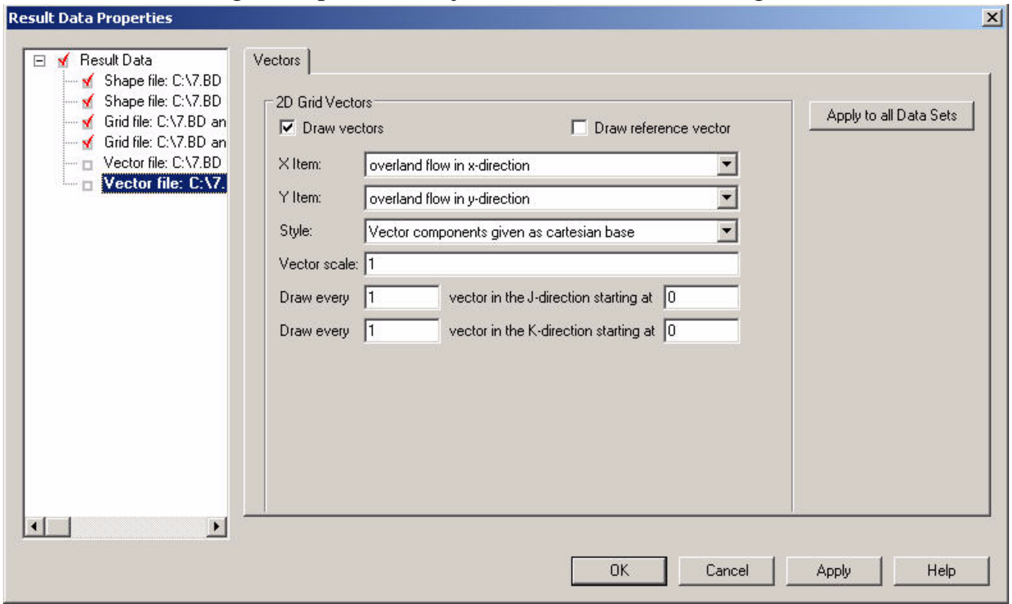
8 For an image file (above), you will need to specify the coordinates just like in the Display items in the Setup Tab.

4.2.2 Adding or modifying vectors

Vectors can be added by adding a MIKE SHE results files that contains flow data, which are *project*_overland.dfs2 and the *project*_3DSZFlow.dfs3 files. To add vectors follow these steps:

1 Add the a flow data file to your results view by following the directions in the section Adding additional result files and overlays *(V.1 p. 83)*.

2 After adding the flow file, open the Property dialogue by right-clicking in the results map and selecting Properties from the pop-up menu or by using the top menu Projects / Active View Settings / Horizontal.



3 When you added the flow results file, the grid data is by default displayed - hiding the original grid data in your results view. To turn off this grid date find the grid item for the data file that you just added and click off the Draw Grid checkbox

4 Then you need to find the Vector item for the flow file that you just added and check on the Draw Vectors checkbox

5 From the comboboxes for X and Y Items, select the flow data for the x and y directions.

6 Finally, select a Vector Scale. A suitable scale can only be found by trial and error. Normally, a large number is good to start with. For example, 10000. If the vectors are too large, then reduce the scale. If they are too small, then increase the scale.

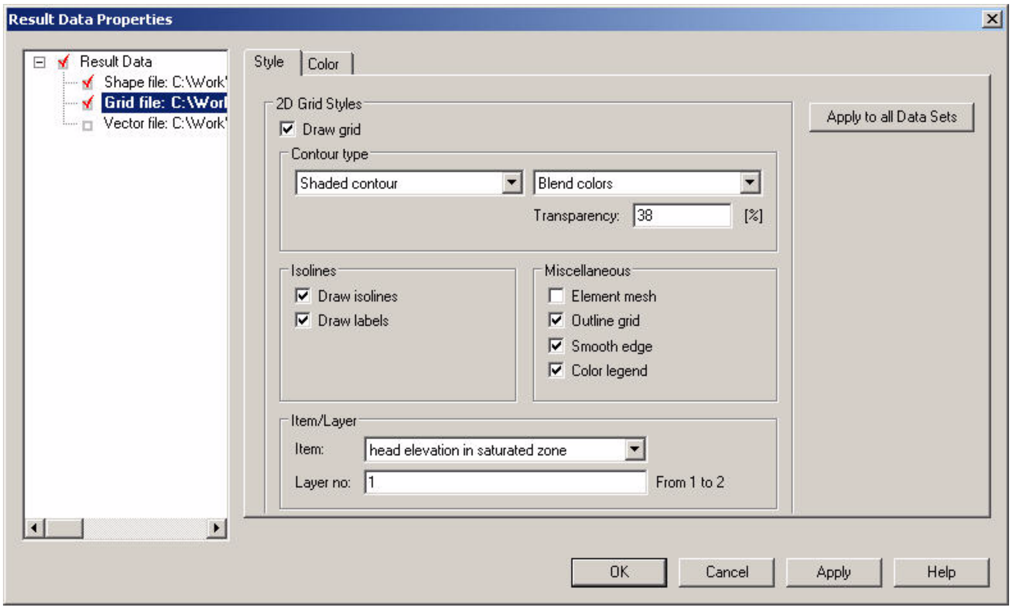
Note. Their is no vector data for the initial time step in the MIKE SHE results files.

7 If your cell size is small or your flows are high you can plot a reduced number of vectors by modifying the Draw every __ vector option.

4.2.3 *Changing the shading and contour settings of gridded data*

Gridded data is visually interpolated to a colour scale. The display of the nodal values can be smoothed to make a more aesthetic plot. Finally, the plot can customized to contain isolines with labels, a colour legend, etc.

The display properties of the gridded results can be modified by right-clicking in the graphical view and selecting Properties from the menu or by selecting Projects/Active View Settings/Horizontal from the top menu. These steps will open the dialogue below



The right hand side of the dialogue lists the available display items. Typically, this is a set of overlays from the MIKE SHE setup tab, plus the grid file that is being displayed. If you have added other output files to the plot, then these will also be listed.

Draw Grid - The Draw grid check box turns the gridded data display on and off.

Contour Type - The interpolation of the gridded values to a colour scale is controlled by the Contour Type. The available interpolation methods include Box contours or Shaded contours. Box contours present a uniform colour for every cell and Shaded contours smooth the colour gradation between and across the cells. The transparency, copy colours and blend colours options control how the display of the other overlays interacts with the gridded data. You should experiment with these settings to get a feeling for how they interact.

Isolines - Isolines can be with or without labels. In the current version, the format of the labels cannot be modified. However, the colour scale settings can be used to change the contour intervals.

Miscellaneous - Additional options are available to display the model mesh and legend. The legend scale is controlled by the Colour tab.

Item/Layer - The Item/Layer section allows you to switch to a different model layer for the gridded data. **Note**. You must remember to manually change the layer number of any other displayed data, such as vectors, in the other display items.

4.2.4 Changing the legend and colour scale

In some cases the default colour scheme may not be appropriate for the intended purpose. The colour scheme and/or contour intervals can be modified by right-clicking in the graphical view and selecting Properties from the pop-up menu or using the Projects / Active View Settings / Horizontal keystrokes and navigating to the grid file entry that you want to modify and the Colour tab for the grid entry (Figure 4.1). Options for modifying the colour scheme and/or contour intervals include making a New scheme/contour interval, Editing the existing scheme/contour interval, Opening an existing scheme/contour interval, Saving the current scheme/contour interval, or Resetting (not implemented yet) the current scheme/contour interval to default values. When making a new scheme/contour interval it is possible to modify the Max and/or Min value(s) used to generate the ranges used in the contour intervals (Figure 4.1).

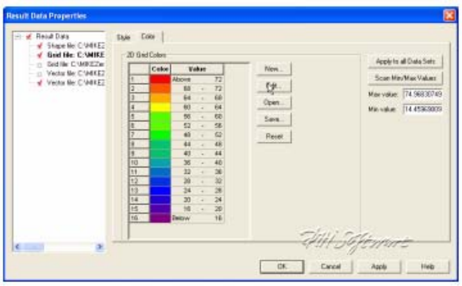


Figure 4.1 Colour modification property tab.

An example of making a new scheme/contour interval using the maximum range is summarized in Figure 4.2 to Figure 4.5. Modification of the number of contour intervals from the default value of 16 to 6 is shown in Figure 4.2. Figure 4.3 shows the available colour schemes that are available and shows use of the Seismic colour scheme. The legend title (Palette title) and palette type (Linear auto Scaled, Fixed, Land/Water Auto Scaled, Land/Water Fixed, and Angle Fixed (Circular)) can also be modified on the first Palette Wizard window (Figure 4.2). Press Next after making the desired changes to move to the next Palette Wizard window.

The colours used for each contour interval (Colour) and the ranges used for each contour interval (Value) can be modified on the second Palette Wizard window (Figure 4.4). Press Next after making the desired changes to move to the next Palette Wizard window.

The third and final Palette Wizard window allows you to review the modified colour scheme and contour intervals before accepting the changes (Figure 4.5). Press the Finish button if all of the modifications are acceptable. Otherwise, press the < Back button to make additional modifications or Cancel button to cancel all changes.

Figure 4.2 Step 1 of 3 - modification of the number of colours used in the colour scheme.

Figure 4.3 Step 1 of 3 - modification of the number of colours model used in the colour scheme.

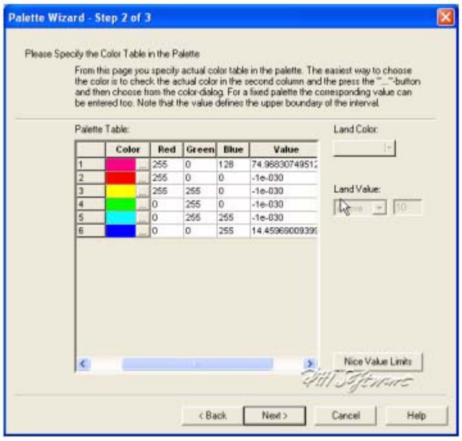


Figure 4.4 Step 2 of 3 - modification of the colours used in the colour scheme and the values colours are applied to.

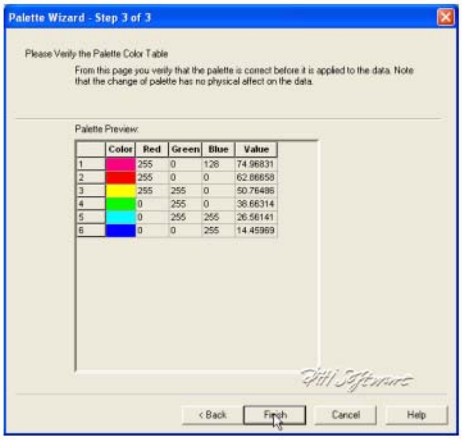


Figure 4.5 Step 3 of 3 - acceptance of the colour scheme modification.

After Accepting the colour scheme/contour interval modifications the Apply button should be pressed on the Result Data Properties window to modify the look of the Result Viewer plot (Figure 4.6). The resulting modified Result Viewer plot is shown in Figure 4.7.

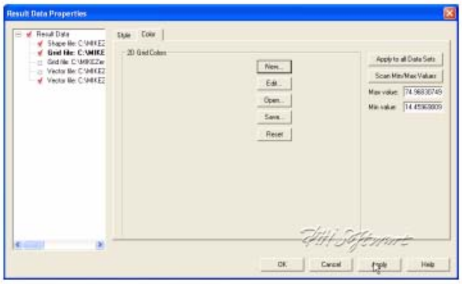


Figure 4.6 Applying the modified colour scheme to the current result viewer file.

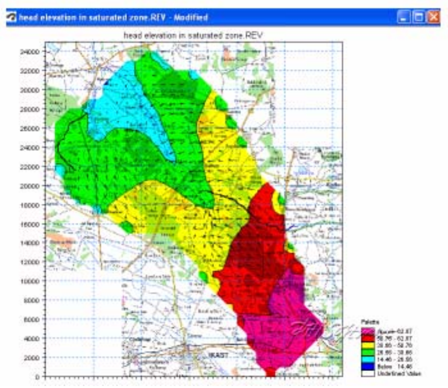


Figure 4.7 Result Viewer file after modification of the default colour scheme.

Users should experiment with the Palette Wizard to develop a better understanding of available functionality than presented in this simple discussion.

4.3 Displaying a time series at a point

The Time Series tool allows you to plot a time series of all the data available in the current view.

Time series data can be selected from multiple locations in the active model area using this tool. A single time series can be selected by double-clicking in the desired location. Time series can be extracted from multiple locations by holding down the Ctrl-key and left clicking on each desired location. When selecting multiple locations the Ctrl-key should be held down while double clicking on the last location.

After selecting the locations of the time series files to extract you have the option to deselect some of the selected points and to accumulate the data over the simulation period (Figure 4.8). After making the appropriate selections/deselections press the OK button to generate the time series plot. The entire extraction process can be stopped by pressing the Cancel button. An example of a time series plot generated in the Results Viewer is shown in Figure 4.9.

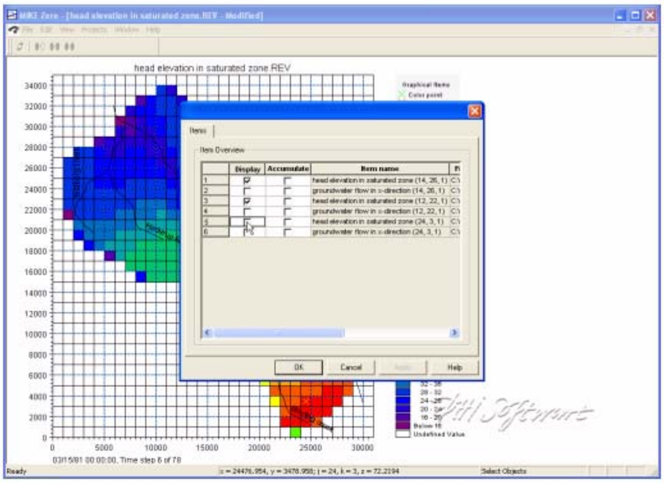


Figure 4.8 Selection of time series items to extract.

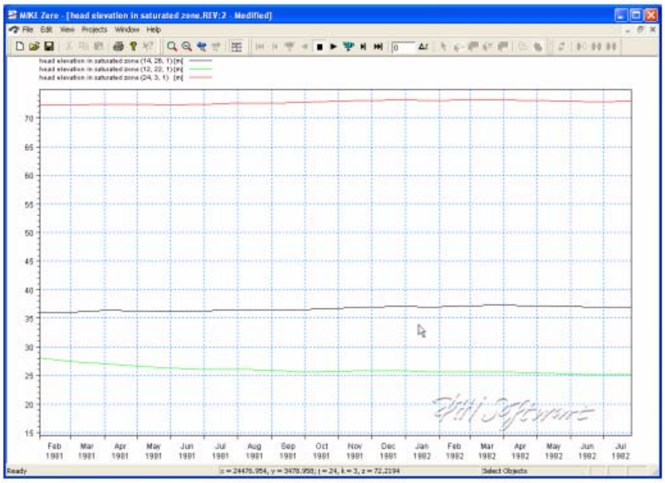


Figure 4.9 Time series plot generated using the time series extraction tool.

Addition graphical functions can be accessed by right-clicking in the graphical view including zooming, exporting images, exporting time series data as dfs0 files, and modification of the time series plot properties (Figure 4.10). Most of the functionality can also be accessed via the menu bar. For example, modification of the time series plot properties can be accessed using Projects/Active View Settings/Timeseries.

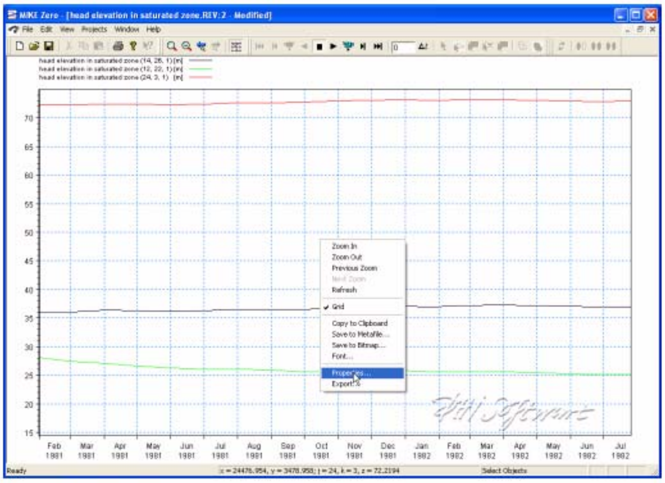


Figure 4.10 Modifying the properties of time series plots in the result viewer.

Because of the rich functionality available in the Result Viewer with respect to time series output, users should experiment with the available options. An example of the available functionality for modification of the time series plot properties is shown in Figure 4.11. For example, as shown in the upper left of Figure 4.11, time series items can be added or deleted from a plot on the items tab.

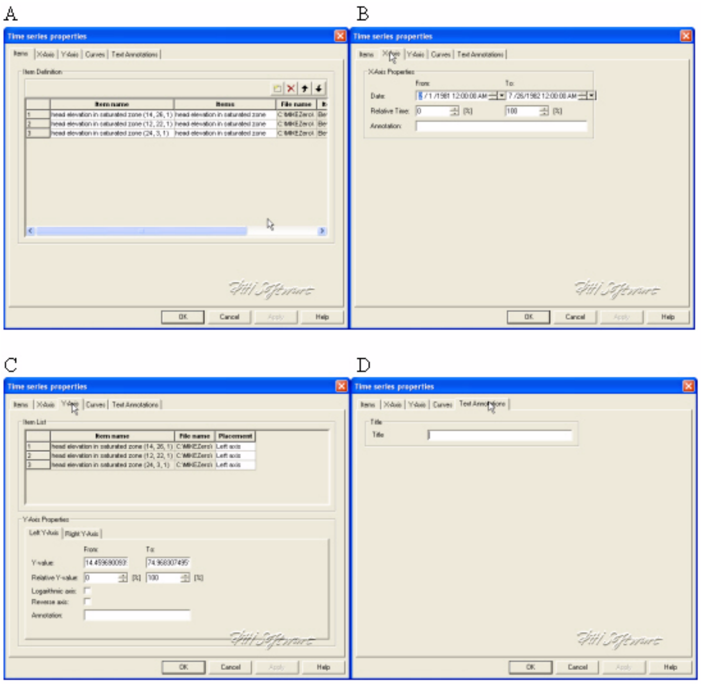


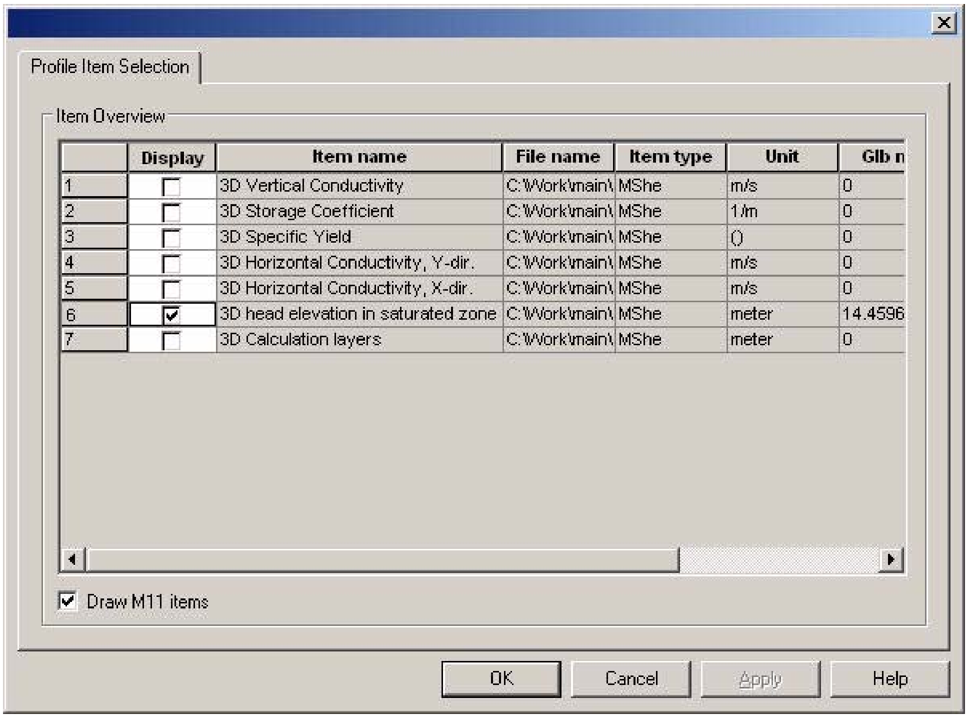
Figure 4.11 Modification of A) items displayed on the time series plot, B) x-axis properties, C) y-axis properties, and D) time series plot title.

4.4 Saturated Zone Cross-section Plots

To display a cross section plot of a set of 3D gridded data, you must click on the Profile icon,  . Clicking on this icon will allow you to interactively define a cross-section by left-clicking at each vertex of the profile line and double-clicking to close the profile.

After closing the profile, the following dialogue will be displayed listing the available output items.

.



Only one of these items can be selected. After selecting your item, click OK and the profile will be displayed.

The profiles exactor tool can be used to extract a cross-section through simulated MIKE SHE and MIKE 11 results. The type of cross-section created is dependent on the simulated data displayed in the result viewer. For example, if the result viewer contains simulated 3D heads and MIKE 11 results then the cross-section will have simulated water levels and simulate MIKE 11 canal stages.

After defining the profile, the items to be displayed on the profile should be selected. The resulting profile is shown in Figure 4.12. As with the other tools, extracted profiles can be animated on the screen and/or exported as avi and image files.

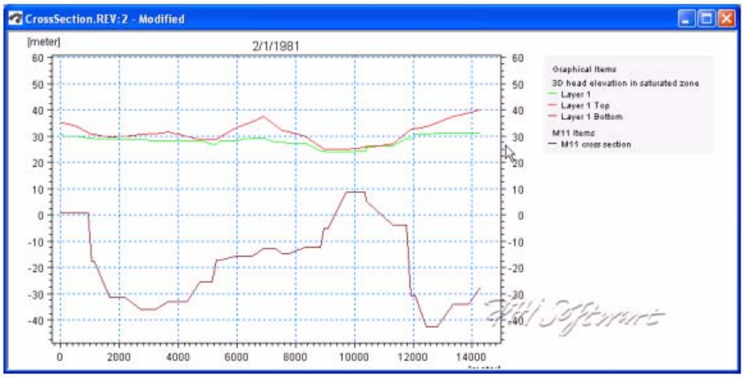
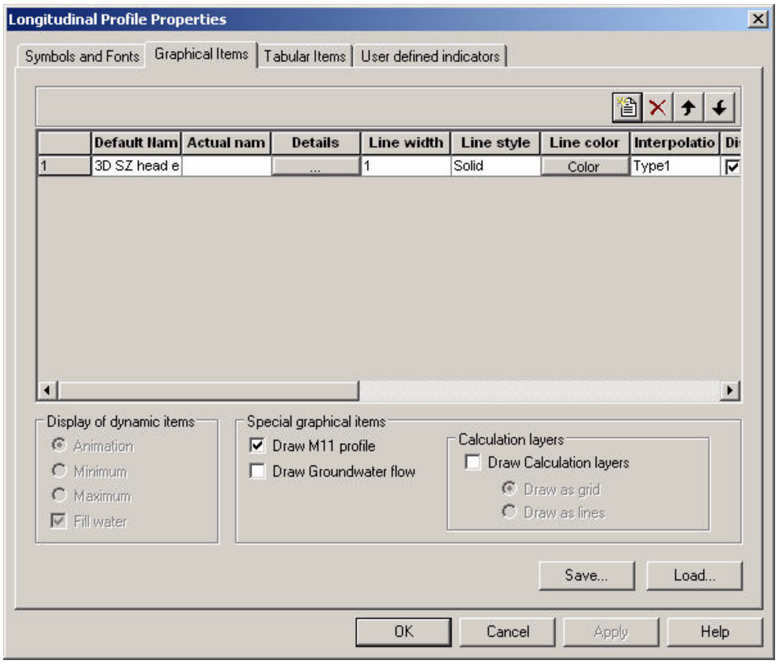
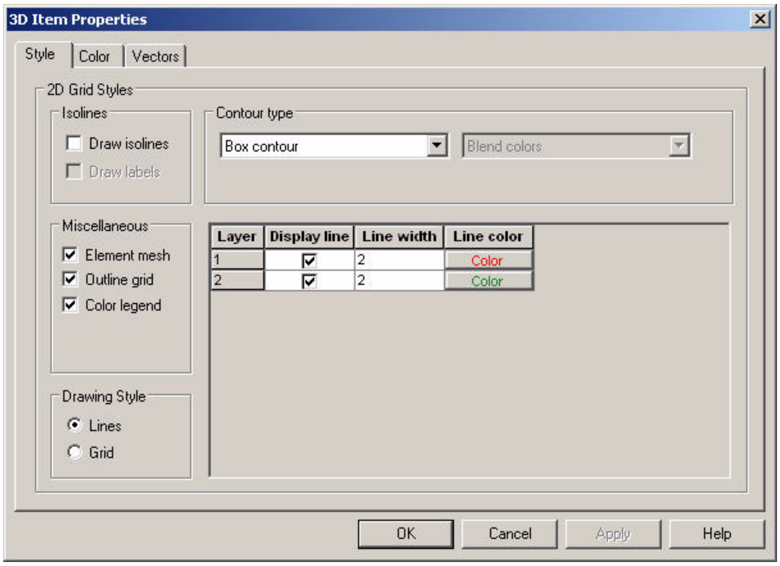


Figure 4.12 Resultant profile generated with the profile extractor tool.

You can modify the plot by right clicking on the plot and selecting Properties form the pop-up menu. In this dialogue, only the Graphical Items Tab is relevant for MIKE SHE results (below).



In this dialogue, if you click on the Details... button for the item, you will get the following dialogue, where you can change the colour scale and plotting characteristics for the cross-section.



As with the other tools available in the result viewer, users should experiment with the available options to learn how to fully use the result viewer profile extractor.

4.4.1 Saving and loading profiles

If you have a profile open, under the View/Profile item in the top menu bar, you can save the current profile location. This allows you to create standard profiles for comparing scenarios.

To load a saved profile, make the plan view plot active, by either minimising or closing open profile plots. The View/Profile/Load option becomes active and you can load a saved profile and select the profile item normally.

4.5 Displaying a MIKE 11 cross-section

MIKE 11 results can also be added to the result viewer and simulated canal water levels can be displayed using the cross-section extractor. The cross-section extractor shows simulated stages and the geometry of the cross-section being viewed. The process of adding MIKE 11 results to the result viewer are given in the section Adding additional result files and overlays.

After selecting the cross-section extractor tool, move the cursor over the location you want to extract the MIKE 11 results from (Figure 4.13). The simulated results are displayed along with the cross-section geometry

(Figure 4.14). As with the other tools, extracted profiles can be animated on the screen and/or exported as .avi and image files.

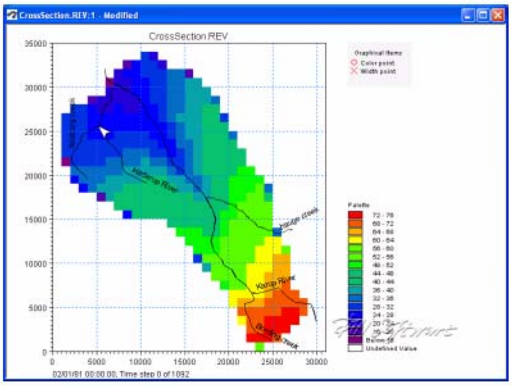


Figure 4.13 Selection of a MIKE 11 cross-section location in the result viewer.

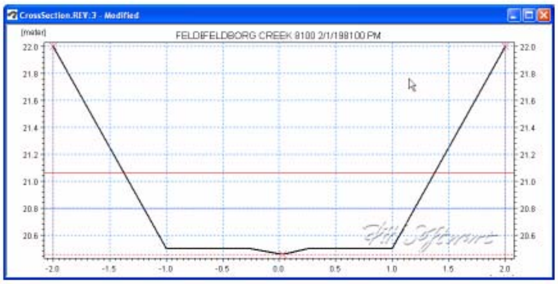


Figure 4.14 Resultant MIKE 11 cross-section plot.

Addition graphical functions can be accessed by right-clicking in the graphical view (Figure 4.15). Modification of the profile properties is one functionality available using the right-click. Since the cross-section plots are relatively simple, modifications are limited to changing line and marker properties, cross-section markers, etc. (Figure 4.16).

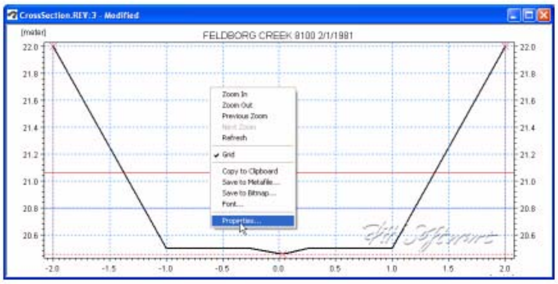


Figure 4.15   *Accessing addition functionality in the extracted MIKE 11 cross-section plot.*

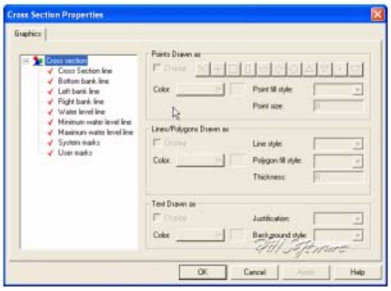


Figure 4.16   *MIKE 11 cross-section properties that can be edited.*

4.6 UZ Specific Plots

4.6.1 UZ Scatter and Filled Plots

For unsaturated zone results, scatter or filled plots can be generated. UZ Scatter and Filled Plots are only different for simulations that do not use the “calculation in all cells” UZ module option.

Scatter plots only show simulated results for UZ calculation cells. The number of UZ calculation cells may be less than the total number of active model domain used by the overland and saturated zone modules if the UZ module for the simulation is not using the “calculation in all cells” option. An example of when use of the UZ scatter plot is useful is shown in Figure 4.17.

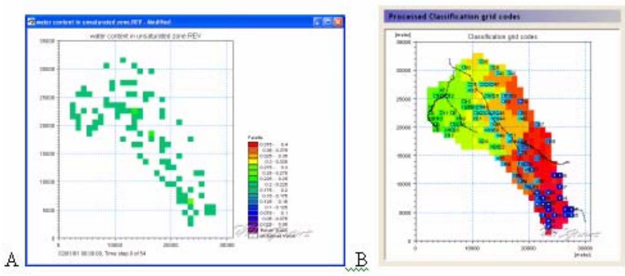


Figure 4.17 A) UZ Scatter Plot and its relationship to B) the UZ calculation cells.

An example of a UZ Filled Plot is shown in Figure 4.18. In cases where the UZ module for the simulation is not using the "calculation in all cells" option the Result Viewer interpolates values from the calculation cells to adjacent inactive UZ cells.

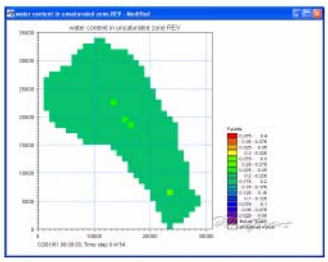


Figure 4.18 Filled UZ plot.

4.6.2 UZ Plot

UZ Plots can only be extracted from simulated unsaturated zone water contents and flow. This is because UZ plots display results for a single column for all of the UZ calculation nodes in the column. Other simulated UZ results show net values for the entire UZ (i.e., infiltration, recharge to the SZ, etc.).

After selecting the UZ Plot extractor tool move the cursor over the column you want to extract the results from and double-click (Figure 4.19). Results from multiple UZ columns cannot be displayed on the same UZ Plot.

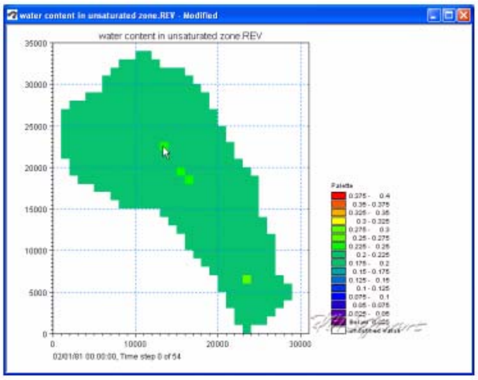


Figure 4.19 Extracting a UZ Plot from simulated unsaturated water contents and flow.

The simulated water content results for the selected column are displayed in Figure 4.20. The UZ Plots show either water content or unsaturated zone flow for each node in the column (y-axis) for the entire simulation (x-axis).

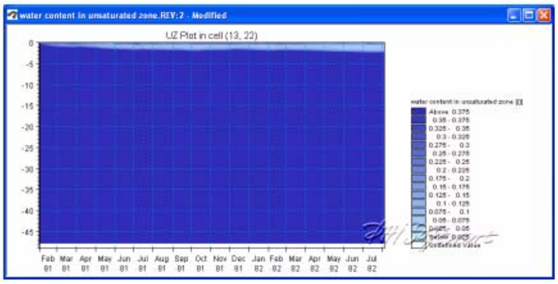


Figure 4.20 Example UZ plot of unsaturated zone water content.

Addition graphical functions can be accessed by right-clicking in the graphical view. Modification of the UZ Plot properties is one functionality available using the right-click (Figure 4.21). Modifications that can be made include changing the interpolation methods, adding the mesh, adding isolines, changing the colour schemes, etc. (Figure 4.22).

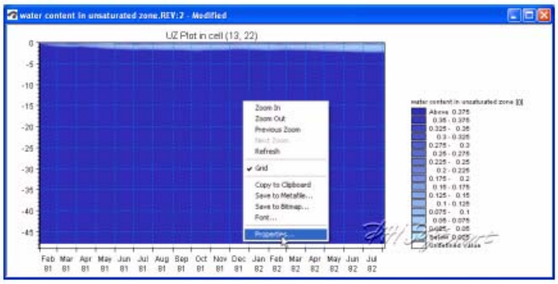


Figure 4.21 Modification of the UZ plot properties.

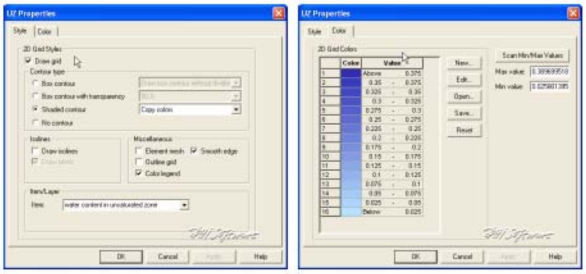


Figure 4.22 Available UZ plot properties that can be modified.

An example of a modified UZ plot with the mesh displayed and only showing the upper five meters of the soil column is shown in Figure 4.22. Additional information on modifying the interpolation and colour scheme are given in the sections Changing the shading and contour settings of gridded data and Changing the legend and colour scale.

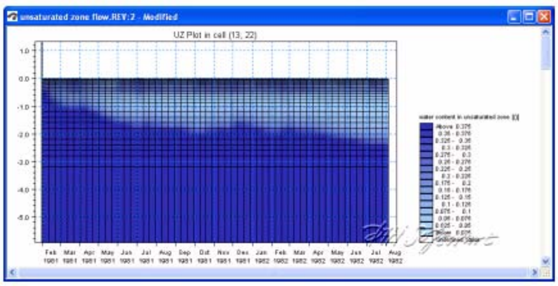


Figure 4.23 Figure 25Close up of upper 5 meters of soil column with the calculation grid displayed.

5 USING THE WATER BALANCE TOOL

The water balance utility is a post-processing tool for generating water balance summaries from MIKE SHE simulations. Water balance output can include area normalized flows (storage depths), storage changes, and model errors for individual model components (e.g., unsaturated zone, evapotranspiration, etc.).

A water balance can be generated at a variety of spatial and temporal scales and in a number of different formats, including dfs0 time series files, dfs2 grid series files, and ASCII text output suitable for importing to Microsoft Excel. You can also automatically create a picture that visualizes the interrelationships between the various water balance components (see Figure 5.1).

The water balance utility can be run from within the MIKE Zero interface or from a MSDOS batch file. The batch functionality allows you to calculate water balances automatically after a MIKE SHE simulation that is also run in batch mode. Alternatively, you can also calculate water balances as part of an AUTOCAL simulation and use the results as part of an objective function

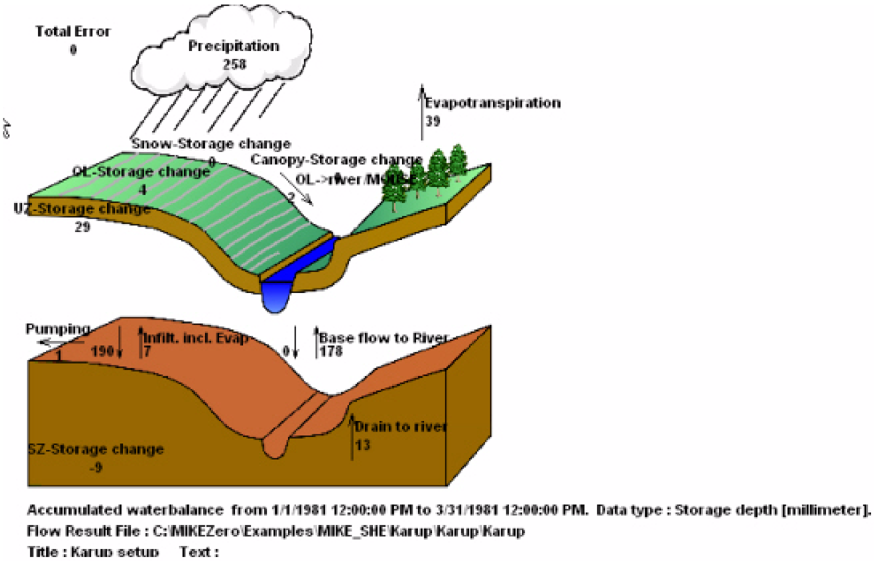
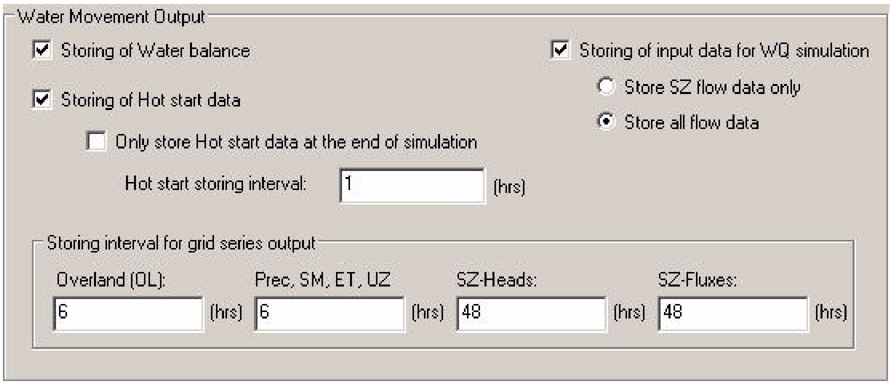


Figure 5.1 Graphical water balance output example

5.1 Creating a water balance

Before you can create a water balance for a MIKE SHE WM simulation, you must have saved the water balance data during the simulation. Saving of the water balance data is specified in the Storing of Results *(V.2 p. 183)* dialogue. If you have forgotten to save the water balance data, then you will need to re-run your simulation.



After you have run your WM simulation, creating and running a water balance in MIKE SHE is quite simple, following these steps

1 Create a new water balance document *(V.1 p. 106)*,

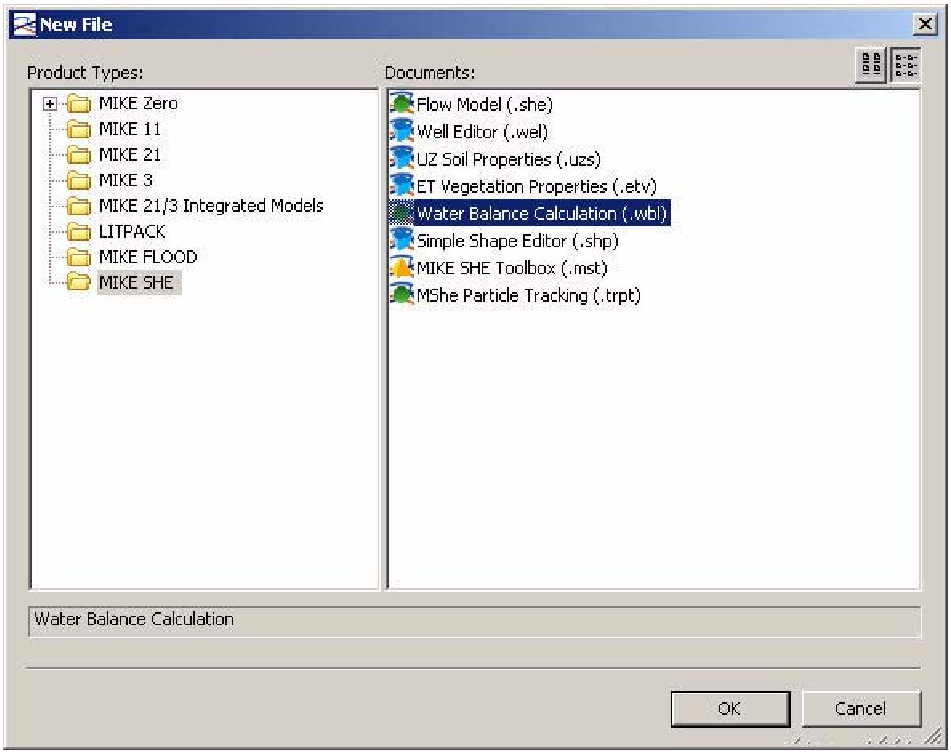
2 Extract the water balance data *(V.1 p. 107)*

3 Specify your water balance *(V.1 p. 109)*, and

4 Calculate and View the Water Balance *(V.1 p. 113)*.

5.1.1 Create a new water balance document

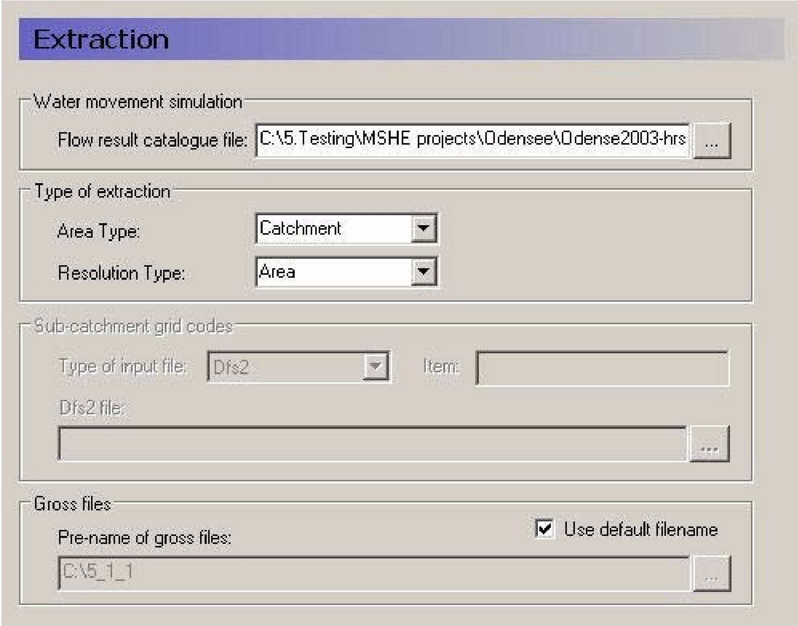
The new water balance document is created by selecting the File/New item in the top menu, or clicking on the New icon in the top menu bar. In the dialogue that appears, select MIKE SHE and Water Balance Calculations in the right hand box, as shown below.



5.1.2 Extract the water balance data

To extract the water balance data, specify the MIKE SHE simulation by selecting the simulation catalogue file (.sheres file), then specify the area of your model that you want the water balance for, and, finally, extract the MIKE SHE water balance data from the results files.

Once you have created a new water balance document, the first tab is as shown below.



Flow result catalogue file

A MIKE SHE simulation generates various output files depending on the options and engines selected for the MIKE SHE simulation. The .sheres file is a catalogue of all the various output files generated by the current MIKE SHE run. When you select the .sheres file, you are not specifying the particular output, but actually just a set of pointers to all the output files.

The extraction process reads all of the output files and makes itself ready to produce specific water balances. In the extraction dialogue, you specify the .sheres file for the simulation that you wish to calculate the water balance for. The .sheres file is located in the same directory as your results.

Note Although, this is an ASCII file, you should be careful not to make any changes in the file, or you may have to re-run your simulation.

Type of Extraction

You can choose to calculate the water balance on the entire model domain or in just a part of the domain. By default the calculation is for the entire domain, or catchment. If you choose the subcatchment area type, they you will be able to use a dfs2 integer grid code file to define the areas that you want individual water balances for.

If you use an area resolution, then the water balance will be a summary water balance for either the entire catchment or the sub-areas that you define.

If you use a single-cell resolution, you will be able to generate dfs2 maps of the water balance.

Sub-catchment grid codes

The subcatchment integer grid code file is only used if you have selected the sub-catchment water balance type. You can specify a delete value to exclude areas from the water balance. The grid spacing and dimensions in this dfs2 file do not have to match the model grid exactly. However, the sub-catchment grid must be both coarser than and aligned with the original grid.

You can also specify a polygon shape file to define the sub-catchment areas. The shape file may contain multiple polygon, with the same or different codes. Further, the shape file length units do not have to be the same as the model length units (e.g. feet vs. meters).

Gross files

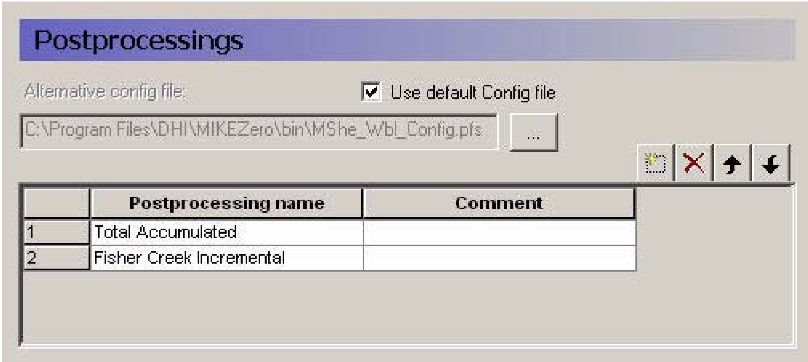
The pre-processor extracts the water balance data from the standard MIKE SHE output files and saves the data in a set of “gross” files. The file names of the gross files is built up from the project name and prefix specified here. The default value is normally fine.

Run the extraction

To run the extraction, you simply have to click on the Run Extraction icon, , or the Run/Extraction top menu item.

5.1.3 Specify your water balance

After you have extracted the water balance data from the MIKE SHE results files, then you can switch to the post-processing tab. Here you can create any number of individual water balances by simply clicking on the Add item icon and specifying the water balance parameters in the parameter dialogue.

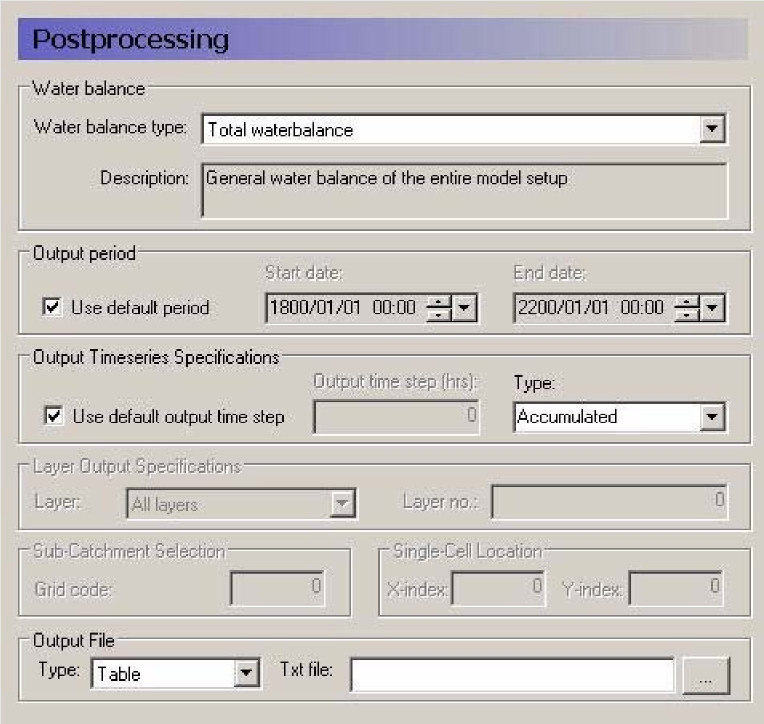


A single Postprocessing item is created by default when the water balance file is created. The default Postprocessing name can be change to a more appropriate name. Postprocessing items that are no longer needed can be deleted using the Delete button.

Use default Config file

Unchecking the Use default Config file checkbox, allows you to specify the location of a custom water balance Config file. Development of custom water balance configuration files is described in detail in Making Custom Water Balances *(V.1 p. 144)*.

For each item in the Postprocessing list above, a new item will be added to the data tree. If you expand the data tree, each will have the following dialogue.



Water Balance

Multiple postprocessings can be run on each water balance extraction. More detail on the types of available water balances data are discussed in the Available Water Balance Items *(V.1 p. 114)* section. In brief, the available types include

- The total water balance of the entire model catchment or sub-catchments in an ASCII table, a dfs0 file, a dfs2 map file, or a graphical chart (also by layer),
- Model errors for each hydrologic component (overland, unsaturated zone, etc.) in an ASCII table, a dfs0 file, or a dfs2 map file (also by layer),
- The snow melt and canopy/interception water balance in an ASCII table, or a dfs0 file,
- An abbreviated or detailed water balance for overland or unsaturated flow in an ASCII table, or a dfs0 file, and
- An abbreviated or detailed water balance by layer for saturated flow in an ASCII table, or a dfs0 file.

Output Period

An output period different from the total simulation period can be specified by unchecking **Use default period** and setting the **Start date** and **End date** to the period of interest

Output Time Series Specification

Incremental or **Accumulated** water balances can be calculated. An incremental water balance is calculated (summed) for each output time step in the Output period. An accumulated water balance each output time step is accumulated over the Output period

Layer Output Specifications

If you are using water balance types that calculate data on a layer basis, you can specify whether you want **All layers** or just the **Specified layer**, where you also must specify a layer number.

Sub-catchment Selection

If you extracted sub-catchment data from the WM results, then you must specify a subcatchment number or the name of the polygon for which you want the water balance for. The combobox contains a list of valid ID numbers or polygon names.

Single Cell Location

If you extracted the WM data by cell and you are not creating a map output, you have to specify a cell location for which you want a water balance.

Output File

If you are creating a table or time series water balance, then you can write the output to either a dfs0 file or to a tab-delimited ASCII file for import to MSExcel, or other post-processing tool. If you are creating a map, then the output will be to a dfs2 file, with the same grid dimensions and spacing as the model grid. If you are creating a chart, then the output will be written to an ASCII file, with a special format for creating the chart graphic.

Run the Post Processing

To run the post processing, you have two choices. You can click on the Run Selected Post-Processing icon, , which runs only the current post-processing item. Or, you can click on the Run All Post-Processing icon, , which runs all of the post-processing items in the list. These two options are also available in the Run top menu.

5.1.4 Calculate and View the Water Balance

The data tree for the results tab lists all of the calculated water balances. The dialogue for each item, includes the file name and an Open button that will open an editor for the file. For ASCII output, this will be your default ASCII editor - usually Notepad. For dfs0 and dfs2 files, the DHI Time Series Editor or Grid Editor will be opened. For the chart output, the graphic will be displayed by the program WblChart.

Units for the water balance

The values in the water balance are in the EUM unit type Storage Depth. This normalization allows water balances for different models or model areas to be more easily compared. The Storage Depth values can be converted to volume by multiplying by the internal model area. The number of internal model cells can be found in the _WM_PRINT.LOG file. Thus, the internal area is the number of cells times the area per cell. If you have calcuated a water balance on a sub-area, the volumes must be calculated based on the number of internal cells in the sub-area.

The default units are [mm], but this can be changed to any length unit (e.g. inches) by changing the EUM unit of the variable Storage Depth.

5.2 Calculating Water Balances in Batch Mode

Like most DHI software, the water balance utility can be run in batch mode. Some possible ways to run the water balance utility in batch mode are:

- Running the water balance utility immediately after completion of a MIKE SHE simulation run in batch mode.
- Running the water balance utility for a MIKE SHE simulation without using the water balance utility graphical users interface.
- Running multiple water balance Postprocessing stages automatically without using the water balance utility graphical users interface.

The water balance graphical utility stores all of its information in a .wbl file. The .wbl file is an ASCII file that can be edited with Notepad or other text editor, but the format of the water balance file must be preserved. For more information on editing the .wbl file and creating custom water balances, see Making Custom Water Balances *(V.1 p. 144)*.

To run the water balance utility in batch mode, the .wbl file must be created prior to executing it and all file names in the .wbl file need to be valid. If during calibration the same MIKE SHE file name is used for each simulation then the same water balance file can be used for all calibration runs. If the MIKE SHE simulation to be evaluated is different from the MIKE SHE simulation used to set up the water balance file, you will have to edit the water balance file.

To run the Extraction and Postprocessing steps in batch mode, the your PATH statement needs to include directory where MIKE SHE was installed. The default directory is

```
C:\Program Files\DHI\MIKEZero\bin
```

An example is shown below of a batch file that generates water balance data for three postprocessing steps, using a water balance utility file named *WaterConservationAreas.WBL*.

```
rem -------------------------------------------
MSHE_Wbl_Ex.exe WaterConservationAreas.WBL
MSHE_Wbl_Post.exe WaterConservationAreas.WBL 1
MSHE_Wbl_Post.exe WaterConservationAreas.WBL 2
MSHE_Wbl_Post.exe WaterConservationAreas.WBL 3
```

The `MSHE_Wbl_Ex.exe` command runs the Extraction phase of the water balance utility. The command

```
MSHE_Wbl_Post.exe WaterConservationAreas.WBL 1
```

runs the first Postprocessing item in the water balance file, `WaterConservationAreas.WBL`. The number after the water balance file name in the Postprocessing command indicates which Postprocessing item to run and this number must consistent with the water balance utility file (i.e., the number cannot be greater than the number of Postprocessing items in the file). Otherwise, the program will terminate with an error. The Postprocessing step cannot be executed before an Extraction step but only one Extraction step needs to be run for a single water balance utility file.

The water balance batch file can contain Extraction and Postprocessing steps from multiple water balance utility files.

5.3 Available Water Balance Items

The .shres file contains a list of all of the simulation output files generated during the WM or WQ simulation. When you use the water balance extraction utility, all of these files are processed and a special set of water balance files are created - the .wblgross files. One file is created for each of the water balance components:

- Snowmelt and precipitation - projectname_sm.wblgross
- Canopy interception - projectname_ci.wblgross
- Ponded surface water - projectname_ol.wblgross
- Unsaturated zone - projectname_uz.wblgross
- Saturated zone - projectname_sz.wblgross

The contents of each of these files can be output using the “Detailed” water balances. All of the items in these files are listed and described in the following tables:

- Table 5.1 SM - Precipitation and snowmelt items *(p. 117)*
- Table 5.2 CI - Canopy interception water balance items *(p. 119)*
- Table 5.3 OL - Overland flow items *(p. 122)*
- Table 5.4 UZ - Unsaturated Zone items *(p. 126)*
- Table 5.5 SZ - Saturated Zone - all layers *(p. 130)*
- Table 5.6 SZ - Saturated Zone - specified by layers *(p. 136)*
- Table 5.7 SZ - Saturated Zone - Linear Reservoir all layers *(p. 137)*

The water balance utility is a very flexible tool that allows you to modify existing Water balance types or create custom Water balance types to suit your needs. The water balance calculations use a water balance Config(uration) file to define Water balance types using the available water balance items and a macro language to control program execution.

To modify existing or custom Water balance types you must understand the available items and what data they contain.

Sign Conventions

MIKE SHE uses a sign convention that is positive in the positive coordinate direction. In other words, water flowing upward in the model is a positive flow in MIKE SHE. Likewise, flow in the direction of increasing x or y is also positive. Boundary flows and other flows that do not have a direction are positive outwards.

However, the water balance utility uses a control volume sign convention, such that all inflows are negative and all outflows are positive. This can cause confusion when calculating a water balance. For example, a vertical downward flow through the unsaturated zone will always be a negative result in MIKE SHE. In the water balance control volume, a downward flow into the unsaturated zone will be a positive outflow in the water balance for ponded water, but a negative inflow into the unsaturated zone water balance.

The sign convention for the water balance error of each storage is such that an increasing storage is positive. Thus, a positive water balance error means that the change in storage plus the total outflows is greater than the total inflows. In other words, the error is positive if your model is creating water.

5.3.1 Snow Storage

The snow storage items include all of the water balance items related to rainfall and the conversion to and from snow.

The items listed in Table 5.1 are those found in the "*Snow Melt component - detailed*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
    Name = 'SM_DETAIL'
    DisplayName = 'Snow Melt component - detailed'
    Description = 'Detailed Snow Melt component water balance'
    NoGroups = 11
    Group = 'Precip and Irr -> Snow(sm.qprecandirrtosnow)'
    Group = 'AirTemp Freezing(sm.qfreezing)'
    Group = 'AirTemp Melting(sm.qthawing)'
    Group = 'Radiation Melting(sm.qradmelting)'
    Group = 'Rain Melting(sm.qrainmelting)'
    Group = 'Snow -> OL(sm.qsnowtool)'
    Group = 'Snow Evap(sm.qesnow)'
    Group = 'Dry Snow Stor.Change(sm.dsnowsto-sm.dwetsnowsto)'
    Group = 'Wet Snow Stor.Change(sm.dwetsnowsto)'
    Group = 'Total Snow Stor.Change(sm.dsnowsto)'
    Group = 'Error(sm.smwblerr)'
  EndSect  // WblTypeDefinition
```

The sign convention is such that precipitation is negative (inflow) and melting is positive (outflow). All of the noted tems together should add to zero. The freezing and thawing items are not included in the error term because they are internal transfers of water between dry snow and wet snow storages.

The snow storage items are found in the projectname_sm.wblgross file. This file also contains the terms sm.qP, sm.qPad, and sm.PIrrSprinkler, which are not included in the detailed water balance output because they are included in the term sm.qPrecAndIrrToSnow.

Table 5.1 SM - Precipitation and snowmelt items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sm.qPrecAndIrr-ToSnow	Precipitation plus irrigation added to snow storage when the air temperature is below the freezing temperature	Inflow - negative	yes
sm.qFreezing	Amount of wet snow converted to dry snow due to freezing	Negative	no
sm.qThawing	Amount of water removed from dry snow storage due to temperature melting	Positive when melting occurs	no
sm.qRadMelting	Amount of water removed from dry snow storage due to radiation melting	Positive when melting occurs	no
sm.qRainMelting	Amount of water removed from dry snow storage due to melting from rain	Positive when melting occurs	no
sm.qSnowToOL	Amount of wet snow storage transfered to interception storage. Actually, this amount is added to qPad, which is the input to canopy interception. Then the water is added to ponded water via interception throughfall. **Note**: Freezing of ponded water to snow storage is not accounted for in MIKE SHE	Outflow - positive when water is added to canopy interception	yes

Table 5.1 SM - Precipitation and snowmelt items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sm.qESnow	Amount of evaporation from snow. This is a combination of sublimation from dry snow and evaporation from wet snow. Evaporation is removed first from wet snow storage. When wet snow storage is zero, then sublimation from dry snow is removed because of the higher energy required for sublimation.	Outflow - positive when evaporation/sublimation occurs	yes
sm.dWetSnowSto	Change in wet snow storage	Positive when wet snow storage increases	no
sm.dSnowSto	Change in total snow storage **Note**: Change in dry snow storage is (dSnowSto - dWetSnowSto)	Positive when total snow storage increases	yes
sm.smWblErr	Snow storage water balance error. Sum of marked items.	Positive if water generated (Δstorage + Outflow > Inflow)	
sm.qP	Total precipitation (not used in detailed SM WB output)	Inflow - negative	no
sm.qIrrSpinkler	Total Irrigation (not used in detailed SM WB output)	Inflow - negative	no
sm.qPad	Total precipitation reaching the canopy (Precipitation + sprinkler irrigation + snowmelt to ponded water). Same as ci.qPad. (not used in detailed SM WB output)	Outflow - positive	no

5.3.2 Canopy interception storage

The canopy interception is a seperate storage on the leaves of the vegetation. If the LAI is zero, then the canopy interception will be zero, as will all of the items in this storage.

The items listed in Table 5.2 are those found in the "*Canopy Interception component*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
    Name = 'CI'
    DisplayName = 'Canopy Interception component'
     Description = 'Canopy Interception component waterbalance items'
     NoGroups = 5
     Group = 'Precip(ci.qpad)'
     Group = 'Can. ThroughFall(-ci.qpnet)'
     Group = 'Evaporation(ci.qeint)'
     Group = 'Can.Stor.Change(ci.dintsto)'
     Group = 'Error(ci.ciwblerr)'
EndSect  // WblTypeDefinition
```

The sign convention in the water balance is such that precipitation is negative (inflow) and evaporation is positive (outflow). All of the items together should add to zero.

Note, however, the negative sign in front of the ci.qpnet term in the water balance definition above. This is because the canopy throughfall is a vertical downward flow in MIKE SHE - making it a negative value in the MIKE SHE results files. Whereas, it must be a positive outflow in the water balance calculation.

Table 5.2 CI - Canopy interception water balance items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ci.qPad	Total precipitation reaching the canopy (Precipitation + sprinkler irrigation + snowmelt to OL - precipitation converted to snow)	Inflow - negative	yes
-ci.qPnet	Canopy throughfall to ponded water	Outflow - positive (Note sign change in water balance definition)	yes

Table 5.2 CI - Canopy interception water balance items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ci.qEInt	Evaporation from intercepted storage	Outflow - positive	yes
ci.dIntSto	Change in interception storage	Positive when interception storage increases	yes
ci.ciWblErr	Interception storage water balance error	Positive if water generated (Δstorage + Outflow > Inflow)	

5.3.3 Ponded water storage

Water on the ground surface belongs to the ponded water storage. Rainfall is added to ponded storage. Ponded storage evaporates, infiltrates or flows to MIKE 11.

The items listed in Table 5.3 are those found in the "*Overland flow - detailed*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
    Name = 'OL_DETAIL'
    DisplayName = 'Overland flow - detailed'
    Description = 'Detailed Overland component water balance'
    NoGroups = 23
    Group = 'qpnet(ol.qpnet)'
    Group = 'qirrdrip(ol.qirrdrip)'
    Group = 'qeol(ol.qeol)'
    Group = 'qh(ol.qh+ol.qhmp)'
    Group = 'qolszpos(-ol.qolszpos)'
    Group = 'qolszneg(-ol.qolszneg)'
    Group = 'qsztofloodpos(-ol.qsztofloodpos)'
    Group = 'qsztofloodneg(-ol.qsztofloodneg)'
    Group = 'qolin(ol.qolin)'
    Group = 'qolout(ol.qolout)'
    Group = 'qolrivpos(ol.qolrivpos)'
    Group = 'qolrivneg(ol.qolrivneg)'
    Group = 'qocdr(ol.qocdr)'
    Group = 'qocdrtoM11HPoint(ol.qocdrtoM11HPoint)'
    Group = 'qfloodtorivin(ol.qfloodtorivin)'
    Group = 'qfloodtorivex(ol.qfloodtorivex)'
    Group = 'qoldrtoMouse(ol.qoldrtoMouse)'
```

```
    Group = 'qolMousepos(ol.qolMousepos)'
    Group = 'qolMouseneg(ol.qolMouseneg)'
    Group = 'qOlExtSink(ol.qOlExtSink)'
    Group = 'qOlExtSource(ol.qOlExtSource)'
    Group = 'dolsto(ol.dolsto)'
    Group = 'olwblerr(ol.olwblerr)'
EndSect  // WblTypeDefinition[WblTypeDefinition]
```

The sign convention for a ponded water control volume is such that precipitation is negative (inflow), and boundary outflow, infiltration and evaporation are all positive (outflow). All of the Wbl Error items together should add to zero.

Note, however, the negative sign in front of some of the terms in the water balance definition above. This is because the SZ exchange to ponded storage is an upwards positive flow in MIKE SHE - making it a positive value in the MIKE SHE results files when flowing to ponded water and a negative value when infiltrating to SZ. Whereas, these flows must be the opposite sign in the water balance calculation.

Special considerations for water balances in Flood Code cells

Water on the ground surface belongs to ponded storage - except in active flood code cells. Active flood code cells are those where the cell is flooded and the water level is controlled by the water level in MIKE 11.

There are four terms in the water balance related to flood codes: qSZToFloodPos/Neg, and qFloodToRivIn/Ex.

When the groundwater table is at or above the land surface, water can exchange directly between ponded water and the saturated zone. The unsaturated zone does not exist. If the land surface is an active flood code cell, then then the water is added to or removed from the storage available for exchange with MIKE 11 and the two terms qSZFloodPos and qSZFloodNeg may be non-zero.

The exchange between ponded water and MIKE 11 in active flood code cells is calculated based on the change of storage due to the various source/sink terms over the MIKE SHE overland time step. This includes overland flow between flooded and non-flooded cells, rainfall, evaporation, infiltration to UZ, direct flow between SZ and flooded cells when the groundwater table is above ground. Thus, in a flood code cell

1 At the beginning of the overland time step, the ponded water level is set equal to the corresponding water level in MIKE 11 (if this is above the MIKE SHE ground level) and the status of the cell is set to active.

2 At the end of the overland flow time step, MIKE SHE calculates the change in ponded water level and adds or subtracts this as lateral inflow to MIKE 11 over the next MIKE 11 time step(s), covering the period of the MIKE SHE Overland time step.

Thus, qsztoflood is not directly added as lateral inflow to MIKE 11. But it’s one of the source/sink terms that contribute to the change in storage in flooded cells – which is then added to MIKE 11 as qfloodtoriv

The two terms, qFloodToRivIn and qFloodToRivEx, together are the net lateral inflow to MIKE 11 from active flood code cells. In other words, summed together, they are the actual exchange between flood code areas and MIKE 11.

Table 5.3 OL - Overland flow items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ol.qPnet	Canopy throughfall to ponded water. This is the same value as ci.qPnet, but with the opposite sign.	Inflow - negative	yes
ol.qIrrDrip	Irrigation added to ponded water. This includes both drip irrigation and sheet irrigation, since both are added directly to ponded storage.	Inflow - negative	yes
ol.qEOL	Direct evaporation from ponded water	Outflow - positive	yes
ol.qH	Infiltration from ponded water into the UZ	Outflow - positive	yes
ol.qHmp	Infiltration from ponded water into the UZ macropores	Outflow - positive	yes
-ol.qOLSZpos	Direct flow up from SZ to OL. This is a positive upwards flow in the MIKE SHE results files.	Inflow - negative (Note sign change in water balance definition)	yes

Table 5.3 OL - Overland flow items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
-ol.qOLSZneg	Direct flow down from OL to SZ. This is a negative downwards flow in the MIKE SHE results files.	Outflow - positive (Note sign change in water balance definition)	yes
-ol.qSZToFloodPos	Direct flow upwards from SZ to an active flood code cell (active means that it is actually flooded and the water level is controlled by the water level in MIKE 11). This is a positive upwards flow in the MIKE SHE results files. Only non-zero when the groundwater table is at or above the ground surface.	Inflow - negative (Note sign change in water balance definition)	yes
-ol.qSZToFloodNeg	Direct flow downwards from an an active flood code cell to SZ. (active means that it is actually flooded and the water level is controlled by the water level in MIKE 11). This is a negative downwards flow in the MIKE SHE results files. Only non-zero when the groundwater table is at or above the ground surface.	Outflow - positive (Note sign change in water balance definition)	yes
ol.qOLin	Inflow to overland storage across the boundary of the model, or inflow across the boundary of the water balance sub-area	Inflow - negative	yes
ol.qOLout	Outflow from overland storage across the boundary of the model, or outflow across the boundary of the water balance sub-area	Outflow - positive	yes

Table 5.3 OL - Overland flow items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ol.qOLRivPos	Overland outflow to MIKE 11	Outflow - positive	yes
olqOLRivNeg	Inflow from MIKE 11 to overland storage	Inflow - negative	yes
ol.qOCDr	Overland flow in paved areas that is added to the SZ drainage network and thus directly to MIKE 11	Outflow - positive	yes
ol.qOCDrToM11HPoint	Overland flow in paved areas that is added to the SZ drainage network and then directly to a specified MIKE 11 h-point	Outflow - positive	yes
ol.qOLDrToMouse	Overland flow in paved areas that is added to the SZ drainage network and then directly to a specified Mouse/MIKE Urban manhole	Outflow - positive	yes
ol.qFloodToRivIn	Net lateral inflow exchange between active flood code cells and MIKE 11 nodes that are inside the current water balance area	Inflow (negative) or Outflow (positive)	yes
ol.qFloodToRivEx	Net lateral inflow exchange between active flood code cells and MIKE 11 nodes that are outside the current water balance area This is always zero unless the water balance is being calculated on a sub-area.	Inflow (negative) or Outflow (positive)	yes
ol.qOLMousePos	Outflow from overland storage to Mouse/MIKE Urban	Outflow - positive	yes
ol.qOLMouseNeg	Inflow from Mouse/MIKE Urban to overland storage	Inflow - negative	yes
ol.qOLExtSink	Outflow to OpenMI sink	Outflow - positive	yes
ol.qOLExtSource	Inflow from OpenMI sink	Inflow - negative	yes

Table 5.3 OL - Overland flow items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ol.dOLSto	Change in overland storage	Positive if storage increases	yes
ol.OLWblErr	OL water balance error	Positive if water generated (Δstorage + Outflow > Inflow)	

5.3.4 Unsaturated Zone Storage

Unsaturated zone storage includes all the water between the ground surface and the water table. Thus, all water stored in the root zone is also included here.

The items listed in Table 5.4 are those found in the "*Unsaturated Zone - detailed*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
    Name = 'UZ_DETAIL'
    DisplayName = 'Unsaturated Zone - detailed'
    Description = 'Detailed Unsaturated zone component water balance'
    NoGroups = 10
    Group = 'qh(uz.qh)'
    Group = 'qhmp(uz.qhmp)'
    Group = 'qeuz(uz.qeuz)'
    Group = 'qtuz(uz.qtuz)'
    Group = 'qrech(-uz.qrech)'
    Group = 'qrechmp(-uz.qrechmp)'
    Group = 'qgwfeedbackuz(-uz.qgwfeedbackuz)'
    Group = 'duzdef(-uz.duzdef)'
    Group = 'uzszstocorr(uz.uzszstocorr)'
    Group = 'uzwblerr(uz.uzwblerr)'
EndSect  // WblTypeDefinition
```

The sign convention in the UZ water balance is such that infiltration from the surface is negative (inflow) and recharge to SZ is positive (outflow). All of the items together should add to zero.

Note, however, the negative sign in front of some of the terms (e.g. uz.qRech) in the water balance definition above. This is because the recharge to SZ is a vertical downward flow in MIKE SHE - making it a negative value in the MIKE SHE results files. The negative sign in the water balance conforms the sign to the water balance sign convention of positive outflows.

Table 5.4 UZ - Unsaturated Zone items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
uz.qH	Infiltration from ponded water into the UZ matrix	Inflow - negative	yes
uz.qHmp	Infiltration from ponded water into the UZ macropores	Inflow - negative	yes
uz.qEuz	Direct evaporation from the top UZ node when using the Richards or Gravity flow finite-difference method	Outflow - positive	yes
uz.qTuz	Transpiration from the root zone	Outflow - positive	yes
-uz.qRech	Recharge out of the bottom of the soil column to SZ via the UZ soil matrix. In the MIKE SHE results, recharge is a vertical downward flow (in the negative direction). In the UZ water balance it is an outflow and must be a positive value.	Outflow - positive (Note sign change in water balance definition)	yes
-uz.qRechMp	Recharge out of the bottom of the soil column to SZ via the UZ macropores. In the MIKE SHE results, recharge is a vertical downward flow (in the negative direction). In the UZ water balance it is an outflow and must be a positive value.	Outflow - positive (Note sign change in water balance definition)	yes

Table 5.4 UZ - Unsaturated Zone items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
-uz.qGWFeed-BackUZ	Feedback from LR to UZ This value is only non-zero if the Linear Reservoir groundwater option is used. In this case, the baseflow reservoirs will add water to the UZ as a fraction of the discharge to MIKE 11. In the MIKE SHE results, the feedback to UZ is a positive value. But, in the water balance it is an inflow and must have a negative sign.	Inflow - negative (Note sign change in water balance definition)	yes
-uz.dUzDef	Change in UZ deficit. The UZ deficit is essentially the amount of air in the profile. It is the opposite of the UZ storage. A decreasing deficit means that the soil is getting wetter, which equals increasing UZ storage. An increasing deficit means that the soil is getting drier, which equals decreasing UZ storage. Internally in MSHE, the value of dUzDef is calculated as a change in storage. The negative sign is added to convert the change in storage to a change in deficit.	Negative for increasing UZ deficit (Note sign change in water balance definition)	yes, but in the error term calculation the negative sign is not used

Table 5.4 UZ - Unsaturated Zone items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
uz.UzSzStorCorr	Water balance correction to account for changing thickness of the UZ zone as the groundwater table rises and falls.	Positive for a falling groundwater table, because the amount of UZ storage is increasing. Negative for a rising groundwater table, because the amount of UZ storage is decreasing.	yes
uz.uzWblErr	UZ Water balance error	Positive if water generated (Δstorage + Outflow > Inflow)	

5.3.5 Saturated Zone Storage

The saturated zone storage includes all water below the water table. All groundwater pumping is from the saturated zone, including irrigation extraction from groundwater.

The items listed in Table 5.5 are those found in the "*Saturated Zone - detailed*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
    Name = 'SZ_DETAIL'
    DisplayName = 'Saturated Zone - detailed'
    Description = 'Detailed Saturated ... balance (depth-integrated)'
    NoGroups = 28
    Group = 'qszprecip(sz.qszprecip)'
    Group = 'qrech(uz.qrech)'
    Group = 'qrechmp(uz.qrechmp)'
    Group = 'qolszpos(sz.qolszpos)'
    Group = 'qolszneg(sz.qolszneg)'
    Group = 'qetsz(sz.qetsz)'
    Group = 'qszin(sz.qszin)'
    Group = 'qszout(sz.qszout)'
    Group = 'dszsto(sz.dszsto)'
    Group = 'qszabsex(sz.qszabsex)'
    Group = 'qszdrin(sz.qszdrin)'
    Group = 'qszdrout(sz.qszdrout)'
    Group = 'qszdrtorivin(sz.qszdrtorivin)'
```

```
    Group = 'qszdrtorivex(sz.qszdrtorivex)'
    Group = 'qszdrtoM11HPoint(sz.qszdrtoM11HPoint)'
    Group = 'qszrivneg(sz.qszrivneg)'
    Group = 'qszrivpos(sz.qszrivpos)'
    Group = 'qszfloodneg(ol.qsztofloodneg)'
    Group = 'qszfloodpos(ol.qsztofloodpos)'
    Group = 'qgihbpos(sz.qgihbpos)'
    Group = 'qgihbneg(sz.qgihbneg)'
    Group = 'qirrwell(sz.qirrwell)'
    Group = 'qszdrtoMouse(sz.qszdrtoMouse)'
    Group = 'qszMousepos(sz.qszMousepos)'
    Group = 'qszMouseneg(sz.qszMouseneg)'
    Group = 'qSzExtSink(sz.qSzExtSink)'
    Group = 'qSzExtSource(sz.qSzExtSource)'
    Group = 'Error(sz.szwblerrtot)'
EndSect  // WblTypeDefinition
```

The sign convention in the SZ water balance is such that infiltration from the unsaturated zone is negative (inflow) and discharge to overland flow is positive (outflow). All of the items together should add to zero.

The use of negative signs in the SZ water balance is avoided by explicitly including both inflow (negative) and outflow (positive) terms. For example, sz.qOlSzPos is the flow from the saturated zone directly to ponded water when the groundwater table is at or above the ground surface. In the MIKE SHE results, this is a positive upwards flow, and in the water balance it is a positive outflow. Similarly, sz.qOlSzNeg is the downward flow from ponded water directly to the saturated zone, which is a negative downward flow and a negative water balance inflow to SZ.

Table 5.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sz.qSzPrecip	Precipitation added directly to the SZ layer. This can only be non-zero when the simulation does not include UZ. If UZ is included, but the groundwater table is at the ground surface (no UZ cells), the precipitation to SZ is included in the term sz.qOlSzNeg. Can be an outflow if the negative precipitation option specified in the Extra Parameters (Negative Precipitation *(V.1 p. 300)*). In this case, negative precipitation can be removed from multiple SZ layers.	Inflow - negative Can be positive (outflow) if negative precipitation option specified.	yes
uz.rech	Recharge out of the bottom of the UZ soil column to SZ via the UZ soil matrix. In the MIKE SHE results, recharge is a vertical downward flow, thus in the negative direction. This is the same sign as the water balance convention of negative inflow.	Inflow - negative	yes

Table 5.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
uz.rechmp	Recharge out of the bottom of the UZ soil column to SZ via the UZ macropores or by-pass flow. In the MIKE SHE results, recharge is a vertical downward flow, thus in the negative direction. This is the same sign as the water balance convention of negative inflow.	Inflow - negative	yes
sz.qOlSzPos	Upward flow directly from SZ to ponded water. This is non-zero only when the groundwater table is at or above the ground surface. The sign is positive upwards which is the same as the positive outflow water balance sign convention.	Outflow - positive	yes
sz.qOlSzNeg	Downward flow directly from ponded water to SZ. This is non-zero only when the groundwater table is at or above the ground surface. The sign is positive upwards which is the same as the negative inflow water balance sign convention.	Inflow - negative	yes
sz.EtSz	Evapotranspiration directly from SZ.	Positive - outflow	yes
sz.qSzIn	Inflow to SZ storage across the boundary of the model, or inflow across the boundary of the water balance sub-area. Inflow from internal fixed head cells is also included in this term.	Inflow - negative	yes

Table 5.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sz.qSzOut	Outflow from SZ storage across the boundary of the model, or outflow across the boundary of the water balance sub-area. Outflow to internal fixed head cells is also included in this term, as well as drainage to local depressions that contain a fixed head boundary condition.	Outflow - positive	yes
sz.dSzSto	Change in SZ storage	Positive when storage increases	yes
sz.qSzAbsEx	Groundwater pumping from SZ. This does not include irrigation wells and shallow irrigation wells, but includes outflow to fixed head drain internal boundary conditions.	Outflow - positive Can be negative (Inflow) if injection specified for wells.	yes
sz.qSzDrIn	SZ drainage to local depressions in the current water balance area from areas outside of the current water balance sub-area. This term also includes inflow to the SZ drainage system added via OpenMI.	Inflow - negative	yes
sz.qSzDrOut	SZ drainage to the model boundary, SZ drainage removed directly from the model. This term also includes SZ drainage to local depressions located outside of the current water balance sub-area.	Outflow - positive	yes
sz.qSzDrToRivIn	SZ drainage to MIKE SHE River Links inside of the water balance sub-area.	Outflow - positive	yes

Table 5.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sz.qSzDrToRivEx	SZ drainage to MIKE SHE River Links outside of the water balance sub-area. This can only be non-zero if the water balance is calculated for a sub-area.	Outflow - positive	yes
sz.qSzDrToM11HPoint	SZ drainage to specified MIKE 11 h-points. These are specified in the Extra Parameter option in SZ Drainage to Specified MIKE 11 H-points *(V.1 p. 316)*	Outflow - positive	yes
sz.qSzRivPos	Baseflow from SZ to MIKE River Links	Outflow - positive	yes
sz.qSzRivNeg	Infiltration from MIKE SHE River Links to SZ	Inflow - negative	yes
ol.qSZToFloodPos	Direct flow upwards from SZ to an active flood code cell (active means that it is actually flooded and the water level is controlled by the water level in MIKE 11). This is a positive upwards flow in the MIKE SHE results files. Only non-zero when the groundwater table is at or above the ground surface.	Outflow - positive (Note sign change compared to detailed Ponded Storage water balance)	yes

Table 5.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ol.qSZToFloodNeg	Direct flow downwards from an an active flood code cell to SZ. (active means that it is actually flooded and the water level is controlled by the water level in MIKE 11). This is a negative downwards flow in the MIKE SHE results files. Only non-zero when the groundwater table is at or above the ground surface.	Inflow - negative (Note sign change compared to detailed Ponded Storage water balance)	yes
sz.qGihbPos	Outflow from SZ storage to internal general head boundaries (GHB cells)	Outflow - positive	yes
sz.qGihbNeg	Inflow from internal general head boundaries (GHB cells) to SZ storage	Inflow - negative	yes
sz.qIrrWell	Groundwater pumping from irrigation wells. This includes both specified irrigation wells and shallow wells.	Outflow - positive	yes
sz.qSzDrToMouse	SZ drainage to specified MOUSE/MIKE Urban manholes. These are specified in the Extra Parameter option in SZ Drainage to MOUSE *(V.1 p. 319)*	Outflow - positive	yes
sz.qSzMousePos	Outflow from SZ storage to Mouse/MIKE Urban pipes	Outflow - positive	yes
sz.qSzMouseNeg	Inflow from Mouse/MIKE Urban pipes to SZ storage	Inflow - negative	yes
sz.qSzExtSink	Outflow to external sinks specified via OpenMI	Outflow - positive	yes

Table 5.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sz.qSzExtSource	Inflow from external sources specified via OpenMI	Inflow - negative	yes
sz.szWblErrTot	Aggregated SZ water balance error for all layers	Positive if water generated (Δstorage + Outflow > Inflow)	

Saturated zone layers

The saturated zone water balance can also be calculated by numerical layer. This means that all of the items in Table 5.5 are repeated for each numerical layer. However, in this case the water balance error term, sz.szWblErrTot is replace by a water balance error for each layer.

The layer water balance is slightly more complicated. It includes terms for the exchange between layers, and the upper layer includes the terms for the exchange with UZ and ponded water.

In particular, the output for each SZ layer water balance only includes the exchange with the layer above. This is found in the two additional layer water balance terms qSzZpos and qSzZneg.

The first term, qSzZpos, is the flow from the current layer upwards to the layer above. In the results files, this term is in the positive (upwards) direction. In the water balance, the term is also a positive outflow.

The second term, qSzZneg, is the flow from the layer above downwards into the current layer. In the results files, this term is in the negative (downwards) direction. In the water balance, the term is also a negative inflow to the current layer.

Note: The layer water balance error includes the flows to and from the layers above and below. However, when summing up the flows, the sign must be changed for the qSzZpos and qSzZneg terms that originate from the layer below.

Table 5.6 SZ - Saturated Zone - specified by layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sz.qSzZpos	Upward SZ flow from the current layer to the layer above Only available for LAYER water balances	Outflow - positive	yes
sz.qSzZNeg	Downward SZ flow from the layer above to the current layer. Only available for LAYER water balances	Inflow - negative	yes
sz.szWblErr	SZ water balance error for the current layer only available for LAYER water balances	Positive if water generated (Δstorage + Outflow > Inflow)	

Saturated Zone Linear Reservoir water balance

If the linear reservoir method is used for the saturated zone, the water balance terms are basically the same but are slightly less transparent.

The layer output for the linear reservoir method divides the SZ into two layers - the interflow reservoirs and the baseflow reservoirs. For the linear reservoir layers, there is no distinction between the two parallel baseflow reservoirs, or the cascading interflow reservoirs.

The items listed in Table 5.7 are those found in the "*Saturated Zone - layers(Linear Reservoir)*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
   Name = 'SZ_LAYER_LR'
   DisplayName = 'Saturated Zone - layers(Linear Reservoir)'
   Description = 'Saturated zone water balance for linear reservoir'
   NoGroups = 13
   Group = 'recharge(uz.qrech+uz.qrechmp)'
```

```
        Group = 'evapotranspirationSZ(sz.qetsz)'
        Group = 'lateral IN(sz.qszin)'
        Group = 'lateral OUT(sz.qszout)'
        Group = 'percolation(sz.qszzneg)'
        Group = 'To river(sz.qszrivpos)'
        Group = 'From river(sz.qszrivneg)'
        Group = 'storagechange(sz.dszsto)'
        Group = 'deadzonestoragechange(sz.dszsto_dead)'
        Group = 'pumping(sz.qszabsex)'
        Group = 'Irr.pumping(sz.qirrwell)'
        Group = 'feedbackUZ(sz.qUZfeedback)'
        Group = 'Error(sz.szwblerr)'
    EndSect  // WblTypeDefinition
```

Table 5.7 SZ - Saturated Zone - Linear Reservoir all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
recharge (uz.qrech+uz.qrechmp)	This is the total recharge into the interflow reservoirs. If UZ is not simulated, then uz.qrech is still calculated based on the infiltration from OL.	Inflow - negative	yes
evapotranspirationSZ (sz.qetsz)	This is the direct ET from the water table. In the LR SZ method, the water table is constant and fixed at the beginning of the simulation. If the root zone reaches the water table, then ET will be taken from the water table as an infinite sink when the reference ET is not satisfied by the other sources.	Outflow - postitive	yes

Table 5.7 SZ - Saturated Zone - Linear Reservoir all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
lateral IN (sz.qszin)	In the LR SZ model, infiltration to the interflow reservoirs and percolation to the baseflow reservoirs is distributed equally to the entire reservoir. When you calculate the water balance in a sub-area, sz.qszin is the amount of recharge/percolation that is distributed into the sub-area. For example, if all your recharge occurs outside of your sub-area, this is the increase in groundwater storage that occurs inside your sub-area. This can only be non-zero for sub-area water balances.	Inflow - negative	yes
lateral OUT (sz.qszout)	In the LR SZ model, infiltration to the interflow reservoirs and percolation to the baseflow reservoirs is distributed equally to the entire reservoir. When you calculate the water balance in a sub-area, sz.qszout is the amount of recharge/percolation that is distributed to areas outside of the sub-area. For example, if all your recharge occurs inside your sub-area, this is the increase in groundwater storage that occurs outside your sub-area. This can only be non-zero for sub-area water balances.	Outflow - positive	yes

Table 5.7 SZ - Saturated Zone - Linear Reservoir all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
percolation (sz.qszzneg)	Infiltration from interflow reservoirs to baseflow reservoirs. This is defined only for the lower (baseflow) layer in the water balance output, but is used in the water balance error calculation of the interflow reservoirs with the opposite sign. The term sz.qszzpos is not included here because the LR method does not allow any transfer of water from the baseflow reservoir upwards to the interflow reservoir.	Inflow - negative (to the baseflow reservoir)	yes
To river (sz.qszrivpos)	Outflow from interflow and baseflow reservoirs to MIKE SHE River Links.	Outflow - positive	yes
From river (sz.qszrivneg)	Inflow from MIKE SHE River Links to the baseflow reservoir. For the interflow reservoirs, this is always zero because MIKE 11 only discharges to the baseflow reservoirs.	Inflow - negative (to the baseflow reservoir)	yes
storagechange (sz.dszsto)	Change in storage in the interflow and baseflow reservoirs.	Positive if storage increases	yes
deadzonestoragechange (sz.dszsto_dead)	Change in storage in the deadzone storage. This is calculated as a change in storage, but it is equal to the outflow to dead zone storage because there is no option in MIKE SHE to reduce the dead zone storage.	Outflow - Positive	yes

Table 5.7 SZ - Saturated Zone - Linear Reservoir all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
pumping (sz.qszabsex)	Groundwater pumping from the baseflow reservoirs. This is always zero for the interflow reservoirs.	Outflow - positive But, can be negative if injection rates specified in wells	yes
Irr.pumping (sz.qirrwell)	This is the sum of groundwater pumping for irrigation - irrigation wells + shallow irrigation wells.	Outflow - positive	yes
feedbackUZ (sz.qUZfeedback)	This is a fraction of the discharge from the baseflow reservoirs to MIKE 11 to account for discharge to riparian zones that is lost to ET.	Outflow - positive (from baseflow reservoirs only)	yes
Error (sz.szwblerr)	SZ water balance error for the current layer only available for LAYER water balances	Positive if water generated (Δstorage + Outflow > Inflow)	
Error (sz.szwblerrtot)	SZ water balance error for the both the interflow and baseflow reservoirs combined. This is only available for the total water balance option.	Positive if water generated (Δstorage + Outflow > Inflow)	

5.3.6 Limitations for Linear Reservoir and Sub-catchment OL Water Balance

The water balance calculations have the following restrictions on **single-cell, sub-catchment** water balances, with the SZ Linear Reservoir and Simple OL:

- single-cell : won't be correct for TOTAL, OL, SZ water balances. But can be used for UZ and others.

- sub-catchment: For TOTAL and OL water balances the smallest valid water balance sub-catchment is one Overland flow zone (i.e. topographical zone) within one hydrological sub-catchment. If a water balance sub-catchment excludes part of an Overland flow zone within one hydrological sub-catchment, the water balances will be wrong in many cases because the OL storage is not necessarily uniformly distributed over one Overland flow zone, while there is only one value for flows between OL flow zones, source/sink terms, etc.
- For TOTAL and SZ water balances: Same restrictions apply, but here with the interflow reservoirs.

There are no restrictions with respect of the baseflow reservoir distributions.

The pre-processor warns in case the above restrictions are violated. It can't give an error, because this program doesn't know which type (TOTAL/OL/SZ/...) the user will specify in the water balance Post-processor.

Basically, sub-catchment water balances can be misleading when using the linear reservoir method. For example, a baseflow reservoir receiving percolation from several subcatchments only "sees" the total amount of percolation. If you make a sub-catchment water balance for one of the sub-catchments, then the water balance program will return the amount of percolation for the subcatchment. However, the baseflow reservoir only received the "average" over the area (total percolation/baseflow res. area).

The difference between these two values will be reflected in the water balance as a "boundary flow" for the sub-catchment, which is obviously not really correct. The same situation applies for river link infiltration to baseflow reservoirs.

5.4 Standard Water Balance Types

Table 5.8 summarizes the 31 standard water balance types defined in the water balance configuration file. Some of the water balances cannot be used in certain conditions and these restrictions are listed in the table.

Table 5.8 Water balance types available in the default configuration files.

Water balance type	Description
Total waterbalance	General water balance of the entire model setup
Error of each component	The water balance error of each model component
Snow Melt component	Snow Melt component water balance items
Canopy Interception component	Canopy Interception component water balance items
Overland flow	Overland component water balance
Overland flow - detailed	Detailed Overland component water balance
Unsaturated Zone	Unsaturated zone component water balance
Unsaturated Zone - detailed	Unsaturated zone component water balance
Saturated Zone	Saturated zone component water balance (depth-integrated)
Saturated Zone - layer(s)	Saturated zone component water balance (each or specified layer)
Saturated Zone - detailed	Detailed Saturated zone component water balance (depth-integrated)
Saturated Zone - detailed - layer(s)	Detailed Saturated zone component water balance (each or specified layer)
Saturated Zone (Linear Reservoir)	Saturated Zone component water balance for the linear reservoir
Saturated Zone -layers (Linear Reservoir)	Saturated Zone component water balance for the linear reservoir
Irrigation component	Irrigation component water balance

Table 5.8 Water balance types available in the default configuration files.

Water balance type	Description
MOUSE-coupling terms	MIKE SHE - MOUSE exchange (depth-integrated)
MOUSE-coupling terms, Saturated zone - layer(s)	MIKE SHE sat.zone - MOUSE exchange (each or specified layer)
Map output: Total error	Distributed output: Total water balance error
Map output: Overland flow error	Distributed output: Overland water balance error
Map output: Unsat. Zone error	Distributed output: Unsat.zone water balance error
Map output: Sat. Zone error	Distributed output: Saturated zone water balance error (depth-integrated)
Map output: Sat. Zone error - layer(s)	Distributed output: Saturated zone water balance error (each or specified layer)
Map output: Total irrigation	Distributed output: Total irrigation
Chart output: Total water balance	Chart output: General water balance of the entire model (depth-integrated)
Chart output: Total + each SZ layer	Chart output: General water balance of the entire model (each SZ layer)
Chart output: Total water balance TEXT IN DANISH	Chart output: Generel vandbalance for hele modellen (dybde-integreret)
Chart output: Total + each SZ layer TEXT IN DANISH	Chart output: Generel vandbalance for hele modellen (hvert SZ-lag)
Saturated Zone	StorageSaturated zone Storage (depth-integrated)
Saturated Zone Storage - layer(s)	Saturated zone Storage (each or specified layer)
Map output: Saturated Zone Storage	Distributed output: Saturated zone Storage (depth-integrated)
Map output: Saturated Zone Storage - layer(s)	Distributed output: Saturated zone Storage (each or specified layer)

5.5 Making Custom Water Balances

The first combobox in the Post-processing dialogue contains a list of all the available water balance types. This list is read from the water balance configuration file, `MSHE_Wbl_Config.pfs`, which is found in the MIKE SHE installation \bin directory. The default location of this directory depends on the operating system of your computer.

You can add extra items to the list of available water balance types by defining additional water balances at the end of the configuration file.

To illustrate how you could add an additional water balance type, the table below describes the format for each line of the water balance type definition. The example is for an extra water balance type to calculate the net vertical flow in a specified SZ layer. This water balance type can only be used with the single-cell resolution and specified output layers options.

Line item	Comment

`// Created: 2004-06-2 16:28:48`
`// DLL id : C:\WINOWS\System32\pfs2000.dll`
`// PFS version: Mar 3 2004 21:35:12`
 	
`[MIKESHE_WaterBalance_ConfigFile]`
`FileVersion = 3`
`NoWblTypes = 31`
	File header
	
	`NoWblTypes` = the number of water balance types in the configuration file. Remember to change this number if you add a water balance item to the file

`[WblTypeDefinition]`
`Name = 'TOTAL'`
`...`
`Group = 'SZ Storage(sz.szsto)'`
`EndSect // WblTypeDefinition`
	Existing water balance definitions

`[WblTypeDefinition]`	First line of the water balance definition
`Name = 'SZ_LAYER_NET_VERT_FLOW_MAP'`	Internal name. No spaces allowed
`DisplayName = 'Map output: Net Vertical Saturated Zone Flow - layer(s)'`	Name displayed in the combobox
`Description = 'Distributed output: Saturated zone Storage (specified layer)'`	Description displayed under the combobox
`NoGroups = 1`	Number of calculation groups in the output file

Line item	Comment
`Group = 'SZ Vertical Flow(sz.qszzpos+ sz.qszzneg)'`	Definition of the calculation group, consisting of a name and a sum of the particular water balance items (no spaces) from Table 5.1 to Table 5.7. Map items can only have one group (NoGroups = 1)
`EndSect // WblTypeDefinition`	
`EndSect // MIKESHE_WaterBalance_ConfigFile`	last line in the file

When making custom water balance types the format of the default water balance configuration file must be maintained. Variable names, including names in square brackets, are case sensitive and the number of spaces in variable names must be consistent with the default configuration file.

5.5.1 Customizing the chart output

The chart water balance is a special water balance function that creates an ASCII file that is read by another program to generate the graphic in Figure 5.1.

The default setup of the items in the chart output do not follow the typical sign convention of the water balance. The sign convention has been adjusted to make the chart output more logical. Thus, in the chart output both precipitation and evapotranspiration are positive values. Whereas, in the standard water balance, precipitation is negative.

The items included in the graphic are in the water balance configuration file. The Group sections include a range of options for displaying the output on the graphic, including arrow directions and locations.

.

Line item	Comment

`// Created: 2004-06-2 16:28:48`
`// DLL id : C:\WINOWS\System32\pfs2000.dll`
`// PFS version: Mar 3 2004 21:35:12`
 	
`[MIKESHE_WaterBalance_ConfigFile]`
`FileVersion = 3`
`NoWblTypes = 31`
	File header
	
	`NoWblTypes` = the number of water balance types in the configuration file. Remember to change this number if you add a water balance item to the file

`[WblTypeDefinition]`
`Name = 'TOTAL'`
`...`
`Group = 'SZ Storage(sz.szsto)'`
`EndSect // WblTypeDefinition`
	Existing water balance definitions

`[WblTypeDefinition]`	First line of the water balance definition
`Name = 'TOTAL_CHART'`	Internal name. No spaces allowed
`DisplayName = 'Chart output: Total Water balance'`	Name displayed in the combobox
`Description = 'Chart output: General water balance of the entire model (depth-integrated)'`	Description displayed under the combobox
`NoGroups = 23`	Number of calculation groups in the output file
`Group = 'SKY TV 45 40 Precipitation(-sm.qp)'`	Various display items for the arrows and items
`EndSect // WblTypeDefinition`	
`EndSect // MIKESHE_WaterBalance_ConfigFile`	last line in the file

RUNNING MIKE SHE

6 RUNNING YOUR MODEL

In the top icon bar, there is a three-button set of icons for running your model. 

.

PP - The PP button starts the preprocessing. You must first **Pre**P**rocess** your model data to create the numerical model from your grid independent data. See Preprocessing your model *(V.1 p. 149)*.

WM - The WM button starts the **W**ater **M**ovement simulation. You can only run your water movement simulation after you have preprocessed your data. See Running your Model *(V.1 p. 149)*.

WQ - The WQ button starts the **W**ater **Q**uality simulation. After you have successfully run a water movement simulation to completion, you can run a water quality simulation.

In addition to the three icon buttons, there is a Run menu. In this menu, you can check on and off all three of the above options. Finally, there is an Execute... menu sub-item that runs only the checked items above it. The Execute option can also be launched using the Alt - R - E hot-key sequence.

6.1 Preprocessing your model

In the Setup Tab, you specify the input data required by the model - including the size of the model and the numerical grid. However, most of the setup data is independent of the model extent and grid. When you pre-process you model set up, MIKE SHE's pre-processor program scans through your model set up and interpolates all spatial data to the specified model grid. This interpolated set up data is stored in a .fif file, which is read during the simulation by the MIKE SHE engine. However, the pre-processed data does not include any time information. All time series information must be interpolated dynamically during the run because MIKE SHE dynamically changes the time step during the simulation in response to stresses on the system.

The Preprocessed Data Tab is used to display the pre-processed data.

Before you run your simulation, you should carefully check the preprocessed data for errors. Errors found in the preprocessed data are typically related to incorrectly specified parameters, file names, etc. in the Setup Tab.

On the main pre-processed dialogue, there is a uneditable text box containing the file and location of the pre-processed data. This is a .pfs ASCII file containing the file references for all of the data. The actual data is stored in a .fif file, as well as a number of dfs2 and dfs3 files.

After you have successfully preprocessed your model, the pre-processed data will be automatically loaded when you expand the data tree. The data tree reflects all the spatial data defined in the model set up tab. In other words, if the overland flow is not included in the Simulation Specification *(V.2 p. 27)* dialogue, then the Overland item will not be included in the pre-processed data tree.

Note If you change your model setup data, the pre-processed data will not reflect the changes until you pre-process your model again.

6.1.1 Viewing the pre-processed data

In all map and time series views, there is a View button. This view button will open the dfs0, dfs2 or dfs3 file that was generated by the pre-processor in either the Grid Editor or the Time Series Editor. However, each of these files usually contains a large number of data items. The Grid or the Time Series Editor opens at the first item, so you must use the scrolling function in the editor to find the data item that you want.

6.1.2 Editing the pre-processed data

MIKE SHE reads only the .fif file during the simulation. The .dfs2 and dfs3 files are created to make it easier to view and plot the preprocessed data. If you edit the dfs2 or dfs3 files, the changes will not be used in the simulation.

If you want to change the pre-processed data and use the changed data in the simulation, you have a couple of options.

Option 1

1 Right click on the map view and save the data to a new dfs2 file,

2 open the new dfs2 file in the Grid Editor, and

3 make the changes in the new dfs2 file and save the file.

Option 2

1 Use the View button to open the dfs2 or dfs3 pre-processed file in the Grid Editor,

2 make your changes in the file, and

3 save the file with a new name.

In both options above, you then use the new dfs2 or dfs3 file as input in the Setup tab.

6.2 Pre-processed data items

The following sections describe in more detail some of the pre-processed data items.

6.2.1 MIKE 11 coupling

The coupling between MIKE 11 and MIKE SHE is made via river links, which are located on the edges that separate adjacent grid cells. The river link network is created by the pre-processor, based on the MIKE 11 coupling reaches. The entire river system is always included in the hydraulic model, but MIKE SHE will only exchange water with the coupling reaches.

The location of each of MIKE SHE river link is determined from the co-ordinates of the MIKE 11 river points, where the river points include both digitised points and H-points on the specified coupling reaches. Since the MIKE SHE river links are located on the edges between grid cells, the details of the MIKE 11 river geometry can be only partly included in MIKE SHE, depending on the MIKE SHE grid size. The more refined the MIKE SHE grid, the more accurately the river network can be reproduced. This also leads to the restriction that each MIKE SHE grid cell can only couple to one coupling reach per river link. Thus, if, for example, the distance between coupling reaches is smaller than half a grid cell, you will probably receive an error, as MIKE SHE tries to couple both coupling reaches to the same river link.

The river links are shown on Rivers and Lakes data tree pages, as well as the SZ Drainage to River page.

Related Items:

- Surface Water in MIKE SHE *(V.1 p. 171)*
- Coupling of MIKE SHE and MIKE 11 *(V.1 p. 199)*

6.2.2 Land Use

The vegetation distribution is displayed on a map, but if you use the vegetation database for specifying the crop rotation, this information will not be displayed in the pre-processor.

Shape files

If you have used shape files for the Land Use distribution, then the PP output order may not reflect the input order if the polygons are labeled with text strings. In this case, the PP program reads the polygons and orders them in the order that they are encountered during the pre-processing. The

6.2.3 Unsaturated Flow

The Unsaturated Flow data tree in the pre-processed data contains a two noteworthy data items.

Soil profiles

Under the unsaturated zone, you will find a map with the grid codes for each of the soil profiles used. Accompanying this map is a text page containing the details of all the soil profiles. At the top of this page is the path and file name of the generated text file, which you can open in any text editor.

Note If you are using one of the finite difference methods, the pre-processor modifies the vertical discretisation wherever the vertical cell size changes. Thus, if you have 10 cells of 20cm thickness, followed by 10 cells of 40cm thickness, the location of the transition will be moved such that the two cells on either side will be have an equal thickness. In this case, cells 10 and 11 will both be 15cm.

UZ Classification Codes

If certain conditions are met, then the flow results for a 1D unsaturated zone column can be applied to columns with similar properties. If you chose to use this option, then a map will be generated that shows the calculation cells and the corresponding cells to which the results will be copied.

The cell with a calculation is given an integer grid code with a negative value. The flows calculated during the simulation in the cells with the negative code, will be transferred to all the cells with the same positive grid code value. For example, if an UZ recharge to SZ of 0.5 m^3/day is calculated for UZ grid code -51, then all the SZ cells below the UZ cells with a grid code of +51 will also be given the same recharge.

Tip This map can be difficult to interpret without using the Grid Editor.

Related Items:

- Unsaturated Zone *(V.2 p. 127)*
- Soil Profile Definitions *(V.2 p. 132)*

- Partial automatic classification *(V.2 p. 137)*
- Specified classification *(V.2 p. 138)*

6.2.4 Saturated Flow

The saturated zone data is generally written to a dfs3 file. In the map view, there is a combo box where you can specify the layer that you want to view.

Specific Yield of upper SZ layer

MIKE SHE forces the specific yield of the top SZ layer to be equal to the “specific yield” of the UZ zone as defined by the difference between the specified moisture contents at saturation, θ_s, and field capacity, θ_{fc}.This correction is calculated from the UZ values in the UZ cell in which the initial SZ water table is located. This is reflected in the pre-processed data.

For more information on the SZ-UZ specific yield see Specific Yield of the upper SZ numerical layer *(V.1 p. 252)*.

Saturated Zone Drainage

The rate of saturated zone drainage is controlled by the drain elevation and the drain time constant. However, the destination of the drainage water is controlled by the drain levels and the drain codes, which determine if the water flows to a river, a boundary, or a local depression. The algorithm for determining the drainage source-recipient reference system is described in Groundwater Drainage *(V.1 p. 60)*.

During the preprocessing, each active drain cell is mapped to a recipient cell. Then, whenever drainage is generated in a cell, the drain water will always be moved to the same recipient cell. The drainage source-recipient reference system is displayed in the following two grids

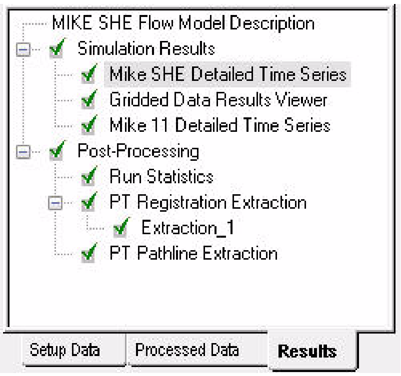
Drainage to local depressions and boundary - This grid displays all the cells that drain to local depressions or to the outer boundaries. All drainage from cells with the same negative value are drained to the cell with the corresponding positive code. If there is no corresponding positive code, then that cell drains to the outer boundary, and the water is simply removed from the model. Cells with a value of zero either do not generate drainage, or they drain to a river link.

Drainage to river - This grid displays all of the cells that drain to river links. All drainage from cells with the same negative value are drained to the cell with the corresponding positive code. Cells with a value of zero either do not generate drainage, or they drain to a the outer boundary or a local depression.

Related Items:

- Groundwater Drainage *(V.1 p. 60)*
- Drainage *(V.2 p. 173)*
- Drain Level *(V.2 p. 176)*
- Drain Time Constant *(V.2 p. 177)*
- Drain Codes *(V.2 p. 178)*
- Option Distribution *(V.2 p. 179)*

6.3 The Results Tab



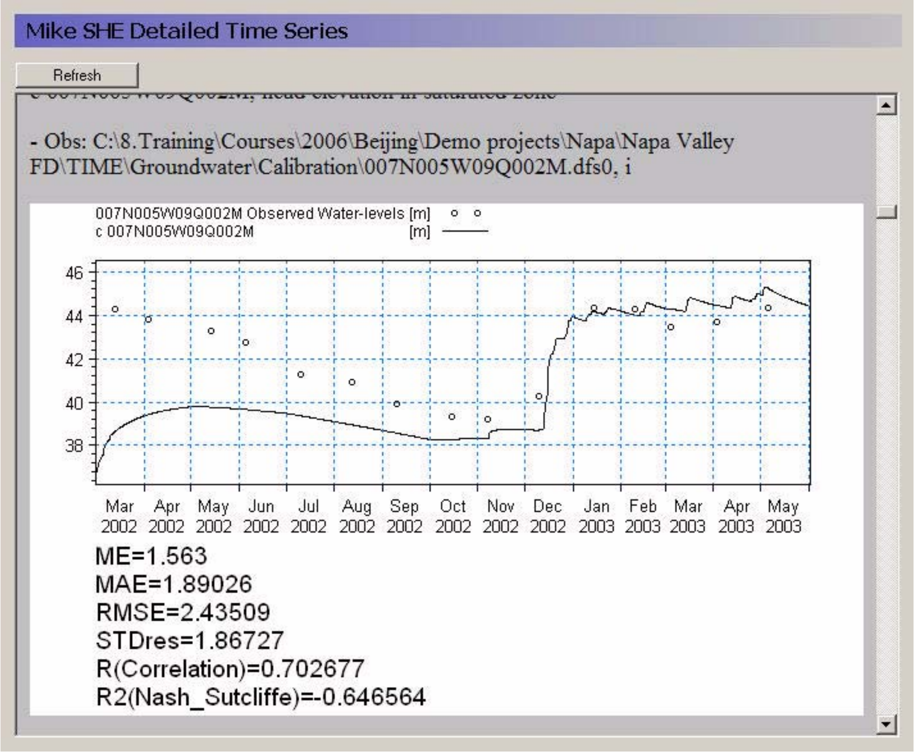
All the simulation results are collected in the Results tab. This includes Detailed time series output for both MIKE SHE and MIKE 11, as well as Grid series output for MIKE SHE.

A Run Statistics tool is available for helping you assimilate the calibration statistics for each of the detailed time series plots.

A link to the GeoScene3D program is also included, where you can visualize your results in a dynamic 3D environment. This program is widely used in Denmark and can be independently purchased from www.i-gis.dk.

The Results post-processing section contains options for post-processing the random walk particle tracking results.

6.3.1 Detailed Time Series Results

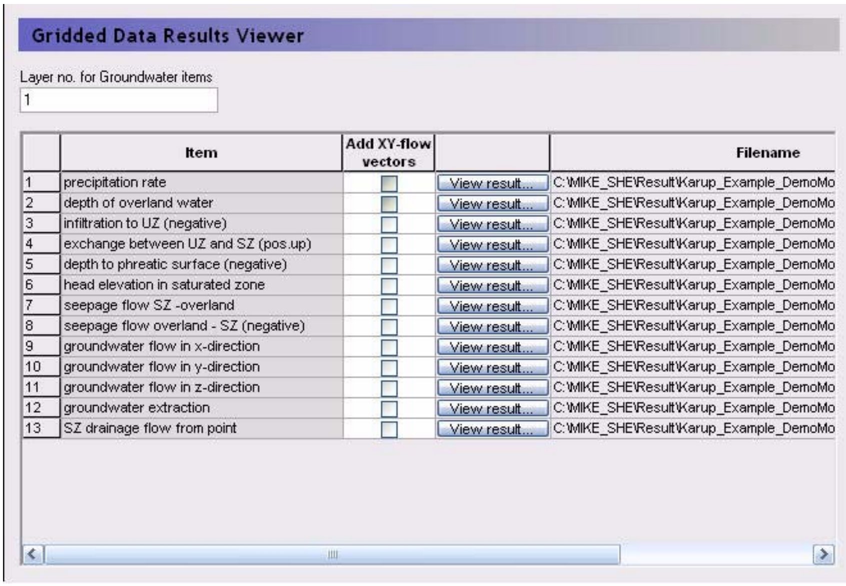


The MIKE SHE Detailed time series tab includes an HTML plot of each point selected in the Setup Editor. The HTML plots are updated during the simulation whenever you enter the view. Alternatively, you can select the Refresh button to refresh the plot.

Note: The HTML plot is regenerated every time you enter the Detailed Time Series page. So, if you have a lot of plots and a long simulation, then the regeneration can take a long time.

For information on the statistics see Statistic Calculations *(V.2 p. 217)*.

6.3.2 Gridded Results



Gridded data results for MIKE SHE can be viewed by selecting the Gridded Data Results Viewer item on the Results tab. The table is a list of all gridded data saved during a MIKE SHE simulation. The items in this list originate from the list of items selected in the Grid series output *(V.2 p. 192)* dialogue from the Setup tab.

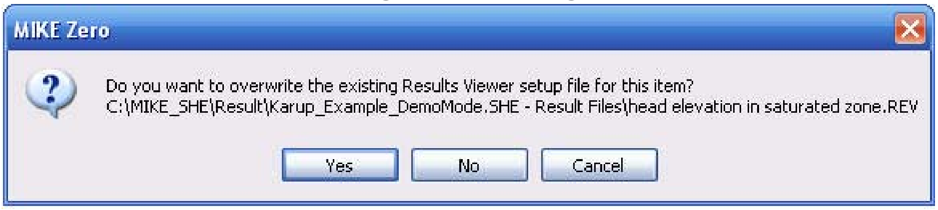
Clicking on the View result button will open the Results Viewer to the current item. All overlays from MIKE SHE (e.g. shape files, images, and grid files) will be transferred as overlays to the result view. However, the MIKE 11 river network is not transferred as an overlay.

Layer number - For 3D SZ data files, the layer number can be specified at the top of the table. However, the layer number can be changed from within the Results Viewer (see Adding additional result files and overlays *(V.1 p. 83)*) By default the top layer is displayed.

Vectors - Vectors can be added to the SZ plots of results, by checking the *Add X-Y flow vectors* checkbox. These vectors are calculated based on the *Groundwater flow in X-direction* and *Groundwater flow in Y-direction* data types if they were saved during the simulation.

In the current version, velocity vectors cannot be added for overland flow output.

The “Overwrite existing file” warning

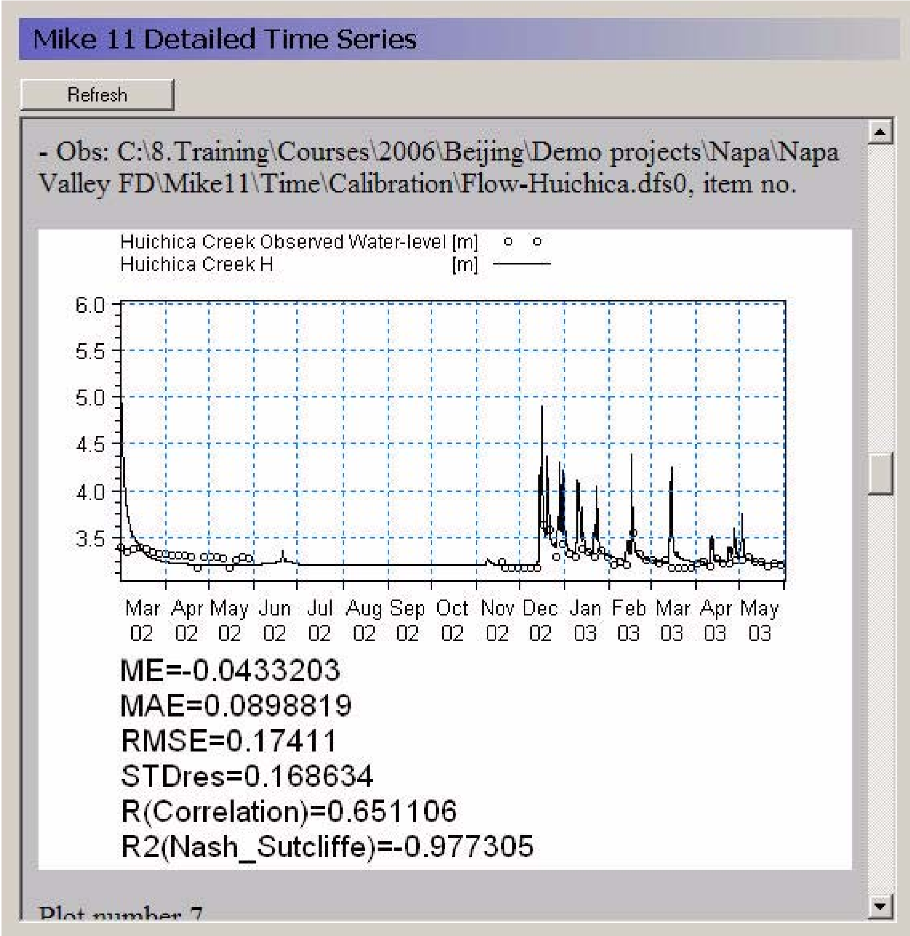


When the Result Viewer opens one of the items in the table, it creates a setup file for the particular view with the extension .rev. The name of the current .rev file is displayed in the title bar of the Results Viewer.

Initially, the .rev file includes the default view settings and the overlay information from MIKE SHE. However, if you make changes to the view, such as changes in the way contours are displayed, when you close the view, you will be asked if you want to save your changes. The .rev file can be opened directly at any time and your results will be displayed using the saved settings.

However, the next time you open the item in the table, you will be asked if you want to overwrite the existing .rev file. If you click on “Yes”, then a new .rev file will be created. If you click on “No”, then your previous settings will be re-loaded, and your results will open with the settings from the previous time you opened these results.

6.3.3 MIKE 11 Detailed Time Series



The MIKE 11 Detailed time series tab includes an HTML plot of each point selected in the Setup Editor. The HTML plots are updated during the simulation whenever you enter the view. Alternatively, you can select the Refresh button to refresh the plot.

Note: The HTML plot is regenerated every time you enter the Detailed Time Series page. So, if you have a lot of plots and a long simulation, then the regeneration can take a long time.

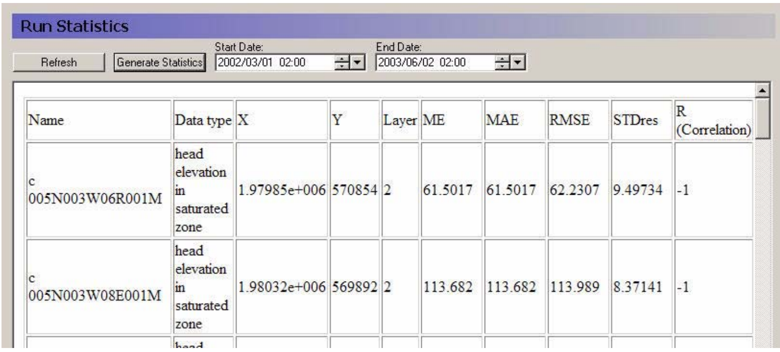
For information on the statistics see Statistic Calculations *(V.2 p. 217)*.

6.3.4 Run Statistics

Run statistics can be generated in HTML format for a MIKE SHE simulation. The run statistics table information can be copied and pasted directly into any word processing program, such as Microsoft Word, or spreadsheet, such as Microsoft Excel. The Run Statistics HTML document includes MIKE SHE and MIKE 11 results for all items included in the MIKE SHE and MIKE 11 detailed time series sections that also include observation data.

To calculate Run Statistics for a simulation, navigate to the Results Tab and the Run Statistics item on the menu tree. Press the Generate Statistics button on the Run Statistics window to perform the statistical calculations. For some simulations with long simulation periods and/or a lot of calibration data it can take a while to generate the run statistics.

After successful completion of the Generate Statistics phase, the Run Statistics HTML document will be displayed in the window on the Run Statistics page (see below).



Similar to the detailed time series output, the Run Statistics can be viewed during a simulation. Press the Refresh button on the Run Statistics page to update the Run Statistics using the most recent model results during a simulation

For information on the statistics see Statistic Calculations *(V.2 p. 217)*.

Shape file output for run statistics

A shape file of statistics is also generated when the html document is generated. The shape file contains all of the information contained in the HTML document and can be used to generate maps of model errors that can be used to evaluate spatial bias. The shape file is created in the simulation directory and is named *ProjectName*_Stat.shp where *ProjectName* is the name of the *.she file for the simulation. Note: the Run Statistics shape file does not have a projection file associated with it and this file should be created using standard ArcGIS methods.

The statistics contained in the HTML document and the shape file are calculated using the same methods used to calculate statistics for the detailed time series output. The reader is referred to the Detailed Time Series Output section for more information on how the statistics are calculated.

6.4 Controlling your simulation

6.4.1 Model Limits

Although, there are no physical limits to the size of your model, there are practical limits and hardware limits.

The practical limits are generally related to run time. We all want the model to be a little bit bigger or more detailed. However, that little extra detail or slightly smaller grid size can quickly lead to long run times.

The physical limits are generally related to memory size. If you model requires more memory than is physically installed on the computer, then the computer will start to swap data to the hard disk. This will vastly slow down your simulation. The section, Hardware Requirements *(V.1 p. 24)*, outlines some hardware considerations when using MIKE SHE.

If your model reaches the practical or physical limits of your computer, then may we suggest the following:

1 Critically evaluate your model to see if you really need such a large, complex model. For example, you may be able to reduce the number of UZ elements or the slightly increase the grid size.

2 Do a rough calibration with a smaller model first. The model independent structure of MIKE SHE makes it reasonable to refine your model later with a minimum of effort. For example, you can use Gravity flow instead of Richards equation, double the grid spacing, or shorten the calibration period, during the initial calibration and switch back to the original during the final calibration. You might even be surprised that the rougher model is actually good enough.

6.4.2 Speeding up your simulation

In most cases, the best way to speed up your model is to make it simpler. You should look very carefully at your model and ask yourself the following questions, for example:

- **Do you really need a fine discretisation during calibration?** A coarser grid may allow you to do many more calibration runs. Then when the model is calibrated, you can refine the grid for the final simulations - but remember to check you calibration first.

- **Do you really need the Richards equation for unsaturated flow?** For regional models, the two layer water balance method is usually sufficient, which is very fast. The gravity flow method is also, typically 2-5 times faster than the Richards equation method. Again during the calibration it can be a good idea to use one of the simpler methods and the more detailed method for the final simulations.
- **Is your MIKE 11 simulation too detailed?** If your MIKE 11 cross-sections are too close together, MIKE 11 will run with a very short time step. Regional models can often be run with the simple routing methods in MIKE 11, which are very fast.

If your simulation is still too slow, then several sections in the manual might be of help. In particular,

- Hardware Requirements *(V.1 p. 24)* contains information on different hardware configurations,
- Controlling the Time Steps *(V.1 p. 161)* contains information on how the dynamic time step control works,
- Overland Flow Performance *(V.1 p. 178)* contains information on how to improve the efficiency of the overland flow solution, which can be very time consuming if you have permanently ponded water,
- Parallelization of MIKE SHE *(V.1 p. 167)* contains information on the using MIKE SHE with multi-core PCs and 64-bit operating systems.

6.4.3 Controlling the Time Steps

Each of the main hydrologic components in MIKE SHE run with independent time steps. Although, the time step control is automatically controlled, whenever possible, MIKE SHE will run with the maximum allowed time steps.

The component time steps are independent, but they must meet to exchange flows, which leads to some restrictions on the specification of the maximum allowed time steps.

- If MIKE 11 is running with a constant time step, then the Max allowed Overland (OL) time step must be a multiple of the MIKE 11 constant time step. If MIKE 11 is running with a variable time step, then the actual OL time step will be truncated to match up with the nearest MIKE 11 time step.
- The Max allowed UZ time step must be an even multiple of the Max allowed OL time step, and

- The Max allowed SZ time step must be an even multiple of the Max allowed UZ time step.

Thus, the overland time step is always less than or equal to the UZ time step and the UZ time step is always less than or equal to the SZ time step.

If you are using the implicit solver for overland flow, then a maximum OL time step equal to the UZ time step often works. However, if you are using the explicit solver for overland flow, then a much smaller maximum time step is necessary, such as the default value of 0.5 hours.

If the unsaturated zone is included in your simulation and you are using the Richards equation or Gravity Flow methods, then the maximum UZ time step is typically around 2 hours. Otherwise, a maximum time step equal to the SZ time step often works.

Groundwater levels react much slower than the other flow components. So, a maximum SZ time step of 24 or 48 hours is typical, unless your model is a local-scale model with rapid groundwater-surface water reactions.

Precipitation-dependent time step control

Periods of heavy rainfall can lead to numerical instabilities if the time step is too long. To reduce the numerical instabilities, the a time step control has been introduced on the precipitation and infiltration components. You will notice the effect of these factor during the simulation by suddenly seeing very small time steps during storm events.

The parameters controlling the time step adjustment are in the Time Step Control *(V.2 p. 31)* dialogue. In particular, the following three parameters control the time step during rainfall events:

Max precipitation depth per time step If the total amount of precipitation [mm] in the current time step exceeds this amount, the time step will be reduced by the increment rate. Then the precipitation time series will be resampled to see if the max precipitation depth criteria has been met. If it has not been met, the process will be repeated with progressively smaller time steps until the precipitation criteria is satisfied. Multiple sampling is important in the case where the precipitation time series is more detailed than the time step length. However, the criteria can lead to very short time steps during short term high intensity events. For example, if your model is running with maximum time steps of say 6 hours, but your precipitation time series is one hour, a high intensity one hour event could lead to time steps of a few minutes during that one hour event.

Max infiltration amount per time step If the total amount of infiltration due to ponded water [mm] in the current time step exceeds this amount, the time step will be reduced by the increment rate. Then the infiltration will be recalculated. If the infiltration criteria is still not met, the infiltration will be recalculated with progressively smaller time steps until the infiltration criteria is satisfied.

If your model does not include the unsaturated zone, or if you are using the 2-Layer water balance method, then you can set these conditions up by a factor of 10 or more. However, if you are using the Richards equation method, then you may have to reduce these factors to achieve a stable solution.

Input precipitation rate requiring its own time step If the precipitation rate [mm/hr] in the precipitation time series is greater than this amount, then the simulation will break at the precipitation time series measurement times. This option is added so that measured short term rainfall events are captured in the model.

For example, assume you have hourly rainfall data and 6-hour time steps. If an intense rainfall event lasting for only one hour was observed 3 hours after the start of the time step, then MIKE SHE would automatically break its time stepping into hourly time steps during this event. Thus, instead of a 6-hour time step, your time steps during this period would be: 3 hours, 1 hour, and 2 hours. This can also have an impact on your time stepping, if you have intense rainfall and your precipitation measurements do not coincide with your storing time steps. In this case, you may see occasional small time steps when MIKE SHE catches up with the storing time step.

Actual time step for the different components

As outlined above the overland time step is always less than or equal to the UZ time step and the UZ time step is always less than or equal to the SZ time step. However, the exchanges are only made at a common time step boundary. This means that if one of the time steps is changed, then all of the time steps must change accordingly. To ensure that the time steps always meet, the initial ratios in the maximum time steps specified in this dialogue are maintained.

After a reduction in time step, the subsequent time step will be increased by

$$timestep = timestep \times (1 + IncrementRate) \qquad (6.1)$$

until the maximum allowed time step is reached.

Relationship to Storing Time Steps

The Storing Time Step specified in the Detailed time series output *(V.2 p. 186)* dialogue, must also match up with maximum time steps. Thus,

- The OL storing time step must be an integer multiple of the Max UZ time step,
- The UZ storing time step must be an integer multiple of the Max UZ time step,
- The SZ storing time step must be an integer multiple of the Max SZ time step,
- The SZ Flow storing time step must be an integer multiple of the Max SZ time step, and
- The Hot start storing time step must be an integer multiple of the maximum of all the storing time steps (usually the SZ Flow storing time step)

For example, if the Maximum allowed SZ time step is 24 hrs, then the SZ Storing Time Step can only be a multiple of 24 hours (i.e. 24, 48, 72 hours, etc.)

6.5 Using Batch Files

A 'batch' file contains native DOS commands in a programming structure. When executed each of the DOS commands in the batch file is executed sequentially. Since, most MIKE Zero and MIKE SHE programs can be executed in this way, a properly constructed batch file allows you to run multiple models sequentially when you are not at the computer, such as over night.

Basically, to run MIKE SHE in batch mode, you must

1 Setup the different models with different names using the Setup Editor
2 Create a .BAT file containing the DOS commands to run the models
3 Run the .BAT file and analyse the results using the standard MIKE Zero analysis tools (e.g. the Results Viewer)

Setup the different models

Your original model can be saved to a new name and the necessary changes made in the new set up. We highly recommended that you create and set up the different models in the MIKE SHE Setup Editor. In principle, you could edit the .SHE file, which is a text file containing all of the information on the model set up, but the file is typically very large and confusing, and the format of this file must be preserved exactly.

Create the batch file

To create a batch file, you must create a text file with the extension .BAT. Then add the DOS commands in the order that you would like them executed. But, before you can run the MIKE SHE executables, you must add the MIKE SHE installation directory to your PATH variable. The default installation directory depends on your operating system. For example, for MS Vista (64-bit) the default directory is:

```
C:\Program Files(x86)\DHI\2011\bin\
```

The DOS command to add the default path to the PATH variable is:

```
Set PATH=%PATH%,C:\Program Files(x86)\DHI\2011\bin\
```

To run MIKE SHE from the batch file you must add the following two DOS command lines after the PATH statement above:

```
MSHE_PreProcessor MyModel.she
MSHE_watermovement MyModel.she
```

The above two lines will run both the preprocessor and the water movement engine separately. If you want to run them together, then you can replace the two lines with

```
MSHE_Simulation MyModel.she
```

The examples above will run silently. That is, no progress information will be displayed. If you want to display progress information, then you should use the MzLaunch utility. Using

```
MzLaunch.exe MyModel.she -e MSHE_Simulation
```

will leave the MzLaunch utility open when the simulation finishes, whereas

```
MzLaunch.exe MyModel.she -e MSHE_Simulation -exit
```

will close the MzLaunch utility when the simulation finishes.

Analyse the Results

The MSHE_watermovement.exe program automatically generates all of the output asked for in the Setup Editor. Thus, to look at your output, you only need to open the model at look at your results in the normal way.

If you want to run the water balance program, which is described in the Using the Water Balance Tool chapter, you can add the following lines to you batch file:

```
MSHE_Wbl_Ex.exe //apv My_WB_areas.WBL

MSHE_Wbl_Post.exe //apv My_WB_areas.WBL 1

MSHE_Wbl_Post.exe //apv My_WB_areas.WBL 2
```

In the above, the first command runs the Extraction phase of the water balance utility, while the subsequent commands run the Post-processing items in the water balance file. The number after the water balance file name indicates which Post-processing item to run. Post-processing steps cannot be executed before an Extraction step but only one Extraction step needs to be run for a each water balance utility file.

6.6 OpenMI

OpenMI stands for Open Modelling Interface. OpenMI is a standard, which facilitates the linking of simulation models and model components of environmental and socio-economic processes. It thus enables managers to more fully understand and predict the likely impacts of their policies and programmes.

The OpenMI Association is the organisation responsible for the development, maintenance, and promotion of OpenMI. DHI active in the OpenMI Association and was one of the original founding members. On the OpenMI Association web site at ***www.openmi.org***, you can learn which models are already OpenMI compliant, get help on OpenMI model migration, request new features, exchange opinions and provide feedback related to OpenMI implementations.

MIKE SHE is OpenMI compliant. That is, MIKE SHE can be linked to other OpenMI compliant programs. If you have specific questions on using MIKE SHE with OpenMI, please contact your local support centre.

Linking MIKE SHE with OpenMI

If you want to link MIKE SHE to another program using OpenMI, then you will need to initialize MIKE SHE to produce the required OpenMI linkages. This is done using the Extra Parameter option: Including OpenMI *(V.1 p. 324)*.

OpenMI limitations of MIKE SHE

The OpenMI GUI has been compiled for "any CPU". So, if you are using a 64-bit CPU, the OpenMI GUI will act like a 64-bit application - and expect the OpenMI components to also be 64-bit applications.

When using MIKE SHE on a 64-bit machine and adding a MIKE SHE model in the OpenMI GUI, an error will be generated. This is a limitation of the current version of OpenMI.

The workaround is to download the source code of the OpenMI editor, change the setting from "any CPU" to "x86", recompile and use the new .exe file instead.

6.7 Parallelization of MIKE SHE

For Release 2011, the MIKE SHE solvers have been parallelized as much as possible and have been updated for 64-bit operating systems. Also, significant improvements in the memory and calculation efficiency have been made. However, the scalability of the parallelization is dependent on the individual modules. Thus, every model will scale differently with respect to the running time.

The unsaturated module is highly scalable because each UZ column is completely independent. The saturated zone and overland flow modules, on-the-other-hand, are not nearly as scalable because of the connections between the cells. As an approximation, a typical model with a mix of modules will probably run between 1.8 and 2.5 times faster on a four-core computer.

The AUTOCAL program has also been updated to take advantage of multi-core computers. In this case, multiple simulations are sent automatically to each of the cores.

In all cases, the use of additional cores is restricted by the available licenses. The default number of run-time licenses is limited to four, which means that the parallelization and AUTOCAL will support up to four cores. If you want to use more than four cores, then you must contact your local DHI office for additional run-time licenses.

S U R F A C E   W A T E R

7 SURFACE WATER IN MIKE SHE

Hydrologically, surface water can occur in defined channels or distributed as ponded water in lakes or on the flood plain. Surface water interacts with the rest of the hydrologic cycle through evaporation and exchange to/from groundwater.

In MIKE SHE, ponded surface water can be simulated directly using the 2D overland flow module. The water flow on the ground surface is calculated by MIKE SHE's Overland Flow Module, using the diffusive wave approximation of the Saint Venant equations, or using a semi-distributed approach based on the Mannings equation. This chapter concentrates on the diffusive wave, finite-difference method.

Historically, MIKE SHE also included its own 1D channel flow module, but this was replace several years ago by MIKE 11.

This chapter describes the interaction and coupling between MIKE 11 and MIKE SHE, as well as guidance on the modelling of both channel flow and flooding in MIKE SHE.

For detailed technical information on 2D surface water flow, see the Overland Flow - Reference *(V.2 p. 265)* chapter.

For detailed technical information on MIKE 11, refer to either the .pdf versions of the MIKE 11 Reference Manuals that are installed with MIKE 11, or the MIKE 11 documentation in the on-line help. These can be easily accessed from your Windows Start menu under MIKE by DHI.

7.1 Overland Flow

When the net rainfall rate exceeds the infiltration capacity of the soil, water is ponded on the ground surface. This water is available as surface runoff, to be routed downhill towards the river system. The rate of overland flow is controlled by the surface roughness and the gradient between cells. The direction of flow is controlled by the gradient of the land surface - as defined by the topography. The quantity of water available for overland flow is the available ponded depth minus the detention storage, as well as the losses due to evaporation and infiltration along the flow path.

Overland flow can be a very time consuming part of the simulation. There are many ways to reduce this burden - often without significantly impacting the accuracy of the results.

7.1.1 *Parameters*

The main overland flow parameters are the surface roughness which controls the rate of flow, the depth of detention storage which controls the amount of water available for flow, plus the intial and boundary conditions.

Surface Roughness

The Stickler roughness coefficient is equivalent to the **Manning M**. The Manning M is the inverse of the commonly used **Mannings n**. The value of **n** is typically in the range of 0.01 (smooth channels) to 0.10 (thickly vegetated channels), which correspond to values of **M** between 100 and 10, respectively.

If you don't want to simulate overland flow in an area, a Mannings M of 0 will disable overland flow. However, this will also prevent overland flow from entering into the cell.

Detention Storage

Detention Storage is used to limit the amount of water that can flow over the ground surface. The depth of ponded water must exceed the detention storage before water will flow as sheet flow to the adjacent cell. For example, if the detention storage is set equal to 2mm, then the depth of water on the surface must exceed 2mm before it will be able to flow as overland flow. This is equivalent to the trapping of surface water in small ponds or depressions within a grid cell.

Water trapped in detention storage continues to be available for infiltration to the unsaturated zone and to evapotranspiration.

Detention storage also affects the exchange with MIKE 11. Only ponded water in excess of the detention storage will flow to MIKE 11. Also, flooding from MIKE 11 will only happen when the water level in the river link is above the topography plus detention storage.

The paved area drainage is also linked to the detention storage. Only the available ponded water will be routed to the SZ drainage network - that is the ponded depth above the detention storage. If you want to route all of the ponded water to the SZ drainage network, then you can use the Extra Parameter: Paved routing options *(V.1 p. 309)*.

Initial and Boundary Conditions

In most cases it is best to start your simulation with a dry surface and let the depressions fill up during a run in period. However, if you have significant wetlands or lakes this may not be feasible. However, be aware that stagnant ponded water in wetlands may be a significant source of numerical instabilities or long run times.

The outer boundary condition for overland flow is a specified head, based on the initial water depth in the outer cells of the model domain. Normally, the initial depth of water in a model is zero. During the simulation, the water depth on the boundary can increase and the flow will discharge across the boundary. However, if a non-zero initial condition is used on the boundary, then water will flow into the model as long as the internal water level is lower than the boundary water depth. The boundary will act as an infinite source of water.

Time varying OL boundary conditions

If you need to specify time varying overland flow boundary conditions, you can use the Extra Parameter option Time-varying Overland Flow Boundary Conditions *(V.1 p. 302)*.

7.1.2 Reduced OL leakage to UZ and to/from SZ

The Surface-Subsurface Leakage Coefficient *(V.2 p. 121)* reduces the infiltration rate at the ground surface. It works in both directions. That is, it reduces both the infiltration rate and the seepage outflow rate across the ground surface.

Conceptually, the leakage coefficient is used to account for soil compaction and fine sediment deposits on flood plains in areas that otherwise have similar soil profiles.

If the groundwater level is at the ground surface, then the exchange of water between the surface water and ground water is based on the specified leakage coefficient and the hydraulic head between surface water and ground water. In other words the UZ model is automatically replaced by a simple Darcy flow description when the profile becomes completely saturated.

If the groundwater level is below the ground surface, then the vertical infiltration is determined by the minimum of the moisture dependent hydraulic conductivity (from the soils database) and the leakage coefficient.

This option is often useful under lakes or on flood plains, which may be permanently or temporarily flooded, and where fine sediment may have accumulated creating a low permeable layer (lining) with considerable flow resistance.

The value of the leakage coefficient may be found by model calibration, but a rough estimate can be made based on the saturated hydraulic conductivities of the unsaturated zone or in the low permeable sediment layer, if such data is available.

The specified leakage coefficient is used wherever it is specified. In areas where a delete value is specified, the vertical hydraulic conductivity of the top SZ layer is used.

In the processed data, the item, Surface-subsurface Exchange Grid Code, is added, where areas with full contact are defined with a 0, and areas with reduced contact are defined with a 1.

7.1.3 Separated Flow areas

The Separated Flow Areas *(V.2 p. 123)* are typically used to prevent overland flow from flowing between cells that are separated by topographic features, such as dikes, that cannot be resolved within a the grid cell.

In many detailed models, surface drainage on flood plains and irrigation areas is highly controlled. The Seperated Flow Areas option allows you to define these drainage control land features in the model.

If you define the separated flow areas along the intersection of the inner and outer boundary areas, MIKE SHE will keep all overland flow inside of the model - making the boundary a no-flow boundary for overland flow.

However, seperated flow areas are not respected by the other hydrologic processes, such as the SZ drainage function. Thus, lateral flow out of the model may still occur via SZ drainage, SZ boundary conditions, MIKE 11, irrigation control areas, etc. even when the seperated flow areas are defined. Therefore, if you use seperated flow areas, you should carefully evaluate your results, for example, by using the water balance tool, to make sure that the water flow is behaving as you expect.

Also, you should note that Overland flow cannot cross a river link. So, the cell faces with river links always define a seperated flow boundary.

7.1.4 Paved Area Drainage

In MIKE SHE, there is a paved area function to account for the increased runoff in urban areas. However, the paved area function is rather complex and currently two limitations.

The paved area function in MIKE SHE relies on the saturated zone drainage reference system to move drainage to the river links. Thus, an important limitation is that paved area drainage can only be simulated if you are using the finite difference SZ model.

The second limitation is that the SZ drainage reference is used for routing paved drainage to river links, specified MIKE 11 h-points and MOUSE manholes. Paved drainage is not routed to SZ boundaries or internal SZ depressions.

The paved area drainage is calculated from **ponded water**. It is not calculated from rainfall directly. However, the order of operations in the three UZ solvers (Richards, Gravity, and 2Layer Water Balance) is such that rainfall is added to existing ponded storage before ET and infiltration are calculated, and, thus, before the paved drainage is calculated. Therefore, the paved drainage effectively acts on rainfall.

The amount of paved drainage is calculated based on the available ponded water. That is, a specified fraction of the amount of available ponded water is routed directly to the SZ drainage system. The SZ drainage system immediately discharges this water into the MIKE 11 river network. This is analagous to a full-pipe of water. That is, for any inflow an equal amount of outflow is generated instantaneously.

Rainfall is added to the ponded depth and then the drainage fraction is removed. However, if at the end of the time step, there is still ponded water in the cell, the paved fraction will be applied to the remainder again in the next time step. Thus, if your cell is half paved and half permanently ponded, the permanently ponded half will eventually drain away.

The paved area drainage is also linked to the detention storage. Only the available ponded water will be routed to the SZ drainage network - that is the ponded depth above the detention storage. If you want to route all of the ponded water to the SZ drainage network, then you can use an Extra Parameter found in the Paved routing options *(V.1 p. 309)*.

The SZ drainage parameters, drain level and drain time constant, do not impact the paved area drainage function. However, the paved area drainage does use the SZ drainage reference system, which means that the SZ drainage levels may play a role in defining the receiving river link.

The paved area drainage does not check to make sure that you do not create any physically impossible feedback loops. So, flood code cells, and overbank spilling from MIKE 11 should be directed to cells that where paved area drainage is active. If this happens, you may encounter excessive feedback between MIKE SHE overland flow and MIKE 11.

Finally, the paved area function has a time step dependency on the UZ time step length. If the UZ time step is less than the maximum UZ timestep length, then the paved area drainage will be reduced. To ensure that the paved area drainage is as expected, you should adjust your precipitation controls such that the UZ time step is not reduced.

Maximum discharge rate

If a cell is 100% paved, then the paved drainage function will remove all of the ponded water in the time time step. That is, by default the rate of drainage is not controlled. This does not reflect the conveyance restrictions of the drainage network. This is addressed by an option for specifying a maximum discharge rate from paved areas. The maximum discharge rate can be specified as a constant value or a distributed value.

The maximum discharge rate can be used to control the inflow to the SZ drainage system. At every time step, the available drainage volume will be checked against the maximum drainage rate. Thus, ponded water will be retained on a grid cell and drained at a controlled rate into the river system. While the water is ponded it will be subject to infiltration and ET. The rate of leakage below the cell can be controlled by the Surface-Subsurface Leakage Coefficient *(V.2 p. 121)*.

If the maximum discharge rate is set high, then the paved area fraction can be used to route a fraction of rainfall directly to the river network. This is reasonable when the travel time in the drainage network is similar to the time step length, and losses in the drainage network are minimal.

If the maximum discharge rate is set low, then the paved area fraction can be used to control inflow to and outflow from, for example, small scale surface impoundments.

The combination of maximum discharge rate and the OL leakage coefficient, along with the multi-cell OL, allows you to simulate distributed on grid surface water storages. You can use the combination to define, for example, distributed farm dams that release water to streams at a fixed rate. The volume of the storage is defined using the multi-grid OL. The ponded water is subject to evaporation, and you can use the OL leakage coefficient to control leakage to groundwater.

Note: When the Multi-cell Overland Flow *(V.1 p. 181)* option is used, a uniform value for the maximum discharge rate will be used within each coarse cell. Further, the depth of ponded water is calculated on the subscale, and used to calculate the paved area flow for each sub-scale cell

7.1.5 Overland Flow Velocities

MIKE SHE does not calculate overland flow velocities. The OL velocity can be calculated based on the water depth and the OL flow in the X- and Y- directions.

However, the calculation is not straight forward. The x- and y- flow is the flow across the cell boundary in the positive x and y directions. That is, the flow across the right and top cell boundaries. The average flow in a cell is, in fact, the mean of the inflow and the outflow in the x and y directions.

Further complicating the calculation is the fact that the x- and y- flow is saved for calculating the water balance - not for velocity calculation. Thus, the flow is the mean-step accumulated flow over the storing time step. If this were not true then it could not be used for the water balance. However, the ponded depth is an instantaneous value at the time it is saved. Thus, the depths and flows are not consistent in time, which has serious implications for the velocity calculation. For example, if your storing time step is a month, your flows will be a monthly average. The velocity will be saved at midnight on the last day of the month. Depending on the timing of your events, you could easily have a high average flow and a zero depth.

In principle, you could compensate for the above limations by using a very short time storing time step, and averaging the flows across the cell. However, then you have to ask yourself, why you want these flows.

MIKE SHE calculates overland flow based on the diffusive wave approximation, which neglects the momentum. Further, the depth and flow rates are averages for a cell, which does not take into account the actual distribution of velocities and water depths in a natural topography. Finally, if the area of interest is next to a river, then the physical exchange with the river depends on the calculation method used. Even in the best case, exchange between the river and the flood plain is conceptual. There is no velocity calculated for the river-OL exchange. Water is simply taken from the river and put on the flood plain cell, or vice versa. The rate of exchange depends the water level difference and the weir coefficients used.

Thus, the calculated velocity is probably not very useful for things like damage assessment. If velocities are important, then MIKE FLOOD is a much better tool. MIKE SHE, on the other hand, is good at calculating overland water depths, general flow directions and the exchange of ponded water with the subsurface and rivers.

7.2 Overland Flow Performance

The overland flow can be a significant source of numerical instabilities in MIKE SHE. Depending on the setup, the overland flow time step can become very short - leading to very long.

Overland flow has both an Implicit and Explicit solver. Your choice of solver affects both the accuracy of your results and the simulation run time.

The Implicit solver is faster than the Explicit solver because it can run with longer time steps. However, the it must iterate to converge on a solution. Thus, if each time step takes several iterations because of the dynamics of the overland flow, then the implicit solver can become slow. The most obvious sign of poor convergence is the presence of warning messages in the *projectname_WM_Print.log* file about the overland flow solver not converging. You may be able to live with a few warning messages, but the if the Implicit solver frequently fails to converge then this will significantly slow down your simulation. If this happens, then you have a few options.

The first option is to reduce your OL time step. This make increase the stability of the solver and actually reduce your run times. You can also increase the convergence criteria. This will decrease the accuracy, but if there is a troublesome area outside of your area of interest, then this may be acceptable.

If you switch to the Explicit solver, then the time step becomes dynamic depending on the Courant Criteria. This will likely reduce your numerical instabilities because the courant criteria is very restrictive, but the simulation is likely to be slower. However, the difference may not be that great if you are having a lot of convergence problems.

7.2.1 Stagnant or slow moving flow

The solution of overland flow is sensitive to the surface water gradient. If the surface water gradient is very small (or zero), then a numerically stable solution will generally require a very short time step.

Slow moving flow is a problem when you have long term ponded water, for example in wetlands. If you are only interested in the water levels in the wetland areas, but not the flow velocity and flow directions, then solving the overland flow equations is not necessary for decision making.

If you want to turn off the overland flow solver in slow moving or stagnant areas, then you can convert these areas to flood codes and allow

MIKE 11 to control the water levels. Lateral overland runoff to these areas will still be calculated, as will evapotranspiration and infiltration. For more detail on using Flood Codes, see Overland Flow Exchange with MIKE 11 *(V.1 p. 212)*.

An alternative is to turn off the overland flow calculation in these cells. You can turn off the overland flow in a cell by setting the Mannings M number to zero. However, this also turns off the lateral overland inflow into the cell, as well.

Another option is to use the detention storage parameter to restrict the amount of available water. In this case, overland flow is allowed into and out of the cell, but overland flow is not actually calculated until the depth of water in the cell exceeds the detention storage.

The Threshold gradient for overland flow (see next Section) is also a way to reduce the influence of stagnant water on the time step. However, you cannot specify a spatially varying threshold. So, the appropriate value may be difficult to select if you want to restrict flow in one area, yet keep surface flow in other less stagnant areas.

If you are using the Explicit OL solver, there are several dfs2 output options that make it easier to find the model areas that are contributing to reducing the time step. These include:

- Mean OL Wave Courant number,
- Max OL Wave Courant number, and
- Max Outflow OL-OL per Cell Volume.

7.2.2 Threshold gradient for overland flow

In flat areas with ponded water, the head gradient between grid cells will be zero or nearly zero. As the head gradient goes to zero, Δt must also become very small to maintain accuracy. To allow the simulation to run with longer time steps and dampen any numerical instabilities in areas with low lateral gradients, the calculated intercell flows are multiplied by a damping factor when the gradients are close to zero.

Essentially, the damping factor reduces the flow between cells. You can think of the damping function as an increased resistance to flow as the gradient goes to zero. In other words, the flow goes to zero faster than the time steps goes to zero. This makes the solution more stable and allows for larger time steps. However, the resulting gradients will be artificially high in the affected cells and the solution will begin to diverge from the Mannings solution. At very low gradients this is normally insignificant, but as the gradient increases the differences may become noticeable. Therefore, the damping function is only applied when the gradient between cells is below a user-defined threshold.

The details of the available functions can be found in the section Low gradient damping function *(V.2 p. 272)* found in the Reference manual.

For both functions and both the explicit and implicit solution methods, each calculated intercell flow in the current timestep is multiplied by the local damping factor, F_D, to obtain the actual intercell flow. In the explicit method, the flow used to calculate the courant criteria are also corrected by F_D.

The damping function is controlled by the user-specified threshold gradient (see Common stability parameters *(V.2 p. 38)* for the Overland Flow), below which the damping function becomes active.

The choice of appropriate threshold value depends on the slope of the flow surface. Based on both actual model tests in Florida and synthetic setups, the following conclusions can be reached:

- A Threshold gradient greater than the surface slope can lead to excessive OL storage on the surface that takes a long time to drain away.
- A Threshold gradient equal to the surface slope is often reasonable, but there may still be some excess storage on the surface.
- Threshold values less than the surface slope typically cause rapid drainage and give nearly the same answers.
- Threshold values below 1e-7 do not significantly improve the results even if the topography is perfectly flat.
- In general, you should used the highest value possible. Lower values may increase accuracy but at the expense of run time.

Therefore, we can safely recommend a Threshold gradient in the range of 1e-4 to 1e-5, with a default value of 1e-4. For many floodplains, 1e-4 or 1e-5 should be sufficient. In flood plains with very flat relief, 1e-6 may be used. Lower values are probably never necessary.

Since most discharge happens during and immediately after an event, the Threshold gradient is likely to be most important when there is significant ponding that lasts over several time steps and drains to a boundary or MIKE 11. Ponded water that infiltrates or evaporates and experiences limited lateral flow will not be affected by the Threshold value.

If the topography slope requires a low Threshold, but the solution is unstable at low threshold values, solution stability may be improved with the Explicit solver by reducing the Maximum Courant number until the solution becomes stable. With the Implicit solver, you may need to change the solver parameters.

Performance Impact

A low Threshold gradient will increase your simulation time. So, the final value that you use, may be a compromise between simulation length and accuracy of the flow in low gradient conditions.

If you have stagnant ponded water in your model, then the intercell gradient in these areas will be nearly zero. If you lower your Threshold gradient, your simulation performance may be adversely impacted, simply because the OL solver will begin to calculate flow sloshing back and forth in these areas. Not only will the OL solver have to work harder, the OL time step will likely also decrease because of the very low gradients. Thus, the Threshold gradient effectively reduces intercell flow in stagnant areas to zero allowing the Courant criteria to be satisfied at much higher time step lengths. See Figure 11.4 in Threshold gradient for overland flow *(V.1 p. 179)* in the Reference manual.

7.3 Multi-cell Overland Flow

The main idea behind the 2D, multi-grid solver is to make the choice of calculation grid independent of the topographical data resolution. The approach uses two grids:

- One describing the rectangular calculation grid, and
- The other representing the fine bathymetry.

The standard methods used for 2D grid based solvers do not make a distinction between the two. Thus, only one grid is applied and this is typically chosen based on a manageable calculation grid. The available topography is interpolated to the calculation grid, which typically does not do justice to the resolution of the available data. The 2D multi-grid solver in MIKE SHE can, in effect, use the two grids more or less independently.

In the Multi-cell overland flow method, high resolution topography data is used to modify the flow area used in the St Venant equation and the courant criteria. The method utilizes two grids - a fine-scale topography grid and a coarser scale overland flow calculation grid. However, both grids are calculated from the same reference data - that is the detailed topography digital elevation model.

In the Multi-cell method, the principle assumption is that the volume of water in the fine grid and the coarse grid is the same. Thus, given a volume of water, a depth and flooded area can be calculated for both the fine grid and the coarse grid. See Figure 7.1.

In the case of detention storage, the volume of detention storage is calculated based on the user specified depth and OL cell area.

During the simulation, the cross-sectional area available for flow between the grid cells is an average of the available flow area in each direction across the cell. This adjusted cross-sectional area is factored into the diffusive wave approximation used in the 2D OL solver. For numerical details see Multi-cell Overland Flow Method *(V.2 p. 275)* in the Reference manual.

The multi-grid overland flow solver is typically used where an accurate bathymetric description is more important than the detailed flow patterns. This is typically the case for most inland flood studies. In other words, the distribution of flooding and the area of flooding in an area is more important than the rate and direction of ingress.

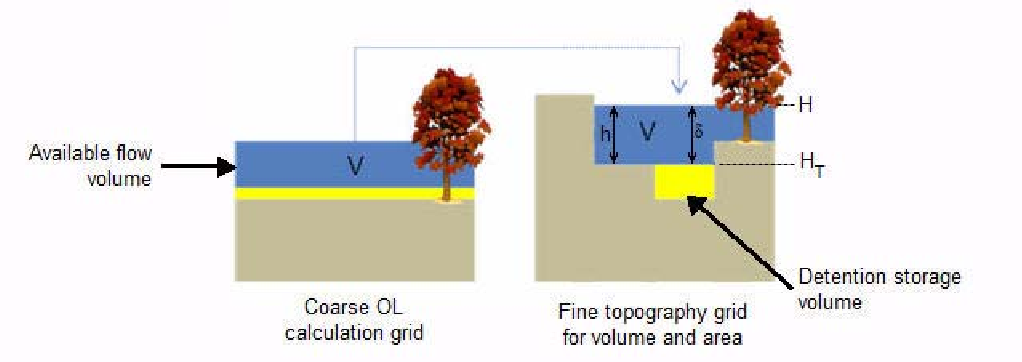


Figure 7.1 The constant volume from the coarse grid is transfered to the fine scale grid.

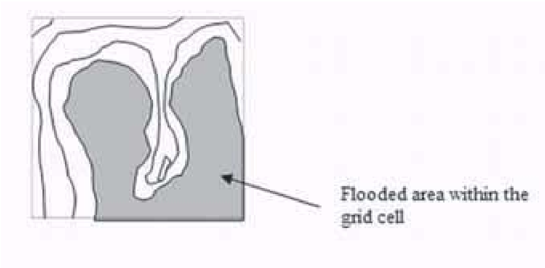


Figure 7.2 Flooded area is a function of the surface water level in the grid cell.

Elevations

The elevation of the coarse grid nodes and the fine grid nodes are calculated based on the input data and the selected interpolation method. However, the coarse grid elevation is adjusted such that it equals the average of the fine grid nodal elevations. This provides consistency between the coarse grid and fine grid elevations and storage volumes. Therefore, there may be slight differences between the cell topography elevations if the multi-cell method is turned on or off. This could affect your model inputs and results that depend on the topography. For example, if you initial water table is defined as a depth to the water table from the topography.

7.3.1 Evaporation

Evaporation is adjusted for the area of ponded water in the coarse grid cell. That is, evaporation from ponded water is reduced by a ponded area fraction, calculated by dividing the area of ponding in the fine grid cell by the total cell area.

The ET from soil evaporation is also reduced to the areas where there is not ponding.

Total transpiration does not need to be adjusted for the non-ponded area because evaporation from the ponded area is calculated prior to transpiration, and this is already adjusted for the ponded area in the cell. Thus, if the cell is fully ponded, then the Reference ET will be satisfied from ponded storage and there will be no transpiration. If the cell is only partially ponded, then the area fraction of the RefET will be first extracted from the ponded water and the remainder from the root zone. Since, there is only one UZ column to extract from, the entire root zone will be available for transpiration.

The Extra Parameter option,Transpiration during ponding *(V.1 p. 309)*, allows transpiration from the root zone beneath ponded areas. In this case, transpiration is calculated before evaporation from ponding. This option includes a reduction factor to account for the reduced ET under saturated conditions. The application of this factor will be changed so that it only applies to the ponded fraction of the cell.

7.3.2 Infiltration to SZ and UZ with the Multi-Grid OL

If ponded water is flowing between cells, the multi-scale topography will ensure that only the lowest part of each cell will be flooded, and the rate of flow between the cells will be adjusted for the flooded depth. However, the infiltration also needs to be adjusted to account for the fact that there is a driving pressure head in some parts of the cell.

Infiltration to UZ

The UZ infiltration is calculated based on three UZ calculations: one for the ponded fraction of the cell, one for the non-ponded fraction of the cell and finally a calculation is made using the area weighted infiltration of the two first UZ calculations. The last step is needed as there is only one UZ column below the multi-cells.

In a MIKE SHE simulation without multi-cell infiltration, the engine calculates an average storage depth, which is available for infiltration. This storage depth is then used for the infiltration calculations. The storage depth is calculated as follows:

1 Assuming that the OL depth from the previous OL time step is known,

2 the OL depth is updated using the current net precipitation and any sink and source terms (irrigation, by pass flow, paved area drainage etc.). Noting that:

- Bypass flow is extracted from the net precipitation before the infiltration calculation, and
- Paved area drainage is also extracted before the infiltration calculation. If you want to calculate the infiltration before paved area drainage this is available as an Extra parameter:

Parameter Name	Type	Value
infiltration before paved routing	Boolean	On

- The updated OL depth is used for the infiltration calculation;

- If the reduced contact option is used, the leakage coefficient is used to calculate the maximum infiltration rate.

When using the multi-cell infiltration, the infiltration is calculated based on three cases which depend on the ponded area fraction from the latest OL time step:

1 **Non ponded** (Ponded Area Fraction = 0). Only one infiltration calculation based on the available storage depth. This is done in the same way as a situation without the multi-cell option.

2 **Fully ponded** (Ponded Area Fraction = 1). Only one infiltration calculation based on the available storage depth. This is done in the same way as a situation without the multi-cell option.

3 **Partly ponded** (0 < Ponded Area Fraction < 1). Three infiltration calculations are made; ponded area, non-ponded area and a final calculation using the area weighted storage depth.

In the partly ponded case, it is assumed that the net-precipitation is equally distributed across the whole cell, while ponding from the previous OL time step only occurs in the ponded part of the cell. For the infiltration calculation in the non-ponded are, the available water depth is calculated as

$$DepPrec = precipitation \times dt \tag{7.1}$$

The remaining part of the available water (ponding + precipitation on the ponded part) is scaled to an equivalent water depth in the ponded area:

$$DPonded = (OLDepth + DepPrec) \times (1-PAreaFrac) / PAreaFrac \tag{7.2}$$

where OLDepth is the depth of ponded water from the previous time step, and PAreaFrac is the ponded area from the previous time step.

Disabling multi-cell infiltration

Multi-cell infiltration is automatically activated when the multi-cell option is invoked. However, an Extra Parameter option is available if you want to disable this function - perhaps for backwards compatability with older models.

Parameter Name	Type	Value
disable multi-cell infiltration	Boolean	On

When this is specified, the infiltration will be calculated based on the values of the course grid, and any ponding occurring in any sub-grid cells will not be included.

7.3.3 Reduced Leakage with Multi-cell OL

If reduced contact is specified, Surface-Subsurface Leakage Coefficient *(V.2 p. 121)*, the OL leakage coefficient is used, meaning that:

- Reduced contact only in ponded areas (activated). Leakage coefficient is only used in the ponded areas and not in the non-ponded areas.
- Reduced contact only in ponded areas (not activated). Leakage coefficient is used in both the ponded and the non-ponded case.

The two infiltration calculations (for the ponded and the non-ponded case) result in two infiltration rates from which an area weighted infiltration rate, *QinfAWghtd*, is calculated:

$$QinfAWghtd = Qinfntpd \; x \; (1 - PAreaFrac) + Qinfpd \; x \; (PAreaFrac) \quad (7.3)$$

where *Qinfntpd* is the infiltration rate from the non-ponded area, and *Qinfpd* is the infiltration rate for the ponded area. The area weighted infiltration rate is then used in the final UZ calculation (when reduced contact is not used).

Reduced leakage in ponded areas only

In many cases, the ponded areas will have a lower infiltration rate than the surrounding dry areas. The land surface in the dry areas will tend to be broken up macropores etc. Whereas, surface sealing will occur beneath ponded areas. To yield a more realistic flow surface drainage for flooded areas, an option for reduced contact (Ol leakage coefficient) in only the ponded part of the cell is available.

Note This is only used in the UZ infiltration and NOT in the exchange between SZ and OL.

Activating this option will allow you to include a distributed dfs2 integer grid code file. The reduced leakage will be applied in all areas with a positive integer value. In all other areas (with a negative, zero or delete value), the reduced leakage condition will be applied to the whole cell with the following constraints:

- The option will be applied to the ponded area from the previous time step. This will ensure that rainfall infiltrates normally in the non-ponded areas and currently ponded water will be retained.

- After the rainfall in non-ponded areas is infiltrated, then intercell lateral flow will be calculated and a new ponded area determined.

This method ensures that ponded water is able to flow laterally between cells with limited losses. By adjusting the leakage rate, you can decrease the losses along the OL flow path. This will essentially lead to a sub-grid scale drainage network that will ensure that runoff will eventually reach the river.

However, this option only applies to cells that are ponded, and thereby ensuring that ponded water remains on the surface. During high intensity rainfall in the current time step, this option will not encourage the creation of flooded areas, as the reduced leakage coefficient will first be applied in the following time step if ponded water is present at the end of the time step.

On flood plains, where the ponding occurs from overbank spilling from rivers or streams, the option will likely result in a more realistic description of the flow paths on the flood plain, as it prevents the flooded water from infiltrating.

7.3.4 Multi-cell Overland Flow + Saturated Zone drainage

The topography is often used to define the SZ drainage network. Thus, a refined topography more accurately reflects the SZ drainage network.

The SZ drainage function uses a drain level and drain time constant. The drain level defines the depth at which the water starts to drain. Typically, this is set to some value below the topography to represent the depth of surface drainage features below the average topography. This depth should probably be much smaller if the topography is more finely defined in the sub-grid model. The drain time constant reflects the density of the drainage network. If there are a lot of drainage features in a cell then the time constant is higher and vice versa.

Details related to the use of the Multi-cell OL with the SZ Drainage function are found along with the rest of the user guidance on SZ Drainage in the section: Saturated Zone drainage + Multi-cell Overland Flow *(V.1 p. 259)*.

7.3.5 Test example for impact on simulation time

The increased accuracy of the multi-cell overland flow method does not come for free. There is a performance penalty when you turn on the multi-cell option. However, the penalty relative to the increased accuracy of water depths is small.

A test done on a large, complex model in Florida, USA illustrates the performance penalty of the multi-cell method.

In the model the grid cell size is 457.2 m (1500 ft). However, a high resolution (5-ft) DEM is available for the whole model domain based on LIDAR data. This makes it attractive to use of higher resolution map with the Multi-cell option to account more accurately for the OL flow between 1500-ft grid cells.

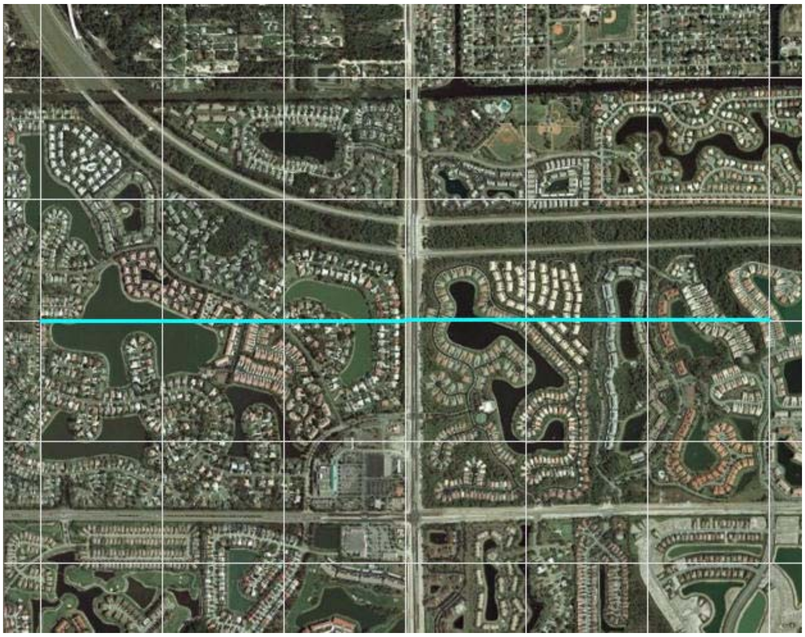


Figure 7.3 Aerial photo of part of the model area.

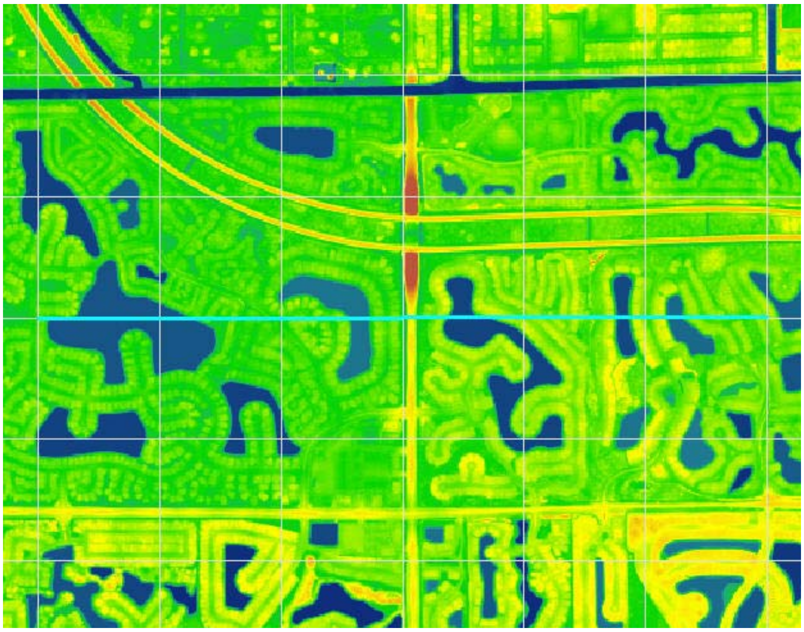


Figure 7.4 5-foot LIDAR data for part of the model area

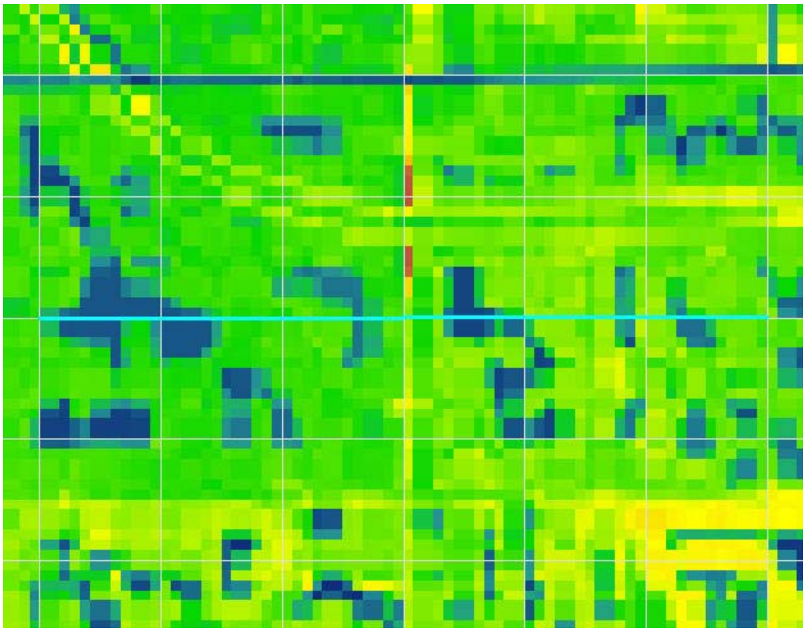


Figure 7.5 Interpolation of the 5-foot LIDAR data to the 125ft model grid

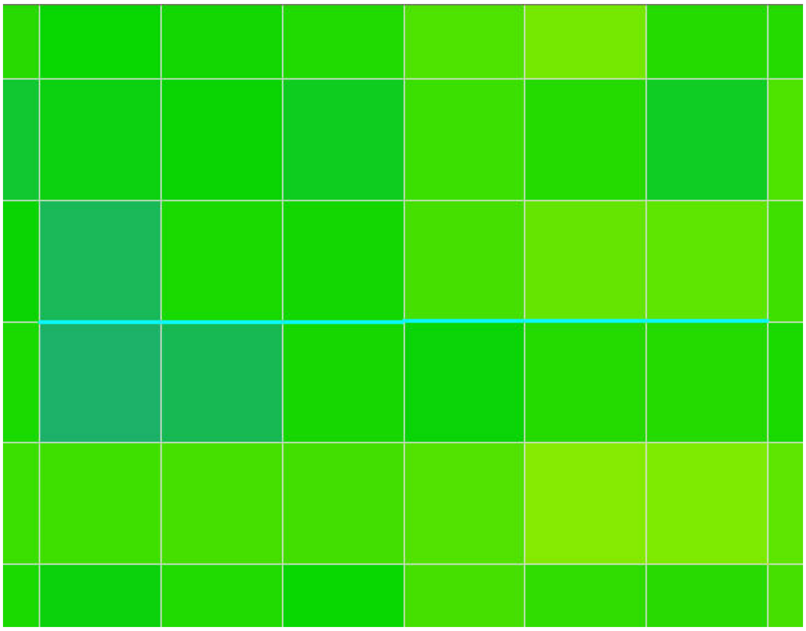


Figure 7.6 Interpolation of the 5-foot LIDAR data to the 1500ft model grid

Impact on simulation time

In this test, we tested multi-cell factors of 1, 2, 3, 4, 6, and 12. The smallest grid size was 125-ft, which is 12 times smaller than the coarse 1500-ft grid.

The following graphs illustrate the impact of the multi-grid option on the running times for the test model. Figure 7.7 shows that the OL run time increases linearly with higher multi-cell factors. In the test model, a multi-cell factor of 12 caused the OL portion of the simulation time took 30 times longer. Figure 7.8 shows that the multi-cell factor also impacts the run time for MIKE 11. However, this impact is not linear, with the impact on MIKE 11 leveling off after a multi-cell factor of four.

The test model run time is dominated by MIKE 11. In this case, the original run time for the OL is not very long and the multi-cell factor increases the OL run time considerably. However, as a fraction of the total run time, the OL is still small. When the OL cells are subdivided, there is probably some significant changes in the lateral inflow to MIKE 11. However, as the multi-cell factor increases, the increased resolution of the inflows is not signficant above a factor of about four.

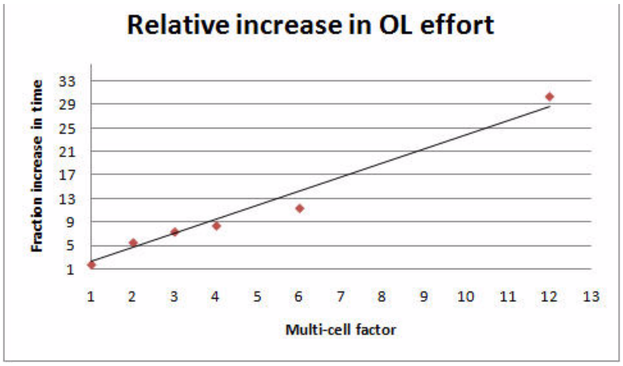


Figure 7.7 Increase in OL run time as a function of multi-cell factor

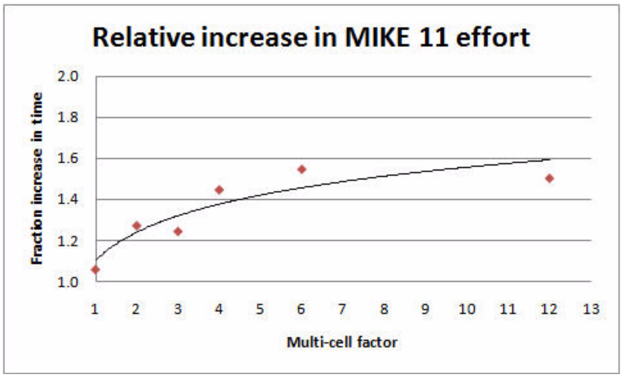


Figure 7.8 Increase in MIKE 11 run time as a function of multi-cell factor.

Impact on model results

The model contains a mix of natural, urban, and agricultural areas. The model also includes a complex river network with the relevant man-made canals and structures, and most of the natural flow ways. However, there is a natural flow way in the southeast part of the model that is not conceptualized in MIKE 11. Since, the surface water flow in this area is relying only on overland flow, the multi-cell option should significant changes in the OL flow prediction in this natural flow way.

In urban and agricultural areas, the drainage and the OL flow components in MIKE SHE route the water into the canal network. The drainage component would keep the water table level below the ground most of the time in those areas. However, it is of interest to test the OL flows predicted with the multi-cell option in those areas during storms events.

7.3.6 *Limitations of the Multi-cell Overland Flow Method*

In principle, all of the exchange terms in MIKE SHE could be adjusted to reflect the fine scale water levels and flooded areas. However, some of these are easier to implement than others and of greater importance. Thus, in current release, the exchange with MIKE 11, as well as UZ and SZ, depend only on the coarse scale grid elevations.

Overland flow exchange with MIKE 11

Overland flow exchange with MIKE 11 does not consider the multi-cell method. That is, flow into and out of the River Links is controlled by the water level calculated from the elevation defined in the coarse grid cell. Likewise the flow area for exchange with MIKE 11 is calculated as the coarse water depth times the overall grid size. Also, the elevation used when calculating flood inundation with flood codes only considers the average cell depth of the coarse grid. See Overland Flow Exchange with MIKE 11 *(V.1 p. 212)*.

However, if you choose to modify the topography based on a bathymetry file, or the MIKE 11cross-sections, then this information will be used when calculating the multi-cell elevations. See Inundation options by Flood Code *(V.1 p. 222)*.

7.3.7 *Setting up and evaluating the multi-grid OL*

The multi-cell overland flow method is activated in the OL Computational Control Parameters *(V.2 p. 36)* dialogue. In this dialogue, you can check on the option and then specify a sub-division factor. The coarse grid will be divided into this number of cells in both directions. That is, for a factor of two, the coarse grid will be divided into four cells. Likewise a factor of five will lead to 25 fine cells per coarse grid cell.

In addition

Pre-processed data

When you enable the Multi-grid OL option the following new items will be available in the preprocessed data:

- Multi-Cell ground levels (subscale topography) In Figure 7.9, two interpolations of the same DEM are shown. The top figure is a plan view of the interpolation to a 100m grid resolution and a 25m grid resolution respectively. The bottom figure is a cross-section across the middle of the top figure, where you can clearly see the more accurate resolution of the drainage features.

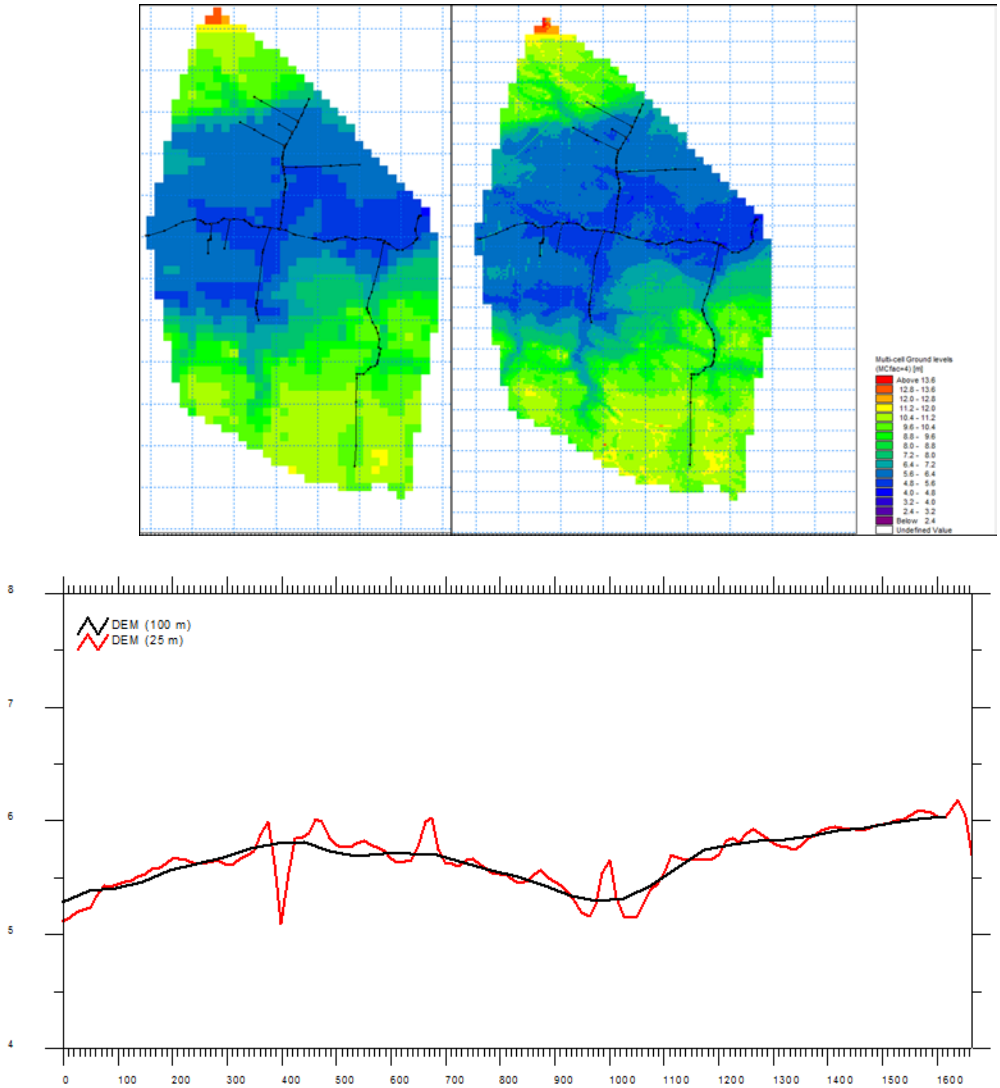


Figure 7.9 Example of preprocessed data - topography using a 100 meter resolution and a sub-scale factor of 4 (25 m sub-scale resolution).

When SZ Drainage is also active, then the following items are also available:

- Max. MC Drain Level - displays the maximum drain level of the sub-cells within each of the model cells

- Min MC Drain level - displays the minimum drain level of the sub-cells within each of the model cells
- Min MC Drain Depth - displays the minimum drain depth of the sub-cells within each of the model cells
- Max MC Drain Depth - displays the maximum drain depth of the sub-cells within each of the model cells

Additional results options

When the Multi-grid OL option is active, the following additional items will be available in the result items:

- Depth of Multi-Cell overland water - displays the depth of overland water using the sub-scale resolution
- Multi-Cell overland water elevation - displays the overland water elevation using the sub-scale resolution

7.4 Channel Flow

7.4.1 MIKE 11 Overview

MIKE 11 is a comprehensive 1D channel flow model for simulating rivers and surface water bodies that can be approximated as 1-dimensional flow (as strict 1-Dimensional flow does not occur in nature). Basically, MIKE 11 can be applied anywhere average values of levels, velocities, concentrations etc. at a point are acceptable, including:

- River hydrodynamics
- Structure/reservoir operational control
- Water quality (e.g. wetlands, salinity)
- Sediment transport & morphology
- Flood studies (e.g. mapping, hazard assessment)
- Flood forecasting (on-line, real-time)
- Dam break
- Sediment transport (e.g. Long term morphology)
- River restoration
- Integrated with groundwater and flooding

MIKE 11 plays a critical role in MIKE SHE Both the overland flow and groundwater flow modules are linked directly to MIKE 11. The MIKE SHE-MIKE 11 coupling enables:

- the one-dimensional simulation of river flows and water levels using the fully dynamic Saint Venant equations.
- the simulation of a wide range of hydraulic control structures, such as weirs, gates and culverts.
- area-inundation modelling, using a simple flood-mapping procedure that is based on simulated river water levels and a digital terrain model.
- dynamic overland flooding flow to and from the MIKE 11 river network.
- the full, dynamic coupling of surface and sub-surface flow processes in MIKE 11 and MIKE SHE.

7.5 Building a MIKE 11 model

Integrating a MIKE SHE and a MIKE 11 model is not very different from establishing a stand-alone MIKE 11 HD model and a stand-alone MIKE SHE model. In principle, there are three basic set-up steps:

1. Build a stand-alone MIKE 11 HD hydraulic model and make a performance test and, if possible, a rough calibration using prescribed inflow and stage boundaries. If needed, you can specify a default groundwater table (e.g. MIKE SHE's initial groundwater level) and leakage coefficients for any leakage calculations.
2. Build a stand-alone MIKE SHE model that includes the overland flow component and (optionally) the saturated zone and unsaturated zone components. An SZ drainage boundary can be used to prevent excessive surface flows in low lying areas and the river flood plain.
3. Couple MIKE SHE and MIKE 11 by defining branches (reaches) where MIKE 11 HD should interact with MIKE SHE. Modify your MIKE SHE and MIKE 11 models so that they work together properly. For example, by removing the specified groundwater table in MIKE 11 and adjusting your SZ drainage elevations if you used these in Step 2.

In the above scheme, the first step in coupling MIKE 11 to MIKE SHE is to create a normal MIKE 11 HD model without coupling it with MIKE SHE. In this regard, a few things should be emphasised:

- In a normal MIKE 11 river model only the river chainage (dx) is important for the results. Geographic positioning of river branches and cross-sections are only important for the graphical presentation. When interfacing MIKE 11 to MIKE SHE geographic positioning is critical, as MIKE SHE needs information on the river location.
- A reasonably high number of river cross-sections should be included to ensure that the river elevations are reasonably consistent with the surface topographic features.

7.5.1 *MIKE 11 network limitations*

There are a few features of MIKE 11 that do not relate well to MIKE SHE.

Short branches

In MIKE 11 there is no restriction on how short your branches are. If you are trying to simulate discontuous lakes or structures on the flood plain, for example. you may have very short branches. However, MIKE SHE does not allow MIKE 11 branches to be shorter than the cell size. Generally, though, short branches are a sign that you should probably reconsider your model conceptualization - or switch to MIKE FLOOD, which allows flood plain structures.

Parallel branches

Like short branches, MIKE SHE does not like it when your branches are too close together. If you have parallel branches that are too close together, then the branches may be mapped to the same river link. However, each river link must be mapped to a unique branch. As a rule of thumb, parallel branches should be greater than a cell width apart. However, this is not uniformly true, since the two close parallel branches may map onto opposite sides of a cell, if they are located on either side of a cell mid-point. Thus, you may have unexpected problems, if you change the cell size in a model that was working and you have branches that are closer together than one cell size.

Long coupling links

MIKE SHE links to MIKE 11 branches. However, when two branches are connected, water is passed between the branches directly. The link has not physical length or storage itself. If your links are too long, there will be an error in the timing of the flows between the two branches. So, the links should be kept short. MIKE 11 does not have any restrictions on how long the links can be, but MIKE SHE will issue a warning if the links are longer than a cell size. The warning is simply to informing you that there is no possibility for groundwater-surface water exchange in the link.

Long distances between cross sections

MIKE 11 controls the distance between the calculation nodes. The properties at the calculation nodes are linearly interpolated from the available cross-sections. This includes geometric properties such as bank and bottom elevations, marker locations, etc. However, linear interpolation can easily result in inconsistences between elevations in MIKE SHE.and marker elevations in MIKE 11. If the bank elevation is higher than the topography, then overland flow into the river will be restricted. If the downstream river bottom elevation is higher than the side branch bottom elevation, then MIKE 11 will likely be unstable.

Long distances between calculation nodes

This is not the same as long distances between cross-sections. MIKE 11 manages the water at the q-points directly linked to the river links. MIKE SHE and the river link system automatically interpolates the nearest river link. However, if the calculation nodes are very far apart or very close together, then the linear interpolation of water volumes between the calculation points may lead to discrepancies in the available water volumes especially if the river links are being used for irrigation or the river is losing water. In this sense, the distance between the calculation nodes, should be similar to the MIKE SHE grid spacing.

7.5.2 MIKE 11 Cross-sections

Whenever there is a significant change in the bed slope there should, in principle, be a cross-section defined in MIKE 11. If only a few cross-sections are available, it may be sufficient to estimate the cross-section shape based on neighbouring cross-sections and estimate the bank/bed elevation based on the surface topographic information in MIKE SHE or other topographic maps.

Cross-sections vs. time step

However, every cross-section in MIKE 11 is a calculation node. The time step in MIKE 11 is sensitive to the Courant number, which is proportional to the distance between calculation nodes. So, if the cross-sections are close together, then you may experience very short time steps in MIKE 11.

Thus, if you are have very short MIKE 11 time steps, then you might want to check your river network to make sure you do not have cross-sections that are too close together. This frequently occurs when the cross-sections have been imported. If you do have cross-sections that are too close together, then you can easily eliminate one or more of them, as long as the conveyance of the different cross-sections is roughly the same. In other words, you can eliminate duplicate cross-sections if their Q/H relationships are roughly the same, even though the physical shape of the two cross-sections may appear quite different. This is often the case in braided stream networks, where the location of the main channels may move left or right, but the overall conveyance of the river bed is relatively constant.

Cross-sections versus MIKE SHE topography

In the absence of flooding, ponded water discharges to the MIKE 11 river as overland flow. As a general rule, the topography must be higher than or equal to the bank elevation. If the bank elevation is higher than the topography, water will not be able to flow into the river in that cell, but will run laterally along the river until it reaches a place for it to flow into the river. An easy trick to see where this is happening is to run a simulation with no infiltration, ET, or detention storage and set the initial water depth at 1m. Then look at the results to find places were the water is piling up against the river links.

In the pre-processor log file, a table is create that lists all the river links where the bank elevation is different than the topography.of the adjacent cell. The critical river links with bank elevations above the topography are highlighted with the ==> symbol. This list can be surprisingly long because the river link bank elevations are interpolated from the neighbouring cross-sections. Whereas the topography is already defined. So, frequently the interpolated bank elevations do not line up precisely with the topography.

If overland flow on the flood plain is essentially absent, for example, due to infiltration or evapotranspiration, then these differences are not relevant and there is no need to modify the topography. However, if the overland to river exchange is important then you may have to carefully modify your topography file or your bank elevations so that they are consistent.

Hint In many cases, your topography is from a DEM that is different from your model grid - either because it is a .shp or xyz file, or if it is a different resolution than your model grid. In this case, it may be easier to save the pre-processed topography to a dfs2 file (right click on the topography map in the pre-processed tab). Then modify and use the new dfs2 file as the topography in your model setup. The disadvantage of this, is that if you change your model domain or grid, then you will have to redo your topography modifications.

Hint You can also use one of the Flood code options to automatically modify your topography, if you have wide cross-sections or a detailed DEM of the floodplain. In this case, after you have set up your MIKE 11 model, you can specify a constant grid code for the whole model and let MIKE SHE calculate a modified topography based on the cross-sections

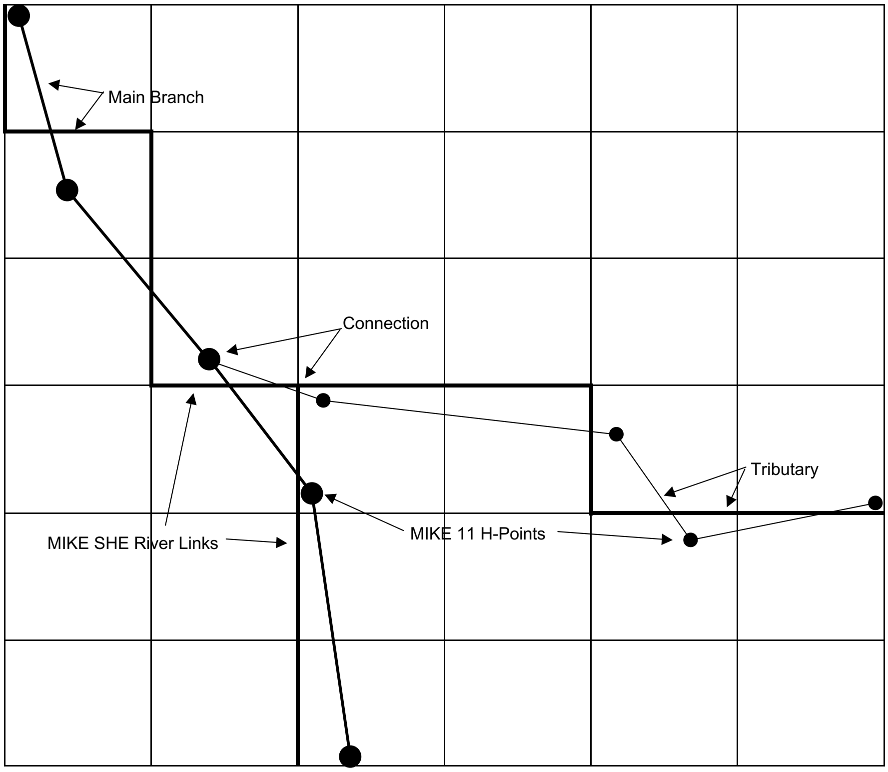


Figure 7.10 MIKE 11 Branches and H-points in a MIKE SHE Grid with River Links

or bathymetry. Then save the topography file as above and then use it as the model topography.

7.6 Coupling of MIKE SHE and MIKE 11

The coupling between MIKE 11 and MIKE SHE is made via **river links**, which are located on the edges that separate adjacent grid cells. The river link network is created by MIKE SHE's set-up program, based on a user-specified sub-set of the MIKE 11 river model, called the **coupling reaches**. The entire river system is always included in the hydraulic model, but MIKE SHE will only exchange water with the coupling reaches. Figure 7.10 shows part of a MIKE SHE model grid with the MIKE SHE river links, the corresponding MIKE 11 coupling reaches, and the MIKE 11 H-points (points where MIKE 11 calculates the water levels).

The location of each of MIKE SHE river link is determined from the co-ordinates of the MIKE 11 river points, where the river points include both digitised points and H-points on the specified coupling reaches. Since the MIKE SHE river links are located on the edges between grid cells, the details of the MIKE 11 river geometry can be only partly included in MIKE SHE, depending on the MIKE SHE grid size. The more refined the MIKE SHE grid, the more accurately the river network can be reproduced.

If flooding is not allowed, the MIKE 11 river levels at the H-points are interpolated to the MIKE SHE river links, where the exchange flows from overland flow and the saturated zone are calculated.

If flooding is allowed, via Flood Codes, then the water levels at the MIKE 11 H-points are interpolated to specified MIKE SHE grid cells to determine if ponded water exists on the cell surface. If ponded water exists, then the unsaturated or saturated exchange flows are calculated based on the ponded water level above the cell.

If flooding is allowed via overbank spilling, then the river water is allowed to spill onto the MIKE SHE model as overland flow.

In each case, the calculated exchange flows are fed back to MIKE 11 as lateral inflow or outflow.

Each MIKE SHE river link can only be associated with one coupling reach, which restricts the coupling reaches from being too close together. This can lead to problems when you have a detailed drainage or river network with branches less than one half a cell width apart. It will also lead to problems if your MIKE 11 branches are shorter than your MIKE SHE cell size.

If you have coupling reaches that are too short or too close together, you will receive an error message. If this happens, you can

- decide not to include one of the branches as a coupling reach (it is still included in the MIKE 11 HD model), or
- remove some of the branches (this error often occurs when you have a detailed looped drainage network), or
- refine your MIKE SHE grid until all coupling reaches are assigned to unique river links.

If you have a regional model with large cells (say 1-2km wide), then you cannot expect the river-aquifer interaction to be accurate at the individual cell level (e.g. all your cell properties – topography, conductivity, ManningsM, etc. – are all average values over 1-4 km^2). Rather, most often you will be interested in having a correct overall water balance along the stream. Typically, this is achieved by calibrating a uniform average river bed leakage coefficient against a measured outflow hydrograph. In such a model, you may also be tolerant of higher groundwater residuals.

On the other hand, if you need more detailed site specific results (and you have data and measurements to calibrate against), then you will use a local scale model, with a smaller grid (say 50-200m) and discrepancies between topography and river bank elevation will largely disappear. In this case, you will be more likely to be able to make accurate local scale predictions of groundwater-surface water exchange.

7.6.1 MIKE SHE Branches vs. MIKE 11 Branches

A **MIKE 11 branch** is a continuous river segment defined in MIKE 11. A MIKE 11 branch can be sub-divided into several coupling reaches.

A **MIKE SHE branch** is an unbroken series of coupling reaches of one MIKE 11 branch.

One reason for dividing a MIKE 11 branch into several coupling reaches could be to define different riverbed leakage coefficients for different sections of the river.

If there are gaps between the specified coupling reaches, the sub-division will result in more than one MIKE SHE branch. Gaps of this type are not important to the calculation of the exchange flows between the hydrologic components (e.g. overland to river, or SZ to river). The exchange flows depend on the water level in the MIKE 11 river, which is unaffected by gaps in the coupling reaches.

However, MIKE SHE can calculate how much of the water in the river is from the various hydrologic sources (e.g. fraction from overland flow and SZ exfiltration). However, this sort of calculation is only possible if the MIKE SHE branch is continuous. If there is a gap in a MIKE SHE branch, then the calculated contributions from the different hydrologic sources downstream of the gap will be incorrect. If there are gaps in the MIKE SHE branch network, then the correct contributions from the different sources must be determined from the MIKE 11 output directly.

Furthermore, the MIKE 11/MIKE SHE coupling for the water quality (AD) module will not work correctly if there are gaps in the MIKE SHE branch network.

There is one further limitation in MIKE SHE. That is, no coupling branch can be located entirely within one grid cell. This limitation is to prevent multiple coupling branches being located within a single grid cell.

Connections Between Tributaries and the Main Branch

Likewise, the connections between the tributaries and the main branch are only important for correctly calculating the downstream hydrologic contributions to the river flow and in the advection-dispersion (AD) simulations. The connections are not important to the calculation of the exchange flows between the hydrologic components (e.g. overland to river, or SZ to river).

In the example shown in Figure 7.10, the river links of the tributary are correctly connected to the main branch. This will happen automatically when

- the hydraulic connection is defined in the MIKE 11 network, AND
- the connection point (the chainage) on the main branch is included in a coupling reach, AND
- the connection point (the chainage) on the tributary is included in a coupling reach.

If the connection does not satisfy the above criteria, then there may be a gap in the MIKE SHE branch network and the limitations outlined above will apply.

7.6.2 The River-Link Cross-section

The MIKE 11(HD) hydraulic model uses the precise cross-sections, as defined in the MIKE 11 *.xns11* (cross-section) file, for calculating the river water levels and the river volumes. However, the exchange of water between MIKE 11 and MIKE SHE is calculated based the river-link cross-section.

The river-link uses a simplified, triangular cross-section interpolated (distance weighted) from the two nearest MIKE 11 cross-sections. The top width is equal to the distance between the cross-section's left and right bank markers. The elevation of the bottom of the triangle equals the lowest depth of the MIKE 11 cross-section (the elevation of Marker 2 in the cross-section). The left and right bank elevations in MIKE 11 (cross-section markers 1 and 3 in MIKE 11) are used to define the left and right bank elevations of the river link (See Figure 7.11).

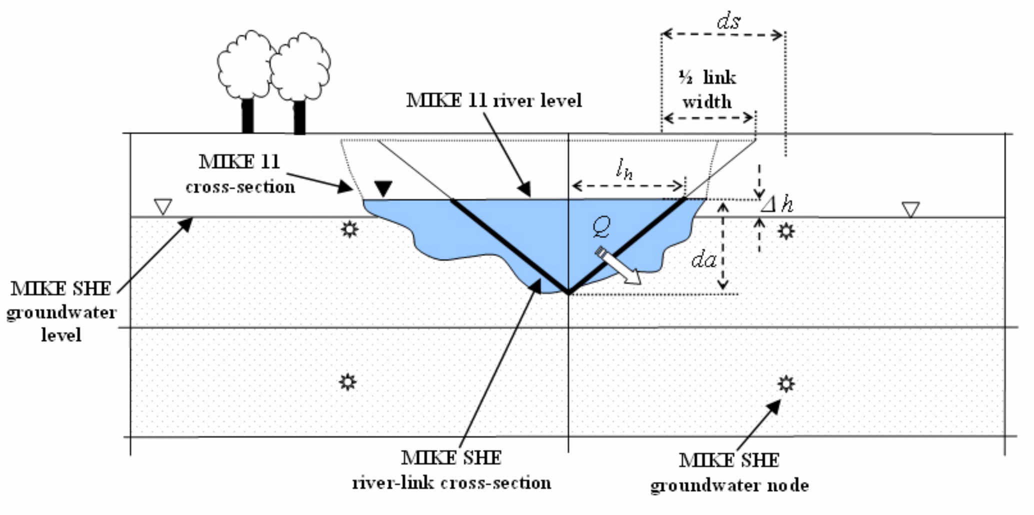


Figure 7.11 A typical simplified MIKE SHE river link cross-section compared to the equivalent MIKE 11 cross-section.

If the MIKE 11 cross-section is wider than the MIKE SHE cell size, then the river-link cross-section is reduced to the cell width.This is a very important limitation, as it embodies the assumption that the river is narrower than the MIKE SHE cell width. If your river is wider than a cell width, and you want to simulate water on the flood plain, then you will need to use either the Flooding from MIKE 11 to MIKE SHE using Flood Codes *(V.1 p. 214)* option or the Direct Overbank Spilling to and from MIKE 11 *(V.1 p. 216)* option.

If you don't want to simulate flooding, then the reduction of the river link width to the cell width will not likely cause a problem, as MIKE SHE assumes that the primary exchange between the river and the aquifer takes place through the river banks. For more detail on the river aquifer exchange see Groundwater Exchange with MIKE 11 *(V.1 p. 207)*.

For more detail on flooding and overland exchange with MIKE 11 see Overland Flow Exchange with MIKE 11 *(V.1 p. 212)*

7.6.3 Connecting MIKE 11 Water Levels and Flows to MIKE SHE

In MIKE 11, every node in the river network requires information on the river hydraulics, such as cross-section and roughness factors. These nodes are known as H-points, and MIKE 11 calculates the water level at every H-point (node) in the river network. Halfway between each H-point is a

Storing Q-point, where MIKE 11 calculates the flow, which must be constant between the H-points.

The water levels at the MIKE 11 H-points are transferred to the MIKE SHE river links using a 2-point interpolation scheme. That is, the water level in each river link is interpolated from the two nearest H-points (upstream and downstream), calculated from the centre of the link. The interpolation is proportionally distance-weighted.

The volume of water stored in a river link is based on a sharing of the water in the nearest H-points. In Figure 7.12, River Link A includes all the water volume from H.points 1 and 2, plus part of the volume associated with H-point 3. The volume in River Link B is only related to the volume in H-point 3. While the volume in River Link C includes water from H-points 3 and 4. This is done to ensure consistency between the river volumes in MIKE 11 and MIKE SHE, as the amount of water that can infiltrate or be transfered to overland flow is limited by the amount of water stored in the river link.

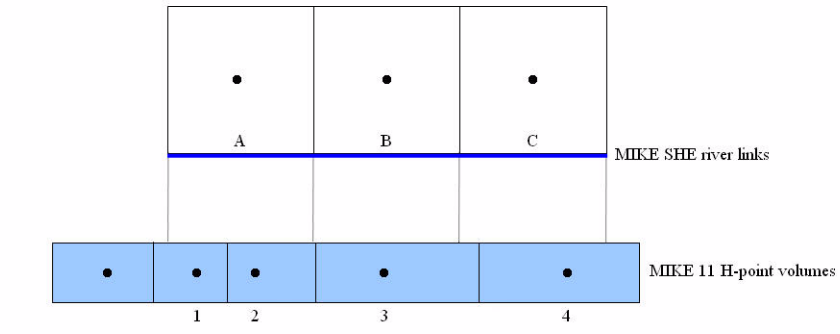


Figure 7.12 Sharing of MIKE 11 H.point volumes with MIKE SHE river links.

The water levels and flows at all MIKE 11 H-points located within the coupling reaches can be retrieved from the MIKE SHE result file.

However, since the MIKE 11 flows are not used by MIKE SHE, the river flows stored in the MIKE SHE result file are not the flows calculated at the MIKE 11 Storing Q-points. Rather, the flows stored in the MIKE SHE result file are the estimated flows at the MIKE 11 H-points. That is, the flows in the MIKE SHE result file have been linearly interpolated from the calculated flows at the Storing Q-point locations to the H-point locations on either side of the Storing Q-point. If the exact Q-point discharges are needed, they must be retrieved or plotted directly from the MIKE 11 result file.

7.6.4 Evaluating your river links

The river links are evaluated during the pre-processing. In the pre-processor log file (*yourprojectnamePP_print.log*), there is a table that contains all of the river link details:

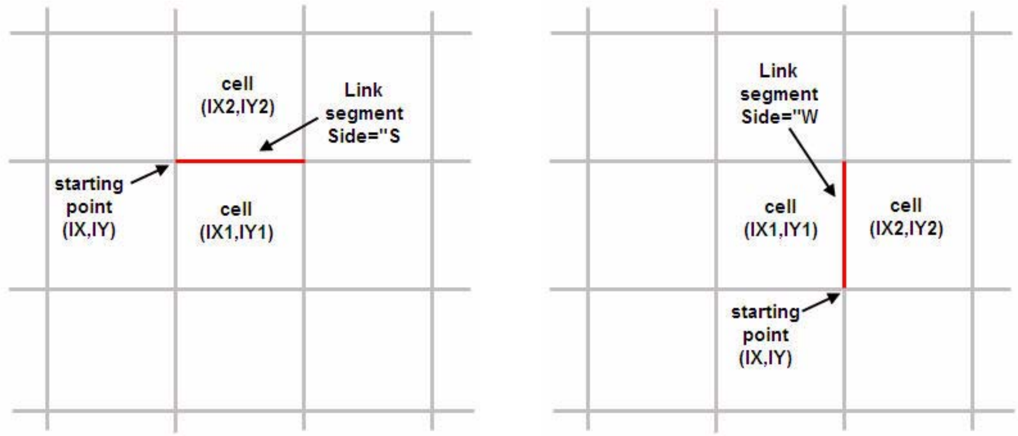
```
MIKE SHE River Link overview
============================

'==>' at start of line indicates that the bank is more than 0.010 m above ground level of a non-flood cell
'***' at start of line indicates that the bank is more than 0.010 m above ground level of a flood (inundation) cell
```

Link	IX	IY	Side	IX1	IY1	Topo1	Bank1	IX2	IY2	Topo2	Bank2	Bed	Width	Leak-opt	Leak-coeff	Spill
1	3	2	S	3	1	0.20	-0.08	3	2	0.22	-0.18	-0.97	71.66	Aq+Bed	0.1000E-04	On
2	3	2	W	2	2	0.12	-0.04	3	2	0.22	-0.14	-0.90	198.97	Aq+Bed	0.1000E-04	On
3	3	3	S	3	2	0.22	-0.10	3	3	0.24	0.00	-0.84	336.07	Aq+Bed	0.1000E-04	On
4	4	3	W	3	3	0.24	0.07	4	3	0.34	-0.03	-0.72	561.31	Aq+Bed	0.1000E-04	On
5	4	4	W	3	4	0.26	0.12	4	4	0.36	0.02	-0.63	741.77	Aq+Bed	0.1000E-04	On
6	3	5	S	3	4	0.26	0.17	3	5	0.28	0.07	-0.54	917.26	Aq+Bed	0.1000E-04	On

Spill	WeirCoeff	HExpo	FullWdepth	ThrVolSpill	Chainage	Branch
On	0.1838E+01	1.500	0.1000E+00	100.00	525.5563	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	453.8689	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	376.6670	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	249.8351	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	148.2199	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	49.40662	BRANCH1

In this table, the locations where the river links are higher than the topography are marked in the outside left column.

The reference system used in the table is illustrated below:



The explanation of the columns is:

Link: River Link ID number. ID starts at 1 and increases by 1.

IX,IY: coordinate of one end of the link segment. They are referred to the preprocessed grid such that (IX,IY)=(1,1) at the left-bottom corner of the model grid. The link segment can be drawn starting from (IX,IY) coordinate, and then following east direction if Side="S" or following the north direction if Side="W".

Side: relative position of the (IX1,IY1) cell with respect to the link segment. "S" stands for south and "W" for west.

IX1,IY1: coordinate of the cell on the south side of the link if Side="S", or the cell on the west side of the link if Side= "W". The left-bottom corner cell of the model grid has coordinates (IX1,IY1)= (1,1).

Topo1: Pre-processed Topo elevation (in meters) of the cell (IX1,IY1).

Bank1: Interpolated cross section bank elevation (in meters) at marker 1 or 3 at the link chainage (last column). The marker (1 or 3) corresponding to Bank1 depends on the position of the cell (IX1,IY1) with respect to the direction of increasing chainage. Marker 1 is the left marker in the increasing chainage direction.

IX2,IY2: Coordinate of the cell on the opposite side to (IX1,IY1). In other words, it is the cell on the north side of the link if Side="S", or the cell on the east side of the link if Side= "W". The left-bottom corner cell of the model grid has coordinates (IX2,IY2)= (1,1).

Topo2: Pre-processed Topo elevation (in meters) of the cell (IX2,IY2).

Bank2: Interpolated cross section bank elevation (in meters) at marker 1 or 3 at the link chainage (last column). The marker (1 or 3) corresponding to Bank2 depends on the position of the cell (IX2,IY2) with respect to the direction of increasing chainage. Marker 1 is the left marker in the increasing chainage direction.

Bed: Interpolated cross section elevation (in meters) at marker 2 at the link chainage (last column). In other words, it is the river bed bottom elevation interpolated at that chainage.

Width: Interpolated cross section width (in meters) at the link chainage (last column). The cross section width is the distance between markers 1 and 3 in the cross section profile.

Leak-opt: The Conductance option used in the coupling reach in which this river link is contained. The value is from in the MIKE SHE links table of the MIKE 11 Coupling Reaches dialogue. The three possible options are "Aq+Bed", "Aq only", and "Bed only". See Groundwater Exchange with MIKE 11 *(V.1 p. 207)* and Figure 7.13.

Leak-coeff: The Leakage Coef. value used in the coupling reach in which this river link is contained found in the MIKE SHE links table of "Coupling Reaches". See Groundwater Exchange with MIKE 11 *(V.1 p. 207)* and Figure 7.13.

Spill: Indicates whether the Allow overbank spilling option is checked for the coupling reach in which the river link is contained. The two possible values are "On" and "Off". See Figure 7.13.

WeirCoeff: The Weir coefficient value used in the coupling reach in which the river link is contained. See Figure 7.13.

HExpo: The Head exponent value used in the coupling reach in which the river link is contained. See Figure 7.13.

FullWdepth: The Minimum upstream height above bank for full weir width value used in the coupling reach in which the river link is contained. See Figure 7.13.

ThrVolSpill: Threshold volume value in cubic meters, which is the product between the Minimum flow are for overbank spilling value (for the coupling reach in which this river link is contained. See Figure 7.13) and the MIKE SHE cell size.

Chainage: Chainage of the MIKE 11 network that corresponds to the center of the link segment. They are sorted from highest to lowest chainage values for the same branch.

Branch: Name of the MIKE 11 Branch. Branches are sorted alphabetically.

7.6.5 Groundwater Exchange with MIKE 11

The exchange flow, Q, between a saturated zone grid cell and the river link is calculated as a conductance, C, multiplied by the head difference between the river and the grid cell.

$$Q = C \cdot \Delta h \tag{7.4}$$

Note that Eq. (7.4) is calculated twice - once for each cell on either side of the river link. This allows for different flow to either side of the river if there is a groundwater head gradient across the river, or if the aquifer properties are different.

Referring to Figure 7.11, the head difference between a grid cell and the river is calculated as

$$\Delta h = h_{grid} - h_{riv} \tag{7.5}$$

where h_{grid} is the head in the grid cell and h_{riv} is the head in the river link, as interpolated from the MIKE 11 H-points.

If the ground water level drops below the river bed elevation, the head difference is calculated as

$$\Delta h = z_{bot} - h_{riv} \tag{7.6}$$

where z_{bot} is the bottom of the simplified river link cross section, which is equal to the lowest point in the MIKE 11 cross-section.

In Eq. (7.4), the conductance, *C*, between the cell and the river link can depend on

- the conductivity of the aquifer material only. See Aquifer Only Conductance *(V.1 p. 208)*, or
- the conductivity of the river bed material only. See River bed only conductance *(V.1 p. 209)*, or
- the conductivity of both the river bed and the aquifer material. See Both aquifer and river bed conductance *(V.1 p. 210)*.

Aquifer Only Conductance

When the river is in full contact with the aquifer material, it is assumed that there is no low permeable lining of the river bed. The only head loss between the river and the grid node is that created by the flow from the grid node to the river itself. This is typical of gaining streams, or streams that are fast moving.

Thus, referring to Figure 7.11, the conductance, *C*, between the grid node and the river link is given by

$$C = \frac{K \cdot da \cdot dx}{ds} \tag{7.7}$$

where *K* is the horizontal hydraulic conductivity in the grid cell, *da* is the vertical surface available for exchange flow, *dx* is the grid size used in the SZ component, and *ds* is the average flow length. The average flow length, *ds,* is the distance from the grid node to the middle of the river bank in the triangular, river-link cross-section. *ds* is limited to between 1/2 and 1/4 of a cell width, since the maximum river-link width is one cell width (half cell width per side).

There are three variations for calculating *da*:

- If the water table is higher than the river water level, *da* is the saturated aquifer thickness above the bottom of the river bed. Note, however, that *da* is not limited by the bank elevation of the river cross-section, which means that if the water table in the cell is above the bank of the river, *da* accounts for overland seepage above the bank of the river.
- If the water table is below the river level, then *da* is the depth of water in the river.
- If the river cross-section crosses multiple model layers, then *da* (and therefore *C*) is limited by the available saturated thickness in each layer. The exchange with each layer is calculated independently, based on the *da* calculated for each layer. This makes the total exchange independent of the number of layers the river intersects.

This formulation for *da* assumes that the river-aquifer exchange is primarily via the river banks, which is consistent with the limitation that there is no unsaturated flow calculated beneath the river.

River bed only conductance

If there is a river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the conductivity of the river bed. However, when the head loss across the river bed is much greater than the head loss in the aquifer material, then the head loss in the aquifer can be ignored (e.g. if the bed material is thick and very fine and the aquifer material is coarse). This is the assumption used in many groundwater models, such as MODFLOW.

In this case, referring to Figure 7.11, the conductance, *C*, between the grid node and the river link is given by

$$C = L_c \cdot w \cdot dx \tag{7.8}$$

where *dx* is the grid size used in the SZ component, L_c is the leakage coefficient [1/T] of the bed material, and *w* is the wetted perimeter of the cross-section.

In Eq. (7.8), the wetted perimeter, *w*, is assumed to be equal to the sum of the vertical and horizontal areas available for exchange flow. From Figure 7.11, this is equal to $da + l_h$, respectively. The horizontal infiltration length, l_h, is calculated based on the depth of water in the river and the geometry of the triangular river-link cross-section.

The infiltration area of the river link closely approximates the infiltration area of natural channels when the river is well connected to the aquifer. In this case, the majority of the groundwater-surface water exchange occurs through the banks of the river and decreases to zero towards the centre of the river. However, for losing streams separated from the groundwater table by an unsaturated zone, the majority of the infiltration occurs vertically and not through the river banks. In this case, the triangular shape of the river link does not really approximate wide losing streams.and the calculated infiltration area may be too small - especially if the MIKE 11 bank elevations are much higher than the river level. This can be compensated for by either choosing a lower bank elevation or by increasing the leakage coefficient.

There are three variations for calculating *da*:

- If the water table is higher than the river water level, *da* is the saturated aquifer thickness above the bottom of the river bed. Note, however, that *da* is not limited by the bank elevation of the river cross-section, which means that if the water table in the cell is above the bank of the river, *da* accounts for overland seepage above the bank of the river.
- If the water table is below the river level, then *da* is the depth of water in the river.
- If the river cross-section crosses multiple model layers, then *da* (and therefore *C*) is limited by the available saturated thickness in each layer. The exchange with each layer is calculated independently, based on the *da* calculated for each layer. This makes the total exchange independent of the number of layers the river intersects.

This formulation for *da* assumes that the river-aquifer exchange is primarily via the river banks, which is consistent with the limitation that there is no unsaturated flow calculated beneath the river.

Both aquifer and river bed conductance

If there is a river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the conductivity of the river bed and can be calculated as a serial connection of the individual conductances. Thus, referring to Figure 7.11, the conductance, *C*, between the grid node and the river link is given by

$$C = \frac{1}{\dfrac{ds}{K \cdot da \cdot dx} + \dfrac{1}{L_c \cdot w \cdot dx}} \tag{7.9}$$

where *K* is the horizontal hydraulic conductivity in the grid cell, *da* is the vertical surface available for exchange flow, *dx* is the grid size used in the

SZ component, *ds* is the average flow length, L_c is the leakage coefficient [1/T] of the bed material, and *w* is the wetted perimeter of the cross-section. The average flow length, *ds,* is the distance from the grid node to the middle of the river bank in the triangular, river-link cross-section. *ds* is limited to between 1/2 and 1/4 of a cell width, since the maximum river-link width is one cell width (half cell width per side).

In Eq. (7.8), the wetted perimeter, *w*, is assumed to be equal to the sum of the vertical and horizontal areas available for exchange flow. From Figure 7.11, this is equal to $da + l_h$, respectively. The horizontal infiltration length, l_h, is calculated based on the depth of water in the river and the geometry of the triangular river-link cross-section.

The infiltration area of the river link closely approximates the infiltration area of natural channels when the river is well connected to the aquifer. In this case, the majority of the groundwater-surface water exchange occurs through the banks of the river and decreases to zero towards the centre of the river. However, in the case of losing streams separated from the groundwater table by an unsaturated zone, the majority of the infiltration occurs vertically and not through the river banks. In this case, the horizontal infiltration area may be too small, if the MIKE 11 bank elevations are much higher than the river level. This can be compensated for by either choosing a lower bank elevation or by increasing the leakage coefficient.

There are three variations for calculating *da*:

- If the water table is higher than the river water level, *da* is the saturated aquifer thickness above the bottom of the river bed. Note, however, that *da* is not limited by the bank elevation of the river cross-section, which means that if the water table in the cell is above the bank of the river, *da* accounts for overland seepage above the bank of the river.
- If the water table is below the river level, then *da* is the depth of water in the river.
- If the river cross-section crosses multiple model layers, then *da* (and therefore *C*) is limited by the available saturated thickness in each layer. The exchange with each layer is calculated independently, based on the *da* calculated for each layer. This makes the total exchange independent of the number of layers the river intersects.

This formulation for *da* assumes that the river-aquifer exchange is primarily via the river banks, which is consistent with the limitation that there is no unsaturated flow calculated beneath the river.

7.6.6 Steady-state groundwater simulations

For steady-state groundwater models, MIKE 11 is not actually run. Rather the initial water level in MIKE 11 is used for calculating *da* in the conductance formulas and h_{riv} for the head gradient.

To improve numerical stability during steady-state groundwater simulations, the actual conductance used in the current iteration is an average of the currently calculated conductance and the conductance used in the previous iteration.

Canyon option for steady-state groundwater simulations

In the case of a deep, narrow channel crossing multiple model layers, the head difference used in Equations (7.4) and (7.5) can optionally be limited by the bottom elevation of the layer. Thus,

$$\Delta h = h_{grid} - max(h_{riv}, z) \tag{7.10}$$

where z is the bottom of the current layer.

The above formulation reduces the infiltration from upper layers by reducing the available gradient. Without the ‘Canyon’ option, MIKE SHE effectively assumes that the river is hydraulically connected to the upper most model layer, since MIKE SHE calculates the exchange flow with all layers that intersect the river based on the difference between the river level and the water table.

Currently, this option is only available for steady-state models. It is activated by means of the boolean Extra Parameter, *Enable Canyon Exchange*. For more information on the use of extra parameters, see Extra Parameters *(V.1 p. 299)*.

7.7 Overland Flow Exchange with MIKE 11

The exchange between overland flow and MIKE 11 rivers can be calculated in three different ways. If the flooding from MIKE 11 to MIKE SHE cells is ignored (the "no flooding" option) then the exchange from overland flow is one way - that is overland flow only discharges to MIKE 11 rivers. If the you want to simulate flooding from MIKE 11 to MIKE SHE then the water can be transfered from MIKE 11 to MIKE SHE using "Flood Codes" or via direct overbank spilling using a wier formula. In principle, the flood code option does not impact the solution time signficantly, is relatively easy to set up for simple cases and is sufficient when detailed flood plain flow is not required. Direct overbank spilling combined with the explicit solution method requires more detailed topopgraphy data and is useful when detailed flood plain flow is required, but can be significantly slower from a numerical perspective.

Flooding with Overbank Spilling

If you are simulating flooding on the flood plain using the overbank spilling option, then the MIKE 11 cross-sections are normally restricted to the main channel. The flood plain is defined as part of the MIKE SHE topography. Since, the bank elevation is used to define when a cell floods, it is more critical that the cross-sections are consistent with your topography, in the areas where you want to simulate flooding. The table in the simulation log file mentioned above is useful to locate these inconsistencies. It is usually necessary to have a very fine grid and a detailed DEM for such simulations, which tends to reduce the inconsistencies because it reduces the amount of interpolation and averaging when creating the model topography.

Flooding with Flood Codes

If you are simulating flooding on the flood plain using the flood code option, then flood plain elevation should be consistent with the cross-sections. Otherwise, the flood plain storage will be inconsistent with the river storage based on the cross-sections.

When you are using Flood Codes, you typically specify wide cross-sections for your rivers. The wide cross-sections can then account for the increased flood plain storage during flood events. MIKE 11 then places water on the MIKE SHE cells that are defined by flood codes - if the water level in the river is above the cell topography. The flood water is then free to infiltrate or evaporate as determined by MIKE SHE.

In such flooded cells, overland flow is no longer calculated, so there is no longer any overland exchange to MIKE 11 in flooded cells. Thus, the bank elevation is not so critical, as long as the cell is flooded. However, when the flood recedes, the cells revert back to normal overland flow cells and the same considerations apply as if the cells were not flooded - namely the bank elevation should be below the topography to ensure that overland flow can discharge to the river link.

Flood codes are also commonly used for lakes and reservoirs. In this case, you specify the lake bed bathymetry as the topography (or using the Bathymetry option). The lake area is defined using flood codes and the MIKE 11 cross-sections stretch across the lake. MIKE 11 calculates the lake level and floods the lake. Overland flow adjacent to the lake intersects the flooded cells and the overland water is added to the lake cell (and to MIKE 11 as lateral inflow). Groundwater exchange to the lake is through the lake bed as saturated zone discharge. In principle, the saturated zone could discharge to the river link, but the local groundwater gradients would probably make this exchange very small.

Combining Flood Codes and Overbank Spilling

Flooding using Overbank spilling and Flood Codes is possible in the same model and even in the same coupling reach. The only restriction is that there is no overland flow calculated in cells flooded by means of Flood Codes. So, in a long coupling reach, you could allow overbank spilling and calculate overland flow using the explicit solver, but define flood codes in the wide downstream flood plain were the surface water gradients are very low during flooding and in the wide shallow reservoir half way down the system.

7.7.1 Lateral inflow to MIKE 11 from MIKE SHE overland flow

MIKE SHE's overland flow solver calculates the overland flow across the boundary of the MIKE SHE cells. If a river link is located on the cell boundary, any overland flow is intercepted by the river link and added to the water balance of the river link. However, two checks are first made to ensure exchange to the river is physically possible. The level of ponded water in the cell must be above the

1. water level in the river link, and
2. bank elevation of the river link.

In the second case, the level of ponded water is checked against the appropriate left and right bank elevations of the river link.

However, there is no mechanism for exchange from MIKE 11 to overland flow. If the water level in the river rises above the bank elevation, then the bank elevation is simply extended vertically upwards.

7.7.2 Flooding from MIKE 11 to MIKE SHE using Flood Codes

The MIKE SHE/MIKE 11 coupling allows you to simulate large water bodies such as lakes and reservoirs, as well as flooded areas. If this option is used, MIKE SHE/MIKE 11 applies a simple flood-mapping procedure where MIKE SHE grid points (e.g. grid points in a lake or on a flood plain) are linked to the nearest H-point in MIKE 11 (where the water levels are calculated). Surface water stages are then calculated in MIKE SHE by comparing the water levels in the H-points with the surface topographic elevations.

Conceptually, you can think of the flooded cells as "side storages", where MIKE 11 continues to route water downstream as 1D flow. But, at the same time, the water is available to the rest of MIKE SHE for evaporation and infiltration.

Determination of the Flooded Area and Water Levels

The flooded area in MIKE SHE must be delineated by means of integer flood codes, where each coupling reach is assigned a flood code.

During the simulation, the flood-mapping procedure calculates the surface water level on top of each MIKE SHE cell with a flood code by comparing the MIKE 11 surface water level to the surface topography in the model grid. A grid cell is flooded when the MIKE 11 surface water level is above the topography. The MIKE 11 water level is then used as the level of ponded surface water.

The actual water level in the grid cell is calculated as a distance weighted average of the upstream and downstream MIKE 11 H-points.

Calculation of the Exchange Flows

After the MIKE SHE overland water levels have been updated, MIKE SHE calculates the infiltration to the unsaturated and saturated zones and evapotranspiration. Thus, MIKE SHE simply considers any water on the surface, including MIKE 11 flood water as ‘ponded water’, disregarding the water source. In other words, ponded rainfall and ponded flood water are indistinguishable.

MIKE SHE does not calculate overland flow between cells that are flooded by MIKE 11. Nor, does MIKE SHE calculate overland exchange to MIKE 11, if the cell is flooded by MIKE 11. However, lateral overland flow to neighbouring non-flooded cells is allowed. Thus, if there is a neighbouring, non-flooded cell with a topography lower than a flooded cell’s water level , then MIKE SHE will calculate overland flow to the non-flooded cell as normal.

The calculated exchange flow between the flooded grid cells and the overland, saturated, unsaturated zone or other source/sink terms is fed back to MIKE 11 as lateral inflow or outflow to the corresponding H-point in the next MIKE 11 time step.

In terms of the water balance, the surface water in the inundated areas belongs to the MIKE 11 water balance. In other words, if there is ponded water on the surface when the grid cell floods, the existing ponded water is added to the MIKE 11 water flow in the river. As long as the element is flooded, any exchange to or from the surface water is managed by MIKE 11 as lateral inflow and regular overland flow is not calculated.

If the element reverts back to a non-flooded state, then any subsequent ponded water is again treated as regular overland flow and the water balance is accounted for within the overland flow component.

7.7.3 Direct Overbank Spilling to and from MIKE 11

If you want to calculate 2D overland flow on the flood plain during a storm event, then you cannot use the Flooding from MIKE 11 to MIKE SHE using Flood Codes *(V.1 p. 214)* method. The Area Inundation method is primarily used as a way to spread river water onto the flood plain and make it available for interaction with the subsurface via infiltration and evapotranspiration.

The Overbank spilling option treats the river bank as a weir. When the overland flow water level or the river water level is above the left or right bank elevation, then water will spill across the bank based on the standard weir formula

$$Q = \Delta x \cdot C \cdot (H_{us} - H_w)^k \cdot \left[1 - \left(\frac{H_{ds} - H_w}{H_{us} - H_w}\right)^k\right]^{0.385} \tag{7.11}$$

where Q is the flow across the weir, Δx is the cell width, C is the weir coefficient, H_{us} and H_{ds} refer to the height of water on the upstream side and downstream side of the weir respectively, H_w is the height of the weir, and k is a head exponent.

The units of the weir coefficient depend on the exponent. In MIKE SHE, the default exponent is 1.5, which means that the weir coefficient has units of $m^{1/2}/s$.

If the water levels are such that water is flowing to the river, then the overland flow to the river is added to MIKE 11 as lateral inflow. If the water level in the river is higher than the level of ponded water, then the river water will spill onto the MIKE SHE cell and become part of the overland flow.

If the upstream water depth over the weir approaches zero, the flow over the weir becomes undefined. Therefore, the calculated flow is reduced to zero linearly when the upstream height goes below a threshold.

If you use the overbank spilling option, then you should also use the Explict Numerical Solution *(V.2 p. 271)* for overland flow.

7.7.4 Converting from Flood Codes to Overbank Spilling

The explicit solver and overbank spilling from MIKE 11 to overland flow are new in the 2007 Release. In principle, if you were careful setting up your flood codes, then the conversion to overbank spilling should result in the same flooded area, with similar depths. The only difference will be that the water on the flooded area is flowing.

However, in practice the conversion is not likely to be this smooth. Flood code setups are typically done manually and the topography is typically not very closely controlled - as long as it was inundated when it was supposed to be. Furthermore, the need for detailed surface roughness (ManningsM) will require additional data. Finally, the complication of fully dynamic (diffusive wave) 2D flow can lead to complicated water flows across the flood plain. So, there is likely to be substantial adjustment and re-calibration to get the flooding right.

Fortunately, you can mix Flood codes and Overbank spilling in the same model and even in the same coupling reach. This allows you to update only the parts of your model where the overbank spilling is important and leave the Flood code option intact elsewhere.

7.8 Unsaturated Flow exchange with MIKE 11

Direct exchange between MIKE 11 and the unsaturated zone is not currently supported. Groundwater exchange is assumed to be a line source and sink at the boundary between cells and the exchange mechanism assumes that the primary exchange takes place along the river banks. This is a suitable assumption when the river is well connected to the aquifer.

However, when MIKE 11 can exchange water with overland flow via overbank spilling or flood codes, then river water is added to the ponded water on a MIKE SHE cell, which can then infiltrate to the unsaturated zone.

7.9 Water balance with MIKE 11

The water balance tool in MIKE SHE (Using the Water Balance Tool *(V.1 p. 105)*) includes the exchange with MIKE 11, but it does not include the water balance within MIKE 11. In other words, once water enters MIKE 11 it is no longer part of the MIKE SHE water balance. Thus, there are numerous water balance items that detail the different exchanges to and from MIKE 11.

Water exchanges within MIKE 11 can be evaluated using the MIKE View tool. In some cases, this may require you to include the additional output for MIKE 11, which is selected in the Additional Output tab in MIKE 11's HD editor.

Note: output in MIKE 11 is instantaneous, whereas the output in MIKE SHE is generally accumulated within a time step. Therefore, a flow at a rate at a point in MIKE 11 (e.g. a weir) will be the instantaneous flow at the end of the time step. In MIKE SHE, however, the flow into a cell will be the average flow over the time step.

7.10 Coupling MIKE SHE Water Quality to MIKE 11

Detailed information on the MIKE 11 Water Quality modules are found in the MIKE 11 documentation.

The coupling between MIKE 11 and the rest of MIKE SHE's hydrologic processes is relatively automatic. You must set up a MIKE 11WQ model independent of MIKE SHE and specify this .sim11 file in the Rivers and Lakes dialog. This .sim11 file must only have the same network geometry as the WM .sim11 file. It does not have to be the same .sim11 file.

The MIKE 11 WQ model can also include EcoLab, which will allow you simulate eutrophication, etc. in the surface water.

There are a few caveats/limitations that you need to be aware of:

- Species names must be identical in MIKE SHE and MIKE 11. If they are not identical, then the solutes will be transfered to the river as an infinite sink, but will not be transported in MIKE 11.
- The overland WQ must be included if you want to simulate water quality coupled to MIKE 11.
- Recycling of WM results is not supported in MIKE 11. This means that if you want to simulate the coupling between MIKE 11 and the rest of MIKE SHE, your WQ simulation must be continuous.
- There is no solute transfer **from** MIKE 11 to MIKE SHE via overbank spilling or flood codes. Only the water is transfered to flood codes and overbank spilling. Any solutes will remain in MIKE 11. Thus, solute transfer from MIKE 11 to MIKE SHE's SZ is the only transfer supported. Solute transfer from MIKE SHE to MIKE 11 is supported for both overland and saturated flow.

7.11 MIKE 11 User Interface

The following section provides additional information for the MIKE 11 dialogues that are commonly used with MIKE SHE.

7.11.1 MIKE SHE Coupling Reaches

Each MIKE 11 branch that exchanges water with MIKE SHE is called a coupling reach. A MIKE 11 branch can be sub-divided into several coupling reaches. A reason for doing so could be to allow different riverbed leakage coefficients for different parts of the river.

The upper half of the dialogue displays the properties of the current coupling reach. While, the bottom half of the dialogue is a table listing all of the coupling reaches defined.

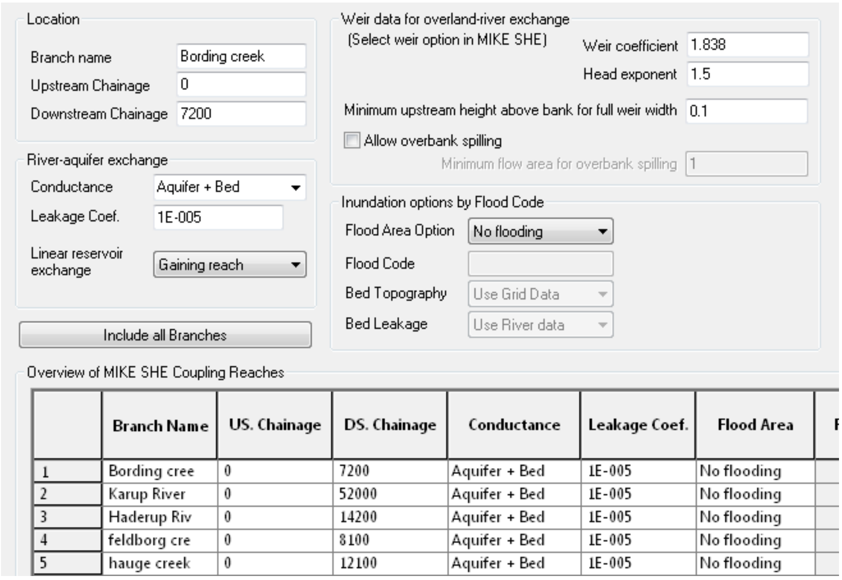


Figure 7.13 MIKE SHE River Links dialogue in the tabular view of the MIKE 11 Network Editor

Include all branches button

If the Include all branches button is pressed all the branches in the MIKE 11 setup will be copied to the MIKE SHE Links table. Branches that should not be in the coupling can subsequently be deleted manually and the specifications for the remaining branches completed. Thus, you may have a large and complex hydraulic model, but only couple certain reaches to MIKE SHE. All branches will still be in the hydraulic MIKE 11 model but MIKE SHE will only exchange water with branch reaches that are listed in the MIKE SHE links table.

Note The Include all branches button will erase all existing links that have been specified.

Location

The **branch name**, **upstream chainage** and **downstream chainage** define the stretch of river that can exchange water with MIKE SHE. A MIKE 11 branch can be sub-divided into several coupling reaches, to allow, for example, different riverbed leakage coefficients for different parts of the river.

River Aquifer Exchange

Conductance

The river bed conductance can be calculated in three ways.

Aquifer only - When the river is in full contact with the aquifer material, it is assumed that there is no low permeable lining of the river bed. The only head loss between the river and the grid node is that created by the flow from the grid node to the river itself. This is typical of gaining streams, or streams that are fast moving. More detailed information on this option can be found in Aquifer Only Conductance *(V.1 p. 208)*.

River bed only - If there is a low conductivity river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the conductivity of the river bed. However, when the head loss across the river bed is much greater than the head loss in the aquifer material, then the head loss in the aquifer can be ignored (e.g. if the bed material is thick and very fine and the aquifer material is coarse). This is the assumption used in many groundwater models, such as MODFLOW. More detailed information on this option can be found in River bed only conductance *(V.1 p. 209)*.

Aquifer + Bed - If there is a low conductivity river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the total conductivity of the between the river and the adjacent groundwater can be calculated as a serial connection of the individual conductances. This is commonly the case, when the aquifer material presents a significant head loss. For example, when the aquifer is relatively fine and the groundwater cells are quite large.More detailed information on this option can be found in Both aquifer and river bed conductance *(V.1 p. 210)*.

Leakage Coefficient - [1/sec]

This is the leakage coefficient for the riverbed lining in units of [1/seconds]. The leakage coefficient is active only if the conductance calculation method includes the river bed leakage coefficient.

Linear Reservoir Exchange

If you are using the Linear Reservoir method for groundwater in MIKE SHE, then by default the Interflow and Baseflow reservoirs discharge uniformly to all the river links within the reservoir. This is generally true in the lower reaches. However, in the upper reaches many rivers discharge to the groundwater system.

In this dialogue, you can define whether or not a branch is a Gaining branch (default) or a Losing branch. If the branch is a:

- Gaining branch, then the leakage coefficient and wetted area are ignored and the rate is discharge from the Baseflow reservoir to the river is calculated based on the Linear Reservoir method.
- Losing branch, then the rate of discharge from the river to the Baseflow reservoir is calculated using:

  Q=water depth*bank width* branch length* leakage coefficient.

The gaining and losing calculations are done in MIKE SHE for every river link within the Baseflow reservoir. For the losing river links, the water level is interpolated from the nearest H-points, the bottom elevation and bank width is interpolated from the nearest cross-sections. The length is simply the cell size. MIKE SHE keeps track of the inflow volumes to ensure that sufficient water is available in the river link.

Weir Data for overland-river exchange

The choice of using the weir formula for overland-river exchange is a global choice made in the MIKE SHE OL Computational Control Parameters *(V.2 p. 36)* dialogue. If the weir option is chosen in MIKE SHE, then all MIKE 11 coupling reaches will use the weir formula for moving water across the river bank.The weir option is typically used when you want to simulate overbank spilling and detailed 2D surface flow in the flood plains. The following parameters and options are available when you specify the weir option in MIKE SHE. If you chose the Manning equation option in MIKE SHE, then these parameters are ignored.

Weir coefficient and Head exponent

The Weir coefficient and head exponent refer to the C and k terms respectively in Equation (7.11). The default values are generally reasonable. Both the weir coefficient and the head exponent are dimensionless.

Minimum upstream height above bank for full weir width

In Equation (7.11), when the upstream water depth above the weir approaches zero, the flow over the weir becomes undefined. To prevent numerical problems, the flow is reduced linearly to zero when the water depth is below the minimum upstream height threshold. The EUM data type is Water Depth.

Allow overbank spilling

This checkbox lets you define which branches are allowed to flood over their banks. Thus, you can allow flooding from MIKE 11 only in branches with defined flood plains, or only in areas of particular interest.

If overbank spilling is not allowed for a particular branch, then the overland-river exchange is still calculated using the weir formula, but the exchange is only one way - that is from overland flow to the river.

Minimum flow area for overbank spilling

The minimum flow area threshold prevents overbank spilling when the river is nearly dry. The flow area is calculated by dividing the volume of water in the coupling reach by the length of the reach. The EUM data type is Flow Area, which by default is m^2.

The default value is 1 m^3/m length of river. This is quite a small amount of water for most reasonable rivers and should be adjusted based on the river width. For example, if your river is 10m wide, then spilling will occur when the water level is 10cm above the bank elevation. However, if your river is 200m wide, then spilling would start when the water level is only 5mm above the bank elevation.

The cell size also plays a role here. When a cell is flooded, the entire cell is covered by water. If the cell size is 1000m x 1000m, then a flood of 1 m^3/m of river will be only 1mm deep across the cell.

Inundation options by Flood Code

The Inundation method allows specified model grid cells to be flooded if the MIKE 11 water level goes above the topography of the cell. In this case, water from MIKE 11 is “deposited” onto the flooded cell. The flood water can then infiltrate, or evaporate. However, overland flow between flooded cells and to the river is not calculated. Also, the flooded water remains as part of the MIKE 11 water balance and is only transferred to MIKE SHE when it infiltrates.

Inundation areas and their associated Flood codes are specified on a coupling reach basis.

Flood Area Option

The following three options are available for the Flood Area Option:

- **No Flooding** (default) With the No flooding option, the MIKE 11 river is confined between the left and right banks. If the water level goes above the bank elevation, then the river is assumed to have vertical banks above the defined left and right bank locations. No flooding via flood codes will be calculated.

  Note If neither inundation nor overbank spilling is allowed, then the overland flow exchange to the river is one way only. The only mechanism for river water to flow back into MIKE SHE is through baseflow infiltration to the groundwater. If overland flow does spill into the river, there is first a check to make sure that the water level in the river is not higher than the ponded water.

- **Manual** If the Manual option is selected, then you must supply a Flood code map in MIKE SHE. This Flood code map is used to established the relationship between MIKE 11 h-points and individual model grids in MIKE SHE. MIKE SHE then calculates a simple flood-mapping during the pre-processing that is used during the simulation to assign river water stages to the MIKE SHE cells if the river level is above the topography.
- **Automatic** The automatic flood mapping option is useful if the river network geometry is not very complex or for setting up the initial flood mapping, for later refinement. The automatic method, maps out a polygon for each coupling reach based on the left and right bank locations of all the cross-sections along the coupling reach. All cells within this polygon are assigned an integer flood code, unique to the coupling reach. The automatic method works reasonably well along individual branches with cross-sections that represent the flood plain. At branch intersections the assigned flood code may not be correct. However, this is often not serious because at river confluences the water levels in the different branches are roughly the same anyway. In any case, the flood code map is available in MIKE SHE’s preprocessed tab, where you can check its reasonableness. Right clicking on the map will give you the option of saving the map to a dfs2 file, which you can then correct and use with the Manual option.

Flood Code

If the Manual option is selected, then you must specify a Flood code for the coupling reach. The flood code is used for mapping MIKE SHE grids to MIKE 11 h-points. You must click on the Flood Code checkbox in Figure 7.13, and then specify an integer flood code file in MIKE SHE. The specified flood code for the coupling reach must exist in the dfs2 Flood Code file. It is important to use unique flood codes to ensure correct flood-mapping.

Bed Topography

Since the flood mapping procedure will only flood a cell when the river water level is above the cell's topography, accurate flood inundation mapping requires accurate elevation data. If one of the flood options are selected, then you have the option to refine the topography of the flood plain cells based on the actual cross-section elevations or on a more detailed local-scale DEM, if it exists.

- **Use Grid Data** (default) If Grid Data option is selected, the MIKE SHE topography value is used to determine whether or not the cell is flooded. However, the program first checks to see if a Bathymetry file has been specified.

  If a Bathymetry file is available, the topography values of the cells with flood codes are re-interpolated based on the bathymetry data. The bathymetry option is useful when a more detailed DEM exists for the flood plain area compared to the regional terrain model.

- **Use Cross-section** If the Cross-section option is specified the topography values of the cells with flood codes are re-interpolated based on the cross-section data.

  When the cross-section option is selected, the pre-processor maps out a flood-plain polygon for the coupling reach, based on the left and right bank locations of all the cross-sections along the coupling reach. Interpolated cross-sections are created between the available actual cross-sections, if the cross-section spacing is greater than ½ Δx (grid size). All the cross-sections (real and interpolated) are sampled to obtain a set of point values for elevation in the flood plain. The topography values of all cells with the current flood code that are within the flood-plain polygon are re-interpolated using the bilinear interpolation method to obtain a new topography value.

  In principle, the Cross-section option ensures a good consistency between MIKE SHE grid elevations and MIKE 11 cross-sections. There will, however, often be interpolation problems related to river meandering, tributary connections, etc., where wide cross-sections of separate coupling reaches overlap. Thus, you can make the initial MIKE SHE set-up using the Cross-section option and then subsequently retrieve and check the resulting ground surface topography, from the pre-processed data. If needed, the pre-processed topography can be saved to a .dfs2 file (right click on the map), modified and then used as input for a new set-up, now using the Use Grid Data option.

Bed Leakage

If one of the flood options are selected, then you must also specify if and how the leakage coefficient will be applied on the flooded cells. The infiltration/seepage of MIKE SHE flood grids is calculated as ordinary overland exchange with the saturated or unsaturated zone. That is, the leakage coefficient, if it exists, is applied to both saturated exchange to and from the flooded cell and unsaturated leakage from the flooded cell. In the case of the unsaturated leakage, the actual leakage is controlled by either the leakage coefficient or the unsaturated zone hydraulic conductivity relationship - which yields the lowest infiltration rate.

- **Use grid data** In this case, the leakage coefficient specified in Surface-Subsurface Leakage Coefficient is used. If this item has not been specified, then the leakage coefficient will be calculated based on the aquifer material only.
- **Use river data** (default) In this case, the Leakage Coefficient - [1/sec] for the coupling reach is actually copied to the flooded cell and used for all flood grid points of the coupling reach. This makes sense if the flood plain is frequently flooded and covered with the same sediments as the river bed. However, in many cases the flood plain material is not the same as the river bed and the infiltration rate can be substantially different.

7.12 Common MIKE 11 Error Messages

There are a number of common MIKE 11 error messages that you are likely to encounter when using MIKE 11 with MIKE SHE.

7.12.1 Error No 25: At the h-point ____ the water depth greater than 4 times max. depth

This error message essentially says that your MIKE 11 model is unstable. It frequently occurs when there is an inconsistency in your bed elevations at the branch junctions. For example, if the bed elevation of the main branch is much greater than the side branch, then the water piles up and causes this error.

7.12.2 Warning No 47: At the h-point ____ the water level as fallen below the bottom of the slot x times

This warning message essentially says that your MIKE 11 model is unstable. The slot is a numerical trick that keeps a very small amount of water in the MIKE 11 cross-section when the river is dry. So, when the water level falls below the slot, it implies that your river has dried out. This warning frequently occurs when there is either an inconsistency in your bed elevations or there is an error in your boundary conditions that is keeping water from entering the system.

7.12.3 Warning No __: Bed levels not the same

This warning message is issued when the bed elevation of a side branch is not the same as the main branch. If the difference is small (say a few cm) it can usually be ignored. However, if the side branch is much lower than the main branch then this warning will often be accompanied by Error No 25: At the h-point ____ the water depth greater than 4 times max. depth, as the water will pile up and not be able to flow into the main branch. If the side branch is only slightly lower than the main branch or even if they are the same, then backward flows can occur in the side branch when the water level in the main branch rises. If this is realistic fine, but often it is not. More typically, the side branch is slightly higher than the main branch.

DRAINAGE MODELLING WITH MIKE URBAN

8 USING MIKE SHE WITH MIKE URBAN

Coupling MIKE URBAN and MIKE SHE allows you to simulate the effect of urban drainage and sewer systems on the surface/subsurface hydrology.

The use of the integrated MIKE SHE/MIKE URBAN system is not very different from establishing a stand-alone MIKE URBAN model and a stand-alone MIKE SHE model. In principle there are three basic set-up steps to have a coupled MIKE SHE-MIKE URBAN model:

1. Establish a MIKE URBANMIKE URBAN hydraulic model as a stand-alone model, make a performance test and, if possible, a rough calibration using prescribed inflow and boundaries.
2. Establish a MIKE SHE model that includes the overland flow component and (optionally) the saturated zone and unsaturated zone components.
3. Couple MIKE SHE and MIKE URBAN by defining the locations where MIKE URBAN should interact with MIKE SHE.

When MIKE SHE runs, it will call MIKE URBAN and ask it to perform a MIKE URBAN time step. If the end of the MIKE SHE time step has not yet been reached, MIKE SHE will ask MIKE URBAN to calculate the next MIKE URBAN time step. The MIKE URBAN model will run normally if it is launched directly from MIKE URBAN.

Note: The MIKE URBAN coupling was originally developed for the stand-alone sewer modelling product called MOUSE, which was later incorporated into MIKE URBAN. Thus, references in this chapter to MIKE URBAN can largely be substituted by “MOUSE”. Further, older MOUSE models can be coupled to MIKE SHE using the same method described here.

Important: In the command lines in the input files, the word “mouse” must still be used. For example, the Extra Parameters option to activate the MIKE URBAN coupling must be “mouse coupling”.

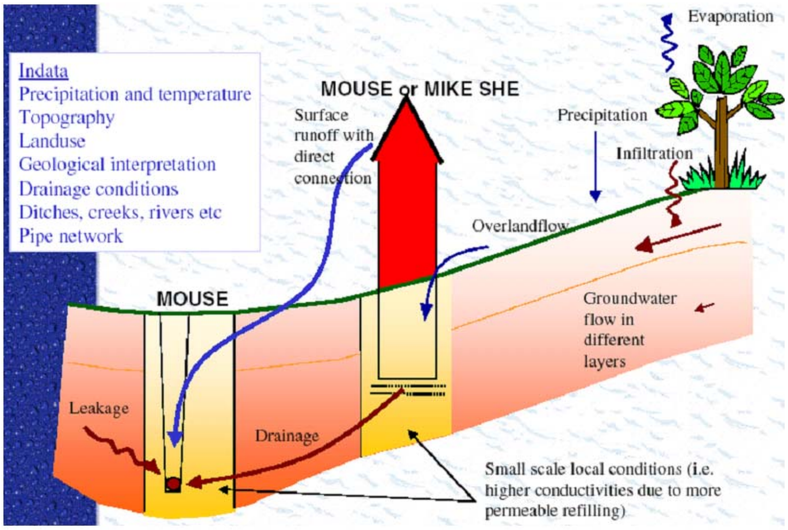


Figure 8.1 MIKE SHE to MIKE URBAN coupling linkages

The exchange between MIKE URBAN and MIKE SHE is calculated based on the following equation

$$Q = C \cdot (H_{SHE} - H_{MOUSE})^k \tag{8.1}$$

where Q is the exchange between MIKE URBAN and MIKE SHE, C is the exchange coefficient, k is a head difference exponent and

$$H_{SHE} = Max(H_{cell}, Z_T, Z_M) \tag{8.2}$$

$$H_{MOUSE} = Max(H_{pipe}, Z_T, Z_M) \tag{8.3}$$

where H_{cell} is the head in the MIKE SHE cell, H_{pipe} is the head in the MIKE URBAN pipe, Z_T is the topographic elevation in the cell and Z_M is the elevation of the manhole.

There are five variations on how to calculate the exchange based on above equations:

MIKE SHE SZ to MIKE URBAN LINKS

This is a leakage-based solution in which the head difference exponent is 1 and the exchange coefficient in Equation (8.1) for the flow to or from the pipe is calculated by

$$C = C_L \cdot R_H \cdot L \tag{8.4}$$

where C_L is the leakage coefficient (see below), R_H is the hydraulic radius for the flow (see below), and L is the length of the MIKE URBAN pipe (link) in the MIKE SHE cell.

Leakage Coefficient - The leakage coefficient can be defined in two ways.

Option 1 is the simple method, which is to use the pipe leakage coefficient specified in the MIKE URBAN .ADP file. See Telling MIKE URBAN that it is coupled to a MIKE SHE model *(V.1 p. 234)*.

Option 2 uses a combination of the pipe leakage coefficient and the aquifer hydraulic conductivity. In this case, the leakage coefficient is calculated as a series connection of the pipe leakage coefficient (C_p) and the “average” leakage coefficient of the aquifer grid cell (C_{aq}). The average leakage coefficient of the grid cell is calculated assuming that the exchange of water between the pipe and the grid cell is both vertical and horizontal. The leakage coefficient calculation does not calculate a detailed flow path based on a geometric calculation, since a MIKE URBAN pipe can be located anywhere in a grid cell. Instead, an average vertical and horizontal flow distance is used based on 1/4 of the vertical and horizontal cell dimensions. Thus,

$$C_{aq} = C_{aqH} + C_{aqV} = \frac{K_x}{(\Delta x)/4} + \frac{K_z}{(\Delta z)/4} \tag{8.5}$$

where K_x and K_z are the horizontal and vertical hydraulic conductivities respectively and Δx and Δz are the horizontal and vertical cell dimensions.

The final leakage coefficient is then calculated as the harmonic mean of both the aquifer leakage coefficient and the pipe leakage coefficient:

$$\frac{1}{C_L} = \frac{1}{C_{aq}} + \frac{1}{C_p} \tag{8.6}$$

Hydraulic Radius - MIKE SHE uses the inner hydraulic radius if the flow is from MIKE URBAN to MIKE SHE. Whereas, it uses the outer hydraulic radius if the flow is from MIKE SHE to MIKE URBAN. The hydraulic radii are calculated by MIKE URBAN.

MIKE SHE Overland flow to MIKE URBAN LINKS

If a MIKE URBAN link is defined as link type ***CRS*** or ***Natural Channel*** and has a cross section which is "open", then MIKE SHE can exchange overland flow with it in both directions. In this case, the exchange coefficient in Equation (8.1) is defined as

$$C = C_L \cdot L \tag{8.7}$$

where C_L is the conductance and L is the length of the MIKE URBAN pipe (link) in the MIKE SHE cell.

If the exponent Equation (8.1) is 1.0, then this is a simple drain formulation and the conductance is per length with units of [m/s]. If the exponent is 1.5, then this is a weir formulation and the units of the conductance term are [$m^{1/2}$/s].

MIKE SHE Overland flow to MIKE URBAN Manholes

If the MIKE URBAN manholes are not sealed, then MIKE SHE can discharge overland flow into the MIKE URBAN manholes. In this case, the exchange coefficient in Equation (8.1) is defined as

$$C = C_L \tag{8.8}$$

where C_L is the conductance.

If the exponent Equation (8.1) is 1.0, then this is a simple drain formulation and the conductance, C_L, is per length with units of [m/s]. If the exponent is 1.5, then this is a weir formulation and the units of the conductance term are [$m^{1/2}$/s].

MIKE SHE SZ drain flow to MIKE URBAN Manholes

If drain flow is specified in MIKE SHE, then the drainage can be discharged to a MIKE URBAN manhole. The flow in the drain is calculated by MIKE SHE based on the groundwater height above the drain level. In MIKE SHE the distributed drainage option must be chosen (see Drainage *(V.2 p. 173)*) and the cells that drain to a manhole must have an option value of 4 (see Option Distribution *(V.2 p. 179)*). The references between the MIKE SHE drain codes and the MIKE URBAN manholes are defined in the *MsheMouse.pfs* file (see Creating a MsheMouse.pfs file *(V.1 p. 235)*).

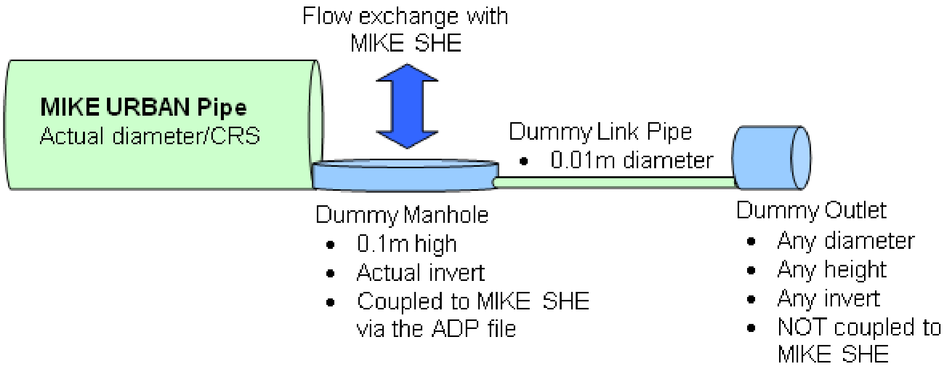
MIKE SHE Paved Areas to MIKE URBAN Manholes

If the paved area option (see Land Use *(V.2 p. 93)*) is used in MIKE SHE, then the flow generated on the paved areas can be discharged to a MIKE URBAN manhole. MIKE SHE's paved area flow module uses the same reference system as the drain component. This option is automatically activated when the MIKE SHE drains in the paved areas point to a MIKE URBAN manhole.

MIKE URBAN Outlets to MIKE SHE

MIKE URBAN outlets cannot directly discharge to MIKE SHE's overland flow. To work around this, you can add a dummy manhole to your MIKE URBAN pipe and then couple the pipe to the outlet via a small diameter dummy pipe (See Figure 8.2). This will force most of the water out of the manhole and into MIKE SHE's overland flow. Downside of this method, is that the head loss at the outlet is over estimated, because the discharge velocity is zero at a manhole.

Figure 8.2 Work around for discharging MIKE URBAN outlets to MIKE SHE



8.1 Coupling MIKE SHE and MIKE URBAN

The MIKE URBAN coupling in MIKE SHE has not yet been added to the MIKE SHE user interface. Thus, to couple the models together, you must:

1 tell MIKE SHE to look for a MIKE URBAN model,

2 tell MIKE URBAN that it is coupled to a MIKE SHE model

3 create an *MsheMouse.pfs* file to define where and how the two models are coupled.

8.1.1 Telling MIKE SHE to couple to MIKE URBAN

To tell MIKE SHE that it needs to couple to a MIKE URBAN model, you must add the following two items in the Extra Parameters *(V.2 p. 193)* section of the MIKE SHE Setup Editor.

Parameter Name	Type	Value
mouse coupling	Boolean	On
mouse coupling file	file name	the file name of the MIKE URBAN coupling .pfs input file

Note, that the parameter names must be spelled exactly as shown. For more information on the use of extra parameters see Extra Parameters *(V.1 p. 299)*.

8.1.2 Telling MIKE URBAN that it is coupled to a MIKE SHE model

To couple a MIKE URBAN model to MIKE SHE, MIKE URBAN must be supplied with some extra information. This information is found in MIKE URBAN’s .ADP file.

Line item	Comment
[MOUSE_COUPLING] SYNTAX_VERSION = 1 UNIT_TYPE = 1 CALLER = 'MSHE'	
// LineHeader = 'ID', 'LinkType', 'C ','OLExp', 'SzLeakageCoef'	Comment line for headers
COUPLINGMMSHE= 'NODE1', 1, 0.001, 2, ,	One line for each coupling item: ID = Link name LinkType = 1 for node; 2 for link C = conductance for Overland flow to MIKE URBAN, units depend on OLExp and whether it is a pipe or a manhole SzLeakageCoeff = leakage coefficient; needed only when the saturated zone is coupled to a link
COUPLINGMMSHE= 'LINK1', 2, 0.001, 2, 0.2	
[Endsect]	

8.1.3 Creating a MsheMouse.pfs file

The MsheMouse.pfs file is an ASCII file that includes all of the specifications for the coupling. The following table defines the structure of the file, along with some information on the parameters. When the MIKE URBAN coupling has been added to the user interface, the creation of this file will be automatic.

Note: The pfs format must be adhered to exactly. There is a small utility (pfsEditor.exe) in the installation \bin directly that you can use for editing and testing pfs files that you create.

Table 8.1 MsheMouse.pfs file format and description

Line item	Comment
[MIKESHE_MOUSE_Specifications] FileVersion = 2	
Link_SZ_Exchange_Option = 2	1 = Leakage coefficient based only on MIKE URBAN pipe leakage coefficient 2 = Leakage coefficient based on a series connection of the MIKE URBAN pipe leakage coefficient and the MIKE SHE aquifer properties
Mouse_MPR_file name = |.\MOUSE_NASSJO\handskeryd.mpr|	**(See Note below this table)** Name of the MOUSE .mpr file or the MIKE URBAN .mex file. The MIKEZero file name format (| |) indicates that the file name is relative to the location of this document.
SZ_Coupling = 1	1 or 0 to include/exclude SZ<->MIKE URBAN coupling
OL_Coupling = 1	1 or 0 to include/exclude Overland<->MIKE URBAN coupling
Dynamic_Coupling = 1	1 for dynamic coupling. Otherwise the initial MIKE URBAN conditions will be used.

Table 8.1 MsheMouse.pfs file format and description

Line item	Comment
Drainage_To_Manholes = 1	1 to include SZ (and paved area) drain to manholes. In this case the SZ drain option must be Levels and Codes (should rather be named Distributed Option). In the areas with drain to MIKE URBAN the Distributed option code must be 4. For each drain code value found in areas with Distributed code 4 a reference from the code to a MIKE URBAN manhole must be defined in the Drainage_Manholes section (see below).
Smooth_SZ_Inflow = 1 Smooth_OL_Inflow = 1	Ensures a more smooth calculation of flows to MIKE URBAN when the MIKE SHE time steps are large compared to the MIKE URBAN time step. The MIKE URBAN coupling is only made at every integer multiple of the MIKE SHE time step. If the Smooth option is not activated, the flows to MIKE URBAN can stop after a number of MIKE URBAN time steps because the calculated flow volume exceeds the volume of the MIKE SHE SZ/Overland grid cells. The Smooth option tries to use a reduced flow rate which equals the available volume / coupling time.
[Dynamic_Coupling_Specifications]	
Limit_Inflow = 0:	Specify 1 if the inflow to MIKE URBAN should be limited so the MIKE URBAN volume + inflow does not exceed a specified fraction of the maximum MIKE URBAN volume. This is used to avoid instabilities due to high pressure.
Limit_Outflow = 0:	Specify 1 if the outflow from MIKE URBAN should be limited so the MIKE URBAN volume - outflow doesn't come below a specified fraction of the maximum MIKE URBAN volume. This is used to avoid instabilities due to drying / negative volume.

Table 8.1 MsheMouse.pfs file format and description

Line item	Comment
[Inflow_Limitations] MaxVolFac_Links = 0.99 MaxVolFac_Manholes = 0.99 EndSect // Inflow_Limitations [Outflow_Limitations] MinVolFac_Links = 0.05 MinVolFac_Manholes = 0.05 EndSect // Outflow_Limitations	The inflow and outflow fractions are specified here:
EndSect // Dynamic_Coupling_Specifications	
No_Of_Storing_reaches = 2 [Storing_Reaches] [Storing_Reach_1] No_Of_Links = 2 LinkName_1 = 'Dike_0111' LinkName_2 = 'Dike_0311' EndSect // Storing_Reach_1 [Storing_Reach_2] No_Of_Links = 1 LinkName_1 = 'Dike_0411' EndSect // Storing_Reach_2 EndSect // Storing_Reaches	When No_Of_Storing_reaches is greater than 0, the [Storing_Reaches] section must be specified, and inside this the [Storing_Reach_1], [Storing_Reach_2], ... defining the no. of links and link names for each reach.
[Drainage_Manholes] No_Of_DrainCodes = 8 [Draincode_1] Draincode= 12 ManholeName='DNB3182' Endsect // Draincode_1 . . Endsect // Draincode_8 EndSect // Drainage_Manholes	When No_Of_Storing_reaches is greater than 0, the [Storing_Reaches] section must be specified, and inside this the [Storing_Reach_1], [Storing_Reach_2], ... defining the no. of links and link names for each reach.
EndSect // MIKESHE_MOUSE_Specifications	

Note on file names:

The pfs file line item is always “Mouse_MPR_file name =”

When coupling MIKE SHE to an old MOUSE model, the MOUSE file name has the extension “.mpr”.

When coupling MIKE SHE to MIKE URBAN, the equivalent file is the “.mex” file. This file contains all the necessary information for the coupling and is generated automatically by MIKE URBAN.

To create the .mex file, you must start a sewer simulation from MIKE URBAN. However, since the .mex file is only created when the simulation is launched, if you make changes to the sewer network, then you must re-create the .mex file by first restarting the sewer simulation in MIKE URBAN. Otherwise, your changes to the sewer network will not be reflected in the coupled models.

8.1.4 Output Files

Output from the coupled run is written to a number of .dfs0 results files- all located in the standard results directory. In the case of storing reaches, there is one item in the .dfs0 file for each storing reach.

Table 8.2 File names and conditions for output for the MIKE SHE-MIKE URBAN coupling. 'setupname' refers to the name of the model setup file.

file name	The file is created when...
.*setupname**setupname*_SZ2MouseReaches.dfs0	...the MIKE SHE SZ coupling is included.
.*setupname**setupname*_OL2MouseReaches.dfs0	...the MIKE SHE Overland coupling is included.
.*setupname**setupname*_OL2MouseManholes.dfs0	...the MIKE SHE Overland flow coupling to manholes is included.
.*setupname**setupname*_SZDrain2MouseManholes.dfs0	...the MIKE SHE SZ drain coupling to manholes is included.
.*setupname**setupname*_PavedDrain2MouseManholes.dfs0	...the MIKE SHE SZ paved areas to manholes is included.

8.2 Warning messages

Exchange inflows reduced

```
Warning: Exchange inflows from Overland to MOUSE
reduced by Overland house-keeping in order to avoid
instabilities
No. of time steps: 27000 of 27000
Total a priori inflows: 1332286 m3
Total reduced inflows:  920643.0 m3 (69.10%)
```

MIKE SHE calculates tine in/out flows after an overland time step and feeds them to MIKE URBAN for one or more MIKE URBAN time steps. The calculations of these flows are not included in the implicit overland flow solver. Thus, the "Total a priori flows" are the rough inflows calculated using Equation (8.1). However, to prevent water balance errors, MIKE SHE checks the volume of water available in the grid cell. If the volume is insufficient, then the inflow is reduced to the available amount. The final value of inflows is the “Total reduced inflows”. Note though that the total NET inflow to MIKE URBAN will be less than this value if the flow goes from MIKE URBAN to MIKE SHE in other grid cells or other time steps.

Ideally, the Total reduced inflow should be 100%, but in practice this is rarely achieved.

8.3 Water Balance Limitations

The interaction with MIKE SHE is not included in the MIKE URBAN Summary HTM file. Thus, the water added from MIKE SHE appears as an error (i.e. 6: Continuity balance in MIKE Urban).

GROUNDWATER

9 UNSATURATED GROUNDWATER FLOW

Unsaturated flow is one of the central processes in MIKE SHE and in most model applications. The unsaturated zone is usually heterogeneous and characterized by cyclic fluctuations in the soil moisture as water is replenished by rainfall and removed by evapotranspiration and recharge to the groundwater table. Unsaturated flow is primarily vertical since gravity plays the major role during infiltration. Therefore, unsaturated flow in MIKE SHE is calculated only vertically in one-dimension.

Vertical, 1D unsaturated flow is sufficient for most applications. However, this assumption may not be valid in some situations, such as on very steep hill slopes with contrasting soil properties in the soil profile.

MIKE SHE includes an iterative coupling procedure between the unsaturated zone and the saturated zone to compute the correct soil moisture and the water table dynamics in the lower part of the soil profile.

There are three options in MIKE SHE for calculating vertical flow in the unsaturated zone:

- the full Richards equation, which requires a tabular or functional relationship for both the moisture-retention curve and the effective conductivity,
- a simplified gravity flow procedure, which assumes a uniform vertical gradient and ignores capillary forces, and
- a simple two-layer water balance method for shallow water tables.

The full Richards equation is the most computationally intensive, but also the most accurate when the unsaturated flow is dynamic. The simplified gravity flow procedure provides a suitable solution when you are primarily interested in the time varying recharge to the groundwater table based on actual precipitation and evapotranspiration and not the dynamics in the unsaturated zone. The simple two-layer water balance method is suitable when the water table is shallow and groundwater recharge is primarily influenced by evapotranspiration in the root zone.

9.0.1 UZ Classification

Calculating unsaturated flow in all grid squares for large-scale applications can be time consuming. To reduce the computational burden MIKE SHE allows you to optionally compute the UZ flow in a reduced subset of grid squares. The subset classification is done automatically by the preprocessing program according to soil and, vegetation distribution, climatic zones, and depth to the groundwater table.

Column classification can decrease the computational burden considerably. However, the conditions when it can be used are limited. Column classification is either not recommended or not allowed when

- the water table is very dynamic and spatially variable because the classification is not dynamic,
- if the 2 layer UZ method is used because the method is fast and the benefit would be limited,
- if irrigation is used in the model because irrigation zones are not a classification parameter, and
- if flooding and flood codes are used, since the depth of ponded water is not a classification parameter

Thus, **the column classification should probably be avoided today** because the models have become more complex, MIKE SHE has become more efficient and computers have become faster.

If the classification method is used, then there are three options for the classification:

- **Automatic classification** The automatic classification requires a distribution of groundwater elevations (see Groundwater Depths used for UZ Classification). This can be either the initial depth to the groundwater based on the initial heads, or you can supply a .dfs2 map of the groundwater elevations. In both cases, you must supply a table of intervals upon which the classification will be based. The number of computational columns depends on how narrow the intervals are specified. If, for example, two depths are specified, say 1 m and 2 m, then the classification with respect to the depth to groundwater will be based on three intervals: Groundwater between 0 m and 1 m, between 1 m and 2 m, and deeper than 2 m.

  One tip is to extract a map of the calculated potential head in the very upper saturated zone layer from a previous simulation. The map should represent the time of the year when the largest variations of the groundwater table are expected (deep groundwater in the hills and shallow groundwater close to the rivers). Repeat the procedure as calibration improves.

  If the Linear Reservoir method is used for the groundwater, then the Interflow reservoirs are also used in the classification. However, since feedback to the UZ only occurs in the lowest Interflow reservoir of each subcatchment, the Interflow reservoirs are added to the Automatic

Classification in two zones - those that receive feedback and those that don’t.

- **Specified classification** Alternatively a data file specifying Integer Grid Codes, where UZ computations are carried out can be specified, with grid codes range from 2 up to the number of UZ columns (see Specified classification). The location of the computational column is specified by a negative code and the simulation results are then transferred to all grids with the an equivalent positive code. For example, if a grid code holds the value -2 a UZ computation will be carried out for the profile located in that grid. Simulation results will subsequently be transferred to all grid codes with code value 2. An easy way to generate a .dfs2 file to be used for specification of UZ computational columns is to let the MIKE SHE setup program generate an automatic classification first, and subsequently extract the UZ classification grid codes. The extracted .dfs2 file can be edited in the 2D editor as desired and used to specify UZ computational grids.
- **Calculated in all Grid points (default)** For most applications you should specify that computations are to be carried out in all soil columns.
- **Partial Automatic** Finally a combination of the Automatic classification and the Specified classification is available. If this option is chosen an Integer Grid Code file must be provide (see Partial automatic classification) with the following grid codes: In grid points where automatic classification should be used the grid code 1 must be given. In grid points where computation should be performed for all cells the grid code 2 must be given.

9.0.2 Coupling Between Unsaturated and Saturated Zone

The following procedure should be used to ensure that the unsaturated zone does not drop below the bottom of the first calculation layer of the saturated zone:

- After a simulation, create a map of grid statistics of the potential head in the first calculation layer of the saturated zone
- Subtract the map of the minimum potential head from the map of the bottom level of the first calculation layer of the saturated zone.
- View the difference map. If the difference is very small in some areas of the map (e.g. <0.5 m), it is strongly advised to move the bottom level of the first calculation layer of the saturated zone downwards.
- Repeat this procedure until there are no small differences.

The water balance program can be used to get an overview of errors due to a bad setup of the unsaturated zone. The follow procedure can be used to make a map of UZ-errors:

- Create a sub catchment map by retrieving UZ-classification codes from the input file.
- Replace negative values of the classification code map by positive values in the 2D graphical editor.
- Use the sub catchment map in the water balance setup file to make a UZ map of the water balance, which will create your map of UZ-errors.

Vertical discretisation - The vertical discretisation of the soil profile typically contains small cells near the ground surface and increasing cell thickness with depth. However, the soil properties are averaged if the cell boundaries and the soil boundaries do not align. .

The discretisation should be tailored to the profile description and the required accuracy of the simulation. If the full Richards equation is used the vertical discretisation may vary from 1-5 cm in the uppermost grid points to 10-50 cm in the bottom of the profile. For the Gravity Flow module, a coarser discretisation may be used. For example, 10-25 cm in the upper part of the soil profile and up to 50-100 cm in the lower part of the profile. Note that at the boundary between two blocks with different cell heights, the two adjacent boundary cells are adjusted to give a smoother change in cell heights.

Specific Yield of upper SZ layer

MIKE SHE forces the specific yield of the top SZ layer to be equal to the "specific yield" of the UZ zone as defined by the difference between the specified moisture contents at saturation, θ_s, and field capacity, θ_{fc}.This correction is calculated from the UZ values in the UZ cell in which the initial SZ water table is located. For more information on the SZ-UZ specific yield see Specific Yield of the upper SZ numerical layer *(V.1 p. 252)*.

Limitations of the UZ - SZ coupling

The coupling between UZ and SZ is limited to the top calculation layer of the saturated zone. This implies that:

- As a rule of thumb, the UZ soil profiles should extend to just below the bottom of the top SZ layer.
- However, if you have a very thick top SZ layer, then the UZ profiles must extend at least to below the deepest depth of the water table.

- If the top layer of the SZ model dries out, then the UZ model usually assumes a lower pressure head boundary equal to the bottom of the uppermost SZ layer.
- All outflow from the UZ column is always added to the top node of the SZ model.
- UZ nodes below the water table and the bottom of the top SZ layer are ignored.

10 SATURATED GROUNDWATER FLOW

The saturated groundwater component of MIKE SHE includes all of the water below the water table. If the water table is at or above the ground surface then the unsaturated zone is turned off this this cell.

The unsaturated zone geology is not related to the saturated zone geology. Instead the unsaturated zone geology is essentially independent of the saturated zone geology.

10.1 Conceptualization of the Saturated Zone Geology

The development of the geological model is probably the most time consuming part of the initial model development. Before starting this task, you should have developed a conceptual model of your system and have at your disposal digital maps of all of the important hydrologic parameters, such as layer elevations and hydraulic conductivities.

In MIKE SHE you can specify your subsurface geologic model independent of the numerical model. The parameters for the numerical grid are interpolated from the grid independent values during the preprocessing.

The geologic model can include both geologic layers and geologic lenses. The former cover the entire model domain and the later may exist in only parts of your model area. Both geologic layers and lenses are assigned geologic parameters as either distributed values or as constant values.

The alternative is to define the hydrogeology based on geologic units. In this case, you define the distribution of the geologic units and the geologic properties are assigned to the unit.

Each geologic layer can be specified using a dfs2 file, a .shp file or a distribution of point values. However, you should be aware of the way these different types of files are interpolated to the numerical grid.

The simplest case is that of distributed point values. In this case, the point values are simply interpolated to the numerical grid cells based on the available interpolation methods.

In the case of shp files, at present, only point and line theme .shp files are supported. Since lines are simply a set of connected points, the .shp file is essentially identical to the case of distributed point values. Thus, it is interpolated in exactly the same manner.

The case of .dfs2 files is in fact two separate cases. If the .dfs2 file is aligned with the model grid then the cell value that is assigned is calculated using the bilinear method with the 4 nearest points to the centre of the cell. If the .dfs2 file is not aligned with the model grid then the file is treated exactly the same as if it were a .shp file or a set of distributed point values.

The geologic model is interpolated to the model grid during preprocessing, by a 2 step process.

1 **The horizontal geologic distribution is interpolated to the horizontal model grid.** If Geologic Units are specified then the integer grid codes are used to interpret the geologic distribution of the model grid. If distributed parameters are specified then the individual parameters are interpolated to the horizontal model grid as outlined above.

2 **The vertical geologic distribution is interpolated to the vertical model grid.** In each horizontal model grid cell, the vertical geologic model is scanned downwards and the soil properties are assigned to the cell based on the average of the values found in the cell weighted by the thickness of each of the zones present. Thus, for example, if there were 3 different geologic layers in a model cell each with a different Specific Yield, then the Specific Yield of the model cell would be

$$S_y = \frac{S_{y1} \cdot z_1 + S_{y2} \cdot z_2 + S_{y3} \cdot z_3}{z_1 + z_2 + z_3} \tag{10.1}$$

where z is the thickness of the geologic layer within the numerical cell.

Conductivity values

Hydraulic conductivity is a special parameter because it can vary by many orders of magnitude over a space of a only few meters or even centimeters. This necessitates some special interpolation strategies.

Horizontal Interpolation - The horizontal interpolation of hydraulic conductivity interpolates the raw data values. Thus, in Step 1 above, when interpolating point values that range over several orders of magnitude, such as hydraulic conductivity, the interpolation methods will strongly weight the larger values. That is, small values will be completely overshadowed by the large values.

In fact, the interpolation in this case should be done on the logarithm of the value and then the cell values recalculated. Until this option is available in the user interface, you should interpolate conductivities outside of MIKE SHE using, for example, Surfer. Alternatively, the point values could be input as logarithmic values and the Grid Calculator Tool in the MIKE SHE Toolbox can be used to convert the logarithmic values in the .dfs2 file to conductivity values.

Vertical Interpolation - In Step 2 above, the geologic model is scanned down and interpreted to the model cell. Although, horizontal conductivity can vary by several orders of magnitude in the different geologic layers that are found in a model cell, the water will flow horizontally based on the highest transmissivity. Thus, the averaging of horizontal conductivity can be down the same as in the example for Specific Yield above. Vertical flow, however, depends mostly on the lowest hydraulic conductivity in the geologic layers present in the model cell. In this case a harmonic weighted mean is used instead. For a 3 layer geologic model in one model cell, the vertical conductivity would be calculated by

$$K_z = \frac{z_1 + z_2 + z_3}{\frac{z_1}{K_{z1}} + \frac{z_2}{K_{z2}} + \frac{z_3}{K_{z3}}} \tag{10.2}$$

where z is the thickness of the geologic layer within the numerical cell.

10.1.1 Lenses

In building a geologic model, it is typical to find discontinuous layers and lenses within the geologic units. The MIKE SHE setup editor allows you to specify such units - again independent of the numerical model grid.

Lenses are also a very useful way to define a complex geology. In this case, the lenses are used to define the subsurface geologic units within a larger regional geologic unit.

Lenses are specified by defining either a .dfs grid file or a polygon .shp file for the extents of the lenses. The .shp file can contain any number of polygons, but the user interface does not use the polygon names to distinguish the polygons. If you need to specify several lenses, you can use a single file with many polygons and specify distributed property values, or you can specify multiple individual polygon files, each with unique property values.

In the case of lenses, an extra step is added to the beginning of the 2-step process outlined in the previous section. The location of the lenses is first interpolated to the horizontal numerical grid. Then the lenses become essentially extra geologic layers in the columns that contain lenses. However, there are a number of special considerations when working with lenses in the geologic model.

- **Lenses override layers -** That is, if a lense has been specified then the lense properties take precedence over the layer properties and a new geologic layer is added in the vertical column.
- **Vertically overlapping lenses share the overlap -** If the bottom of lense is below the top of the lense beneath, then the lenses are assumed to meet in the middle of the overlapping area.
- **Small lenses override larger lenses -** If a small lense is completely contained within a larger lense the smaller lense dominates in the location where the small lense is present.
- **Negative or zero thicknesses are ignored -** If the bottom of the lense intersects the top of the lense, the thickness is zero or negative and the lense is assumed not to exist in this area.

10.2 Numerical Layers

10.2.1 Specific Yield of the upper SZ numerical layer

The specified value for specific yield is not used for the specific yield of the upper most SZ numerical layer if UZ is included in the simulation.

By definition, the specific yield is the amount of water release from storage when the water table falls. The field capacity of a soil is the remaining water content after a period of free drainage. Thus, specific yield is equal to the saturated water content minus the field capacity.

To avoid water balance errors at the interface between the SZ and UZ models, the specific yield of the top SZ layer is set equal to the he saturated water content minus the field capacity. The value is determined once at the beginning of the simulation. The water content parameters are taken from the UZ layer in which the initial SZ water table is located.

In principle, having different values between the SZ and UZ models does not directly cause a water balance error, but it may cause numerical problems that could lead to water balance errors. By definition, the steady-state water table location will be identical in both the SZ and UZ models. Pumping from the SZ will lower the SZ water table by an amount equal to the specific yield divided by the cell area times the pumping rate. However, if the field capacity is not correlated to the specific yield, then the amount of water released from storage in the UZ will be more or less than the amount extracted from the SZ cell. This will result in different water tables in the SZ and UZ models. If pumping stops, the system will again reach an equilibrium with the same water table in both the SZ and UZ simply because of the pressure head redistribution.

As mentioned, the upper Sy value is calculated only at the beginning of the simulation based on the UZ layer in which the initial SZ water table is located. If the soil profile has multiple soil types with different field capacities and saturated water contents, then the specific yield in the SZ and UZ model may diverge during the simulation. With slowly moving water tables, the differences may not be that large and the errors generated will likely be tolerable. If the water table drops into a lower SZ layer, then the specified Sy will be used. ,

The actual value used in the model is displayed in the pre-processed tab under Specific Yield.

10.2.2 SZ Boundary Conditions

The upper boundary of the top layer is always either the infiltration/exfiltration boundary, which in MIKE SHE is calculated by the unsaturated zone component or a specified fraction of the precipitation if the unsaturated zone component is excluded from the simulation.

The lower boundary of the bottom layer is always considered as impermeable.

In MIKE SHE, the rest of the boundary conditions can be divided into two types: Internal and Outer. If the boundary is an outer boundary then it is defined on the boundary of the model domain. Internal boundaries, on the

10.3 Groundwater Drainage

Saturated zone drainage is a special boundary condition in MIKE SHE used to defined natural and artificial drainage systems that cannot be defined in MIKE 11. It can also be used to simulate simple overland flow, if the overland flow system can be conceptualized as a shallow drainage network connected to the groundwater table - for example, on a flood plain.

Saturated zone drainage is removed from the layer of the Saturated Zone model containing the drain level. Water that is removed from the saturated zone by drains is routed to local surface water bodies, local topographic depressions, or out of the model.

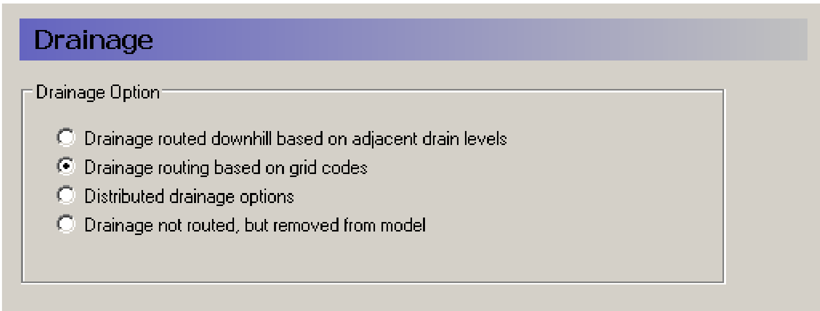
When water is removed from a drain, it is immediately moved to the recipient. In other words, the drain module assumes that the time step is longer than the time required for the drainage water to move to the recipient. This is the same as a “full pipe”. That is, water added to the end of a full pipe of water causes an equal amount of water to immediately flow out the opposite end - regardless of the length of the pipe.

Drain flow is simulated using an simple linear reservoir formula. Each cell requires a drain level and a time constant (leakage factor). Both drain levels and time constants can be spatially defined. A typical drainage level is 1m below the ground surface and a typical time constant is between 1e-6 and 1 e-7 1/s.

Drainage reference system

MIKE SHE also requires a reference system for linking the drainage to a recipient node or cell. The recipient can be a MIKE 11 river node, another SZ grid cell, or a model boundary.

There are four different options for setting up the drainage source-recipient reference system



Drainage routed downhill based on adjacent drain levels

This option was originally the only option in MIKE SHE. The reference system is created automatically by the pre-processor using the slope of the drains calculated from the drainage levels in each cell.

Thus, the pre-processer calculates the drainage source-recipient reference system by

1 looking at each cell in turn and then

2 look for the neighbouring cell with the lowest drain level.

3 If this cell is an outer boundary cell or contains a river link, the search stops.

4 If this cell does not contain a boundary or river link, then the search is repeated with the next downstream neighbour until either a local minimum is found or a boundary cell or river link is found.

The result of the above search for each cell is used to build the source-recipient reference system.

If local depressions in the drainage levels exist, the SZ nodes in these depressions may become the recipients for a number of drain flow producing nodes. This often results in the creation of a small lake at such local depressions. If overland flow is simulated, then the ponded drainage water will become part of the local overland flow system.

If the drain level equals the topography, drainage will be turn off in that cell. Likewise, drain levels above the topography are not allowed. In this case a warning will be written to the PP_Print.log and the drain level will be automatically adjusted to a value just below the topography.

The drain level method is not allowed when using Time varying drainage parameters *(V.1 p. 320)* because the source-recipient reference system is only calculated once at the beginning of the simulation.

The drain-slope based reference system has been used in MIKE SHE for many years and works well in most situations. However, when MIKE SHE is applied where there is very little surface topographic relief, it is often difficult to establish a suitable reference system based on the surface topography/drain slopes. For example, often it is assumed that the drains are located 50 to 100 cm below the terrain. In flat areas, this may generate many undesired local depressions, which may receive drainage water from a large area, thus generating lakes in places where there should not be a lake.

If the drain level is perfectly flat, drainage is turned off. In other words, if the drain-slope method cannot find a downhill neighbour because all the neighbours have the same elevation as the cell, the drain slope method assumes that the cell is a local depression. However, the depression has no sources of drainage except itself. Thus, the drainage function is effectively turned off.

Tip: MIKE SHE considers a grid point to be a local depression even if the drainage level in the four surrounding model grids is only 1 mm higher. The only way to avoid such problems is to create a drain level map that does not contain “wrong” local depressions. For large models this may be difficult and time consuming. In this case, one of the other drainage options may be better.

Remember, the drainage is routed to a destination. It does not phyisically flow downhill. The drain levels are only used to build the drainage source-recipient reference system, and to calculate the amount of drainage.

Drainage routing based on grid codes

This method is often used when the topography is very flat, which can result in artificial depressions, or when the drainage system is very well defined, such as in agricultural applications.

In this method, the drainage levels and the time constants are defined as in the previous method and the amount of drainage is calculated based on the drain levels and the time constant.

If the drainage routing is specified by Drain Codes, a grid code map is required that is used to restrict the search area for the source-recipient reference system. In this case, the pre-processer calculates the reference system within each grid code zone, such that all drainage generated within one zone is routed to recipient nodes with the same drain code value.

When building the reference system, the pre-processor looks at each cell and then

1 looks for the nearest cell with a river link with the same grid code value,

2 if there is no cells with river links, then it looks for the nearest outer boundary cell with the same grid code,

3 if there are no cells with outer boundary conditions, then it looks for the cell with the same grid code value that has the lowest drain level. In this case, the reference system is calculated as if it was based on Drain Levels (see previous section).

The result of the above search for each cell is used to build the source-recipient reference system.

The above search algorithm is valid for all **positive** Drain Code values. However, all cells where

Drain Code = 0 - will not produce any drain flow and will not receive any drain flow, and

Drain Code < 0 (negative) - will not drain to river links, but will start at Step 2 above and only drain to either a outer boundary or the lowest drain level.

Tip: One method that is often used is to specify only one Drain Code value for the entire model area (e.g. Drain Code = 1). Thus, all nodes can drain and any drain flow is routed to the nearest river link. If there are no rivers, the drain flow will be routed to the nearest boundary. If you want to route all drain flow to the boundaries instead of the rivers, a negative drain code can be specified for the entire area (e.g. Drain Code = -1).

Distributed drainage options

Choosing this method, adds the Option Distribution item to the data tree. With the Option Distribution, you can specify an integer grid code distribution that can be used to specify different drainage options in different areas of your model.

Code = 1 - In grid cells with a value of 1, the drainage reference system is calculated based on the Drain Levels.

Code =2 - In grid cells with a value of 2, the drainage reference system is calculated based the Drain Codes.

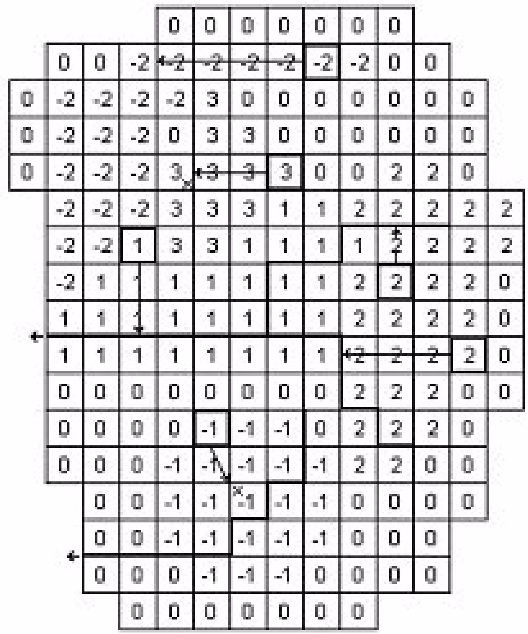
Code = 3 - Drainage in grid cells with a value of 3 is routed to a specified MIKE 11 branch and chainage. At the moment, this options requires the use of Extra Parameters *(V.2 p. 193)* and is described in SZ Drainage to Specified MIKE 11 H-points *(V.1 p. 316)*.

Code = 4 - Drainage in grid cells with a value of 4 is routed to a specified MOUSE man hole. At the moment, this options requires the use of Extra Parameters *(V.2 p. 193)* and is described in the section Using MIKE SHE with MIKE URBAN *(V.1 p. 229)*.

Drain flow not routed, by removed from model

The fourth option is simply a head dependent boundary that removes the drainage water from the model. This method does not involve routing and is exactly the same as the MODFLOW Drain boundary.

Drain Code Example



- The grid cells with Drain Code 3 drain to a local depression since no boundary or river link is found adjacent to a grid with the same drain code.
- The grid cells with Drain Code 1 or 2 drain to nearest river link located adjacent to a grid with the same drain code.
- The grid cells with drain code 0 do not contain drains and thus no drainage is produced.
- The grid cells with Drain Code -1 drains to local depression since no boundary is found adjacent to a grid with the same drain code.
- The grid cells with Drain Code -2 drains to nearest boundary grid with the same drain code.

The Pre-processed Drainage Reference System

During the preprocessing, each active drain cell is mapped to a recipient cell. Then, whenever drainage is generated in a cell, the drain water will always be moved to the same recipient cell. The drainage source-recipient reference system is displayed in the following two grids in the Pre-processed tab, under the Saturated Zone:

Drain Codes - The value in the pre-processed Drain Codes map reflects the Option Distribution specified. For example, those cells with an Option Distribution equal to 1 (Drainage routed based on Drain Levels) will have a pre-processed Drain Code equal to 0, because the Drain Codes are not being used for those cells.

Drainage to local depressions and boundary - This grid displays all the cells that drain to local depressions or to the outer boundaries. All drainage from cells with the same negative value are drained to the cell with the corresponding positive code. If there is no corresponding positive code, then that cell drains to the outer boundary, and the water is simply removed from the model. Cells with a delete value either do not generate drainage, or they drain to a river link.

Drainage to river - This grid displays the river link number that the cell drains to. Adjacent to the river links, the cells are labeled with negative numbers to facilitate the interpretation of flow from cells to river links. Thus, in principle, all drainage from cells with the same positive code are drained to the cell with the corresponding negative code.

However, this is slightly too simple because the cells actually drain directly to the river links. In complex river systems, when the river branches are close together, you can easily have cells connected to multiple branches on different sides. In this case, the river link numbers along the river may not reflect the drainage-river link reference used in the model.

If you want to see the actual river links used in all cells, you can use the Extra Parameter, SZ Drainage River Link Reference Table *(V.1 p. 321)*, to generate a table of all the river link-cell references in the PP_Print.log file.

Cells with a value of zero either do not generate drainage, or they drain to a the outer boundary or a local depression.

10.3.1 Saturated Zone drainage + Multi-cell Overland Flow

The topography is often used to define the SZ drainage network. Thus, a refined topography more accurately reflects the SZ drainage network.

The SZ drainage function uses a drain level and drain time constant. The drain level defines the depth at which the water starts to drain. Typically, this is set to some value below the topography to represent the depth of surface drainage features below the average topography. This depth should probably be much smaller if the topography is more finely defined in the sub-grid model. The drain time constant reflects the density of the drainage network. If there are a lot of drainage features in a cell then the time constant is higher and vice versa.

When using the multi-cell OL, the drainage system is updated in the sense that the drain level will be defined using the sub-scale topography information. The SZ drainage will include the following when using sub-scale:

- Multi-scale SZ drainage supported only in the PCG transient SZ solver
- Each sub-grid cell will have the same drain time constant defined by the value in the coarse grid.
- If the drain level is defined as an elevation, then all sub-grids will have the same drain level.
- If the drain level is defined by depth below the surface, then each sub-grid may have a unique drain level, since each sub-grid can have a different "Each coarse grid cell has a water table that is common for all fine scale grids within the coarse grid.
- If the coarse cell water table is above the fine scale drain level, then drainage is calculated based on the drain time constant and the depth of water above the fine scale drain level.
- Total drainage in a coarse cell is the sum of all the fine scale drainage volumes.
- Drainage routing by levels will be determined by the coarse grid. However to make it more realistic with respect to the fine scale hydrology, the drainage routing by levels will be based on the lowest drain level in a coarse cell.
- Drainage to local depressions will be added to the SZ cell, and resultant ponding will then follow the multi-scale OL flow.

Internal validation of the drainage scheme

MIKE SHE performs an internal validation of the SZ drainage scheme. The following are used in connection with the sub-scale feature:

- Drainage depths of zero are allowed and drainage depths above the topography are set to the topography. This allows drain levels at the ground surface. This check will be done on the coarse grid. That is, if the coarse grid drain level is above the coarse grid topography, a warning will be issued and all the sub-grid drain depths will be set to zero.

  Note for Release 2011 In Release 2011 and prior releases, a drain level of zero turned off SZ drainage, and drain levels above topography were set to the topography (and turned off drainage). For backwards compatibility an Extra Parameter is available.

Parameter Name	Type	Value
disable drains at or above ground	Boolean	On

Drain levels vs River link elevations There is an optional Extra Parameter check in the drainage routing by levels that checks on the river link bottom elevation.

Parameter Name	Type	Value
check drain level against bed level	Boolean	On

If the river link bottom elevation is higher than the drain level, the cell becomes a local depression. However, this will likely create a lot of local depressions beside the rivers.

When using the multi-grid OL option, the drainage in a coarse cell is controlled by the minimum drainage level in the cell. If one sub-grid cell has a drainage level below the bed level then the drainage in the entire cell is transferred to an internal depression.

Note for Release 2011 The check was originally added to prevent the "lifting" of drainage water up to a river link. However, in most cases, such lifting is probably unintentional. That is, the river bed has been poorly interpolated. Prior to Release 2012, this was the default behaviour and the check above has been added for backwards compatability.

- There is a check on the drain levels below the bottom of the model. If the coarse grid drain level is below the coarse grid bottom of the model, then a warning will be printed and the drain level will be adjusted to the bottom of the model. In the sub-grids, you may have the situation where the sub-grid drain level is below the bottom of the model, but the average drain level is above. In this case, the sub-grid drain level will be the maximum elevation of the bottom of the model and the drain level. Meaning if the drain level of a sub-grid is below the bottom of the model, the drain level is adjusted to the maximum value of i) the bottom of the model and ii) the drainage elevation.

Disabling Multi-Cell Drainage

By default, when if the multi-cell OL option is invoked, multi-cell drainage will be active. If you want to disable multi-cell drainage, perhaps for backwards compatability with older models, an Extra parameter option is availble to switch off multi-cell drainage: .

Parameter Name	Type	Value
disable multi-cell drainage	Boolean	On

If this option is used, then the multi-cell drainage is switched off and the drainage will function using the groundwater level and drain level based on the course cells.

10.4 MIKE SHE versus MODFLOW

The MIKE SHE can be used to simulate all of the processes in the land phase of the hydrologic cycle, including overland flow, channel flow, groundwater flow in the unsaturated zone and saturated groundwater flow. MODFLOW, on the other hand, is restricted to simulating flow only in the saturated groundwater zone. Although many of the processes simulated in MIKE SHE are used in a similar way when simulating groundwater flow with MODFLOW, they are not actually “simulated” by MODFLOW.

Let’s take groundwater recharge as an example. MODFLOW allows you to include recharge as an upper boundary condition to the groundwater model, where recharge is defined as the amount of water reaching the groundwater table after accounting for evapotranspiration, surface runoff and changing storage in the unsaturated zone. In MODFLOW, the modeller has to account for these processes herself - usually by applying a constant rule-of-thumb fraction to the measured precipitation data. In most cases, the model results are very sensitive to this fraction and since the modeller has little data on this fraction, she will assume an initial value and use this parameter as a calibration parameter. Thus, she will adjust the amount of recharge during the calibration process until the measured groundwater levels match the calculated values.

However, the fraction of precipitation reaching the groundwater table is constant in neither space nor time. The actual amount of precipitation reaching the groundwater table depends strongly on the maximum rate of infiltration, which is a characteristic of the soil and will vary spatially over the model domain. Further, since the maximum rate occurs when the soil is saturated, different amounts of water will infiltrate during wet periods compared to dry periods. To complicate matters further, the length of the preceding dry period will determine the amount of available storage in the unsaturated zone. For example, if there has been a long dry summer period, then evapotranspiration may have created a large deficit of water in the unsaturated zone that must be satisfied before any water reaches the water table.

This example shows that infiltration of precipitation is a very dynamic process. It depends on a complex interaction between precipitation, unsaturated zone soil properties and the current soil moisture content, as well as vegetation properties.

In MIKE SHE, the saturated zone is only one component of an integrated groundwater/surface water model. The saturated zone interacts with all of the other components - overland flow, unsaturated flow, channel flow, and evapotranspiration.

In comparison, MODFLOW only simulates the saturated flow. All of the other components are either ignored (e.g. overland flow) or are simple boundary conditions for the saturated zone (e.g. evapotranspiration).

On the other hand, there are very few difference between the MIKE SHE Saturated Zone numerical engine and MODFLOW. In fact, they share the same PCG solver. The differences that are present are limited to differences in the discretisation and to some differences in the way boundary conditions are defined.

Setting up the saturated zone hydraulic model involves defining the:

- the geological model,
- the vertical numerical discretisation,
- the initial conditions, and
- the boundary conditions.

In the MIKE SHE GUI, the geological model and the vertical discretisation are essentially independent, while the initial conditions are defined as a property of the numerical layer. Similarly, subsurface boundary conditions are defined based on the numerical layers, while surface boundary conditions such as wells, drains and rivers (using MIKE 11) are defined independently of the subsurface numerical layers.

The use of grid independent geology and boundary conditions provides a great deal of flexibility in the development of the saturated zone model. Thus the same geological model and many of the boundary conditions can be re-used for different model discretisation and different model areas.

10.4.1 Importing a MODFLOW 96 or MODFLOW 2000 Model

A FORTRAN executable is automatically installed with MIKE SHE and located in the MIKE SHE bin directory. The program can be used to read a MODFLOW file set and extract the stationary distributed data to a set of point theme shape files. The shp files can then be used directly in MIKE SHE.

To extract data from a MODFLOW model, open a command prompt in the directory containing the input files. On the command prompt line, type

```
MShe_ModflowExtraction.exe file_name.pfs
```

The extraction will proceed silently - that is without any messages. To run the extraction with the messages, you need to use

```
MZLaunch file_name.pfs -e  MShe_ModflowExtraction.exe
```

which will start the MZLaunch utility. The `file_name.pfs` variable is the input file for the MODFLOW extractor. The input file has the standard MIKEZero Pfs format. The input fields of the file are explained below. Lines starting with '//' are not read, but rather can be used as comment lines.

Table 10.1 is an example .pfs file for the MODFLOW data extractor program:

Table 10.1 MODFLOW Extraction.pfs file format and description

Line item	Comment
[MIKESHE_ModflowExtraction] FileVersion = 3	File version 3 is for Release 2009 and up
ModflowModel = 'MODFLOW-96' \\ModflowModel = = 'MODFLOW-2000'	The ModflowModel variable should be changed to MODLFOW-2000, if the MODFLOW model is a MODFLOW 2000 model.

Table 10.1 MODFLOW Extraction.pfs file format and description

Line item	Comment
NameFileName = |.\Airport5.nam|	The NameFileName is the name of the MODFLOW name file that contains all of the references to the other input files. The '|' around the name-file name and the path of the specified file name must be relative to the location of the pfs file.
XMin = 300. YMin = 400. XMax = 3032. YMax = 1132	The minimum and maximum (X,Y) coordinates are used to determine the exact spatial coordinates of the nodal points. XMin and YMin are the UTM coordinates of lower left MODFLOW corner. Xmax and Ymax are the UTM coordinates of the upper right MODFLOW corner. See figure next page.
TimeUnit = 'DAYS'	The TimeUnit is not currently used, but must be input. Valid values for TimeUnit are DAYS, HOURS, MINUTES and SECONDS.
LengthUnit = 'METER'	The LengthUnit is not currently used, but must be input. Valid values for LengthUnit are METER and FEET.
StartDate = 2005,1,1,0,0	The start date and time of the MODFLOW simulation. Format: YYYY, MM, DD, HH, MM
WellExtraction = 1	Extract well data to a dfs0 file. On: Flag = 1 Off: Flag = 0
RechargeExtraction = 1	Extract recharge input to a dfs2 file. On: Flag = 1 Off: Flag = 0 Note: only works with uniform MODFLOW grids.

Table 10.1 MODFLOW Extraction.pfs file format and description

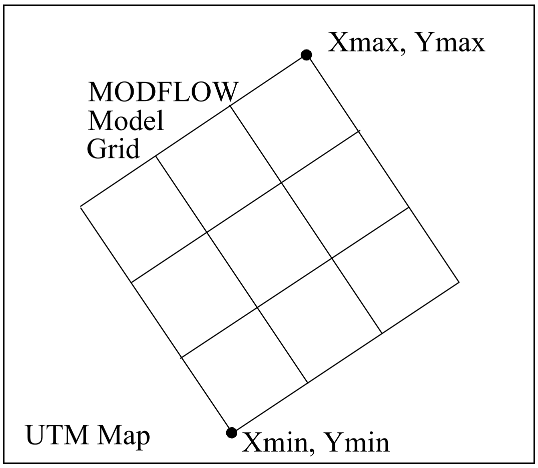
Line item	Comment
HeadExtraction = 1	Extract head results to a dfs2 file. On: Flag = 1 Off: Flag = 0 Note: only works with uniform MODFLOW grids
EndSect // MIKESHE_ModflowExtraction	

Note MODFLOW does not have any internal unit checking. The units written in the MODFLOW file are only for display purposes. Also, the units that you define in your MODFLOW user interface may not be the same as those written to the MODFLOW files. So, you need to be careful of units and know what units the MODFLOW files are written in.

The MODFLOW name file has the usual MODFLOW format. However, you should

- Specify a new name for the LIST file in order not to overwrite the LIST file of an existing simulation, and
- Make copies of, or rename, all output files (lines starting with DATA). Existing result files might otherwise be overwritten during the execution of the extraction routine.

The coordinate information is the UTM coordinates of the lower left and upper right MODFLOW model corners - not the MODFLOW block-centered nodal coordinates.



These coordinates plus the DELR, DELC vectors from the MODFLOW files are used to defined the spatial location of the shape file and dfs2 output.

For a MODFLOW model, the extraction routine reads and outputs the following MODFLOW static parameters to a point theme shape file:

Top, Bot, Shead, Tran, Hy, Vcont, Sf1, and Sf2

Plus, it outputs the Specific storage, which is calculated as Sf1 divided by the layer thickness.

If the well output option is selected, a dfs0 file will be created. In this file, every cell in the MODFLOW file containing a well will have a seperate item in the dfs0 file.

If the recharge data and head results is selected, a dfs2 file will be created for each of these. However, the dfs2 format does not allow for variable grid spacing, which means that variable grid spacing will be ignored. The DELR and DELC for the first column and row will be used as the grid spacing in the dfs2 file. Thus, the recharge and head results output option is really only useful for MODFLOW models with a uniform grid spacing.

The extraction routine outputs point theme shape files -one file per data type - with one item for each extracted layer. The shape file names reflect the MODFLOW manual naming convention (Top.shp, Vcont.shp, etc.). The points represent the centre of each grid square. The model orientation is calculated from the user-specified coordinates of lower left (origin) and upper right corner of the model.

To use the MODFLOW data in MIKE SHE, select the Point/Line .shp option for the static variable. Then browse to the appropriate .shp file. The .shp file will contain one item for each model layer in the MODFLOW model. The appropriate item is selected in the file browse dialogue. Once the file has been assigned, MIKE SHE will automatically interpolate the data to the model grid.

Internal inactive zones

Currently, it is not possible to extract the inactive zones from the MODFLOW model and convert these to inactive cells in MIKE SHE. MODFLOW and MIKE SHE treat internal inactive zones quite differently. In MIKE SHE, the internal inactive zones are simply treated as cells with a very low hydraulic conductivity, whereas, MODFLOW ignores them in the solution. Furthermore, the extraction program only writes points to the .shp file for the active nodes. Thus, when it comes to the interpolation in

MIKE SHE, the interpolation does not know about the inactive zone and interpolates through the inactive zone - there are simply no data points in the inactive zones.

WATER QUALITY

11 SOLUTE TRANSPORT

The complete MIKE SHE advection-dispersion (AD) module is comprised of four independent components, each describing the transport processes in one of the parts of the hydrological cycle. Used in combination they describe solute transport in the entire hydrological cycle. The four components are:

- Overland Transport
- Channel Transport (MIKE 11)
- Unsaturated Zone Transport
- Groundwater Transport

A number of processes relevant for simulating reactive solute transport are included in MIKE SHE including

- Water and solute transport in macro pores,
- Sorption of solutes described by either equilibrium sorption isotherms (Linear, Freundlich or Langmuir) or kinetic sorption isotherms, which include effects of hysteresis in the sorption process,
- Attenuation of solutes described by an exponential decay, and
- Plant uptake of solutes.

Current Limitations

The solute transport module in MIKE SHE currently does not support

- exchange **from** MIKE 11 **to** Overland flow,
- any solute transfer via irrigation,
- any solute transfer via flood codes, and
- solute migration **from** UZ **to** OL.

In the first three cases, the solutes will remain in the source location and only the water will be transfered. This will lead to increasing concentrations at the source.

In the last case, there is no mechanism in MIKE SHE to transfer water from UZ to OL, so there is also no means to move solutes from the UZ cells onto the ground surface. This has implications for salinity modelling, as there is no way for runoff to remove surface salts that migrate upwards due to capillarity and concentrate on the ground surface due to evaporation.

11.1 Flow Storing Requirements

Solute transport calculations in MIKE SHE AD are based on the water fluxes from a MIKE SHE Water Movement (WM) simulation. To ensure that all the needed WM result data types are stored, you have to specify that results should be stored for an AD simulation. See Storing of Results *(V.2 p. 183)*.

The WM data should be stored frequently enough to describe the dynamics of the flow. The selected storing frequencies of flow results will usually be a compromise between limitations in disk space and resolution of the flow dynamics. The maximum computational time steps in a transport simulation are often restricted by advective and dispersive stability criterions. However, the transport time step cannot be greater than the flow storing time step in each component.

11.2 Storing of Results

The simulated concentration distribution in each component as well as the mass balances and fluxes will be stored in dfs2 and dfs3 files with different time steps. Besides these result files, the program also writes output to the error log, which describes errors encountered during execution and a print log which contains execution step information, statistics on the run and a mass balance (if requested).

Normally, the results from the saturated zone (species concentration in each grid) is by far the most disk consuming parameter. So, be careful with the storing time step. Mass balances, which includes time series of mass storage and fluxes between components (and sources, drains, boundaries etc.) can be stored at smaller time steps.

When you select the time step you should also be aware of the time scale of the process. The time scale for transport processes in groundwater is usually much larger than the time scale for transport in a river.

Enter the desired time steps - notice that the unit is hours - in each of the edit fields. There are no limitations on this time step but if you select a time step less than the simulation time step, the storing time step will be the new simulation time step.

11.3 Simulation and Time Step Control

Simulation time steps are to some extent controlled by the user. Several possibilities for time step control exist to make the execution as fast as possible with no numerical dispersion and instabilities.

The first possibility for controlling the simulation time steps in the different components is simply to define the maximum time step in each component. Note that time steps should be given in increasing order i.e. $dt_{RIVER} \leq dt_{OVERLAND} \leq dt_{UZ} \leq dt_{SZ}$. Also note that this is the MAXIMUM time step. That is, the actual simulation time step is controlled by the stability criterions with respect to advective and dispersive transport given below. Furthermore, time steps for transport cannot exceed the storing time step for the relevant data in the flow result file from a MIKE SHE flow simulation.

Enter the maximum allowable Courant number for each component. The Courant number is defined by *V x dt/dx* (velocity times time step divided by “grid size”). This number should normally not exceed 1.0 for one- and two-dimensional transport (UZ, Overland and Channel Flow) and 0.8 for three-dimensional transport (SZ). The maximum time step will be calculated accordingly.

Enter the maximum allowable dispersive Courant number for each component. The dispersive Courant number is defined by *D x dt/dx2* (Dispersion coefficient times time step divided by “grid size” squared). This number should normally not exceed 0.5. The maximum time step will be calculated accordingly.

The transport limits are used to avoid negative concentrations in cases with extreme gradients (e.g. close to sources) or in areas with highly irregular velocity fields. Enter the maximum allowable transport from a node or grid as a fraction of the storage in the node or grid. A recommended value for all components is 0.9, which ensures that this option is in use (the value 0 determines that this option is not in use).

11.3.1 Calibrating and Verifying the Model

The advection-dispersion of solutes depends largely on the simulated flows and fluxes calculated by the MIKE SHE flow model. After your first AD simulations, you will usually have to go back and improve the calibration of your flow model. Rarely, can the simulated concentrations and mass fluxes be calibrated to the measured concentrations by tuning only the solute transport model.

It is important to recognise that a transport model must be calibrated. This is true for all applications larger than the laboratory scale since model output cannot necessarily be compared directly to measured values. Measurements are mostly point measurements at a certain time whereas results often are mean values over larger volumes and longer times.

The purpose of the calibration is to tune the model so that it is able to reproduce measured conditions for a particular period in a satisfactory way. This period - known as the calibration period - should be chosen long enough to include events of similar kind as the ones you are going to investigate.

A satisfactory calibration is reached when the model is able to reproduce the measured values taking the following conditions into account:

- uncertainty in the measurements (time, space, equipment)
- representativeness of measurements (point/average grid values)
- differences between your conceptual model and nature
- uncertainty in other model parameters and data (source description etc.)

In general, it is impossible to specify an exact level of divergence between measured data and computed results before the model is satisfactorily calibrated. In each application you have to consider all factors influencing your result.

After the calibration, you should verify your model by running one or more simulations for which measurements are available without changing your model parameters. If the model is able to reproduce the validation measurements you can consider your calibration to be successful. This ensures that simulations can be made for any period similar to the calibration and the verification period with satisfactory results.

11.4 Executing MIKE SHE WQ

In the top icon bar, there is a three-button set of icons for running your model. 

.

WQ - The WQ button starts the **W**ater **Q**uality simulation. After you have successfully run a water movement (WM) simulation to completion, you can run a water quality simulation.

In addition to the three icon buttons, there is a Run menu. In this menu, you can check on and off all three of the above options. Finally, there is an Execute... menu sub-item that runs only the checked items above it. The Execute option can also be launched using the Alt - R - E hot-key sequence.

MIKE SHE WQ can also be launched from a batch file with or without the MZLaunch function. For more information on this, see Using Batch Files *(V.1 p. 164)*

11.5 Output

The output of the MIKE SHE AD is stored to several dfs0, dfs2 and dfs3 files which can be viewed and processed with the different tools available for these files in MIKE ZERO.

For each species, a concentration file is created for each hydrologic process - a dfs2 file for OL, and a dfs3 file each for UZ and SZ.

For each species, the total WQ mass balance is stored in two dfs0 files. The xx_species_AllItems.dfs0 includes all of the possible water quality mass balance items. The xx_species.dfs0 is a copy of the _AllItems.dfs0 file with all of the non-zero items removed.

The first 20 items in the _AllItems.dfs0 file define the global mass balance (see Table 11.2). There are four items: Storage, Input, Output and Error. Each of these is calculated for each of the five storage items: SZ, Immob(SZ), UZ, MP(UZ), and OL.

The item Immob(SZ) is for solutes stored in the SZ matrix porosity when the dual porosity option in SZ is turned on. In this case, water and solutes move in the fractures and solutes diffuse into the rock matrix. The fractures are then the primary porosity.

The item MP(UZ) is for solutes stored in the UZ macropores when the macropore option in UZ is turned on. In this case, water and solutes move in both the matrix and the macropores, but the volume of water in the macropores is generally much less than the volume of water in the matrix. So, the primary porosity is the UZ matrix.

The Error is calculated for each of the five items as:

$$Error = Output - Input + \Delta Storage$$

However, the mass in the SZ and UZ items includes the mass in both the primary and secondary porosities.

The Output, Input and Storage are all displayed as positive values in the dfs0 file and the WQ log file. A positive change in storage denotes an increase in mass.

Table 11.1 WQ mass balance items in the _AllItems.dfs0 file.

Mass balance item	Component
Storage	SZ, Immob(SZ), UZ, MP(UZ), OL
Input	SZ, Immob(SZ), UZ, MP(UZ), OL
Output	SZ, Immob(SZ), UZ, MP(UZ), OL
Error	SZ, Immob(SZ), UZ, MP(UZ), OL

The rest of the items in the _AllItems.dfs0 file are only non-zero if the item is relevant for the current WQ simulation. Table 11.2 lists all of the rest of the items in the _AllItems.dfs0 organized by the source of the solute.

Table 11.2 Available WQ mass balance items in the _AllItems.dfs0 file

From	To	Comment
Sources	SZ, UZ, OL, River	Note that sources can be specified as positive or negative.
Ext. Sources (OpenMI)	SZ, UZ, OL	This is non-zero only if a model is linked to MIKE SHE by OpenMI that adds mass to the component.
Ext. Input (OpenMI)	SZ Drain	This is non-zero only if a model is linked to MIKE SHE by OpenMI that adds mass to the component. However, this is a special case because you can add mass directly to the SZ drain without it actually becoming part of the SZ model. The mass is then added to the model at the location were the drain discharges (i.e. river link, SZ boundary, or local SZ depression)
Boundary	SZ, UZ, OL, River	The (Boundary to River) item is typically zero, but can be non-zero in a couple of rare cases: If you have 1) a fixed head SZ boundary, or 2) a fixed concentration boundary, next to a river link, then mass from this cell to the river will be added to (Boundary to River).

Table 11.2 Available WQ mass balance items in the _AllItems.dfs0 file

From	To	Comment
Precip	UZ, OL	Mass from precipitation is always added to either OL or UZ, even though precipitation can be added to SZ in the WM module. If you have a SZ-only simulation, then mass from precipitation is included in the (Precip to OL) and then (OL to SZ)
Decay	SZ, Immob(SZ), UZ, MP(UZ), OL	This is the mass that has decayed in each of the processes
Sorp/DeSorp	SZ, Immob(SZ), UZ, MP(UZ), OL	This is the net Sorption and Desorption to the soil matrix in each of the processes. If mass is sorbed to the soil matrix it is removed from solution and this value will be negative. If mass desorbs from the soil matrix it is added to solution and this value will be positive.
Colloid-Sorp/DeSorp	SZ, Immob(SZ), UZ, MP(UZ), OL	This is the net Sorption and Desorption to colloids in each of the processes. If mass is sorbed to the colloids it is removed from solution and this value will be negative. If mass desorbs from the colloids it is added to solution and this value will be positive. However, this is normally zero, because colloid transport is not available in the commercial version of MIKE SHE, only a research version.
EcoLab	SZ, Immob(SZ), UZ, MP(UZ), OL	This is the mass change resulting from passing solutes to and from EcoLab - positive if EcoLab causes the mass to increase and negative if mass decreases.
SZ	UZ, MP(UZ), OL, Sinks, Sources, Ext.Sinks(OpenMI), Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab, Plant Uptake	Note that there is no mass transfer to SZ drains because the SZ drains have not storage. Mass that discharges to SZ drains passes straight through the drain and is added to the end recipient (i.e. a river, boundary, or local depression.
Immob(SZ)	UZ, Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab	
SZ Baseflow	River	This is a special item that includes only the mass from SZ to rivers.
SZ Drain	River, SZ (Local Dep), Boundary	This is a special item that divides up the SZ mass discharge to drains by the end recipient.
SZ Flow	Boundary	This is a special item to distinguish between SZ mass discharge to boundaries via drains and direct discharge to the boundary.
SZ(Fract)	Immob(SZ)	This is a special item that includes only the mass exchange between fractures and the matrix when the dual porosity option is selected in the SZ.

Table 11.2 Available WQ mass balance items in the _AllItems.dfs0 file

From	To	Comment
UZ	SZ, Immob(SZ), OL, Sinks, Sources, Ext.Sinks(OpenMI), Boundary, Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab, Plant Uptake	Note that the (UZ to Boundary) item refers to mass discharge from UZ to SZ boundaries. This can arise, for example, when a UZ column discharges into an SZ cell that contains an internal boundary condition, such as a constant head. Note that the (UZ to Immob(SZ)) item will only be non-zero when the dual porosity option in SZ is turned on. In this case, as the water table increases, there will be a transfer of UZ mass to water in both the SZ fractures and SZ immobile matrix based on the ratios of their porosities. Related to the above, if macropores are active, then mass in the UZ macropores will be distributed to both the SZ matrix and the SZ fractures.
MP(UZ)	SZ, Immob(SZ), UZ(Matr), Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab	
UZ(Matr)	MP(UZ)	This is a special item that includes only the mass exchange between macropores and the matrix when the macropore option is selected in the UZ.
OL	SZ, UZ, MP(UZ), River, Sinks, Sources, Ext.Sinks(OpenMI), Boundary, Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab	
River	SZ, OL, Boundary	(River to OL) is always zero because this exchange has not yet been implemented.

11.6 Coupling MIKE SHE and MIKE 11 WQ

Coupling the WQ modules between MIKE 11 and MIKE SHE is described in the section: Coupling MIKE SHE Water Quality to MIKE 11 *(V.1 p. 218)*.

12 MIKE SHE + ECO LAB

ECO Lab is an open and generic equation solver. ECO Lab is mostly used for water quality and ecosystem modeling, such as modelling eutrophication of lakes, calculating the fate and transport of heavy metals and determining ecology indicators (e.g. distribution of sea grass). Originally, ECO Lab was developed as a tool to simulate water quality reactions in surface water, but has been expanded to include agent based modeling and other more complex reactions and components.

In MIKE SHE, ECO Lab can be used from basic water quality kinetic reactions in surface and groundwater, to complex coupled feedback interactions between nutrients, plant growth and hydrology.

The initial implementation of ECO Lab in MIKE SHE depends on MIKE SHE's WQ module. That is, you must run a WQ simulation after the WM simulation. When running MIKE SHE + ECO Lab, ECO Lab reads concentrations (state variables) from MIKE SHE's WQ module, reads other necessary input data files, generates additional output and passes modified concentrations (state variables) back to MIKE SHE.

ECO Lab acts on a cell basis. That is, it is called for each cell in the model. By default it is called at every time step in the MIKE SHE WQ simulation, but optionally can be called less frequently.

Thus, using ECO Lab is a multi-step process, whereby you:

1 Create an ECO Lab template file that specifies the equations to be solved, including all the forcing (spatially and time varying input), constants (spatially varying constants), state variables (parameters calculated in MIKE SHE, i.e species concentrations), and derived outputs (results).

2 Specify the name of the template file in the MIKE SHE WQ model.

3 Define the links between the template variables and the MIKE SHE parameters.

4 Run the MIKE SHE Water Quality model. During the simulation, MIKE SHE passes the State Variables to ECO Lab. ECO Lab updates the State Variable values and passes them back to MIKE SHE. At the same time, ECO Lab will write to any specified output files.

The output files are standard dfs2 or dfs3 output files. These files can be used as input in subsequent WM or other WQ simulations, or viewed in the Results Viewer, etc.

12.1 ECO Lab Templates

An ECO Lab Template contains the mathematical description of the ordinary differential equations to be solved. These could, for example, describe an ecosystem including the processes affecting the ecosystem.

An ECO Lab template contains six components:

- State Variables - State variables represent the state that the user wants to predict (at least one state variable must be specified).
- Constants - Constants are used as arguments in the mathematical expressions of processes in an ECO Lab model. They are constant in time, but can vary in space.
- Forcings - Forcings are used as arguments in the mathematical expression of processes in an ECO Lab model. They can vary in time and space. They basically represent variables of an external nature that can affect the ecosystem.
- Auxiliary Variables - Auxiliary Variables are arguments in the mathematical expression in an ECO Lab model. They can be used as intermediate calculations that can include any state variable, constant or forcing. They can also be used to specify results directly.
- Processes - Processes describe the transformations that affect the state variables. That means processes are used as arguments in the differential equations that ECO Lab solves to determine the state of the State Variables.
- Derived Outputs - Derived Outputs are output files that are derived based on the model results.

Additional details on developing ECO Lab templates is available in the MIKE ZERO ECO Lab manual.

Important Note: Units

All concentrations passed from MIKE SHE to ECO Lab are in units of [g/m^3], which is equivalent to [mg/L].

Thus all parameters and equations defined in the ECO Lab template must reflect these units - either directly or via an appropriate scaling factor. For example, the correct units for a decay rate constant might be [g/m^3/day] or [mg/L/day].

12.1.1 Developing a Template

Creating and developing an ECO Lab template involves several steps.

Create an ECO Lab template

First you must create an ECO Lab template from the File/New menu or the New File icon. In both cases, you will chose MIKE Zero and then ECO Lab (.ecolab) in the New File dialogue. This will create a new blank ECO Lab template file.

Alternatively, you can copy and edit an existing ECO Lab template.

A few tips will be useful before you start.

- You should try to keep the names of the Constants, Forcings, etc. as short as practical. The names are used when defining Processes, Auxiliary Variables, and Derived Outputs.
- The names used in the definitions are case-sensitive.
- The names must be unique within the list of Constants, Forcings etc.
- To add a new Constant, Forcing, etc. right click on the item and chose the appropriate option.

Add State Variables

In the current coupling to MIKE SHE, the only available State Variables are species concentrations. Thus, you must add one State Variable item for each species in MIKE SHE that you want ECO Lab to modify during the WQ simulation.

An ECO Lab template must include at least one State Variable.

The State Variable name must be exactly the same as the Species name in MIKE SHE.

Dual domain mass transfer

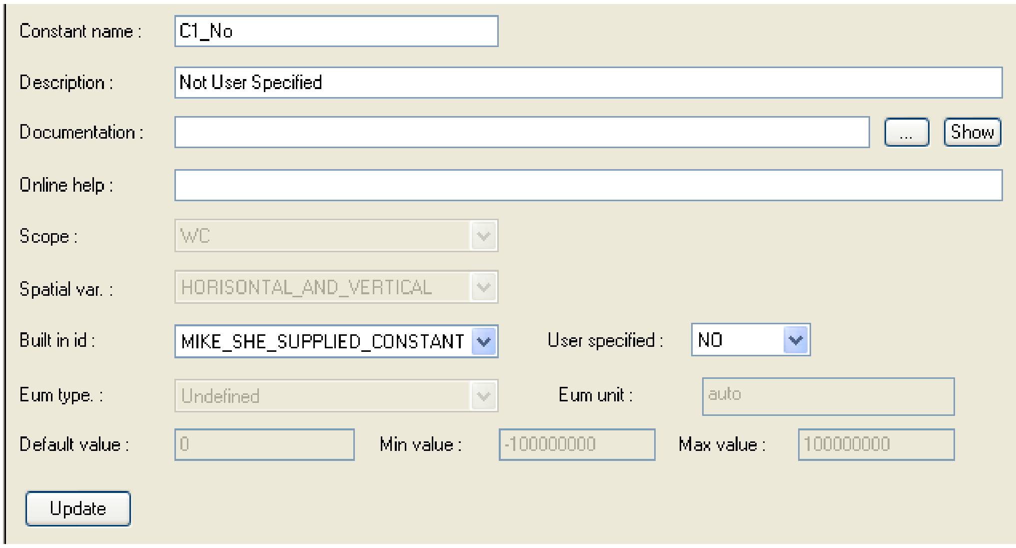
The exception to the exact naming rule is when simulating dual domain mass transfer. In this case, the State Variable name must use the reserved suffix "_2" for the solute in the secondary porosity. For example, OXYGEN and OXYGEN_2 would be the State Variable names for the species OXYGEN in MIKE SHE.

Add one or more Constants

Constants are spatially distributed values that are constant in time. Each constant may or may not be User Defined.

User Defined = NO

If the Constant is a parameter that is pre-defined then the Constant is **not** User Defined (i.e. User Defined = NO). If this is selected, a list of Constants is available in the combo box. However, the only Constant on this list that is relevant for MIKE SHE is "MIKE_SHE_SUPPLIED_CONSTANT". All others will be ignored.



There are a limited number of Constants that can be passed directly from MIKE SHE. These are mostly geometry related parameters, including the grid area, the cell volume, the topography, and the depth to the top and bottom of the cell. However, some specific Constants are also available for the different domains, such as the Detention Storage in the Overland flow, and the Bulk Density and the Porosity in the saturated zone.

User Defined = YES

If your Constant is not in the list of available pre-defined Constants from MIKE SHE, then you must chose User Defined = YES. In this case, you can define any spatially distributed, static parameter.

The actual values and spatial distribution of the Constant will be defined in the MIKE SHE Setup Editor. In the template, the only thing you need to specify is the name of the Constant. The name is then used when defining the Processes, Derived Outputs, etc.

In some cases, the Constant may already be defined in the Setup Editor, but only those in the list will be automatically passed to ECO Lab. So, if your Constant is not available, then you will need to define it again in the list of Constants in the Setup Editor.

Add one or more Forcings

A Forcing is any spatially distributed value that is time varying. You can think of it as a value that is affecting the State Variable during the simulation. For example, air temperature will affect the degradation rate of a solute.

Similar to the Constants, the Forcings can be User Defined = YES, or User Defined = NO.

User Defined = YES

If the forcing is not user defined (“User Defined = NO”), then a long list of available forcings are listed in a combo box. However, the only Forcing on this list that is relevant for MIKE SHE is “MIKE_SHE_SUPPLIED_FORCING”. All others will be ignored.

Note: The various other “MIKE_SHE_” forcings are used by MIKE 11 to define concentrations and mass of solute entering the river.

For all forcings that are not user-defined, there will be a list of pre-defined available forcings in the MIKE SHE Setup editor depending on the domain.

User Defined = NO

If your Forcing is not in the list of available pre-defined Forcings from MIKE SHE, then you must chose User Defined = YES. In this case, you can define any spatially distributed, time-varying parameter.

The actual values and spatial distribution of the Forcing will be defined in the MIKE SHE Setup Editor. In the template, the only thing you need to specify is the name of the Forcing. The name is then used when defining the Processes, Derived Outputs, etc.

In some cases, the Forcing may already be defined in the Setup Editor, but only those in the list will be automatically passed to ECO Lab. So, if your Forcing is not available, then you will need to define it again in the list of Forcings in the Setup Editor.

Create Auxiliary Variables, Processes and Derived Outputs

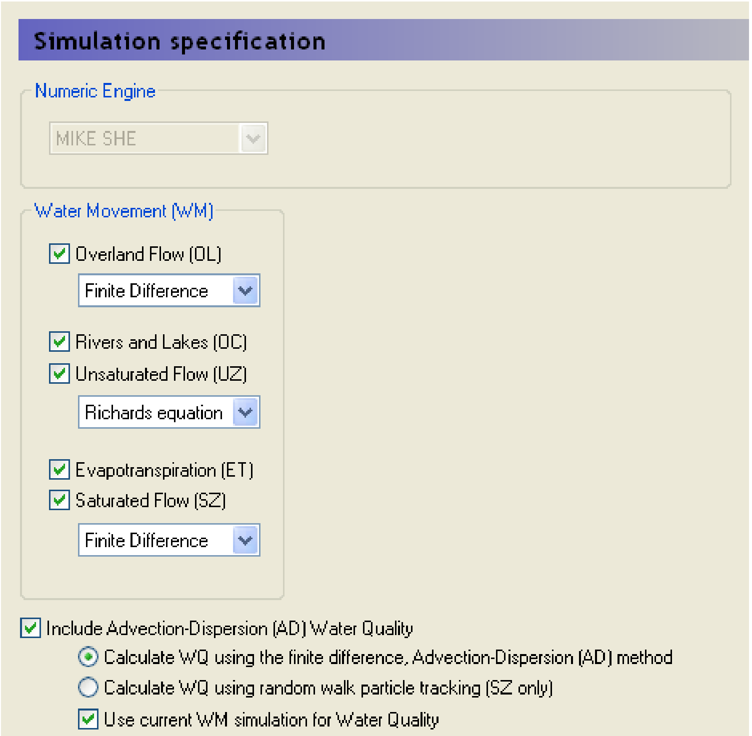
Auxiliary variables are used for intermediate calculations and must be defined as 3D (UZ and SZ) or 2D variables.

Processes transform a State Variable or calculate another result. Spatial variation and type must be defined for each process. Each process can be included in the results file by choosing "YES" in the "Output" box.

Derived Outputs allow the user to define output files based on the process results.

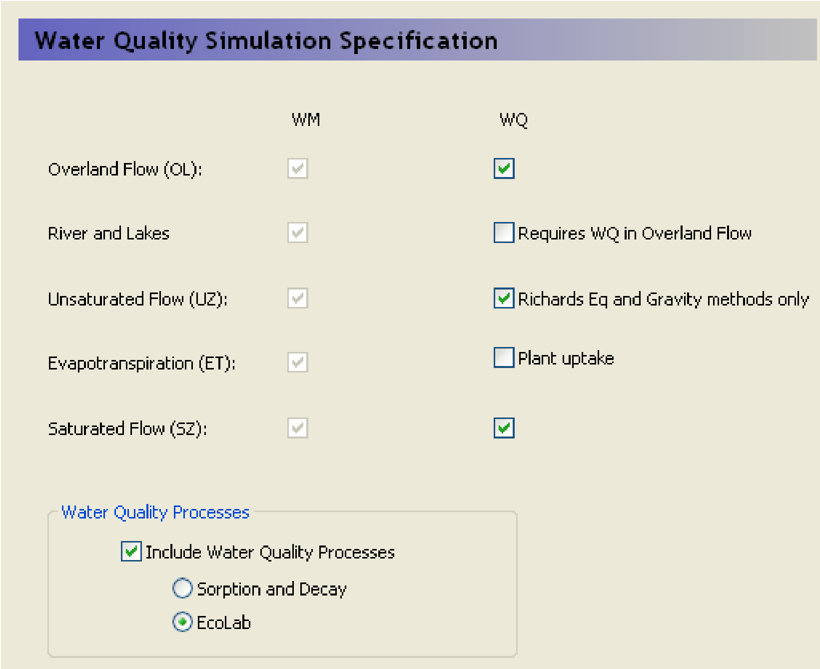
12.1.2 ECO Lab templates in MIKE SHE

ECO Lab is only available when Water Quality is selected. Thus, to be able to use ECO Lab, the Water Quality option in the main Simulation Specification dialogue must be selected.

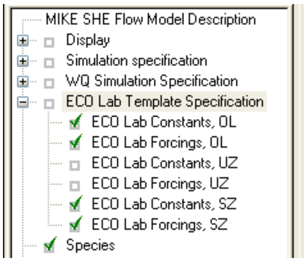


Note also that ECO Lab will only work with the Finite Difference AD method.

After activating the Water Quality module, the ECO Lab option must be selected in the WQ Simulation Specification dialogue.

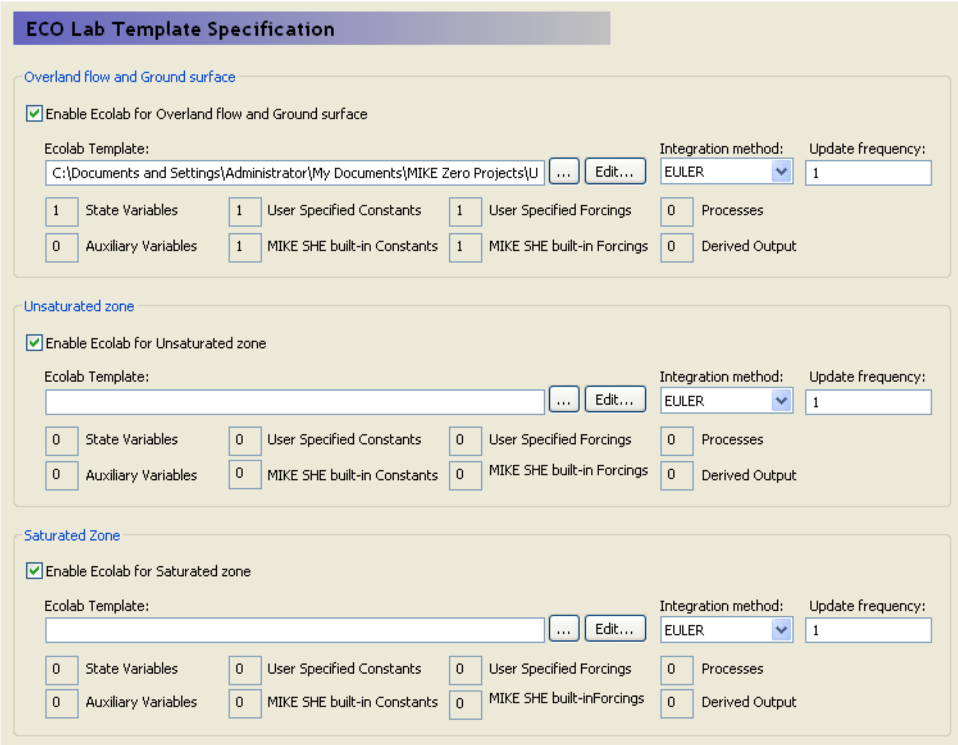


A new data tree branch will appear, where ECO Lab templates can be specified for each of the hydrologic processes - Overland Flow, the Unsaturated Zone, and the Saturated Zone. Separate templates are required for each of these zones because the processes in each of these domains are very different.



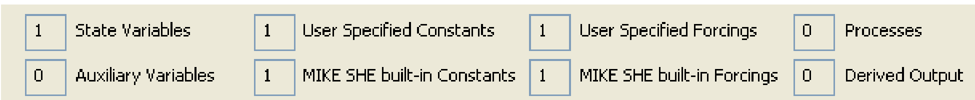
In the ECO Lab Template Specification dialogue, there is a checkbox for each of the processes. These checkboxes are active if the Water Quality is activated for the process in the WQ Simulation Specification dialogue

.

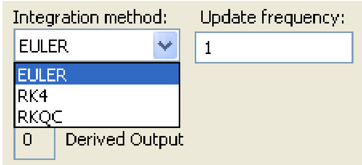


When you enable ECO Lab for the specific process, you will be able to browse to the required ECO Lab template. Specified templates can be directly modified by clicking on the Edit button.

When you browse to a template, the template file is read by the Setup Editor and the number of components (i.e. State Variables, Forcings, Processes, etc.) that have been specified in the template are displayed in the Template summary.



When calculating the concentrations (State Variables) for the next time step, an explicit time-integration of the transport equations is made. Depending on the desired accuracy of this numerical integration, you can chose one of three different integration methods. The methods are in increasing level of accuracy (and numerical effort), starting with the Euler method and finishing with the Runge Kutta 5th Order method).



Finally, for each template you can specify an update frequency (see above). The update frequency tells MIKE SHE how frequently to call ECO Lab. If the water quality processes are slow relative to the simulation time step, you can save considerable simulation time by calling ECO Lab less frequently. For example, to call ECO Lab every second or third simulation time step, you would specify an Update frequency of 2 or 3.

12.1.3 State Variables in MIKE SHE

The State Variables defined in the ECO Lab template are passed to MIKE SHE as Species. This is the only way to pass information from ECO Lab to MIKE SHE.

The Species Name in MIKE SHE must be exactly the same as the State Variable Name used in the ECO Lab Template (except in the case of dual domain mass transfer, which uses the reserved suffix “_2”).

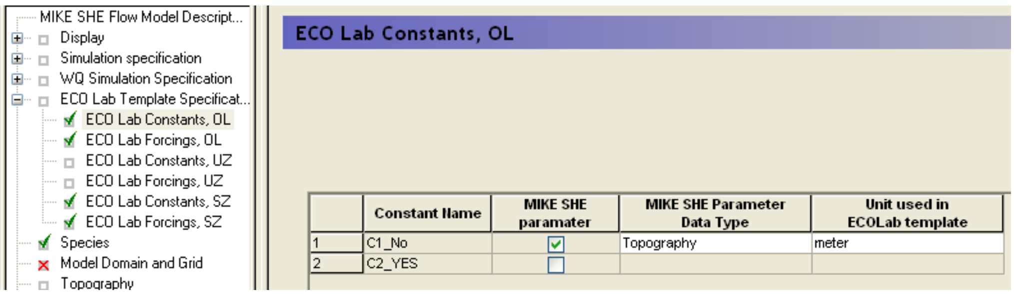
There are four species types in MIKE SHE. Species can be either:

- **Dissolved** - Dissolved species are mobile in the water. They are active in the subsurface and surface water. Disolved species have a default concentration of [μg/m^3].
- **Sorbed** - Sorbed species are only available in the subsurface. They are fixed to the soil matrix and do not move with the water. Sorbed species have a default concentration of [g/g].
- **Suspended** - Suspended species are only available in ponded water. They do not infiltrate to the UZ or SZ, and they cannot become Sorbed species. If the ponded water infiltrates, the species is left behind. Suspended species have a default concentration of [μg/m^3].
- **Fixed -** A fixed species is neither disolved or nor sorbed. It is used as an immobile state variable by ECOLab. This allows ECOLab to read and write arbitrary values to MIKE SHE during the simulation. Fixed species have an undefined unit.

In particular, the Fixed species is especially interesting in MIKE SHE. It is a species type without pre-defined units of concentration. The non-dimensional species cannot be transported with the flow and can be used as a book-keeping mechanism for resulting processes. ECO Lab itself can read a value from any file, update the value and write it back to the file. However, ECO Lab cannot append a new value to a file. That is, ECO Lab cannot read a value, update the value and add the new value as a new timestep in file. The Fixed species type gives you a mechanism for handling various auxiliary user defined quantities during the simulation. In other words, ECO Lab can read the current Fixed species value, and return a new value to MIKE SHE. MIKE SHE moves forward in time, and ECO Lab starts over again. In the mean time, MIKE SHE has saved the value from the previous time step. This mechanism greatly increases the flexibility of the ECO Lab coupling in MIKE SHE. It allows you for example, to create things like ecological indexes that map changing ecohydrologic conditions over time.

12.1.4 Specifying Constants and Forcings in MIKE SHE

When you browse to the ECO Lab template, the template is read by MIKE SHE's Setup Editor to find the MIKE_SHE_SUPPLIED_FORCINGS and MIKE_SHE_SUPPLIED_CONTANTS used in the template. These are separated into Built-in and User Specified Forcings and Constants. In both cases, the Forcing or Constant is added to the appropriate list of Forcings and Constants in the MIKE SHE data tree - separated into the different domains (OL, UZ, and SZ).



Built-in Forcings and Constants

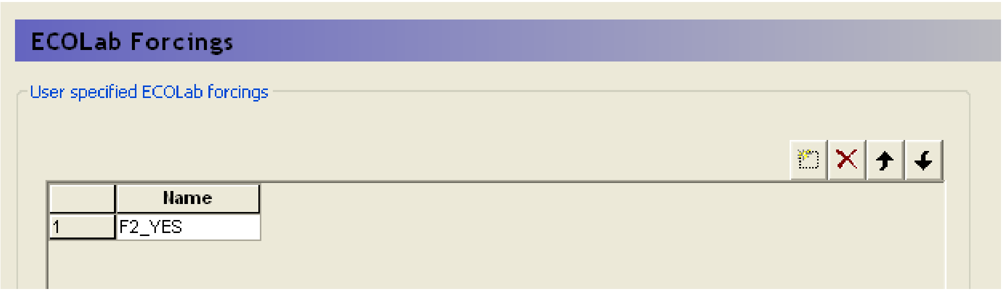
If the Forcing or Constant is not user defined (User Defined = "NO"), then the Forcing or Constant is an internal value in MIKE SHE and will be passed automatically to ECO Lab. The list of available parameters is quite short - primarily geometry related (e.g. cell volume), plus a few domain specific constants (e.g. porosity).

After selecting the parameter from the list of available parameters in the combo box, select the units that are being used in the template. The list of units is taken from the standard available units in the EUM database for the particular item. The Constant or Forcing can be used in various equations in the template. However, there is no check on the units being used. So, it is expected that the Forcing or Constant will used one of the options from the list of units in the EUM database. Otherwise, and appropriate conversion must be done in the template equation.

User Specified Forcings and Constants

If the Forcing or Constant is user defined (User Defined = “YES”), then the Forcing or Constant must be specified explicitly in MIKE SHE.

In this case, there is nothing to specify on the main list of Forcings and Constants, but a Forcing or Constant item is added to the data tree down under the appropriate branch in the data tree. In this branch you will find a table of user specified Forcings and Constants, plus individual sections for each item. Each item can be spacially defined similarly to other constants or time varying values in MIKE SHE.



Note: The Forcings and Constants are defined by Water Quality layer in the Saturated Zone. Thus, you have to define at least one Water Quality Layer in the Saturated Zone.

The list of user defined Forcings and Constants is initially taken from the template. However, the list is not fully dynamic. Thus, if you add items to the template, these will be added to the list. However, if you remove items from the template, or change the name in the template, the item will not be removed from the list. This allows you maintain your data inputs while you are developing your template. Any data that is not used in the template will be pre-processed like any other data, but will not be used in ECO Lab. If you don’t need the items any more, you can delete them from the list

Tip: The fact the list of user specified Forcings and Constants is not permanently linked to the template, allows you to pre-process any static or time-varying data and map it to the numerical grid. You can then use this pre-processed data in other grid operations in MIKE SHE.

12.1.5 Running ECO Lab with MIKE SHE

To run ECO Lab with MIKE SHE, simply pre-process and run the model normally. You can view the user specified Constants and Forcing in the

Pre-processed tab. The Derived Outputs will be written to the default output directly along with all of the regular output from MIE SHE. The Auxiliary Variables and Processes will also be written to this directory, if the Output option is on.

All output can be viewed or processed normally with all of the regular MIKE Zero tools.

13 PARTICLE TRACKING (PT)

The PT module calculates the flow path of a hundreds, thousands, or even millions, of hypothetical particles, which are moved in the three-dimensional, saturated groundwater zone (SZ). The particles are displaced individually in a number of time steps. The movement of each particle is composed of a deterministic part, in which the particle is moved according to the local groundwater velocity calculated by the MIKE SHE water movement (WM) module, and a stochastic part where the particle is moved randomly based on the local dispersion coefficients.

Particle tracking is only calculated for the saturated zone (SZ) and particles that leave SZ are not traced any further. Initially, the user assigns a number of particles to each model grid cell. Particles are added during the simulation from sources, for example solutes in precipitation or from boundary or internal defined concentration cells. Particles leave SZ when they arrive at a boundary or an internal constant concentration cell or when they go to a sink. Possible sinks in the Particle Tracking are wells, rivers, drains, and exchange with the unsaturated zone or overland flow.

All particles are assigned a mass, which means that a number of particles within a specific volume correspond to a solute concentration. The Particle Tracking module can therefore be used for solute transport simulations and is in some cases superior to the conventional numerical solution of the advection-dispersion equation since numerical dispersion is negligible. However, the module is mostly used for delineation of abstraction well capture zones and upstream zones and for determination of groundwater age and conservative solute transport times.

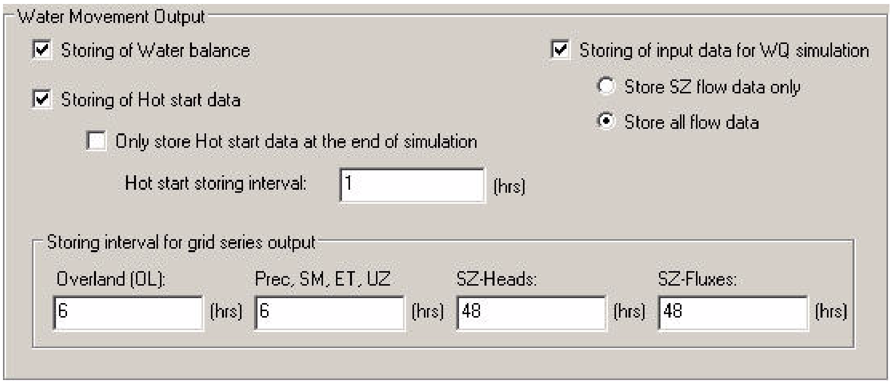
The PT module uses the concept of 'registration' cells. This records particle data when particles enter certain model cells. Registration can be used to delineate capture zones or to observe particles passing through some region of interest, such as a redox layer.

13.1 Requirements in MIKE SHE WM

Prior to running a PT simulation, the MIKE SHE Water Movement (WM) simulation must be run. This section describes what needs to be specified in the WM simulation to run the PT simulation afterwards.

13.1.1 Flow Storing Requirements

Particle transport calculations in MIKE SHE PT are based on the groundwater flows from a MIKE SHE WM simulation. In principle, only groundwater fluxes are needed but to ensure that all the needed WM result data types are stored the user has to specify that results should be stored for an AD run in the WM input. Thus, you must check the appropriate checkbox under “Storing of Results” in the MIKE SHE WM GUI.

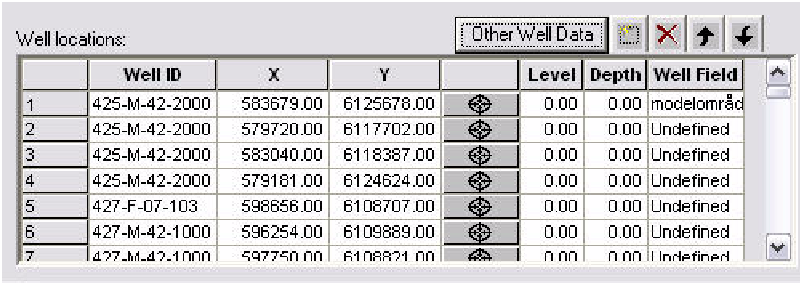


The user can choose between “SZ only” and “All hydraulic components”, however, for PT-simulations “SZ only” will be sufficient, since particle tracking is only calculated for the saturated zone.

The simulated particle distribution is stored with a desired frequency in the MIKE SHE WM GUI under “Storing of results” => “Grid series output”. It is important that the SZ and SZ-flow use the same storing frequency in order to run the following PT simulation. The WM result files to be used in the PT-simulations will be located in a folder with the same name as the *.SHE file.

13.1.2 Specification of Well Fields

To be able to retrieve particle locations based on well fields, it is necessary to specify the well fields in the MIKE SHE well database file .



	Well ID	X	Y		Level	Depth	Well Field
1	425-M-42-2000	583679.00	6125678.00		0.00	0.00	modelområd
2	425-M-42-2000	579720.00	6117702.00		0.00	0.00	Undefined
3	425-M-42-2000	583040.00	6118387.00		0.00	0.00	Undefined
4	425-M-42-2000	579181.00	6124624.00		0.00	0.00	Undefined
5	427-F-07-103	598656.00	6108707.00		0.00	0.00	Undefined
6	427-M-42-1000	596254.00	6109889.00		0.00	0.00	Undefined
7	427-M-42-1000	597750.00	6108821.00		0.00	0.00	Undefined

13.2 Output from the PT simulations

The result files will be located in standard Results directory for your project. The PT result files are:

- ***projectname*.PTRES**: An ASCII file in pfs-format listing the abstraction wells and the computational cells, where abstraction occurs. Used for retrieval of particle location - see PT Registration Extraction *(V.2 p. 220)*.
- ***projectname*.PTREG** and ***projectname*.trf**: Two binary files that cannot be opened directly.
- ***projectname*.PTBIN**: An optional binary file containing all of the particle locations at every saved time step. Individual path lines can be extracted using the Extraction of particle pathlines *(V.1 p. 295)*.
- ***projectname*.PTGross.shp**: An optional point theme shape file containing the path line information of every particle at every saved time step. As part of the shape file, a .shx and a .dbf file are also created. The .dbf file can be opened in Excel if it is less than 65536 lines.
- ***projectname*_AD_3DSZ.dfs3**: Temporal and spatially varying SZ concentrations in the mobile phase based on the mass of the particles.
- ***projectname*_PT_3DSZ.dfs3**: Temporal and spatially varying PT results including:
  - **Number of particles -** this is the actual number of particles in each cell
  - **Accumulated particle count** – this is the number of particles that have entered the cell during the simulation
  - **Number of registered particles** – this is the number of particles that started in this cell that have become registered
  - **Most recent registration zone** – this is the registration zone attached to the last particle to be registered that originated in the cell
  - **Average age** – this is the average age of all the particles in the cell
  - **Average transport time** – this is the average length of time from when the particle was born in this cell until it was registered somewhere

Besides these result files, the program also writes output to two log files. The error log list errors encountered during execution and the print log file contains execution step information, statistics on the run and a mass balance (if requested).

13.3 Extraction of particle registrations

After the particle tracking has been run, the registration information needs to be read from the output files and processed in a useful way. The Results Tab includes an utility to sift through all the particles and their registrations, to find the ones that you want. This is output from this utility is an ArcGIS point-theme, shape file with the starting locations of the all the particles that meet your registration criteria. The extraction utility allows your to filter the results for:

- Destination type:
  - Specific sink types (drain, river, unsaturated zone, well, constant concentration boundary or constant concentration sink)
  - Registration codes specified by the user
  - Wells found in the flow results
  - Well fields found in the flow results*)
- Layer from which the particles originated
- Release (birth) time
- Transport time

Note: To extract particle locations based on well fields requires that different well fields have been defined, see section Specification of Well Fields *(V.1 p. 292)*.

The results can be written to either

- a single shape file where the point attributes allow further selection of the particles in ArcView, or
- separate files for each destination type and optionally for each layer e.g. one file for each sink type/layer combination.

More detailed information on the actual extraction mechanics can be found in the PT Registration Extraction *(V.2 p. 220)* section.

13.3.1 Running from a batch file

The registration extraction can be run from a command line. To execute the program open a command line and type:

```
Ptoutputretrieval.exe projectname.she extraction_num
```

The extraction_num is the item number in the table of extraction items in the PT Registration Extraction *(V.2 p. 220)* dialog. The extraction will proceed silently - that is without any messages. To run the extraction with the messages, you need to use

```
MZLaunch project_name.she -e Ptoutputretrieval.exe
```

which will start the MZLaunch utility

***projectname*_ptoutputretrieval.err**: If errors occur during execution of the program these are written to this log file.

13.3.2 Limitations with the PT registration method

When using registration zones to identify particles that move through certain parts of the model it should be noted that particles can appear more than once in the output. As they move from one zone to the next they are repeatedly registered and are finally also registered when they are removed from the model by a sink. An example would be a particle moving into a registration zone with code 1. The particle is then registered as being in an 'active cell' and the registration zone code and travel time to this zone is memorised. If the particle is at a later time removed by a well it will again be registered but now it will be registered as being removed by the 'Well' sink.

If there are multiple wells within one cell and output for wells is requested then the output can contain the same particle more than once. As the model does not know which of the wells the particle should be assigned to (the program looks at the total well sink for the cell and cannot distinguish individual wells) the particle will be repeated for each of the wells within the cell.

13.4 Extraction of particle pathlines

It is too cumbersome to extract and plot the pathlines for all of the thousands of particles that can be generated during a PT simulation. The PT Pathline Extraction utility allows you to extact the pathlines for specific particles if you have saved the intermediate locations in the Storing of Results *(V.2 p. 183)* dialog.

To extract a particle pathline you need a Particle ID. These can only be found after the simulation by evaluating the PT output. For example, you can find the particle ID by extracting the particle start locations that end in a specific well and then finding the ID numbers of the particles that you want in the shape file that was created.

In the Results tab, the PT Pathline Extraction *(V.2 p. 223)* utility is available to make this extraction.

Running from a batch file

The pathline output retrieval program can be run from a command line. To execute the program, open a command line and type:

```
PtBinRetrieval.exe file_name.she extraction_num
```

The extraction_num is the item number in the table of extraction items in the PT Pathline Extraction *(V.2 p. 223)* dialog. The extraction will proceed silently - that is without any messages. To run the extraction with the messages, you need to use

```
MZLaunch file_name.she -e PtBinRetrieval.exe
```

which will start the MZLaunch utility. Particle IDs can be found by using the PT Output Retrieval utility.

A D D I T I O N A L   O P T I O N S

14 EXTRA PARAMETERS

The Extra Parameters section is a special section of the Setup data tree that allows you to input parameters for options that have not yet been included in the MIKE SHE user interface.

The Extra Parameters are only recognized if the Name (e.g. “sheet piling module”) are spelled exactly correct. After the initial run, you should check in the Preprocessor_print.log file to ensure that the module has actually been activated.

Available Extra Parameters include:

Climate

- Negative Precipitation *(V.1 p. 300)*
- Precipitation Multiplier *(V.1 p. 301)*

Surface water

- Time-varying Overland Flow Boundary Conditions *(V.1 p. 302)*
- Time varying surface infiltration (Frozen soils) *(V.1 p. 303)*
- Simplified Overland Flow Options *(V.1 p. 304)*
- Irrigation River Source Factors *(V.1 p. 306)*
- Explicit Overland Flow Output *(V.1 p. 307)*
- Alternative low gradient damping function *(V.1 p. 308)*
- Paved routing options *(V.1 p. 309)*
- Transpiration during ponding *(V.1 p. 309)*

Unsaturated Zone

- Threshold depth for infiltration (2-Layer UZ) *(V.1 p. 310)*
- Increase infiltration to dry soils *(V.1 p. 311)*

Saturated Zone

- Sheet Pile Module *(V.1 p. 312)*
- SZ Drainage to Specified MIKE 11 H-points *(V.1 p. 316)*
- SZ Drainage Downstream Water Level Check *(V.1 p. 319)*
- SZ Drainage to MOUSE *(V.1 p. 319)*
- Time varying drainage parameters *(V.1 p. 320)*

- SZ Drainage River Link Reference Table *(V.1 p. 321)*
- Canyon exchange option for deep narrow channels *(V.1 p. 322)*

Water Quality

- Disable SZ solute flux to dummy UZ *(V.1 p. 323)*
- SZ boundary dispersion *(V.1 p. 323)*

Miscellaneous

- Including OpenMI *(V.1 p. 324)*
- Plot control for Detailed Time Series Output *(V.1 p. 325)*
- Extra Pre-Processing output *(V.1 p. 325)*
- GeoViewer Output *(V.1 p. 325)*

14.1 Climate

14.1.1 Negative Precipitation

Negative precipitation is sometimes required when net groundwater recharge has been calculated using an external program, such as DAISY GIS. In this case, the evapotranspiration may exceed infiltration leading to a net upward flux of water from the groundwater table. However, the standard precipitation module in MIKE SHE does not recognize negative rainfall. In this case, you must specify the negative rainfall using the following Extra Parameters options:

Parameter Name	Type	Value
use negative precipitation	Boolean	On
If the negative precipitation is uniformly distributed:		
negative precipitation max depth	float	greater than zero
negative precipitation max layer	integer	greater than zero

Parameter Name	Type	Value
If the negative precipitation is spatially distributed:		
negative precipitation max depth dfs2 file	file name	.dfs2 file
negative precipitation max depth dfs2 item	integer	item number in dfs2 file, greater than zero
negative precipitation max layer dfs2 file	file name	dfs2 file
negative precipitation max layer dfs2 item	integer	item number in dfs2 file, greater than zero

Max depth - This represents the depth of the root zone plus the thickness of the capillary fringe and is the maximum depth from which negative precipitation can be extracted.

Max layer - This is the maximum layer depth from which negative precipitation can be extracted.

Note: the negative precipitation option will only work if there is no UZ model active.

14.1.2 Precipitation Multiplier

To facilitate calibration and sensitivity analysis of recharge, in models where measured precipitation is not being used, a multiplication factor has been implemented.

Parameter Name	Type	Value
precipitation factor	float	greater than zero

If this extra parameter is used, then all precipitation values are multiplied by the factor prior to being used in MIKE SHE.

14.2 Surface Water

14.2.1 Time-varying Overland Flow Boundary Conditions

The default boundary condition for overland flow in MIKE SHE is a constant water level on the outer boundary. The value of this boundary condition is determined by the initial water depth on the boundary. In most models the recommended value is a water depth of zero. In this case, if the water level adjacent to the boundary increases, water will discharge across the boundary and out of the model. If you want to prevent overland outflow then you can use the Seperated Flow Areas option to restrict lateral flow out of the model.

If you specify a non-zero value for initial water depth on the boundary, then this value becames a constant for the entire simulation. If the water level inside the model decreases below this value, the boundary will act as an infinite source of inflow to the model.

However, in many models - especially those with significant wetland areas - the constant water level condition on the boundary is too restrictive.

The following extra parameter options allow you to specify a time varying condition for the outer boundary of the overland flow. If you initialize this option, then you must supply a dfs2 integer grid code file that defines the locations at which you want a time varying boundary. The input requirements have been set up such that you can re-use the model domain dfs2 output file from the pre-processor. In the model domain pre-processed output, the outer cells are defined by a value of 2 and the inner cells are defined by a value of 1.

If the grid code value on the boundary is:

- 2 - the cell is a time varying boundary node, or
- 1 - the cell will have a constant water depth equal to the initial water depth.

The second required file is the actual time-varying water level values. These can be obtained from any MIKE SHE simulation, where the overland water elevation has been stored as a grid series output. There is no requirement that they be stored on the same grid. Internally, the actual boundary condition values will be interpolated from the nearest input values. Thus, the OL boundary conditions can be taken from a coarse regional model and applied to a local scale model.

Finally, each filename must be accompanied by an integer item number that defines which item in the dfs2 file should be used.

Parameter Name	Type	Value
time varying ol boundary	Boolean	On
ol boundary code file name	filename	.dfs2 file
ol boundary code item number	integer	item number in dfs2 file, greater than zero
ol boundary head file name	filename	.dfs2 file
ol boundary head item number	integer	item number in dfs2 file, greater than zero

The Hot Start function is not impacted by the time varying OL boundary. If the continuing simulation includes the time varying OL function then it will be used. If the continuing simulation does not include the time varying OL function the head from the hot start time point.

14.2.2 Time varying surface infiltration (Frozen soils)

A common characteristic in cold climates is that infiltration is reduced during the winter months. When the air temperature is cold enough to maintain precipitation as snow, then infiltration will be limited in any case. However, in the spring, when snow storage is melting, then infiltration may still be limited for some period of time.

Although this function was conceived as a way to support reduced infiltration in winter, it can be used any time a time varying leakage is required.

The time varying infiltration function is a modification of the Surface-Subsurface Leakage Coefficient *(V.2 p. 121)* to allow it to be time varying.

Parameter Name	Type	Value
time varying ol leakage coefficient	Boolean	On
leakage coefficient dfs2 file name	filename	.dfs2 file
leakage coefficient item number	integer	item number in dfs2 file, greater than zero
mean step accumulated leakage coefficient	Boolean	On

The time varying leakage coefficient dfs2 file contains a uniform time series of leakage values. By default the leakage values are instantaneous values. However, the last option above allows you to specify mean step accumulated values.

Note that the areas in which these values will be applied has not changed. The areas are defined in the original Surface-Subsurface Leakage Coefficient *(V.2 p. 121)* dialogue. That is, the leakage coefficient is active if a non-delete value is specified in this file.

14.2.3 Simplified Overland Flow Options

Avoiding the redistribution of ponded water

In the standard version of the Simplified Overland Flow solver, the solver calculates a mean water depth for the entire flow zone using the available overland water from all of the cells in the flow zone. During the Overland flow time step, ET and infiltration are calculated for each cell and lateral flows to and from the zone are calculated. At the end of the time step, a new average water depth is calculated, which is assigned to all cells in the flow zone.

In practice, this results in a redistribution of water from cells with ponded water (e.g. due to high rainfall or low infiltration) to the rest of the flow zone where cells potentially have a higher infiltration capacity. To avoid this redistribution, an option has been added where the solver only calculates overland flow for the cells that can potentially produce runoff, that is, only in the cells for which the water depth exceeds the detention storage depth.

Parameter Name	Type	Value
only simple OL from ponded	Boolean	On

Routing simple overland flow directly to the river

In the standard version of the Simplified Overland Flow solver, the water is routed from 'higher' zones to 'lower' zones within a subcatchment. Thus, overland flow generated in the upper zone is routed to the next lowest flow zone based on the integer code values of the two zones. In other words, at the beginning of the time step the overland flow leaving the upper zone (calculated in the previous time step) is distributed evenly across all of the cells in the receiving zone. In practice, this results in a distribution of water from cells in the upstream zone with ponded water (e.g. due to high rainfall or low infiltration) to all of the cells in the downstream zone with potentially a large number of those cells having a higher infiltration capacity. In this case, then, overland flow generated in the upper flow zone may never reach the stream network because it is distributed thinly across the entire downstream zone.

To avoid excess infiltration or evaporation in the downstream zone, an option was added that allows you to route overland flow directly to the stream network. In this case, overland flow generated in any of the overland flow zones is not distributed across the downstream zone, but rather it is added directly to the MIKE 11 stream network as lateral inflow.

Parameter Name	Type	Value
no simple OL routing	Boolean	On

14.2.4 Irrigation River Source Factors

A global "river source volume factor" and "river source discharge factor" are available as extra parameters for increased control of river sources during irrigation.

Parameter Name	Type	Value
river source volume factor	float	positive
river source discharge factor	float	0 or positive

None, one, or both can be specified. If the factor is not specified, then a Volume factor of 0.99 and a Discharge factor of 0.0 will be used.

The factors are used in the calculation of the available water (depth) of a river source:

$$Depth = MIN\left(\frac{C_s \cdot \Delta t}{A}, \frac{F_V \cdot V_L}{A} + \frac{F_D \cdot D_L \cdot \Delta t}{A}\right) \tag{14.1}$$

where *Depth* is the available water depth in the river link, C_s is the source capacity, Δt is the time step length, F_v is the specified volume factor, V_L is the volume of water in the link, F_D is the specified volume discharge, D_L is the river link discharge, and A is the cell area.

The river link discharge is the same as used when checking with the threshold discharge for switching on/off the source. It is the absolute discharge in the middle of the MIKE SHE river link, interpolated between two MIKE 11 H-points.

MIKE SHE prints the following message in the xxx_WM_Print.log file when the parameters are specified:

```
Extra-parameter specified:
river source volume factor
value = 1.500000

Extra-parameter specified:
river source discharge factor
value = 1.000000
```

MIKE SHE also prints the following warnings in the xxx_WM_Init_Messages.log file if one or both of the factors may result in water balance errors or numerical instabilities

```
WARNING: Specified value for river source volume factor
is greater than 1 : 1.500000.
There is a risk of water balance errors and/or insta-
bilities in the coupling between MIKE SHE and MIKE 11.

WARNING: Specified value for river source discharge
factor is greater than 0 : 1.000000.
There is a risk of water balance errors and/or insta-
bilities in the coupling between MIKE SHE and MIKE 11.
```

Note: This option is less useful now that River Sources are defined by both an Upstream and Downstream chainage. The option is maintained for backward compatability.

14.2.5 Explicit Overland Flow Output

If you are using the explicit overland flow solver, the time step depends on the location in the model with the critical courant criteria. The grid series output allows you to save the courant criteria, so that you can see where the critical locations are. However, the grid series output is an average courant number over the storing time step, where there can be hundreds of OL timesteps in a storing time step. If you are experiencing very short time steps due to short duration rainfall events, for example, the critical information can be difficult to distill from the dfs2 grid series output.

To make it easier to find the critical locations, an extra parameter option was added that writes out the critical locations at every time step, if the time step is reduced below a user-defined fraction of the storing time step.

Parameter Name	Type	Value
adaptive OL time step info threshold fraction	Float	between 0.0 and 1.0 Default = 0.01

The default value is 0.01. This means that if the reduced time step is less than 0.1 times the Max OL time step, then a message will be printed in the _WM.log file. Such as:

```
Adaptive time step info from Explictit OL solver:
OL step no: 59: ... Final time step = 1.8108 seconds
```

with the following four reasons:

```
Critical: OL Wave Courant number. Cell (8,21)...
Critical: Net outflow from OL cell to River. Cell (8,21) ...
Critical: Net OL outflow from cell. Cell: (17,19) ...
Critical: Net outflow from River to OL. River link between ...
```

If you experience frequent severe reductions in the OL time step when using the explicit OL solver, then this threshold can cause very large log files to be created. If you are not interested in this information, then you can reduce this threshold to reduce the frequency of the output.

14.2.6 Alternative low gradient damping function

In flat areas with ponded water, the head gradient between grid cells will be zero or nearly zero, which means that as the gradient goes to zero Δt also goes to zero. To allow the simulation to run with longer time steps and dampen any numerical instabilities in areas with low lateral gradients, the calculated intercell flows are multiplied by a damping factor when the gradients are close to zero.

Compared to the default damping function, an alternative damping function is available as an Extra Parameter that goes to zero more quickly and is consistent with the function used in MIKE FLOOD.

The alternative function is a single parabolic function (see Figure 11.5 in the Reference Manual)

To activate the alternate function, you must specify the following boolean parameter in the Extra Parameters *(V.2 p. 193)* dialog:.

Parameter Name	Type	Value
Enable Alternative Damping Function	Boolean	On

For more detail, see the section Low gradient damping function *(V.2 p. 272)* in the Reference manual.

14.2.7 Paved routing options

By default, the paved area function routes the available ponded water to the SZ drainage network. However, the available ponded depth does not include the detention storage.

If you want to route all of the ponded water in a cell - including the water in detention storage.- to the SZ drainage network, then you can define the following Extra Parameter:

Parameter Name	Type	Value
allow paved routing of detention storage	Boolean	On

There is an option to restrict the maximum drainage rate for paved drainage. If this is specified, then the actual drainage rate will not exceed this value. In Release 2012, this has been added to the user interface. .

Parameter Name	Type	Value
max paved flow rate mm/d	Float	greater than zero

14.3 Unsaturated Zone

14.3.1 Transpiration during ponding

In general, plants are not very tolerant of saturated soil in their root zone. Saturated soil is quickly depleted of oxygen and the roots will soon die. MIKE SHE normally takes care of this automatically by removing ET from ponded water before calculating transpiration from the unsaturated or saturated zones. If there is sufficient ponded water then the entire ET will be satisfied from the ponded water.

However, some plants, such as rice, are more tolerant of saturated soils and still extract ET from saturated soils, although normally at a reduced rate. If ET from the soil zone is ignored, then the distribution of water supplied to ET will be incorrect.

The transpiration during ponding option changes the order in which the ET is calculated. In this case, the ET rate is multiplied by an anaerobic tolerance factor and ET is removed from the soil before being removed from the ponded water.

Parameter Name	Type	Value
allow transpiration during ponding	Boolean	On
global anaerobic tolerance factor	Float	Greater than or equal to 0 Less than or equal to 1
optional (instead of global value)		
anaerobic tolerance factor dfs2 file name	file name	.dfs2 file
anaerobic tolerance factor item number	integer	item number in dfs2 file, greater than zero

14.3.2 Threshold depth for infiltration (2-Layer UZ)

The 2-Layer water balance method for the unsaturated zone does not include evapotranspiration from the soil surface. Thus, even a small amount of water on the ground surface will infiltrate. If you use this extra parameter, then you can define a depth of overland water that must be exceeded before infiltration will occur. This keeps small amounts of precipitation from infiltrating and allows them to evaporate instead.

The calculated infiltration is simply reduced if the remaining overland water depth will be smaller than the specified threshold value.

Parameter Name	Type	Value
use threshold depth for infiltration	Boolean	On
threshold depth for infiltration meter	Float	Greater than zero

Note: This option is less useful with the ET Deficit Factor introduced in the 2008 Release, which maintains ET at the full rate until the specified deficit is reached. The option is maintained for backward compatability.

14.3.3 Increase infiltration to dry soils

In dry soils the rate of infiltration can be higher than the saturated hydraulic conductivity because capillarity will draw water into the soil and increase the rate of infiltration. The Increase Infiltration to Dry Soils extra parameter is available to account for this process, when Richards equation is not being used.

If the actual water content in the root zone is below the field capacity, θ_{fc}, then the infiltration capacity is calculated as

$$K_{infiltration} = K_{infiltration} \cdot InfiltrationFactor \tag{14.2}$$

if

$$\theta_{actual} > \frac{\theta_{fc} - \theta_{wp}}{InfiltrationFactor} + \theta_{wp} \tag{14.3}$$

where θ_{wp} is the wilting point water content.

Otherwise, the infiltration capacity is calculated as

$$K_{infiltration} = K_{infiltration} \cdot \left(\frac{\theta_{fc} - \theta_{wp}}{\theta_{actual} - \theta_{wp}}\right) \tag{14.4}$$

Parameter Name	Type	Value
increase infiltration to dry soils	Boolean	On
max infiltration rate factor	Float	Greater than 1.0

14.4 Saturated Zone

14.4.1 Sheet Pile Module

The Sheet Piling module is not yet included in the MIKE SHE GUI. However, the input for the module is fairly simple and is handled via the Extra Parameters options

The Sheet Piling module is activated by including the following two parameters in the Extra Parameters section of the data tree, and creating the required module input file:

Parameter Name	**Type**	**Value**
sheet piling module	Boolean	On
sheet piling file	file name	the file name of the Sheet Pile input file

Sheet Pile Location

The location of the sheet piles is defined using a dfs2 file with integer grid codes. One file (or item) is required for each computational layer with sheet piling. Each file must have the same grid size as the MIKE SHE model. The grid codes are “composed” of simple sums of 100, 10, 1, 0 where:

100 = a N-S sheet piling “link” between the actual cell and the next cell in positive x-direction,

10 = a E-W sheet piling “link” between the actual cell and the next cell in the positive y-direction,

1 = a Horizontal sheet-piling “surface” between the actual layer and the layer above (ground surface if actual layer is 1), and

0 = no sheet piling.

Thus, for example, a cell containing the code “110” defines the existence of sheet piling along the Eastern and Northern cell boundaries. A cell containing the code “11” defines a sheet piling along the Northern cell boundary and at the top of the layer.

Leakage Coefficient

The Leakage Coefficient is required for flow in the x-, y-, and z-direction for each layer containing sheet piling. The Leakage Coefficient is required in the x-direction if any cell contains a "100" value, in the y-direction if any cell contains a "10" value, and in the z-direction if any cell contains a "1" value.

The leakage coefficients can be specified as a global value (per layer) or as a distribution in a dfs2 file. In the case of a dfs2 file, the values must be specified in the cells where the grid codes are specified. The EUM type (unit) of the dfs2 files must be "Leakage coefficient/Drain time constant" with the unit 1/Time.

Top and bottom levels (optional)

This option can be used when the vertical sheet piling only extends across part of a layer. The levels are specified in the same cells as the leakage coefficients in the x- and y-direction, one set of top and bottom levels for each direction.

The levels can be specified as global values (per layer) or as a distribution in a dfs2 file. Both can be absolute levels or relative to ground. The EUM type of the dfs2 files must be "elevation" for absolute levels, and "depth below ground" (positive values) or "height above ground" (negative values) when specified relative to the ground surface. The type and unit of the global value is "elevation" (m) when absolute, and "height above ground" (m) (negative value) when relative.

In cells where the sheet pile extends across the entire layer, the top and bottom levels should simply be set to large positive and negative values respectively (e.g. 1.0E+30 and -1.0E+30).

Input File for the Sheet Pile Module

The name of the input file is specified in the Extra Parameters section described above. The file has the general MIKEZero parameter file (pfs) format. The exact format of the file is given below, along with a description of the different data items.

Note: The pfs format must be adhered to exactly. There is a small utility (pfsEditor.exe) in the installation \bin directly that you can use for editing and testing pfs files that you create.

.

Line item	Comment
[MIKESHE_SheetPiling_File] FileVersion = 2 [SheetPiling]	FileVersion can be 1 or 2, but must be 2, if you want to check for the SpecifiedXYLevels option
NrOfLayers = 1	Total number of SZ layers with sheet piling
SpecifiedXYLevels = 1	0: not specified. 1: top and bottom levels specified for each layer Note: only checked when FileVersion > 1
[Layer_1]	This section must be repeated for each -NrOfLayers- sheet piling layer. The sections must be named Layer_1, Layer_2, etc.
LayerNumber = 1	The MIKE SHE SZ layer number of the actual sheet piling layer (1 = top layer).
[GridCodes] Type = 1 FixedValue = 0 [DFS_2D_DATA_FILE] FILE_NAME = |.\SPGrid_1.dfs2| ITEM_COUNT = 1 ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // GridCodes	**[GridCodes] section** Specification of grid codes for the current layer. **Type** Normally 1 because a dfs2 file is required. 0 means global value. **FILE_NAME** Name of the dfs2 file with grid codes. The file name is enclosed in "|" which tells the system that the name is relative to the location of this module input file. **ITEM_NUMBERS** : One number (because ITEM_COUNT must be 1) defining the item of the dfs2 file to be used.

Line item	Comment
[X_Leakage] Type = 0 FixedValue = 1.0E-7 [DFS_2D_DATA_FILE] FILE_NAME = |.\maps\SPLeakX_1.dfs2| ITEM_COUNT = 1 ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // X_Leakage	**[X_Leakage] section** Required if there are any cells with N-S sheet piling affecting the flow in the x-direction (codes containing 100). **Type** Set to 0 if a global value is specified and 1 if using a dfs2 file. **FixedValue** The global value (1/s) which is read if Type = 0. **FILE_NAME** and **ITEM_NUMBERS** Dfs2 file name and item number if Type = 1 (relative file name as explained under Grid Codes).
[Y_Leakage] Type = 0 //(0:Fixed value,1:DFS2 file) FixedValue = 2.0E-7 [DFS_2D_DATA_FILE] FILE_NAME = |.\maps\SPLeakY_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 1 1 EndSect // DFS_2D_DATA_FILE EndSect // Y_Leakage	Y_Leakage] section : Required if there are any cells with E-W sheet piling affecting the flow in the y-direction (codes containing 10).
[Z_Leakage] Type = 0 //(0:Fixed value,1:DFS2 file) FixedValue = 3.0E-7 [DFS_2D_DATA_FILE] FILE_NAME = |.\maps\SPLeakZ_1.dfs2| ITEM_COUNT = 1 ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // Z_Leakage	[Z_Leakage] section : Required if there are any cells with horizontal sheet piling affecting the vertical flow (codes containing 1).
[X_TopLevel] RelativeToGround = 0 // 0: no, 1: yes Type = 1 //(0:Fixed value,1:DFS2 file) FixedValue = 0.0 [DFS_2D_DATA_FILE] FILE_NAME = |.\YLevels_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // Y_TopLevel	[X_TopLevel] section : Required if SpecifiedXYLevels=1 and there are any codes containing 100.

Line item	Comment
[X_BottomLevel] RelativeToGround = 0 // 0: no, 1: yes Type = 1 //(0:Fixed value,1:DFS2 file) FixedValue = 0.0 [DFS_2D_DATA_FILE] FILE_NAME = |.\YLevels_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 2 EndSect // DFS_2D_DATA_FILE EndSect // Y_BottomLevel	[X_BottomLevel] section : Required if SpecifiedXYLevels=1 and there are any codes containing 100.
[Y_TopLevel] RelativeToGround = 0 // 0: no, 1: yes Type = 1 //(0:Fixed value,1:DFS2 file) FixedValue = 0.0 [DFS_2D_DATA_FILE] FILE_NAME = |.\YLevels_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // Y_TopLevel	[Y_TopLevel] section : Required if SpecifiedXYLevels=1 and there are any codes containing 10.
[Y_BottomLevel] RelativeToGround = 0 // 0: no, 1: yes Type = 1 //(0:Fixed value,1:DFS2 file) FixedValue = 0.0 [DFS_2D_DATA_FILE] FILE_NAME = |.\YLevels_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 2 EndSect // DFS_2D_DATA_FILE EndSect // Y_BottomLevel	[Y_BottomLevel] section : Required if SpecifiedXYLevels=1 and there are any codes containing 10.
EndSect // Layer_1	
EndSect // SheetPiling	
EndSect // MIKESHE_SheetPiling_File	

14.4.2 SZ Drainage to Specified MIKE 11 H-points

The Reference Drainage (RFD) option allows you to route drainage from the saturated zone drains and paved area runoff directly to MIKE 11 H-points. This is different from the normal drainage function, which routes drainage and paved area discharges to river links rather than directly to H-points. Further, this option can route drainage to MIKE 11 branches that are not defined in the MIKE SHE coupling section of the MIKE 11 network file.

The following steps are required to activate the RFD option:

1 Create a pfs file containing information for each specified drainage area to be routed to the specific MIKE 11 H-points.

Line item	Comment
[MIKESHE_MIKE11DrainageReach_File]	
[SpecifiedMIKE11ReachesForDrainage] NrOfReaches = 1 RiverChainageUnit = 'meter'	NrOfReaches is the number of items specified in the section below
[Reach_1] DrainCode = 1 BranchName = 'Lammehavebækken' Upstream_Chainage = 6000. Downstream_Chainage = 8459 EndSect // Reach_1	For each specified reach, you must include a section specifying the MIKE SHE drain code, and the MIKE 11 branch name and the upstream and downstream chainage.
EndSect // SpecifiedMIKE11ReachesForDrainage	
EndSect // MIKESHE_MIKE11DrainageReach_File	

The drain code references the area that drainage and/or paved area discharge is routed to the specified MIKE 11 branch and chainage. The drain code must be greater than or equal to zero. Drain code values equal to zero (0) are not included in the reference drainage system. Furthermore, an error condition will occur if the specified drain code does not exist in the drainage code file used in MIKE SHE

The branch name must be spelled correctly and include all spaces contained in the name, if any. The branch name should not be enclosed in quotes. An error condition will occur if the specified branch is not present in the MIKE 11 network.

The chainages refer to the starting and ending chainage of the specified branch which drainage and/or paved area discharge is routed to. The interval does not have to correspond exactly to specific MIKE 11 H-points because the MIKE SHE pre-processor finds the closest H-points to the specified interval. If the upstream and downstream chainages are the same, the drainage and/or paved area discharge is routed to the closest H-point.

2 Add the following items to the Extra Parameters list

Parameter Name	Type	Value
use specified reaches for drainage	Boolean	On
specified reaches for drainage	file name	the pfs file name, including the path

3 In the Drainage item under the Saturated Zone, select **distributed drainage options**. See Drainage *(V.2 p. 173)*.

4 Specify drain codes is the same manner as usual. Remember that all drain codes in the RFD option pfs file must exist in the active domain of the model or you will get an error.

5 Specify where the RFD option should be used in Drainage Distribution item in the data tree under the Saturated Zone. The RFD option will be used in all cells with a value of 3. If a combination of the original drainage method and the RFD option is going to be used, 2 should be used for areas using the original drainage option and 3 should be used where you want the RFD option to be used.

6 Pre-process and run your MIKE SHE model normally.

If the MIKE SHE setup does not successfully preprocess you should review the above steps to see if you have any error in the setup. The *projectname*_PreProcessor_Messages.log file (where *projectname* is the name of your *.she file) in your simulation subdirectory should help you identify why the MIKE SHE setup failed to preprocess.

If the MIKE SHE setup successfully preprocesses you should also look at the preprocessed data (on the Processed data tab) and the YourSetup_PreProcessor_Print.log file in your simulation subdirectory to make sure you are comfortable with how the preprocessor has set up the drainage reference system. You can search for *Making setup of Specified MIKE 11 Reaches For Drainage* in the YourSetup_PreProcessor_Print.log file to find the start of the section that details the drainage reference system.

Water balance

The water balance utility (e.g., Saturated zone - detailed) can be used to look at differences between drainage discharges from areas using the original drainage option and the RFD option. The MIKE SHE water balance configuration file (MSHE_Wbl_Config.pfs in the installation directory) should be reviewed to see which water balance types segregate standard drainage flow (data type sz.qszdrtorivin) and RFD drainage flow (data type sz.qszdrtoM11Hpoint) (see Using the Water Balance Tool *(V.1 p. 105)*)

14.4.3 SZ Drainage Downstream Water Level Check

In Release 2011, you can optionally check the downstream water level before calculating SZ drainage. This prevents drainage from being added to rivers during a flood, for example. It also prevents recirculation of SZ drainage water when using Flood Codes.

Testing has shown that the test on drainage to local depression can negatively impact runtimes because the number of outer iterations in the PCG solver may increase. Thus, the downstream check has been seperated into two Extra Parameters.

Parameter Name	**Type**	**Value**
check gradient for drainage to river or mouse	Boolean	On
check gradient for drainage to local depression	Boolean	On

14.4.4 SZ Drainage to MOUSE

The MOUSE coupling in MIKE SHE has not yet been added to the MIKE SHE user interface. Thus, to couple the models together, use the Extra Parameters options, along with creating a *MsheMouse.pfs* file to define where and how the two models are coupled.

To tell MIKE SHE that it needs to couple to a MOUSE model, you must add the following two items:.

Parameter Name	**Type**	**Value**
mouse coupling	Boolean	On
mouse coupling file	file name	the file name of the MOUSE coupling .pfs input file

The MIKE SHE - MOUSE coupling is described fully in Using MIKE SHE with MIKE URBAN *(V.1 p. 229)*.

14.4.5 Time varying drainage parameters

In projects where you want to simulate the build out of a drainage network over time, or changes in the drainage time constants over time, then you can use this set of extra parameters. Without this set of extra parameters you would have to hot start your simulation at regular time intervals with the new drainage parameters.

The time varying drains are also allowed to shift between layers. However, if the drainage level goes above or below the model, the level will be adjusted and a warning is issued to the log file.

Note: The SOR solver does not allow drainage in any layer except the top layer and the drain level will be adjusted accordingly.

Note: If you specify time varying drainage parameters, you will not be able to use any of the drainage routing methods that depend on the drain level. The preprocessor checks this and gives an error if you have specified

– option 1 (routing based on levels), or

– option 3 (distributed options) AND any of the distributed option codes are 1 (routing based on levels in these cells).

To activate time varying drainage parameter options, you must specify the following extra parameters

Parameter Name	Type	Value
time varying drainage levels	Boolean	On
time varying drainage constants	Boolean	On
time varying drainage level dfs2 file name	file name	.dfs2 file
time varying drainage level item number	integer	item number in dfs2 file, greater than zero

Parameter Name	Type	Value
time varying drainage time constant dfs2 file name	file name	.dfs2 file
time varying drainage time constant item number	integer	item number in dfs2 file, greater than zero
Optional if mean step accumulated values instead of instantaneous values:		
mean step accumulated drainage levels	Boolean	On
mean step accumulated drainage time constants	Boolean	On

The dfs2 Drain Level is an elevation that can be specified using the following three EUM Data Units *(V.1 p. 329)*:

- Elevation
- Depth Below Ground (i.e. positive values)
- Height Above Ground (i.e. negative values)

By default, the Time Series Types *(V.1 p. 342)* is Instantaneous, but their is an extra option that allows you to used Mean Step Accumulated values if you want.

Note The code does not check for the time series type.

All specifications are printed to the *projectname*_PreProcessor_Print.log and *projectname*_WM_Print.log files.

14.4.6 SZ Drainage River Link Reference Table

In the pre-processing tab, the Drain to River grid displays the river link number that the cell drains to. Adjacent to the river links, the cells are labeled with negative numbers to facilitate the interpretation of flow from cells to river links. Thus, in principle, all drainage from cells with the same positive code are drained to the cell with the corresponding negative code.

However, this is slightly too simple because the cells actually drain directly to the river links. In complex river systems, when the river branches are close together, you can easily have cells connected to multiple branches on different sides. In this case, the river link numbers along the river may not reflect the drainage-river link reference used in the model.

If you want to see the actual river links used in all cells, you can use the following Extra Parameter to generate a table of all the river link-cell references in the PP_Print.log file. This table can easily be several thousand lines long. .

Parameter Name	Type	Value
drainage setup test print value	Boolean	On

14.4.7 Canyon exchange option for deep narrow channels

In the case of a deep, narrow channel crossing multiple model layers, the head difference used in Equations (7.5) and (7.6) can optionally be limited by the bottom elevation of the layer. Thus,

$$\Delta h = h_{grid} - max(h_{riv}, z) \tag{14.5}$$

where z is the bottom of the current layer.

The above formulation reduces the infiltration from upper layers by reducing the available gradient. Without the ‘Canyon’ option, MIKE SHE effectively assumes that the river is hydraulically connected to the upper most model layer, since MIKE SHE calculates the exchange flow with all layers that intersect the river based on the difference between the river level and the water table.

Currently, this option is only available for steady-state models.

Parameter Name	Type	Value
enable canyon exchange	Boolean	On

14.5 Water Quality

14.5.1 Disable SZ solute flux to dummy UZ

The following Extra Parameter is useful, if you are using an alternative UZ model, such as DAISY, in MIKE SHE and you are trying to couple it to the WQ.

In this case, you will be typically using the Negative Precipitation *(V.1 p. 300)* option. If you use this option, then you will not use a MIKE SHE UZ, and the UZ-SZ exchange will pass through a “dummy UZ” layer. When this is coupled to the water quality, solutes will also be passed to this dummy UZ layer and removed from the SZ domain and the model.

To prevent the upflow of solutes from SZ to the dummy UZ, you must specify the following Extra Parameter..

Parameter Name	Type	Value
disable sz transport to dummy uz	Boolean	On

14.5.2 SZ boundary dispersion

A detailed test of the MIKE SHE WQ engine comparing an SZ model with fixed concentration at an inflow boundary with an analytical solution for a fixed concentration source, showed that MIKE SHE under-estimates the mass flux into the model when the model includes longitudinal dispersion.

The problem is that the SZ transport scheeme (QUICKEST) doesn't include dispersive transport to/from open boundary cells. This is as designed, but apparently not correct. After including the boundary dispersion, the mass input to the model is within 2 % of the analytical solution.

From Release 2011 and onwards, the boundary dispersion has been made optional for backwards compatibility and is activated with the extra-parameter: .

Parameter Name	Type	Value
enable sz boundary dispersion	Boolean	On

However, the SZ boundary dispersion option (above) does not calculate dispersive transport to an inflow boundary correctly. Again, this problem was identified in the tests of MShe_WQ with ECOLab vs analytical solution. For example:

- Species 1 enters the model via an inflow (flux) boundary with fixed concentration - including dispersive transport due to the new sz boundary dispersion option.
- Species 1 decays to Species 2 which again decays to Species 3.
- The concentrations of Sp2 & Sp3 are too high, especially close to the inflow boundary.

The analytical solution includes dispersive transport of Sp2 & Sp3 against the flow direction because the concentration of these species are 0 at the boundary. However, this dispersive mass flux to the boundary is not included in the SZ solver due to an old check in the code. When mass flux to/from a boundary point is reversed compared to the flow direction, the mass flux is simply reset to 0.

This made sense before the boundary dispersion was implemented because advective transport against the flow direction would be wrong. But, now, when the boundary dispersion is active, this situation is allowed.

14.6 Miscellaneous

14.6.1 Including OpenMI

If you want to link a program to MIKE SHE using OpenMI then you must specify the following Extra Parameter..

Parameter Name	Type	Value
make omi file	Boolean	On

When enabled, an *.omi file for WM is created called MIKESHE_WM_SetupName.omi.

If Water Quality is included, a second *.omi file is created called MIKESHE_WQ_SetupName.omi.

These omi files are to be used in the OpenMI configuration editor.

14.6.2 Plot control for Detailed Time Series Output

On the Results Tab, the Detailed Time Series plots are created in a set of .html files. The default file length is 5 plots per file. However, you can control the number of plots per html file by using the following Extra Parameter. .

Parameter Name	Type	Value
max number of detailed ts plots per html file	Integer	Greater than or equal to 1

Note If the loading of the html file can become very slow if the simulation is long and there are many plots in the file.

14.6.3 Extra Pre-Processing output

The pre-processing log file, *_PP_Print.log, can be very long. To improve the readability of the file, some long tables have been removed, including the tables for drainage references.

To include these tables in the log file, use the following extra parameter: .

Parameter Name	Type	Value
detailed setup test print	Boolean	On

14.6.4 GeoViewer Output

The GeoViewer is a MIKE Zero tool that is used in the MIKE GeoModel product for viewing geologic cross-sections in your conceptual model.

The GeoViewer Output extra parameters will create a set of dfs2 output files during the pre-processing that will allow you to look at your pre-processed model in the GeoViewer.

The GeoViewer Output is activated by

Parameter Name	Type	Value
make SZ level dfs2 files	Boolean	On
Optional		
adjust dfs2 levels	Boolean	On

If this option is active, then the following files will be created:

- *setupname\setupname*_GeoLayers.dfs2 - containing the top and bottom of each geologic layer

If there are lenses:

- *setupname\setupname*_GeoLenses.dfs2 - containing the top and bottom of each geologic lense and delete values where there are no lenses

If the computational layers are not defined by geologic layers:

- *setupname\setupname*_CompLayers.dfs2 - containing the top and bottom of each computational layer

If the optional second parameter is used, then the top and bottom elevations that are written to the files will be adjusted to be confined between the topography and the lowest computational layer.

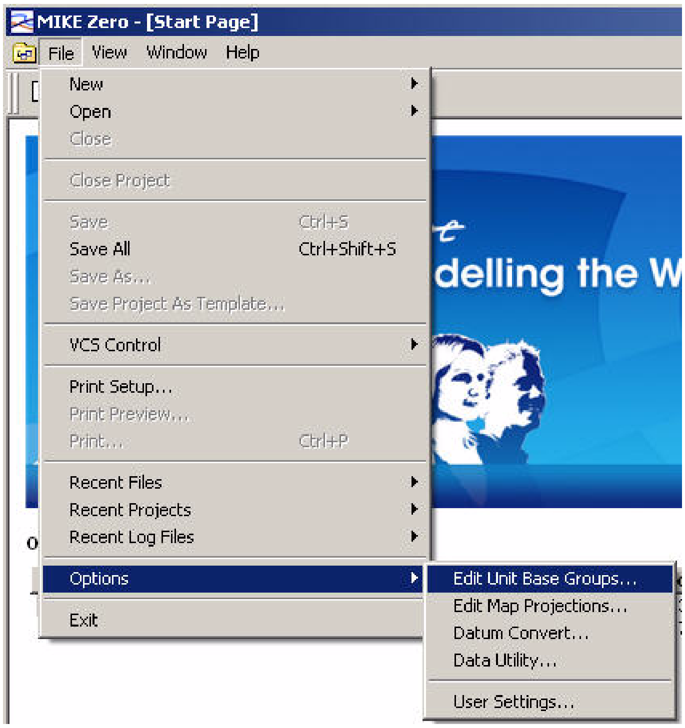
MIKE ZERO OPTIONS

15 EUM DATA UNITS

All MIKE Zero products use a standard library of data units, called the Engineering Unit Management (EUM) library. This allows you to change the displayed units for any value that is included in the library.

Every parameter in MIKE SHE has been added to the EUM library and to change the displayed unit, you must know the EUM Data Type. In most cases, the EUM Data Type is displayed in the fly-over text when you put your mouse cursor in the text field. Alternatively, all items in the on-line help (F1) list the EUM Data Type in the table at the beginning of the section.

To change the display units of any EUM Data Type, you must close all open documents and then select 'Options/Edit Unit Base Groups...' from the File pull down menu.



When you select this menu item, the Unit Base Group Editing dialogue appears. By default all of the data units for each active module are displayed. For a clearer overview of the data types, close all of the model engines that are not relevant.

Next select the data item that you want to change the units of. Then select the new units from the combobox list of available units.

After you have changed the data units, click 'Save and Close'. This saves your changes to the default Unit Base Groups (.ubg) file:

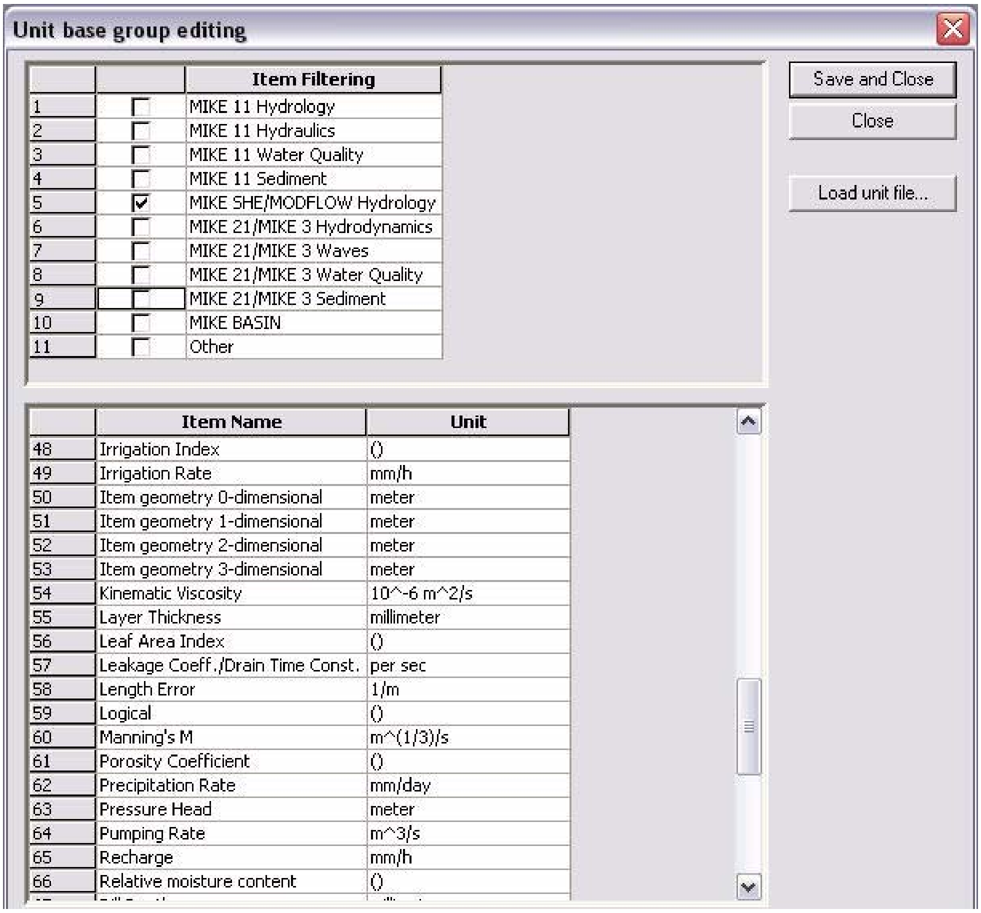
C:\Program Files\Common Files\DHI\MIKEZero\MIKEZero.ubg

which is read every time you open a model.

Note! If you have already added data to your model, changing the Unit Base Group will not convert any of your data. This process simply changes the displayed units in the user interface and the conversion factors used to make the input files internally consistent.

In some cases the relevant data item name is not clear, as there may be several data items with similar names. This is more likely to occur if several modules are selected at the same time. To find out which data item is correct, close the dialogue and re-open your model. Then either move the mouse to the relevant textbox, where a fly-over text box should appear telling you what is the relevant data type for this field. Alternatively, for gridded data, you can use the Create button to create a data file and then notice the data type that is displayed in the dialogue.

Finally, occasionally, you may find that the data unit that you are looking for is not available. In this case, contact your local Technical Support Cen-



tre, who should forward your request to the developer for inclusion in the next release.

15.1 Changing from SI to Imperial (American) data units.

The default Unit Base Groups (.ubg) file,

C:\Program Files\Common Files\DHI\MIKEZero\MIKEZero.ubg

is read every time you open a model.

In the same directory there are two standard Unit Base Group files:

MIKEZero_Default_Units.ubg

MIKEZero_US_Units.ubg

The first is the default file and contains standard SI units for all data items in all of the MIKE Zero products. The second contains standard Imperial (US) units for most data items in all of the MIKE Zero products.

To change the display units for all of your data items to Imperial units, load the MIKEZero_US_Units.ubg file, Save and Close the dialogue and then reopen your model.

If you want to change individual data items to SI or Imperial, you can change the items individually. Then use the Save and Close button to save your changes back to the MIKEZero.ubg file. If you want to create special unit versions, then you can copy the MIKEZero.ubg to a different file name and reload it.

15.2 Restoring the default units

You can return to your default unit specification at any time, by Loading either of the default .ubg files:

MIKEZero_Default_Units.ubg

MIKEZero_US_Units.ubg

which are found in the

C:\Program Files\Common Files\DHI\MIKEZero\

directory.

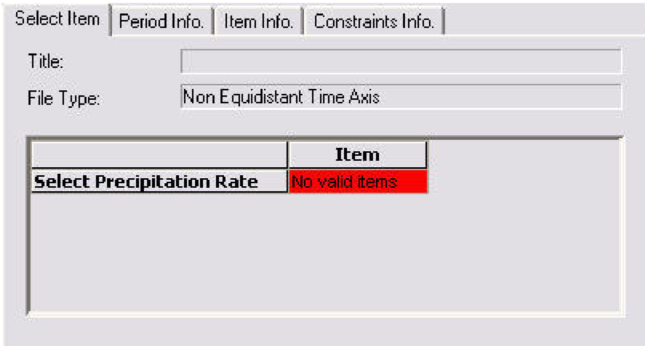
Note! If you want to save any of your model specific changes, then you should first save the MIKEZero.ubg to a new name.

15.3 Changing the EUM data type of a Parameter

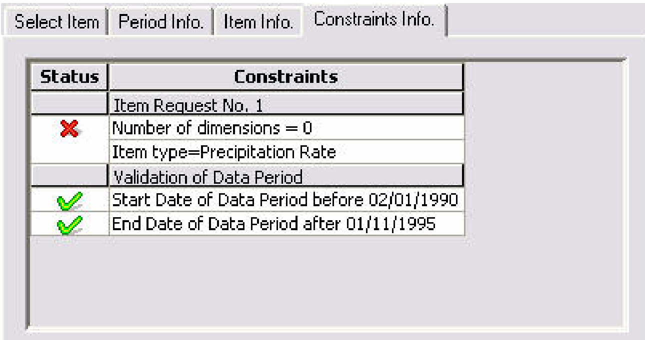
When you create a .dfs0 or .dfs2 parameter file, you must also define the EUM data type for each parameter in the file. When you assign a .dfs0 or a .dfs2 file to a parameter value, then MIKE SHE automatically verifies that the correct EUM data type is being used. If the wrong data type is present then you will not be able to select OK in the file browser dialogue.

For example, in the following set of dialogues, an Evapotranspiration time series was selected instead of the correct Precipitation time series file

The first error is in the Select Item tab, where there is a message that no Valid Items are found.

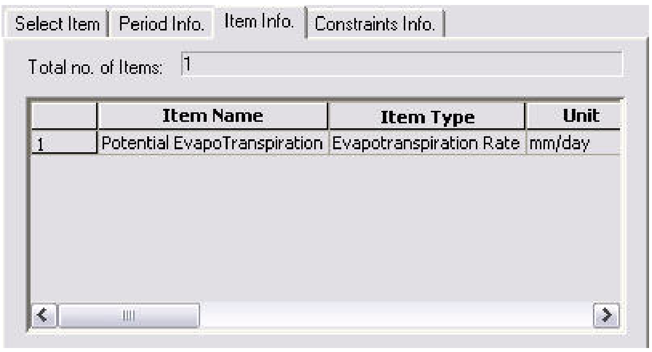


The find out why there is no valid items, you should look in the Constraints Info tab



Here you can see that the Item type is supposed to be Precipitation Rate, but this constraint has failed.

To find out what the Item Type of the selected file is, look at the Item Info tab,

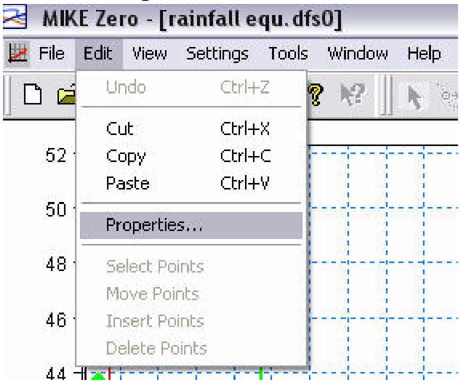


where you can see that the current Item Type is Evapotranspiration Rate.

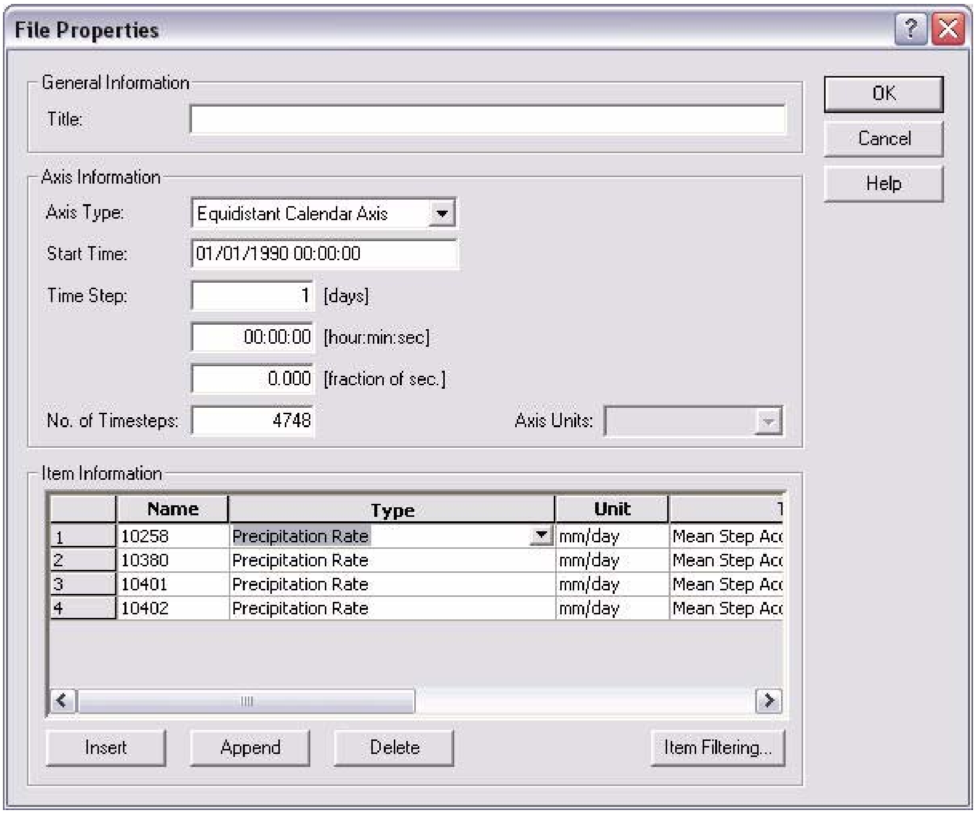
The next two sections outline how to change the EUM Type of an existing file.

15.3.1 Changing the EUM Type of a .dfs0 Parameter

To change the EUM Data Type of a parameter in a .dfs0 file, open the time series in the Time Series Editor and then select the Properties... item from the Edit drop down menu



This opens the item properties dialogue

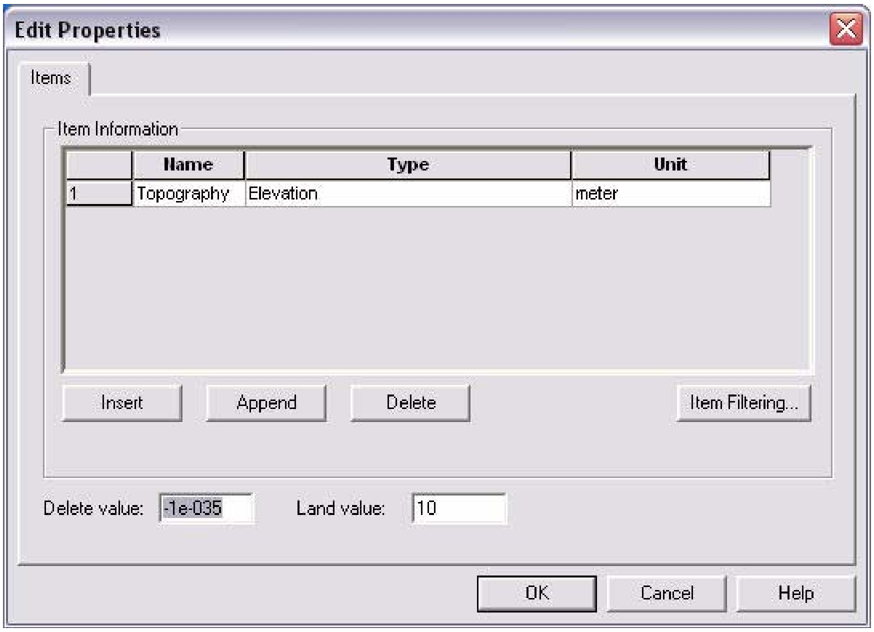


where you can change the EUM Type and the EUM Unit that is assigned for each time series in the file.

15.3.2 Changing the EUM Type of a .dfs2 Parameter

To change the EUM Data Type of a parameter in a .dfs2 file, open the grid file in the Grid Editor and then select the Items... item from the Edit drop down menu

This will open the Edit Properties dialogue for the Grid Editor



where you can change the EUM Type and the associated data EUM Unit of the item.

WORKING WITH DATA

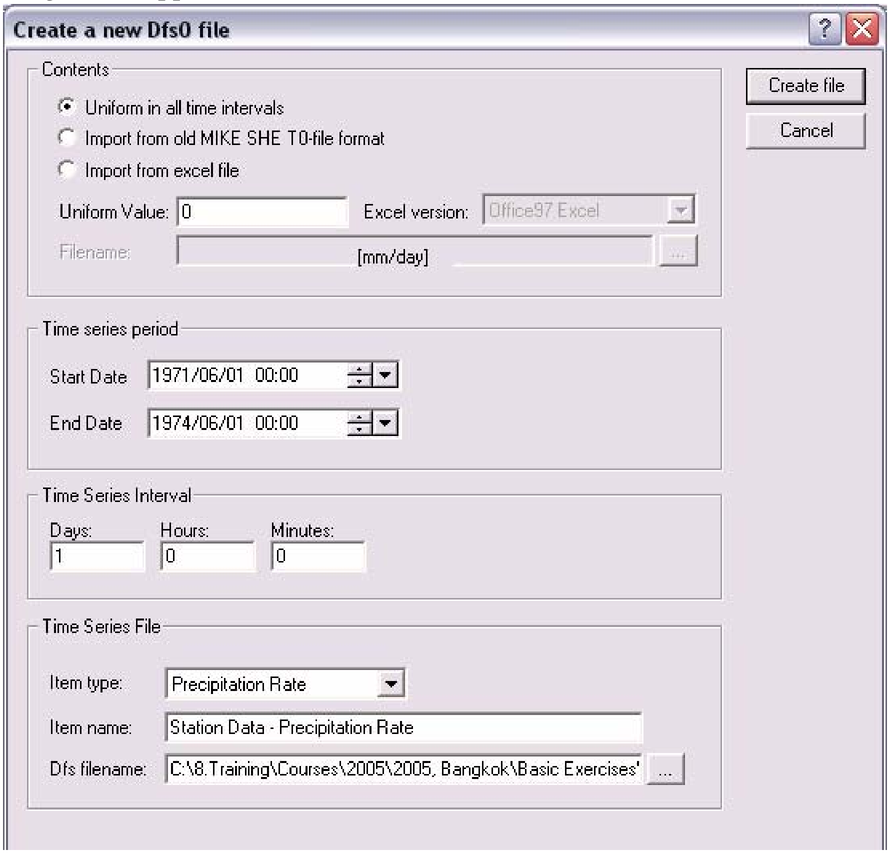
16 TIME SERIES DATA

MIKE SHE uses the dfs0 file format for time series data. Various tools are available for converting ASCII and EXEL time series to the dfs0 file format. Time series data is required as input for most transient simulations, for example, daily records of precipitation. Transient simulations can also generate numerous dfs0 output files.

16.1 Creating Time Series in MIKE SHE

In most cases, you will create dfs0 files using the Create buttons in the MIKE SHE Setup dialogues. In this way, you can avoid the confusing task of assigning the Type of time series (e.g. precipitation) and EUM Unit type (e.g. millimetres) and the TS Type (e.g. reverse step accumulated). Each of these items are specified automatically.

If you create time a time series using a Create button, the following dialogue will appear:



The principle choice in this dialogue is whether to create an initially uniform time series file or to import a time series from an Excel file or from a file with the older .t0 file format.

Uniform time series

In a uniform time series, every time step will have the same value. You should use the uniform time series option if you want to create a time series file where you do not have any data to import.

Time Series Period

The time series period is the extent of the time series. In a MIKE SHE simulation, all the time series files must cover the Simulation Period *(V.2 p. 29)*. The default time series period for a new time series file is the Simulation Period. However, if you change the time series period so that it does not cover the simulation period, you will receive an error message when MIKE SHE tries to run. If you try to add a time series file that does not cover the simulation period, then the OK button will remain greyed out and you will not be able to select the file. The constraints tab in the file selector dialogue gives you the reason that you cannot select the file.

Time Series Interval

The time series interval is the length of the individual time periods. The number of time periods is the length of the time series period divided by the period interval. The last period is shortened if necessary.

Time Series File

Every time series has an **Item Type** which is defined by the valid EUM Data Unit (see EUM Data Units *(V.1 p. 329)*) for the particular variable from which the Create dialogue was launched. In most cases, there is only one valid Type. In some cases you may have a choice. For example, in Precipitation, you can chose between Precipitation Rate, which is the average amount of precipitation per time (e.g. mm/hour) in the time interval, and Rainfall.which is the measured amount of precipitation in the time interval (e.g. mm).

The **Name** is simply the name of the data item in the resulting .dfs0 file.

The **file name** has a default value, that you should change if you will be creating several files of the same type, such as multiple rain gauge time series files. Otherwise you may accidentally overwrite the previous file.

16.1.1 Import from ASCII

The easiest way to import ASCII data into a dfs0 file is via the Windows clipboard. In this case, create a uniform time series file with the correct number of time steps and then highlight all of the data values. Then copy and paste the data from the ASCII file into the table.

However, if you want to import the data from an ASCII file, then you need to create the file from the File/New menu and choose ASCII file. This is part of the Time Series Editor itself.

16.1.2 Import from Excel

Only the first Excel Worksheet will be read when reading the Excel file. However, the worksheet can contain any number of columns of time series data. If there are multiple columns of data, each will be assumed to be the of the same type. If the Excel file columns are of different types, then you can change the data type in the Time Series Editor.

The time series is assumed to have a non-equidistant time axis and the time series period is read from the first column of the Worksheet.

Worksheet Format

The first row is a header containing the names of each of the columns. Each subsequent row contains the data. The first column is the date and time (with DATE or TIME cell format), followed by the data values.

	name1	namel2	name3
01/01/1981 00:00:00	0.1	0.2	0.3
02/01/1981 00:00:00	0.304	0.304	0.304
03/01/1981 00:00:00	0.025	0.025	0.025
04/01/1981 00:00:00	0.604	0.604	0.604

16.1.3 Import from old .t0 file

The old .t0 file format is from the X-Motif version of MIKE SHE that existed before the Windows version was introduced in 2001. The .t0 file format contains all of the relevant time information. For more information on the .t0 file format, please refer to your original MIKE SHE documentation.

16.2 Working with Spatial Time Series

In the MIKE SHE Toolbox, there is a Tool in the File Converter section called ***dfs2+dfs0 to dfs2***. In this utility you specify a dfs2 grid file with integer grid codes and a dfs0 file with time series data, where the dfs2 file grid codes are the item numbers in the dfs0 file.

The utility will read the dfs2 file and for each time step in the dfs0 file, it will substitute the grid code with the time series value.

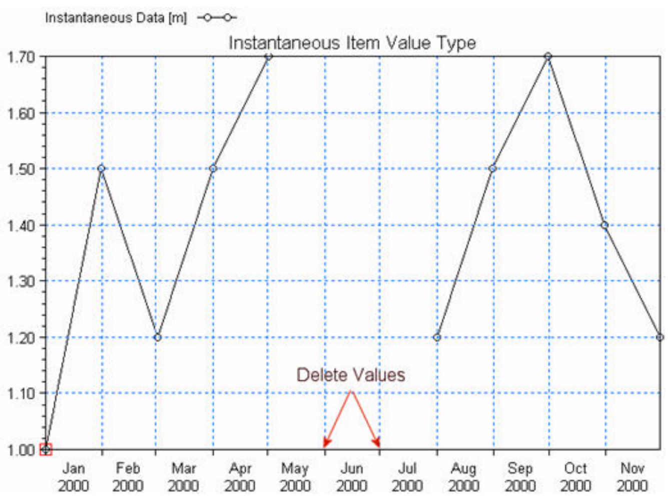
The result is a dfs2 file with one grid for each time step and the grid values are the time series values.

16.3 Time Series Types

Specifies how the time step is being defined and how the measured value is being assigned to the time step. There are five different value types available:

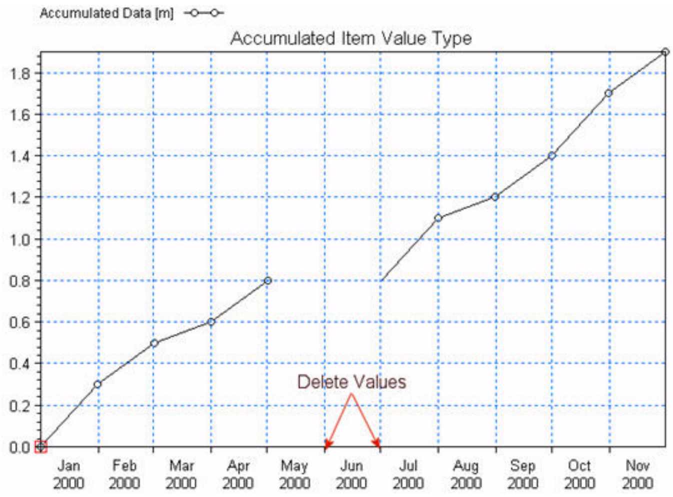
Instantaneous

The values are measured at a precise instant. For example, the air temperature at a particular time is an instantaneous value.



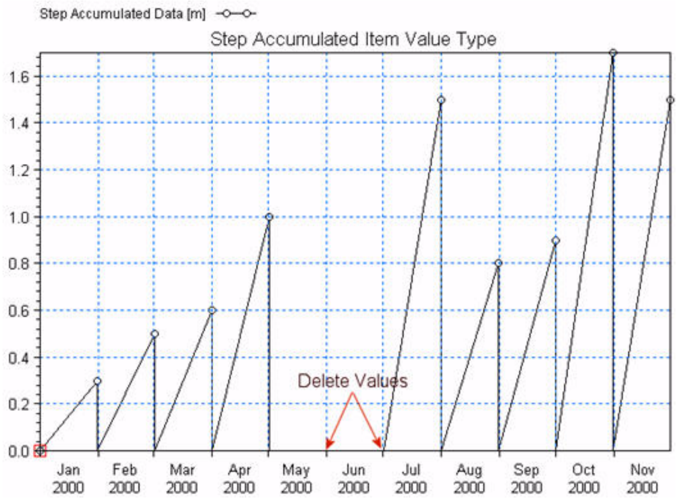
Accumulated

The values are summed over successive intervals of time and always relative to the same starting time. For example, rainfall accumulated over a year with monthly rainfall values.



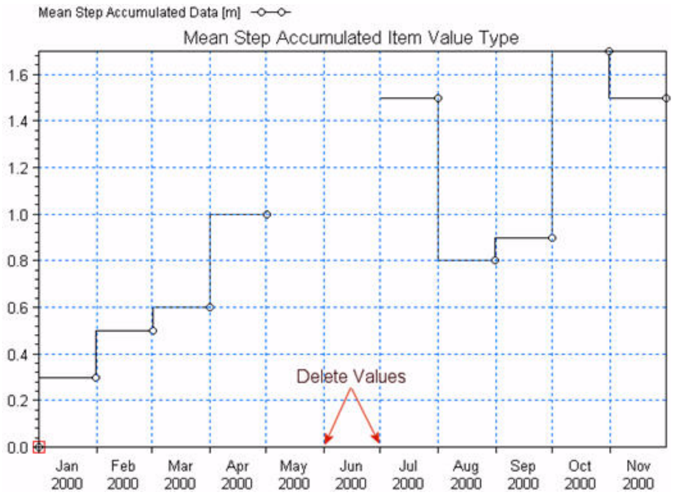
Step Accumulated

The values are accumulated over a time interval, relative to the beginning of the interval. For example, a tipping bucket rain gauge measures step-accumulated rainfall. In this case, the rain gauge accumulates rainfall until the gauge is full, then it empties and starts accumulating again. Thus, the time series consists of the total amount of rainfall accumulated in each time period - say in mm of rainfall.



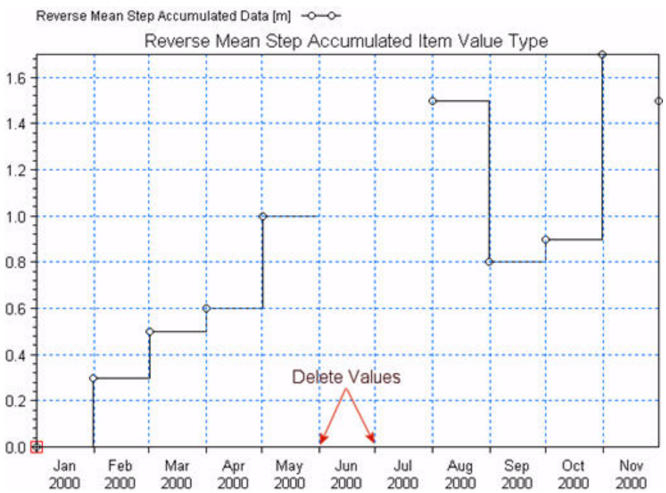
Mean Step Accumulated

The values are accumulated over the time interval as in the Step Accumulated, but the value is divided by the length of the accumulation period. Thus, based on the previous example, the time series consists of the rate of rainfall accumulated in each time period - say in mm of rainfall per hour (mm/hr).



Reverse Mean Step Accumulated

In this case, the values are the same as the Mean Step Accumulated, but the values represent the time interval from now to the start of the next time interval. The Reverse Mean Step Accumulated time series are primarily used for forecasting purposes.



17 USING MIKE SHE WITH ARCGIS

MIKE SHE has been designed to work smoothly with ArcGIS files. In most cases, distributed data can be linked directly to shape files created by ArcGIS or any other application. The type of shape file depends on the type of data. Distributed data, such as initial water levels can be input as point and line themes, whereas spatial data that is referenced to a time series, such as precipitation, can be added as a polygon theme. In this case, each polygon can be assigned a time series of values.

In the reverse direction, all gridded data in the MIKE SHE Setup Editor can be easily saved as a point theme shape file from the pop-menu when you right click on a colour shaded map. This includes both interpolated data in the Setup tab and pre-processed data in the Pre-processed tab.

ArcGIS grids yet cannot be added directly in the MIKE SHE Setup Editor, but they can be converted to the dfs2 file format. Select New, then the MIKE Zero Tool box and choose GIS in the list. The Grid2Mike tool will convert your ArcGIS grid files to the dfs2 file format. Support for native ArcGIS grid files will be available in a service pack later this year.

The MIKE Zero Tool box also contains tools for converting dfs2 files to ArcGIS shape files (Mike2Shp) and Grid files (Mike2Grd). These tools can be useful if you have manipulated your grid files in the MIKE Zero Grid Editor, since it does not directly support shape file export. Alternatively, you can open any dfs2 file in the MIKE SHE Setup Editor, as along as the unit type is the same) and then use the right mouse function to export to a shape file. If you want to convert a dfs3 file to a shape file or a grid file then you will need to extract a dfs2 file from the dfs3 first using the 2D Grid from 3D file tool that is found under the Extraction item in the MIKE Zero Toolbox.

Some items in the MIKE SHE Setup Editor do not support shape files. Mostly these are related to integer grid codes, such as Drain codes. In this case, it is difficult to assign integer values based on grid independent polygons. In a complex setup, it would be very difficult to control which cells are being assigned to which code when the polygons do not coincide with the cell boundaries. In some areas, the model results could be very sensitive to the code assigned.

18 SPATIAL DATA

Spatial data includes all model data that can be location dependent, for example precipitation rates and soil parameters.

18.1 The Grid Editor

The Grid Editor is a generic MIKE Zero grid tool for all MIKE by DHI software. It is the primary means to edit and manipulate gridded data in MIKE SHE.

The Grid Editor 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 nodal system because they are based on a node-centered finite difference scheme. Whereas, MIKE SHE is based on a block-centered finite difference scheme.

Node numbering in the Grid Editor

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

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.

18.2 Gridded Data Types

There are two basic types of spatial data in MIKE SHE - Real and Integer. Real data is generally used to define model parameters, such as hydraulic conductivity. Integer data is generally used to define parameter zones. Thus, model cells with the same integer value can be associated with a time series or other characteristic.

Furthermore, real spatial parameters can be distinguished by whether or not they vary in time. At the moment Integer zones cannot vary with time.

Thus, spatial parameters can be divided into the following:

- Stationary Real Parameters

- Time Varying Real Parameters, and
- Integer Grid Codes

Stationary Real Parameters

Stationary Real Parameters can vary spatially but do not usually vary during the simulation, such as hydraulic conductivity. If such parameters do vary in time, then you must divide the simulation into time periods and run the each time period as a separate simulation, starting each simulation from the end of the previous simulation. This is most easily accomplished using the Hot Start facility, which is found in the Simulation Period dialogue.

The spatial distribution of stationary real parameters are entered using the Stationary Real Data dialogue

Time Varying Real Parameters

Many spatial parameters are time dependent, such as precipitation rate. In this case, both a spatial distribution , as well as a time series for each cell in the model, must be defined. Spatially distributed parameters that also vary in time are entered using the Time-varying Real Data dialogues

18.3 Integer Grid Codes

Integer Grid Codes are required when Real data varies in time or when model functions, such as soil profiles and paved areas, are assigned to particular zones. Integer Grid Codes are always integer values and do not vary with time.

For information on entering Integer Codes see the Integer Grid Codes section.

The following is an outline of the parameters that require Integer Grid Codes.

Model Domain

Integer Grid Codes are used to define the inactive areas both inside and outside the model domain. Inactive areas outside of the model and the edge of the model are defined in the Model Domain and Grid section, while inactive, subsurface areas inside of the model are defined as Internal boundary conditions.

Component Calculations

Integer Grid Codes are used to delineate such things as paved areas. In this case, the integer code acts like a flag and the calculations that are done are different depending on how the flag is set.

Model Properties

Integer Grid Codes are used to delineate areas with similar properties. In this case, the integer value defines the zone to which the cell belongs. Thus, it defines which set of model properties is to be assigned to the particular cell.

For example, a model may be divided into a five zones each with a different soil profile for the unsaturated zone. In this case, the data tree will expand under the model property to include five separate sub-branches, where the soil profile can be defined.

Time Series

Integer Grid Codes are used to define zones for which Real data varies in time. Thus, a time series for a parameter, such as precipitation rate, can be assigned to a model zone. Similarly to the Model Properties above, the model tree will expand under the parameter to include a separate sub-branch for each zone, where the time series file can be defined.

Time Varying Integers

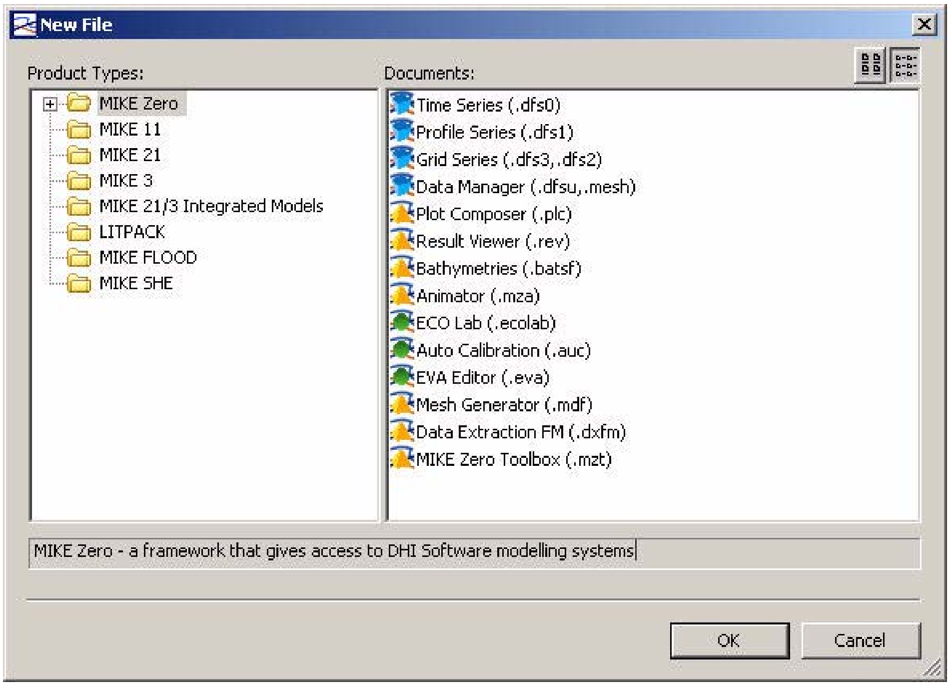
Grid Codes and Integer values do not normally vary with time. If such parameters do vary in time, then you must divide the simulation into time periods and run each time period as a separate simulation, starting each simulation from the end of the previous simulation using the Hot Start options (see Simulation Period).

18.4 Gridded (.dfs2) Data

If the parameter is defined using gridded data, then the data must be in DHI's .dfs2 file format.

The easiest way to create the .dfs2 file is to use the Create... button, which creates a new grid with the proper default values and attribute type. You can then edit this grid in the MIKE Zero Grid Editor, which can be accessed using the Edit... button.

Alternatively, a .dfs2 file can be created using the Grid Series editor, which can be accessed by clicking on File|New in the pull-down menu, or using the New File icon, , in the toolbar, and then selecting Grid Series.



If you create the file from these tools you must be careful to ensure that the EUM Data Type matches the parameter that you are creating the file for. For more information on the EUM data types, see EUM Data Units.

The grid for the .dfs2 file does not have to be the same as the numerical model grid. However, if the grids are not subsets of one another then the grids will be interpolated using the bilinear interpolation during the pre-processing stage.

The parameter grid and the model grid are aligned with one another if the parameter grid or the model grid contain an even multiple of the other grid’s cells. For example, if the parameter grid was two times finer, then every model grid cell must contain exactly four parameter grid cells.

If the grids are aligned then the parameter grid will be averaged to the model grid during the pre-processing stage. However, in some cases it does not make sense to average parameter values. For example, Van Genuchten soil parameters cannot really be averaged, since they are a characteristic of the soil. In such cases, you should ensure that the model grid and the parameter grid file are identical.

18.4.1 *Stationary Real Data*

Spatially distributed Real parameters, such as conductivity or topography, can be defined in three ways, namely they can be defined as a uniform (global) value or they may be distributed and defined using either gridded data (.dfs2 file), GIS points and polygons (ArcView .shp file), or irregularly distributed point data (x, y, value coordinate file).

It does not make sense to interpolate some parameters to the model grid. In such cases, the use of line and point data should be avoided.

Uniform

A uniform, global value means that all the grid cells in the model will have the same value.

GIS point and line data or Distributed point data

If the parameter is defined using irregularly distributed point data or an ArcView shape (.shp) file, then the data will be interpolated to the model grid during the pre-processing stage, using the interpolation method selected.

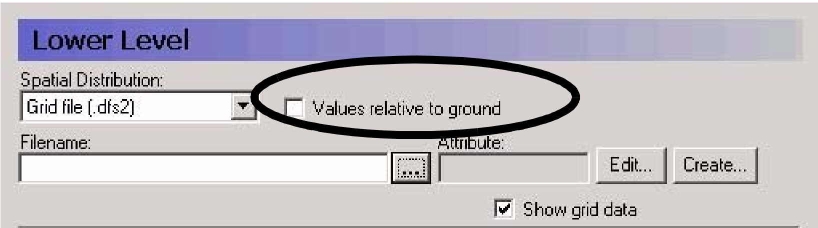
The following interpolation methods are included:

- Bilinear Interpolation *(V.1 p. 355)*, or
- Triangular Interpolation *(V.1 p. 359)*
- Inverse Distance *(V.1 p. 361)*.

It does not make sense to interpolate some parameters to the model grid. In such cases, the use of line and point data should be avoided.

Elevation Data

Elevation data, such as Layer elevations, is handled exactly the same as all other Stationary Real Parameters, except that the value may be optionally specified as a depth below the ground surface rather than absolute elevation above the datum.



Note: The value must be negative if it is below the ground level.

Tip: The current tools do not allow you to specify a polygon shape file with real values. However, this would be desirable in some cases, such as when implementing Mannings M values based on vegetation distributions. A trick to get around this limitation is the following:

1 Temporarily assign an integer grid code to each of the polygons.

2 Specify this file as an input file for one of the data items that needs integer grid codes, such as drain codes.

3 Right click on the map that will be displayed and save the map view to a dfs2 file

4 Open this dfs2 file in the grid editor and use the grid editor tools to replace the integer values with real values

5 In the Grid Editor, change the EUM unit to the appropriate value

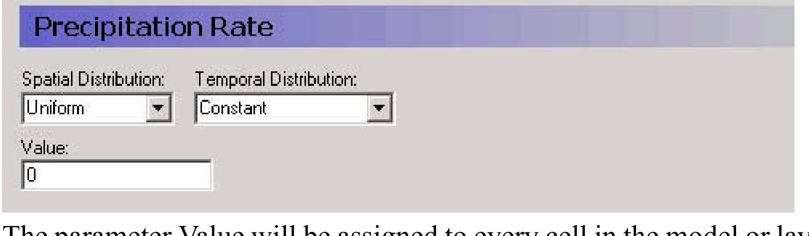
6 Save the file and then load it into the Data item for which you wanted it.

18.4.2 Time-varying Real Data

If the time-varying Real parameter does not vary spatially then the parameter must be defined as Global with either a Fixed or Time-varying value (see Uniform + Constant and Uniform + Time Varying).

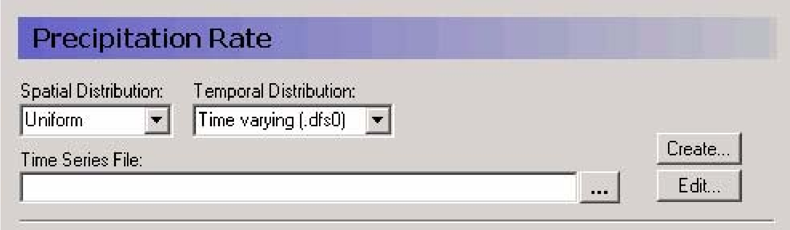
Often, time-varying data, such as precipitation rate, are spatially distributed using measurement stations, which in the model are translated into model zones using, for example, Thiessen polygons. In this case, each station is associated with a .dfs0 time series file that contains the time series of precipitation rate. Station-based zones are defined using Integer Grid Codes in either a .dfs2 file as Grid Codes, or in a Shape (.shp) file as polygons with an Integer Code (see Station-based + Grid Codes or Polygons).

Uniform + Constant



The parameter Value will be assigned to every cell in the model or layer as appropriate and will remain constant throughout the simulation.

Uniform + Time Varying



The time series in the .dfs0 file will be assigned to every cell in the model or layer as appropriate.

Station-based + Grid Codes or Polygons

Station-based time varying data means that the model domain is divided into zones that are defined by an Integer Grid Code.

If a .dfs2 file is used, then the Integer Grid Codes are defined on a regular grid, which is interpreted to the model grid during the Pre-processing stage.

If the Integer Grid Codes are defined using polygons then you must supply an ArcView .shp file containing polygons each with an Integer Grid Code. The item **Fill Gaps with**: allows you to define the Integer Grid Code to use in the event that a cell is not included within one of the polygons.

Once the file containing Integer Grid Codes has been defined, a new level in the data tree will appear below the current level, containing one entry for every unique Integer Grid Code in the file.

On this level, you must then supply a time series values for every Integer Grid Code. However, the time series can also be fixed, in the sense that a constant value over time is used. This makes it easy to use detailed time series for some zones and constant values for zones where little information exists.

The time series dialogue itself includes two graphical views. The upper graphic displays the time series that is being applied and the lower graphic shows where the time series will be applied.

18.4.3 Integer Grid Codes

The dialogues for Integer Codes function essentially same as those for Stationary Real Data, except that interpolation does not make sense for integer grid codes.

If Integer Grid Codes are being used to assign Model Properties, such as soil profiles or time series, then new sub-branches will appear in the data tree corresponding to the number of unique Integer Grid Codes in the .dfs2 file.

Uniform Value

A Uniform, global value means that all the grid cells in the model will have the same value. Thus, all cells would belong to the same zone.

Grid File (.dfs2)

If the Integer Code is defined using a grid file, then the Integer Code is defined on a grid. This grid may be different than the numerical model grid. However, the grids must be subsets of one another. That is, the Integer Code grid and the model grid must be aligned with one another and the Integer Code grid or the model grid must contain an even multiple of the other grid's cells. For example, if the Integer Code grid was two times finer, then every model grid cell must contain exactly four Integer Codes.

Normally, the Integer Code will be assigned to the model grid based on the most prevalent Integer Code in the cell. However, this can lead to problems when the a particular code is both infrequent and widely dispersed. For example, if a model area contained many small wetland areas that were much smaller than a grid cell.

For this reason, a bookkeeping count is kept of the assignments to reduce any bias in the assignment of Integer Codes and ensure that less frequently occurring Integer Codes will be represented in the resulting model grid. For example, if their were two different Integer Codes, A and B, used in the model and A always occurred more frequently in each model cell, the bookkeeping count would ensure that B would actually be assigned to some of the model cells. The final frequency of occurrence of the Integer Codes in the model cells would reflect the underlying frequency of occurrence of the Integer Codes. That is, if A occurred twice as often as B, the model grid would also contain twice as many A's as B's.

Thus, in our widely dispersed wetland example, if every model grid cell contained 9 Integer Codes for Land Use, and 1/9 of the Land Use grid codes were for wetlands, then every ninth Model Cell would be assigned a Land Use grid code for wetlands.

Polygons

In the current version, only some of the parameters are set up to accept .shp file polygons. Currently, .shp file polygons are only allowed in:

- Model Domain and Grid *(V.2 p. 70)*
- Precipitation Rate *(V.2 p. 77)*

- Vegetation *(V.2 p. 94)*,
- Reference Evapotranspiration *(V.2 p. 81)*
- UZ Soil Profile Definitions *(V.2 p. 132)*,
- SZ Internal boundary conditions *(V.2 p. 169)*, and
- Horizontal Extent *(V.2 p. 195)* of SZ Lenses.

Note The Horizontal Extent *(V.2 p. 195)* of SZ Lenses accepts polygons, but the dialogue is still set up for point/line .shp files and an error is given in the Data Verification window.

Model grid codes are assigned based in which polygon the centre of the cell is located in.

18.5 Interpolation Methods

The gap filling is based on the concept that we have to calculate the depth in the point (x_c, y_c). We define this as the function $z_c = f(x_c, y_c)$. If we place our self in this point, we can divide the world up into four quadrants Q1 - Q4. From here it's a matter of finding some points from the raw data set relatively close to this point. The search radius for all possible techniques can be entered - in grid cell distance. Points outside this distance will never be taken into account.

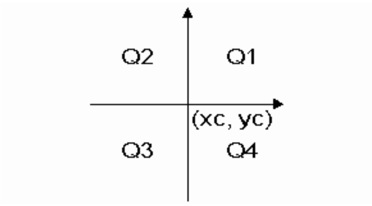


Figure 18.1 Definition of quadrants

18.5.1 Bilinear Interpolation

This technique finds four points from the raw data set - one in each quadrant. The search is done in the following way. A mask of relative indices is created. The cells in this mask are sorted according to the distance. For the quadrant Q1 the cells are sorted in the following way, the grid point it self being excluded.

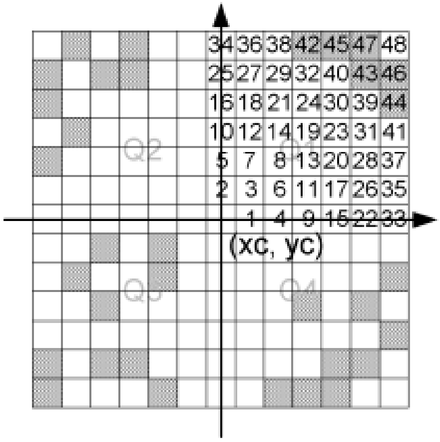


Figure 18.2 Illustration of the neighbouring grid cells being sorted

Note that the grid cells with a crosshatch pattern contain raw data points. When the closest raw data point in each quadrant is found, we have four points that form a quadrangle. This quadrangle contains the centre point, where we want to calculate the z-value. This is illustrated on Figure 18.3.

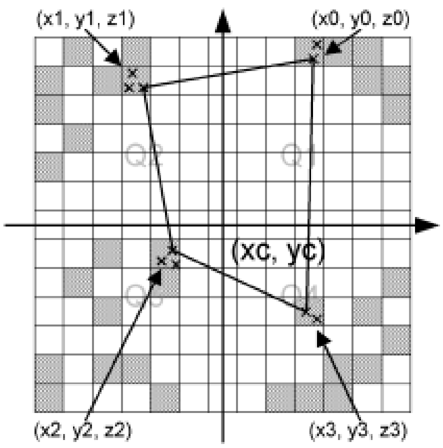


Figure 18.3 Illustration of the closest raw data points in each quadrant.

Note that each grid cell might contain more raw data points. If this is the case, the closest of these is chosen. We now have an irregular quadrangle, where the elevation is defined in each vertex. We need to compute the elevation in (x_c, y_c). If we transform our quadrangle into a square, we can perform bilinear interpolation. This is illustrated on Figure 18.4.

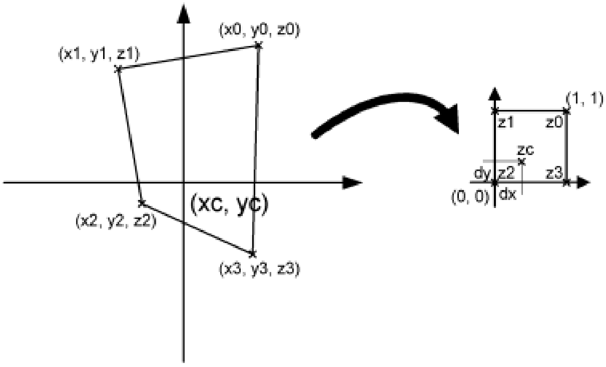


Figure 18.4 Illustration of bilinear interpolation.

First the interpolation requires the transformation from quadrangle to a normalized square. This is done in the by computing 8 coefficients in the following way:

$$
\begin{aligned}
A_1 &= x_0 \\
A_2 &= y_0 \\
B_1 &= x_1 - x_0 \\
B_2 &= y_1 - y_0 \\
C_1 &= x_3 - x_0 \\
C_2 &= y_3 - y_0 \\
D_1 &= x_2 - x_1 + x_0 - x_3 \\
D_2 &= y_2 - y_1 + y_0 - y_3
\end{aligned}
\tag{18.1}
$$

Mapping the coordinates (x_c, y_c) to the normalized square (dx, dy) is done by solving equation (18.2).

$$
ax^2 + bx + c = 0 \tag{18.2}
$$

where the coefficients are

$$
\begin{aligned}
a &= D_1B_2 - D_2B_1 \\
b &= D_2x_c - D_1y_c - D_2A_1 + D_1A_2 + C_1B_2 - C_2B_1 \\
b &= C_2x_c - C_1y_c + C_1A_2 - C_2A_1
\end{aligned}
\tag{18.3}
$$

Solving equation (18.2) gives us *dx*.

$$dx = \frac{-b \pm \sqrt{b^2 - 4ac}}{2a} \tag{18.4}$$

where $0 \le dx \le 1$ is used to choose the correct root. *dy* can now be computed in two ways:

$$dy = \frac{x_c - A_1 - B_1dx}{C_1 - D_1dx} \tag{18.5}$$

or

$$dy = \frac{x_c - A_2 - B_2dx}{C_2 - D_2dx} \tag{18.6}$$

Choosing between (18.5) and (18.6) is done in such a way, that division by zero is avoided. (x_c, y_c) has been mapped to (dx, dy). The task was to compute the elevation in the point (x_c, y_c) and this is done in the following way using regular bilinear interpolation:

$$z_c = (1 - dx)(1 - dy)z_2 + dx(1 - dy)z_3 + (1 - dx)dy\,z_1 + dxdy\ z_0 \tag{18.7}$$

If less than four points are found (if one or more quadrants are empty), the double linear interpolation is replaced with reverse distance interpolation (RDI). This is done according to the following scheme:

$$w_i = \frac{1}{\sqrt{(x_i - x_c)^2 + (y_i - y_c)^2}} \tag{18.8}$$

$$w_s = \sum_{i=1}^{N} w_i \tag{18.9}$$

$$z_c = \frac{1}{w_s} \sum_{i=1}^{N} w_i z_i \tag{18.10}$$

The method works fairly efficiently, but it has one drawback. The quadrant search is heavily dependent on the orientation of the bathymetry. If the bathymetry is rotated 45 degrees 4 completely different points might be used for the interpolation. For this reason there is also a Triangular interpolation method, which can be used, and this method should be direction independent.

18.5.2 Triangular Interpolation

As mentioned previously the 'Bilinear Interpolation' is dependent on the orientation of the bathymetry. The 'Triangular Interpolation' is made as an answer to this problem. First the closest point to (x_c, y_c) is found. The following figure shows this:

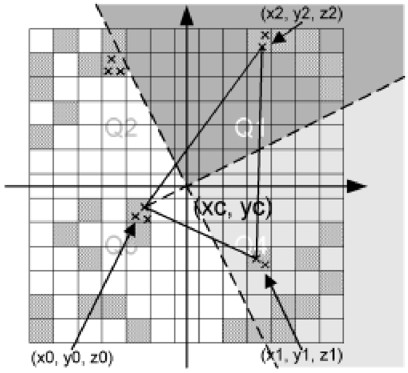


Figure 18.5 Illustration of triangular interpolation

In this example the point (x_0, y_0, z_0) is the closest point. When this point is identified, two quadrants are identified – indicated by the light grey and the dark grey areas. The closest point in these two quadrants are then found. They can be seen on the figure as (x_1, y_1, z_1) and (x_2, y_2, z_2). The interpolation is then done in two steps. First the coefficients describing the plane defined by the 3 found points are computed:

$$A = \frac{-(y_1 - y_0)(z_2 - z_0) + (y_2 - y_0)(z_1 - z_0)}{(x_1 - x_0)(y_2 - y_0) - (x_2 - x_0)(y_1 - y_0)}$$
$$B = \frac{(x_1 - x_0)(z_2 - z_0) + (x_2 - x_0)(z_1 - z_0)}{(x_1 - x_0)(y_2 - y_0) - (x_2 - x_0)(y_1 - y_0)} \tag{18.11}$$
$$C = z_0 - Ax_0 - By_0$$

And secondly the actual interpolation is done:

$$z_c = Ax_c + By_c + C \tag{18.12}$$

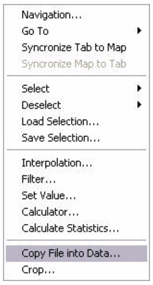
If less than 3 points are found, reverse distance interpolation (RDI) is used. The triangular interpolation is more time consuming due to the more complex direction independent search, but better end results should be achieved with this method.

18.5.3 Inverse Distance

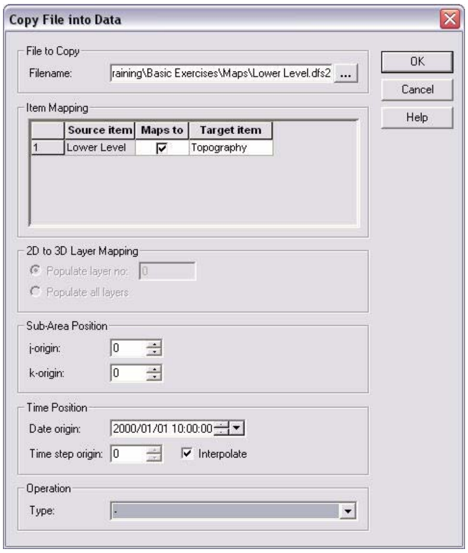
The two inverse distance methods both use the nearest point in each quadrant to calculate the interpolated value, weighted by the distance or the distance squared respectively.

18.6 Performing simple math on multiple grids

In the upper menu of the Grid Editor, under tools, there is an item called **Copy File into Data**.



If you select this item then a dialogue appears where you can insert an existing dfs2 or dfs3 file into the current dfs2 or dfs3 file that you are editing in the Grid editor.



Alternatively, you can define an operation that you want to do with the file. For example, if you were editing a topography file, you could subtract all of the values in a lower elevation file, to obtain a thickness distribution for a layer.

The principle advantage of this tool, is that time varying dfs2 and dfs3 files can be manipulated. However, if the operations are complex, but not time varying then

Target file

The target file is the current file you are editing in the Grid editor. The operations that you do are performed on the target file. So, if you don't want to edit the target file, copy it to a new name first and edit the copy.

File to Copy

The top section of the dialogue is the name of the source file that you want to insert into, subtract from, add to, etc. the target file.

Item mapping

If the target file or the source file has more than one item in it, then all of the items will be listed here and you will be able to choose whether or not to map the various items to one another.

2D to 3D Layer Mapping

If you are mapping a 2D dfs2 file into a 3D dfs3 file, then you can choose to map all of the layers or only a single layer.

Sub-area position

You select to map the source file onto the target file starting at a different location than the origin. In this case, you must specify the coordinates in the target grid where the origin of the source grid should be positioned. For example, if you have a 20x20 grid and we wish to copy data into the 4x4 rectangle given by the four nodes (10,14), (13,14), (13,17) and (10,17), then you should select a 4x4 grid file and specify j-origin=10 and k-origin=17. **Note: the Grid editor starts its nodal numbering at 0,0**.

Time Position

The source grid and target grid do not have to have equal time steps or the same time origin. In this section of the dialogue, you can specify the time at which the source grid should be added to the target grid. In this way, you can add additional time steps to the end of a time varying dfs2 file, or insert hourly information into a monthly time series, for example.

Operation

Finally, you can specify how the source grid file should interact with the target file.

Copy - all values are copied such that they replace the existing data in the data set

Copy if target differs from delete value - values in the source file will be copied into the target file, only if the target value is a delete value

Copy if source differs from delete value - values in the source file will be copied into the target file, only if both the source value and the target values are not delete values

Copy if source AND target differs from delete value - values in the source file will be copied into the target file, only if the source value is not a delete value

+ - the source values will be added to the target values

- - the source values will be subtracted from the target values

***** - the source values will be multiplied by the target values

/ - the source values will be divided by the target values

18.7 Performing complex operations on multiple grids

In the Toolbox, under MIKE SHE/Util, there is a Grid calculator tool, which allows you to perform complex operations on .dfs2 grid files. However, the grid files must have the same grid dimensions and they may not included multiple time steps or multiple items. Thus, this tool is much more restrictive than the grid operations available in the Grid Editor. However, you can make complex chains of operations and save the setup, which can save you a lot of time if you are doing the same operation many times.or after each simulation

The Grid Calculator works like a wizard, with Next and Back buttons to move between dialogues.

INDEX

Symbols

Numerics

A

B

C

D

E