MIKE SHE

Volume 1: User Guide

PLEASE NOTE

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.

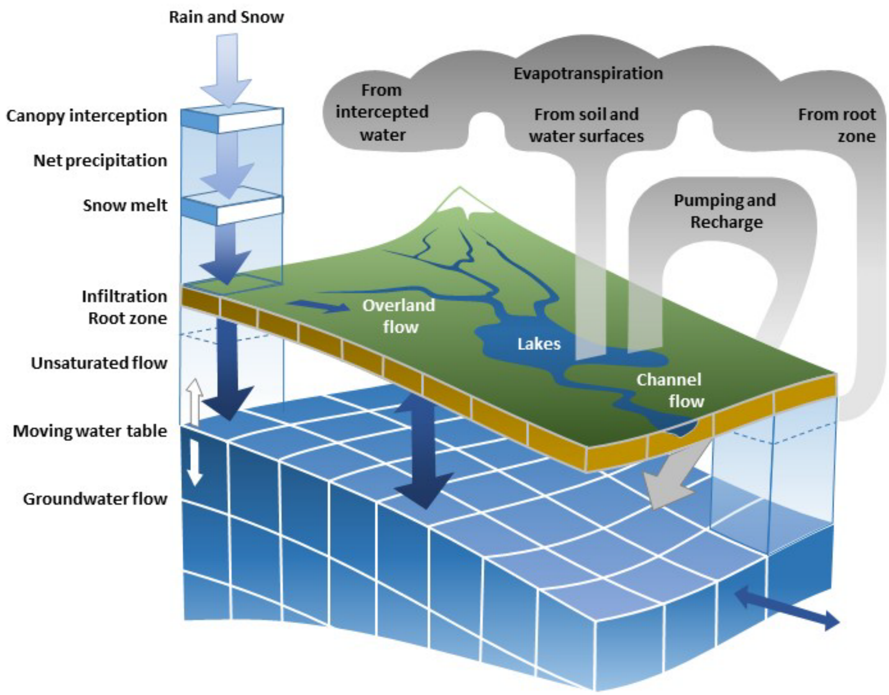


Figure 1.1 Hydrologic processes simulated by MIKE SHE

MIKE SHE uses MIKE Hydro River to simulate channel flow. MIKE Hydro River 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 in many 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, all the hydrologic processes are solved implicitly at a uniform time step, which can lead to intensive computational effort for watershed scale model.

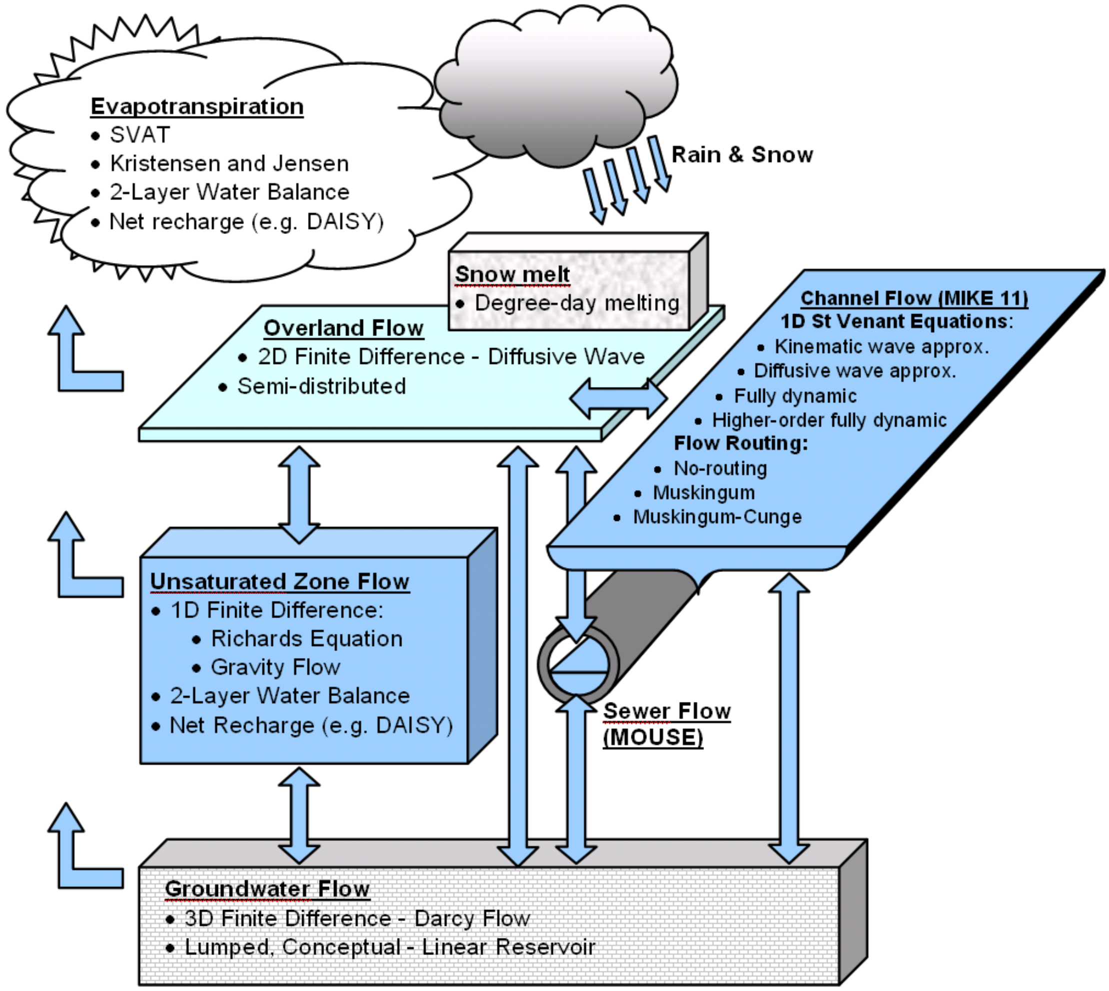


Figure 1.2 Schematic view of the process in MIKE SHE, including the available numeric engines for each process. The arrows show the available exchange pathways for water between the process models. Note: the SVAT evapotranspiration model is not yet available in the commercial version of MIKE SHE

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 (sub-catchment-based)	Surface roughness	Detention storage Slope parameters
River flow	River bed roughness River bed leakage coefficient	

Table 1.1 Principle parameters for MIKE SHE

	Principle calibration parameters	**Other parameters**
Unsaturated flow (finite difference)	Saturated hydraulic conductivity	Soil water contents at saturation, field capacity, and wilting point Soil 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
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 snowmelt. 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.

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 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 valid license 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 time steps: 800
- number of SZ time steps: 200
- no steady-state SZ
- no overbank spilling
- no MIKE 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, 8-16GB of RAM, and 100-500 GB of free disk space.

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

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

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.

MIKE SHE always runs with the maximum allowed number of threads, which will not be slower, but may be less efficient if you are running multiple simulations at the same time. There is an extra parameter option ("max number of threads") that you can use to control the parallelization of each simulation. Also, hyper threading is less efficient than physical cores. So, there might also be some benefit to turning off hyper threading for some simulations (This is a BIOS setting).

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 eight cores/processors. If you want to take advantage of more than eight cores, then you will need to contact your local DHI sales office to obtain additional run-time licenses.

However, the 2017 Release allows you to use all the cores available. There is no longer any restrictions on the number of cores - as long as you have a Corporate License and valid SMA).

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.

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 Hydro River or other MIKE Zero dialogue, you will land in a relevant section of the on-line help.

The 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 checkmark. Thus, the basic work flow for a fully integrated MIKE SHE model is built around the following components:

1. The MIKE SHE User Interface *(V1 p. 29)*
2. Background maps *(V1 p. 32)*
3. Initial model setup *(V1 p. 32)*
4. Simulation parameters *(V1 p. 34)*
5. Model Domain and Grid *(V1 p. 35)*
6. Topography *(V1 p. 37)*
7. Climate *(V1 p. 38)*
8. Channel Flow *(V1 p. 42)*
9. Overland Flow *(V1 p. 43)*
10. Unsaturated Flow *(V1 p. 49)*
11. Saturated Groundwater Flow *(V1 p. 54)*
12. Storing of results *(V1 p. 59)*
13. Preprocessing your model *(V1 p. 65)*
14. Running Your Model *(V1 p. 65)*

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
- Bathymetry (.batsf) - for sea bed elevations
- Animator (.mza) - for 3D visualization of 2D surface water and waves
- **MIKE ECO Lab (.ecolab)** - for detailed water quality processes
- **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, including:

- **MIKE View** - a results evaluation utility for MIKE Hydro and MIKE Urban (sewers) 1D flow results.

- **Image Rectifer** - a simple tool for stretching and georeferencing image files
- **Launch Simulation Engine** - a utility for launching and running MIKE Zero simulation engines independent of the Graphical User Interface.

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 yourself 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 *(V1 p. 167)*.

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

- Working with Evapotranspiration - User Guide *(V2 p. 37)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*
- Working with Rivers and Streams - User Guide *(V2 p. 111)*
- Working with Unsaturated Flow - User Guide *(V2 p. 173)*
- Working with Groundwater - User Guide *(V2 p. 219)*

2.2.1 The Setup Editor

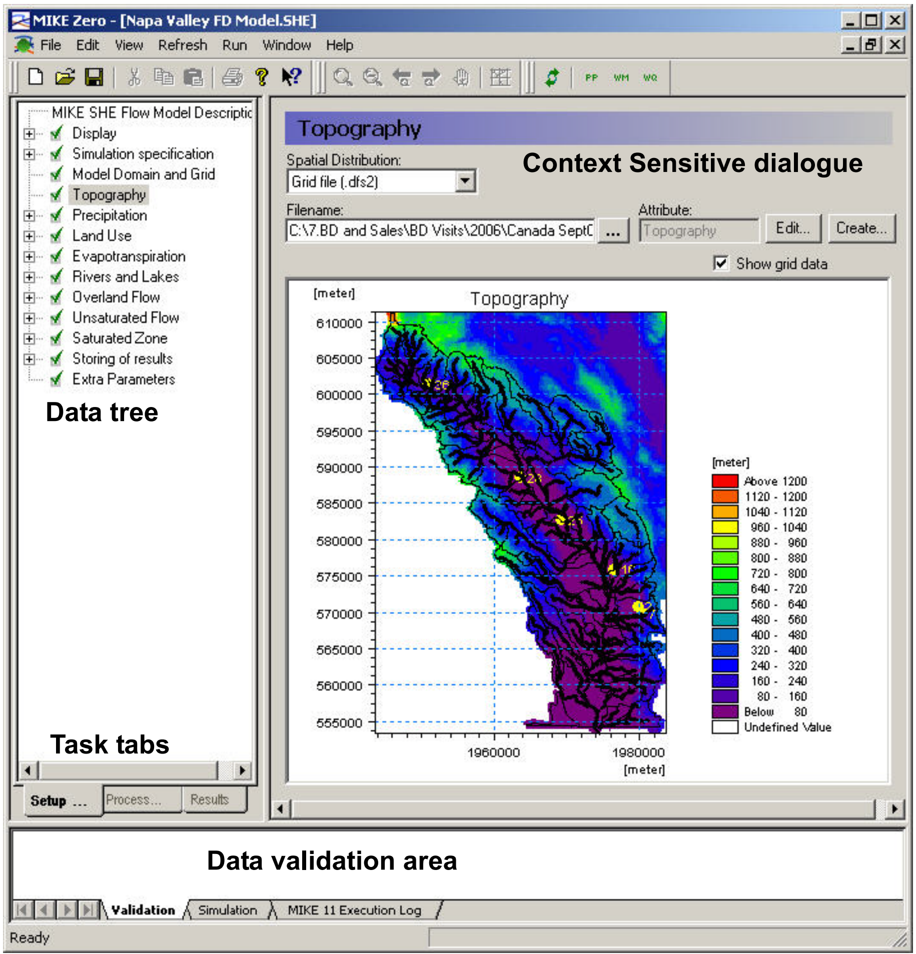


Figure 2.1 Graphical overview of the in the MIKE SHE GUI, without the Project Explorer.

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 time steps.

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.

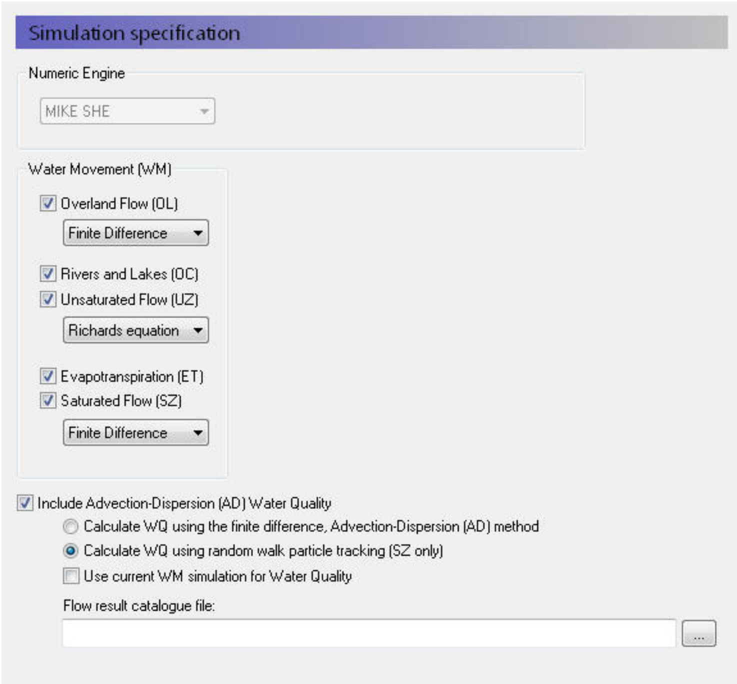
In the event that you are using scanned paper maps, if your maps are not rectilinear, or are not correctly georeferenced, then you can use the Image Rectifier (see on-line help under MIKE Zero) to align your image to the coordinate system you are using.

Note: The display of the river network is not carried over to the Results Viewer, but can be added to the view in the Results Viewer by adding a MIKE Hydro River results file.

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 baseflow 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 - Technical Reference *(V2 p. 51)*
- Channel Flow - Technical Reference *(V2 p. 109)*
- Evapotranspiration - Technical Reference *(V2 p. 17)*
- Unsaturated Zone - Technical Reference *(V2 p. 141)*
- Saturated Flow - Technical Reference *(V2 p. 189)*
- Particle Tracking-Reference *(V2 p. 319)*
- Advection Dispersion - Reference *(V2 p. 255)*

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 *(V1 p. 183)*
- UZ Computational Control Parameters *(V1 p. 188)*
- SZ Computational Control Parameters *(V1 p. 189)*

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 *(V1 p. 178)* dialogue to optimize your simulation time stepping. For more information on time step control, you can see the Controlling the Time Steps *(V1 p. 77)* section.

Note: Although the different hydrologic processes can run on different time steps, the processes exchange water explicitly. 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 time steps, which can each include three 2-hour overland flow time steps. See Time Step Control *(V1 p. 178)* 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 *(V1 p. 325)* dialogue. In this dialogue, you specify the storing interval for hot start data. Then in the Simulation Period *(V1 p. 176)* 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.

- MKE 11 and MIKE Hydro River require an independent hot-start.
- 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 *(V1 p. 157)* 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 *(V1 p. 155)*.

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 *(V1 p. 147)* for more information.

Creating dfs2 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 *(V1 p. 215)* dialogue. See Using MIKE SHE with ArcGIS *(V1 p. 147)* 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 *(V1 p. 336)*,
- Upper Level *(V1 p. 335)*,
- Initial Potential Head *(V1 p. 308)*, and
- Drain Level *(V1 p. 318)*.

Depth parameters, such as ET Surface Depth *(V1 p. 287)*, 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 *(V1 p. 157)* method is useful for interpolating previously gridded DEM data. Whereas, the Triangular Interpolation *(V1 p. 160)* method is useful for contour data digitized from a DEM. Inverse Distance Interpolation *(V1 p. 161)* 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 *(V1 p. 147)* 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 *(V1 p. 131)* must be “Precipitation” and the time series must be Mean Step Accumulated *(V1 p. 146)*.

If you use an amount, then the EUM Data Units must be “Rainfall” and the time series must be Step Accumulated *(V1 p. 145)*.

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 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.

2.5.3 Freezing and Snow melt

MIKE SHE includes a comprehensive snow melt module based on a modified degree-day method. If the Include snow melt *(V1 p. 220)* checkbox is checked then rain accumulates as snow if the Air Temperature *(V1 p. 227)* is below the Threshold Melting Temperature *(V1 p. 231)* (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 *(V1 p. 232)*.

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 *(V1 p. 234)*, 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 online help for the individual dialogues and in the Snow Melt - Technical Reference *(V2 p. 41)* section.

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 *(V1 p. 220)* section of the on-line help and User Interface manual

Frozen Ground

Frozen ground is not directly simulated in MIKE SHE, but can be approximated by using a time varying infiltration rate, where you manually reduce the infiltration rate during the winter. More information on this can be found in the section, Time varying surface infiltration (frozen soils) *(V2 p. 334)*.

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 *(V1 p. 37)*. Related to topography is the definition of Subcatchments *(V1 p. 217)*, 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 Hydro River branches using Flood codes. For details on how use Flood codes see the chapter on Surface Water *(V1 p. 97)*.

Hydraulic properties - The properties related directly to overland sheet flow are found under Overland Flow *(V1 p. 43)*. This includes the Manning number *(V1 p. 258)* or surface roughness and the Detention Storage *(V1 p. 259)*, 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 *(V1 p. 264)*.

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 *(V1 p. 49)* 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 *(V1 p. 260)*.

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 *(V1 p. 57)*

Ponded drainage and Paving - Heavy rainfall will generate ponded areas in the landscape and paving will inhibit infiltration. However, 2D overland flow does not usually travel far. It typically drains into local, small scale ditches and channels. The Ponded Drainage function is used to route local ponded water to urban and natural drainage features. The Ponded Drainage function is calculated before any other function, so it effectively acts on rainfall.

2.6.1 Vegetation

The vegetation properties are used to calculate the Actual evapotranspiration from the 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 *(V1 p. 373)*.

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 soils 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 *(V1 p. 287)*.

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.

2.7 Channel Flow

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

For MIKE 11, 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).

For MIKE Hydro River, the main simulation file is the .hydro file.

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

1. Establish a MIKE Hydro River 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 Hydro River by defining branches (reaches) where MIKE Hydro River should interact with MIKE SHE. Modify your MIKE SHE and MIKE Hydro River models so that they work together properly. For example, by removing the specified groundwater table in MIKE Hydro River 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 Hydro River is in the chapter Surface Water *(V1 p. 97)*.

Additional documentation on MIKE Hydro River can be found in the MIKE Hydro River 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 Hydro River in MIKE SHE, as the overland flow module provides lateral runoff to the rivers.

The Simplified Overland Flow Routing *(V2 p. 68)* 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 *(V2 p. 51)* 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 *(V2 p. 55)*
- Explict Numerical Solution *(V2 p. 56)*

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 the river onto the flood plain. Alternatively, you can use Flood codes *(V1 p. 255)* to inundated flood plain areas based on the water level in the river.

The Multi-grid overland flow option allows you to take advantage of detailed DEM information if it is available. The multi-grid method, sub-divides 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 Hydro River and MIKE SHE and the overbank spilling options, and ways to optimize the calculation of overland flow can be found in the chapters:

- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*
- Working with Rivers and Streams - User Guide *(V2 p. 111)*

Manning's M

The Manning M is equivalent to the Stickler roughness coefficient, the use of which is described in Overland Flow - Technical Reference *(V2 p. 51)*.

The Manning M is the inverse of the more conventional Manning's 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 Manning's M are used for overland flow compared to channel flow.

If you don't want to simulate overland flow in an area, a Manning's 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 *(V2 p. 332)*.

Separated flow areas

The Separated Flow Areas *(V1 p. 264)* 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.

2.8.1 Ponded drainage and paving

Ponded drainage (OL Drainage) is a special boundary condition in MIKE SHE used to defined natural and artificial drainage systems that cannot be defined by the River Network. This is an effective means to specify urban drainage networks and large scale upland drainage areas.

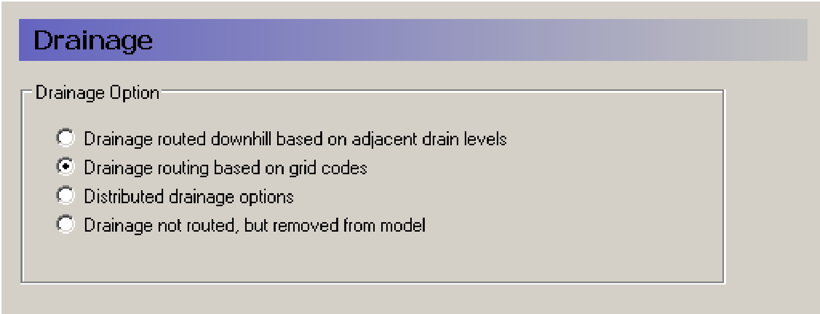
OL Drainage is removed from ponded water on the cell. Water that is removed is routed to local surface water bodies, local topographic depressions, or out of the model. The amount of drainage is calculated based on the depth of ponding and the drain level using a linear reservoir formulation.

When water is removed from a drain, it is immediately moved to the OL Drain Storage. 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 storage. From the OL Drain Storage, the water is released to the recipient at a rate based on the discharge time constant.

Each cell requires a drain level, inflow time constant (which is the same as a leakage factor), and an outflow time constant. Both drain levels and time constants can be spatially defined. The default drainage level is the topography. A typical time constant may be between 1e-4 and 1 e-5 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 Hydro River H-point or MIKE Urban manhole as a recipient.
- **Removed** The fourth option is simply a head dependent boundary that removes the drainage water from the model.

2.8.2 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 *(V2 p. 64)* 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 *(V1 p. 108)*.

2.8.3 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 *(V1 p. 99)* contains many more details on simulating overland flow and the coupling to MIKE Hydro River. In particular the section Overland Flow Performance *(V1 p. 105)* contains detailed information on improving the performance of the overland flow in your model.

2.8.4 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 Hydro River 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 Chapters:

- Working with Evapotranspiration - User Guide *(V2 p. 37)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*
- Working with Unsaturated Flow - User Guide *(V2 p. 173)*

The Technical Reference manual includes detailed information on the calculation methods - Unsaturated Zone - Technical Reference *(V2 p. 141)*.

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) *(V2 p. 156)*.

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 *(V2 p. 157)*.

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 *(V2 p. 162)* in the Reference Guide.

2.9.5 UZ column classification

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.

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

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 *(V1 p. 282)*, is the lower UZ boundary condition.
- **Specified classification** With the specified option, you must supply a list of grid codes, Specified classification *(V1 p. 284)*, 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 *(V1 p. 283)*) 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 *(V1 p. 222)*.

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 Zone - Technical Reference *(V2 p. 141)*. The chapter, Working with Unsaturated Flow - User Guide *(V2 p. 173)*, 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 overrelaxation 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 lens is below the top of the lens beneath, then the lenses are assumed to meet in the middle of the overlapping area.
- **Small lenses override larger lenses** - If a small lens is completely contained within a larger lens the smaller lens dominates in the location where the small lens is present.
- **Negative or zero thicknesses are ignored** - If the bottom of the lens intersects the top of the lense, the thickness is zero or negative and the lens 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 *(V1 p. 222)*.

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 the River Network. It can also be used to simulate simple, lumped conceptual surface water drainage of groundwater.

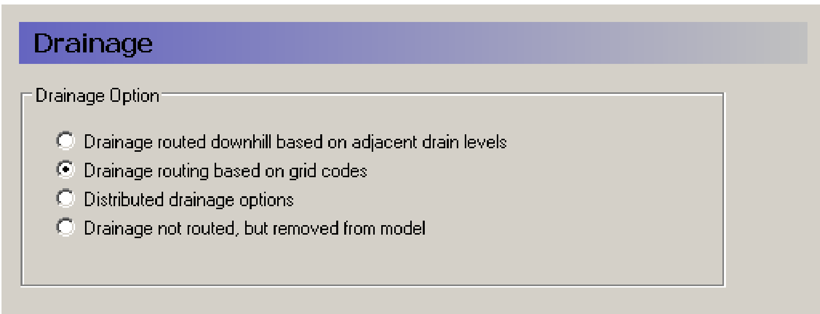
Saturated zone drainage is removed from the layer of the SZ layer containing the drain level. Water that is removed from the saturated zone by drains is routed to local surface water bodies, local topographic depressions, or out of the model. The amount of drainage is calculated based on the groundwater 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 Hydro River 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 *(V1 p. 367)*.

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, see the section Linear Reservoir Method *(V2 p. 205)*.

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 *(V1 p. 91)*.

Hot start output

It is often very useful to be able to start a simulation from a consistent 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 *(V1 p. 88)*.

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 Appendix A, MIKE SHE Output Items *(V1 p. 397)*.

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 sub-surface 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_1201234500.456740.0. 0.\time\obsdata1

Obs_2151239700.458900.10.1.\time\obsdata2

Obs_3160241500.459310.20.1.\time\obsdata3
```

2.11.2 Detailed River Time Series output

River hydraulic output is generally analysed using the MIKE View program, or directly in MIKE Hydro River. However, the default River output is only at specified time intervals. Every item in the Detailed MIKE Hydro River Time Series table is output at every MIKE Hydro River time step. However, there is a user specified minimum output time to prevent excessive output if the time step is very small.

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 than using MIKE VIEW.

Importing ASCII data

Detailed River Time Series specifications can be imported directly into the 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 River Branch (check your spelling!)

Chainage - The location of the MIKE 11 h-point or q-point. The nearest one will be taken within a tolerance. If the nearest one is not close by, a warning will be printed to the log file.

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 *(V1 p. 88)*.

3 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 **P**re**P**rocess your model data to create the numerical model from your grid independent data. See Preprocessing your model *(V1 p. 65)*.

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 *(V1 p. 65)*.

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.

3.1 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 *(V1 p. 174)* 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.

3.1.1 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.

3.1.2 Editing the pre-processed data

MIKE SHE only reads 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.

3.2 Pre-processed data items

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

3.2.1 MIKE Hydro River coupling

The coupling between MIKE Hydro River 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 Hydro River 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 Hydro River points, where the 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 Hydro River network 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 *(V1 p. 99)*
- Working with Rivers and Streams - User Guide *(V2 p. 111)*
- Channel Flow - Technical Reference *(V2 p. 109)*

3.2.2 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.

3.2.3 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 *(V1 p. 273)*
- Soil Profile Definitions *(V1 p. 278)*
- Partial automatic classification *(V1 p. 283)*
- Specified classification *(V1 p. 284)*

3.2.4 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 *(V1 p. 222)*.

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 *(V1 p. 57)*.

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

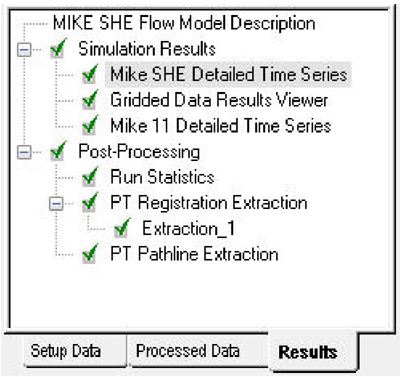
Drainage to local depressions and boundary - This grid displays all the cells that drain to local depressions or to the outer boundaries. All drainage from cells with the same negative value are drained to the cell with the corresponding positive code. If there is no corresponding positive code, then 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 *(V1 p. 57)*
- Drainage *(V1 p. 315)*
- Drain Level *(V1 p. 318)*
- Drain Time Constant *(V1 p. 319)*
- Drain Codes *(V1 p. 319)*
- Option Distribution *(V1 p. 321)*

3.3 The Results Tab

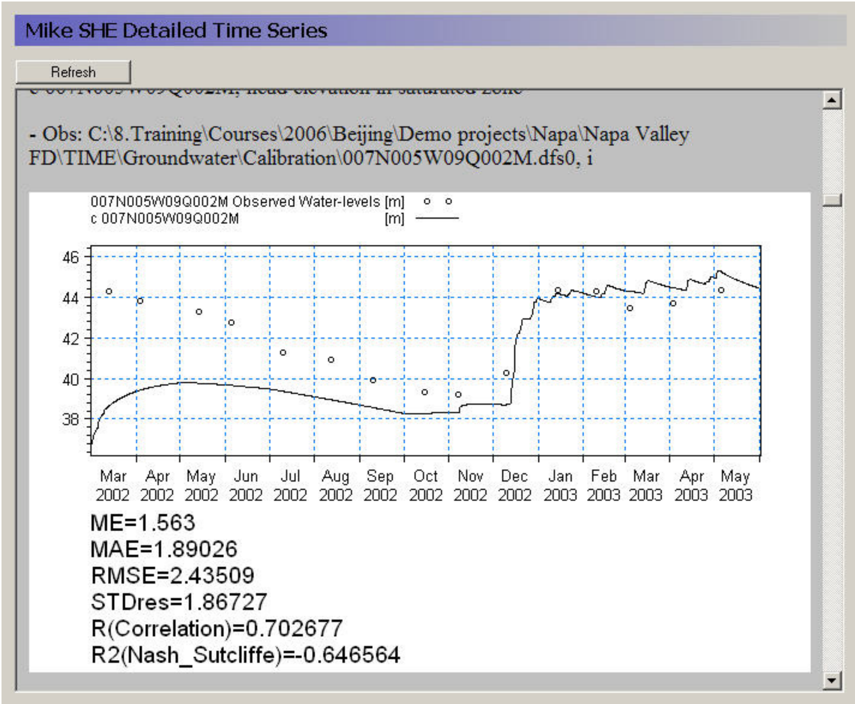


All the simulation results are collected in the Results tab. This includes Detailed time series output for both MIKE SHE and MIKE Hydro River, as well as Grid series output for MIKE SHE.

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

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

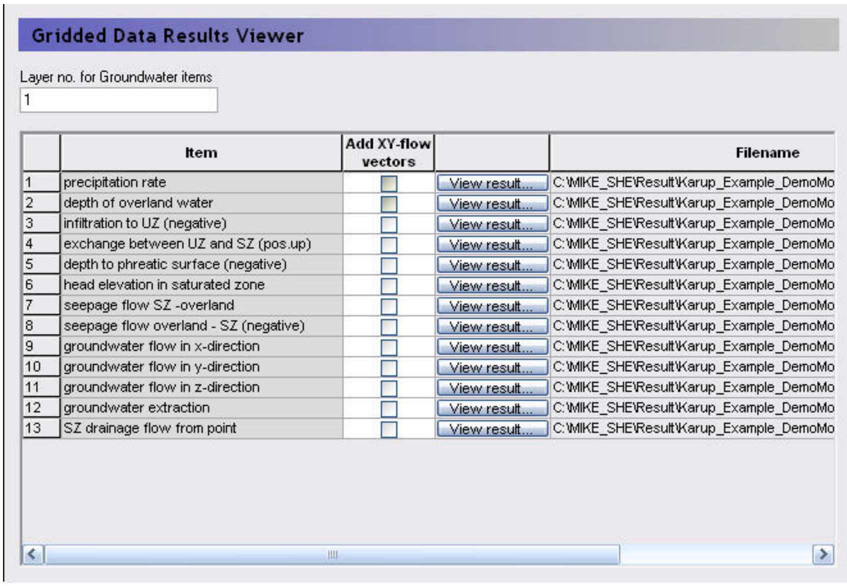
3.3.1 Detailed Time Series Results



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.

For information on the statistics see Statistic Calculations *(V1 p. 355)*.

3.3.2 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 *(V1 p. 333)* dialogue from the Setup tab.

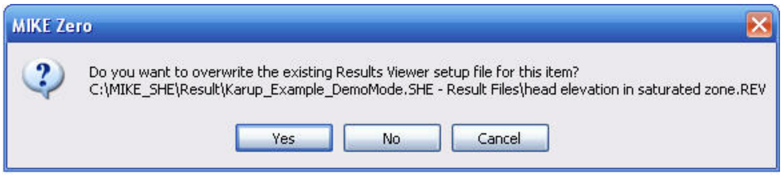
Clicking on the View result button will open the Results Viewer to the current item. All overlays from MIKE SHE (e.g. shape files, images, and grid files) will be transferred as overlays to the result view. However, the River Network is not transferred as an overlay.

Layer number - For 3D SZ data files, the layer number can be specified at the top of the table. However, the layer number can also be changed from within the Results Viewer. 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

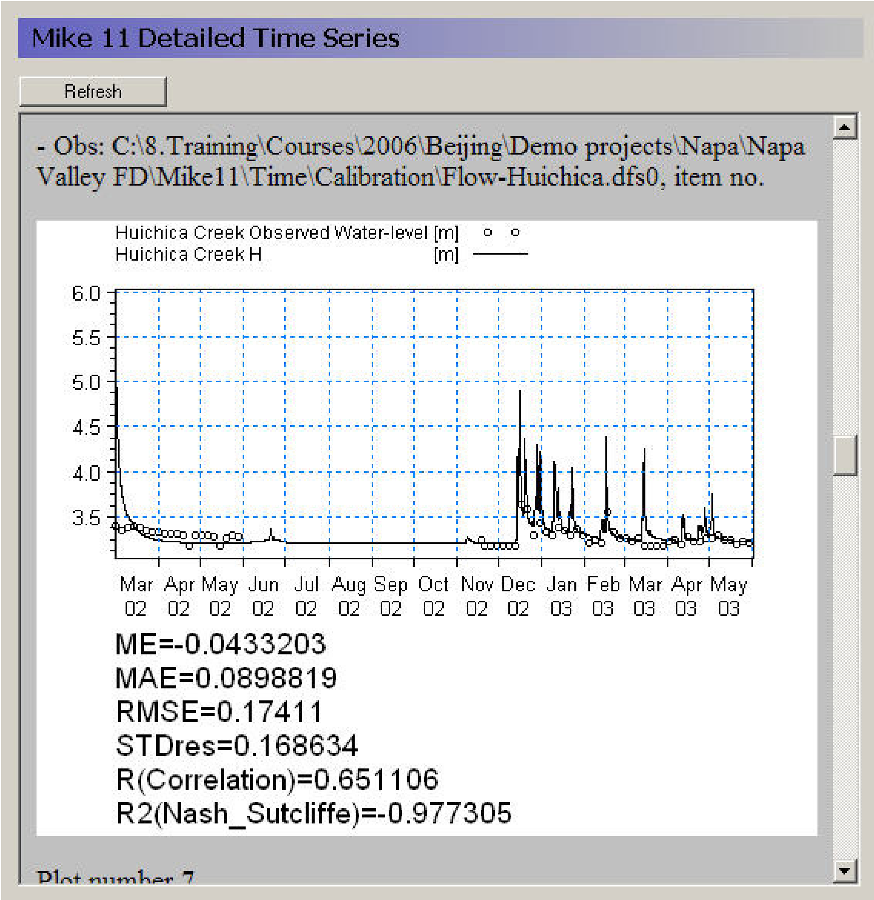


When the Result Viewer opens one of the items in the table, it creates a setup file for the particular view with the extension .rev. The name of the current .rev file is displayed in the title bar of the Results Viewer.

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 reloaded, and your results will open with the settings from the previous time you opened these results.

3.3.3 MIKE Hydro River Detailed Time Series



The MIKE Hydro River 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.

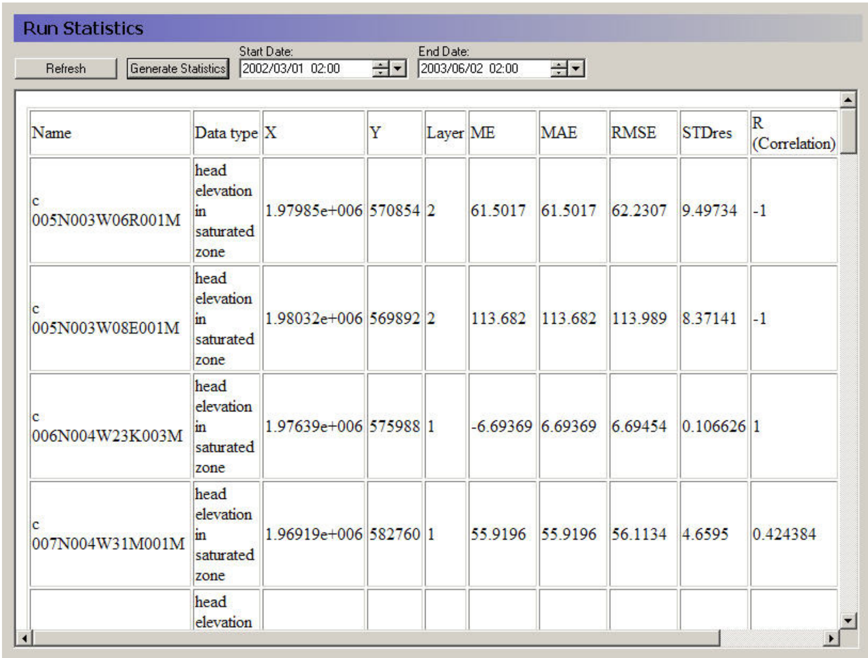
For information on the statistics see Statistic Calculations *(V1 p. 355)*.

3.3.4 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 spread-sheet, such as Microsoft Excel. The Run Statistics HTML document includes MIKE SHE and MIKE Hydro River results for all Detailed Time Series items that have 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).



Name	Data type	X	Y	Layer	ME	MAE	RMSE	STDres	R (Correlation)
c 005N003W06R001M	head elevation in saturated zone	1.97985e+006	570854	2	61.5017	61.5017	62.2307	9.49734	-1
c 005N003W08E001M	head elevation in saturated zone	1.98032e+006	569892	2	113.682	113.682	113.989	8.37141	-1
c 006N004W23K003M	head elevation in saturated zone	1.97639e+006	575988	1	-6.69369	6.69369	6.69454	0.106626	1
c 007N004W31M001M	head elevation in saturated zone	1.96919e+006	582760	1	55.9196	55.9196	56.1134	4.6595	0.424384
	head elevation								

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 *(V1 p. 355)*.

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.

3.4 Controlling Your Simulation

Controlling your simulation is about working with your model such that the simulation runs in a reasonable length of time.

3.4.1 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 *(V1 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.

3.4.2 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 may be 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. However, switching between methods will likely invalidate your UZ calibration, and require additional calibration adjustments.

- **Is your MIKE Hydro River simulation too detailed?** If your MIKE Hydro River cross-sections are too close together, MIKE Hydro River will run with a very short time step. Regional models can often be run with the simple routing methods in MIKE Hydro River, 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 *(V1 p. 24)* contains information on different hardware configurations,

- Controlling the Time Steps *(V1 p. 77)* contains information on how the dynamic time step control works,

- Overland Flow Performance *(V1 p. 105)* 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 *(V1 p. 83)* contains information on the using MIKE SHE with multi-core PCs and 64-bit operating systems.

3.4.3 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 Hydro River is running with a constant time step, then the Max allowed Overland (OL) time step must be a multiple of the MIKE Hydro River constant time step. If MIKE Hydro River is running with a variable time step, then the actual OL time step will be truncated to match up with the nearest MIKE Hydro River 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 *(V1 p. 178)* 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

$$timestep = timestep \times (1 + IncrementRate) \tag{3.1}$$

until the maximum allowed time step is reached.

Relationship to Storing Time Steps

The Storing Time Step specified in the Detailed time series output *(V1 p. 328)* 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.)

3.5 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 Windows 7 (64-bit) the default directory is:

```
C:\Program Files(x86)\DHI\2017\bin\x64
```

The path for the actual installation directory is saved in an environment variable called DHI_MIKE_2017, so the DOS command to add the default path to the PATH variable can be written as:

```
Set PATH=%PATH%;%DHI_MIKE_2017%
```

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.

3.6 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 *(V2 p. 359)*.

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.

3.7 Parallelization of MIKE SHE

The MIKE SHE solvers have been parallelized as much as possible and updated for 64-bit operating systems. Also, significant improvements in the memory and calculation efficiency was 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 eight, which means that the parallelization and AUTOCAL will support up to eight cores. If you want to use more than eight cores, then you must contact your local DHI office for additional run-time licenses.

4 The Results Viewer

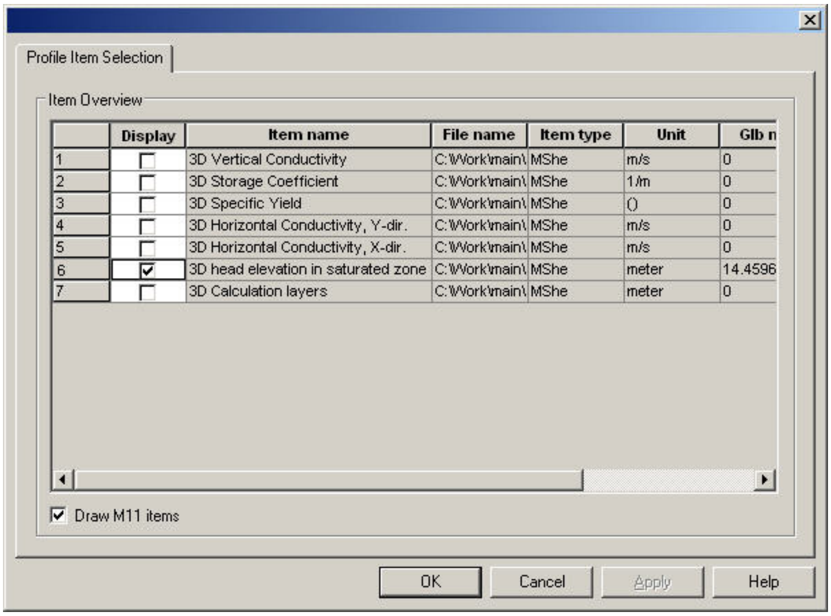
The Results Viewer is a generic MIKE Zero tool for displaying both gridded and unstructured spatial data. For more information on the Results Viewer, see the separate MIKE Zero documentation.

This chapter highlights some of the specific functionality for MIKE SHE.

4.1 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 output item can be selected. After selecting your item, click OK and the profile will be displayed.

You can modify the plot by right clicking on the plot and selecting Properties form the pop-up menu.

4.1.1 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.

4.2 UZ Specific Plots

4.2.1 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.

This option is no longer recommended and is essentially obsolete.

4.2.2 Transient UZ Column 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. The UZ Plot shows either water content, the pressure or the saturation for each node in the column (y-axis) for the entire simulation period (x-axis).

Addition graphical functions can be accessed by right-clicking in the graphical view. Modifications that can be made include changing the interpolation methods, adding the mesh, adding isolines,

Results from multiple UZ columns cannot be displayed on the same UZ Plot.

5 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.

5.1 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 *(V1 p. 85)* for more details on the Results Viewer.

5.1.1 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.

5.2 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 Hydro River or MIKE 11 model, the river results files will be overwritten. To prevent this, you must create different .sim11 or .mhydro files and change the results file name.
- 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 will not be correct and the .frf files can not be concatenated.

5.3 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 Hydro River 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 *(V1 p. 91)*. 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 instantaneous 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 Appendix A.1 MIKE SHE Output Items *(p. 397)*.

Note The **Code** listed in the tables in Appendix A.1 MIKE SHE Output Items *(p. 397)* is the **Data Type Code** that is needed when importing time series items into the Detailed time series output *(V1 p. 328)* dialogue.

5.3.1 Overland flow

The overland flow velocities are discussed in more detail in the section, Output: Overland Flow Velocities *(V2 p. 77)*.

Overland flow in the x- and y-direction

The overland flow in the x- and y- in the list of available output items is used for the water balance calculations.

The cell velocity cannot be directly calculated from these because the overland water depth is an instantaneous value output at the end of storing time step. Whereas, the overland flow in the x- and y- directions are mean-step accumulated over the storing time step. Thus, it is 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.

H Water Depth, P flux and Q flux

The P and Q fluxes are instantaneous fluxes across the positive cell faces of the cells. These are found in a separate *_flood.dfs2* results file, along with the H Water Depth. This file is the same format as the MIKE 21 output files generated by MIKE FLOOD. Thus, you can use this file to generate flood maps etc in, for example, the Flood Modelling Toolbox, or the Plot Composer.

You can also add these values to create flow vectors in the Results Viewer.

TS average, TS min, and TS max

Three calculated depths and velocities are available. These are the Average, Minimum and Maximum velocities and depths over the storing timestep. These values could be useful, for example, when evaluating susceptibility to erosion, or to calculate a flood hazard indicator.

5.3.2 Recharge

The data item Total recharge to SZ (positive for downwards flow) is stored on the UZ storing time step interval. It is a Step Accumulated value stored in the results file: *_2DUZ_AllCells.dfs2*. The calculated Recharge includes 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 (positive drainage, only one way)

The Total recharge to SZ 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.

6 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 6.1).

The water balance utility can be run from within the MIKE Zero interface or from a DOS 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.

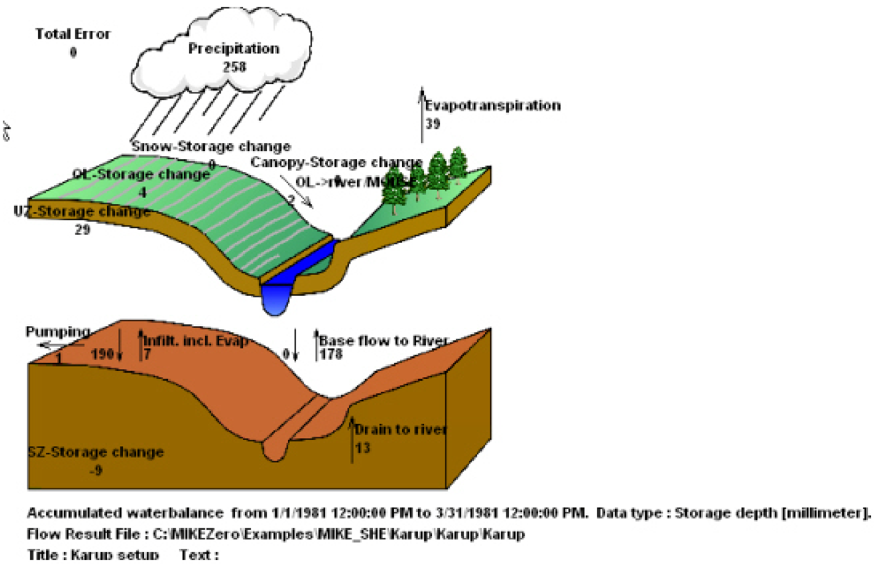
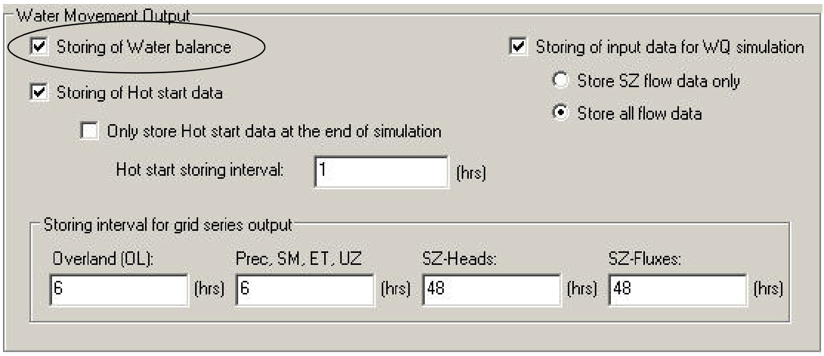


Figure 6.1 Graphical water balance output example

6.1 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 *(V1 p. 325)* dialogue. If you have forgotten to save the water balance data, then you will need to re-run your simulation.



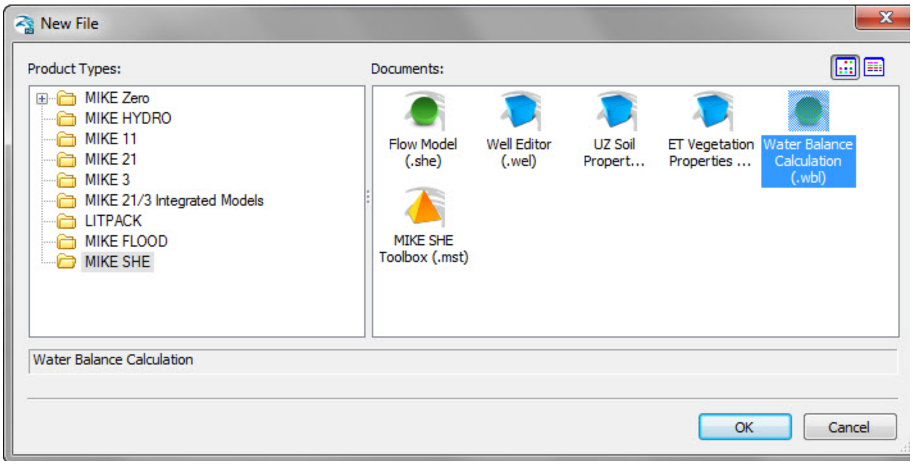
Note: the available time step resolution of the water balance will be the same as the Storing Interval for Grid Series Output. If the storing intervals are different, then it will be the largest Storing Interval.

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 *(V1 p. 92)*,
2. Extract the water balance data *(V1 p. 93)*
3. Specify your water balance *(V1 p. 94)*, and
4. Calculate and View the Water Balance *(V1 p. 97)*.

6.1.1 Create a new water balance document

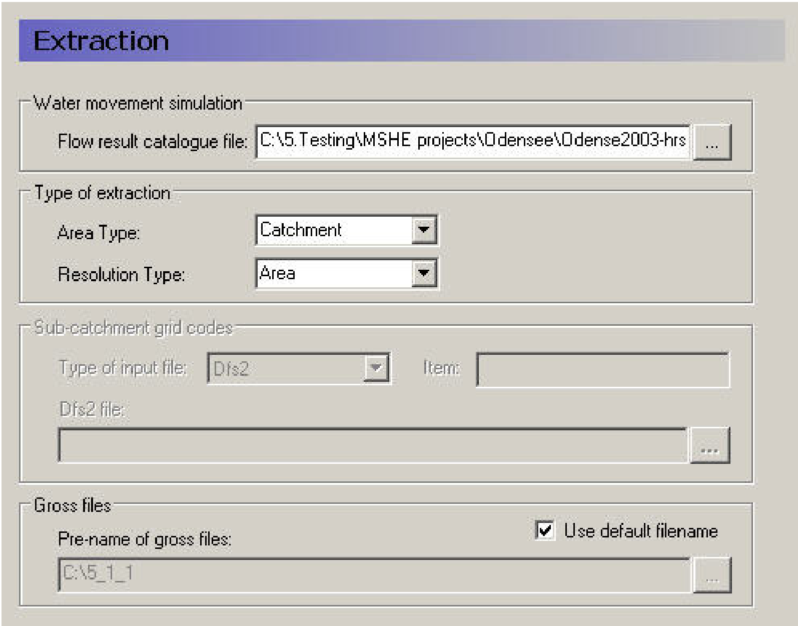
The new water balance document is created by selecting the File/New item in the top menu, or clicking on the New icon in the top menu bar. In the dialogue that appears, select MIKE SHE and Water Balance Calculations in the right hand box, as shown below.



6.1.2 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

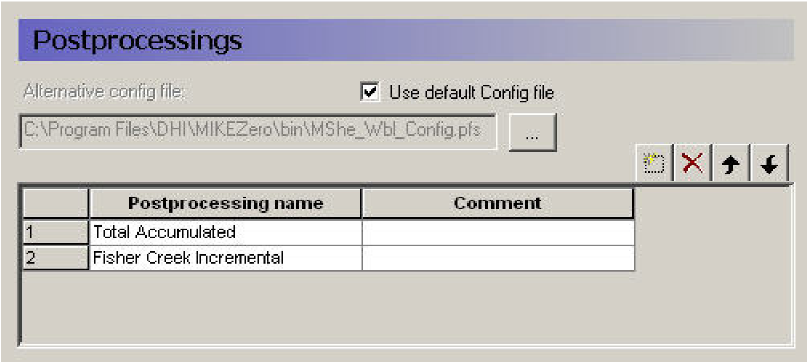
The pre-processor extracts the water balance data from the standard MIKE SHE output files and saves the data in a set of “gross” files. The file names of the gross files is built up from the project name and prefix specified here. The default value is normally fine.

Run the extraction

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

6.1.3 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.

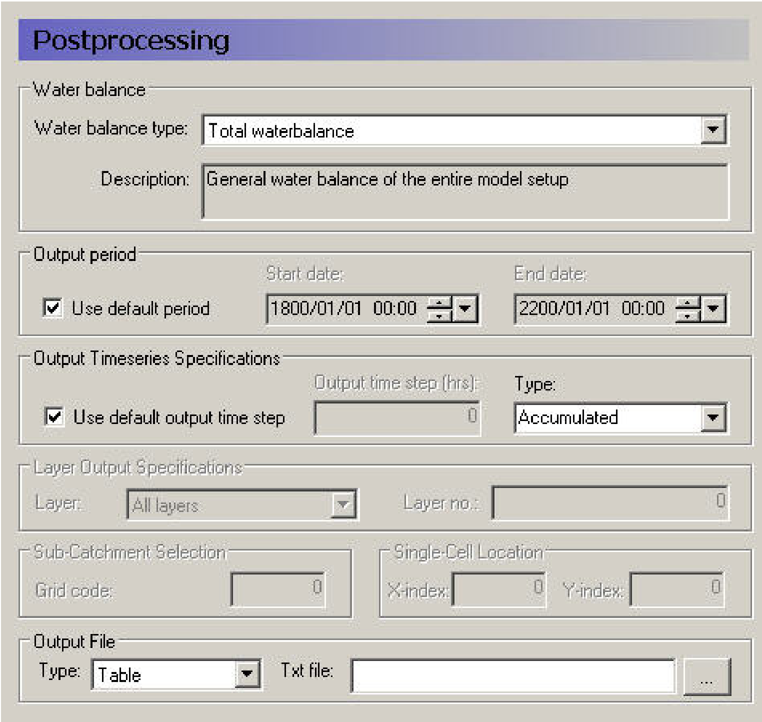


A single Postprocessing item is created by default when the water balance file is created. The default Postprocessing name can be change to a more appropriate name. Postprocessing items that are no longer needed can be deleted using the Delete button.

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 *(V1 p. 124)*.

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 *(V1 p. 99)* 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,
- A graphical cartoon (“Chart” type) of the total water balance (such as Figure 6.1),
- 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.

6.1.4 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.

6.2 Calculating Water Balances in Batch Mode

Like most DHI software, the water balance utility can be run in batch mode. This is useful if you want to run the water balance utility:

- immediately after a simulation that was also run in batch mode, or
- without using the water balance utility graphical user interface.

The water balance 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 *(V1 p. 124)*.

The executables for the water balance utility are found in the installation directory (`..\bin\x64`). There are three executables.

To perform the water balance extraction use:

```
MSHE_Wbl_Ex.exe xxx.wbl
```

where `xxx.wbl` is the water balance .wbl input file. To run an individual water balance use:

```
MSHE_Wbl_Post.exe xxx.wbl num
```

where `num` is the number of the post-processing water balance item that you want to run. To display the chart water balance output use:

```
MSHE_WblChart.exe xxx.txt
```

where `xxx.txt` is the output text file for the water balance chart defined in the post-processor item.

The number in the Postprocessing command must be 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.

To run the water balance utility in batch mode, the .wbl file must be created prior to executing the water balance 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 .wbl 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 .wbl file.

To run the Extraction and Postprocessing steps in batch mode, the PATH statement needs to include the directory where MIKE SHE was installed. For the 2016 Release, the default directory is

```
C:\Program Files (x86)\DHI\2016\bin\x64
```

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

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
```

6.3 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 6.1 SM - Precipitation and snowmelt items *(p. 102)*
- Table 6.2 CI - Canopy interception water balance items *(p. 104)*
- Table 6.3 OL - Overland flow items *(p. 107)*
- Table 6.4 UZ - Unsaturated Zone items *(p. 110)*
- Table 6.5 SZ - Saturated Zone - all layers *(p. 114)*
- Table 6.6 SZ - Saturated Zone - specified by layers *(p. 118)*
- Table 6.7 SZ - Saturated Zone - Linear Reservoir all layers *(p. 120)*

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.

6.3.1 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 6.1 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 6.1 SM - Precipitation and snowmelt items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sm.qPrecAndIrrToSnow	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
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

Table 6.1 SM - Precipitation and snowmelt items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sm.smWblErr	Snow storage water balance error. Sum of marked items.	Positive if water generated (∆storage + 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

6.3.2 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 6.2 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 6.2 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
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)	

6.3.3 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 the River.

The items listed in Table 6.3 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 Hydro River.

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 Hydro River and the two terms qSZFloodPos and qSZFloodNeg may be non-zero.

The exchange between ponded water and MIKE Hydro River 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 Hydro River (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 Hydro River over the next MIKE Hydro River time step(s), covering the period of the MIKE SHE Overland time step.

Thus, qsztoflood is not directly added as lateral inflow to MIKE Hydro River . 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 Hydro river as qfloodtoriv

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

Table 6.3 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
-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 Hydro River). 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

Table 6.3 OL - Overland flow items

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 Hydro River). 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
ol.qOLRivPos	Overland outflow to MIKE Hydro River	Outflow - positive	yes
ol.qOLRivNeg	Inflow from MIKE Hydro River 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 Hydro River	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 Hydro River 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 Hydro River nodes that are inside the current water balance area	Inflow (negative) or Outflow (positive)	yes

Table 6.3 OL - Overland flow items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ol.qFloodToRivEx	Net lateral inflow exchange between active flood code cells and MIKE Hydro River 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
ol.dOLSto	Change in overland storage	Positive if storage increases	yes
ol.OLWblErr	OL water balance error	Positive if water generated (Δstorage + Outflow > Inflow)	

6.3.4 Unsaturated Zone storage

The 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 6.4 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 6.4 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

Table 6.4 UZ - Unsaturated Zone items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
-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
-uz.qGWFeedBackUZ	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 Hydro River. 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 6.4 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 (∆storage + Outflow > Inflow)	

6.3.5 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 6.5 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 6.5 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 *(V2 p. 329)*). 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
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

Table 6.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
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
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

Table 6.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
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
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 Hydro River h-points. These are specified in the Extra Parameter option in SZ Drainage to Specified MIKE Hydro River H-points *(V2 p. 350)*	Outflow - positive	yes
sz.qSzRivPos	Baseflow from SZ to MIKE SHE 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 Hydro River). 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 6.5 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 Hydro River). 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 Time varying SZ drainage parameters *(V2 p. 353)*	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
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 6.5 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 6.6 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 6.7 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 6.7 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 inter-flow 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
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 6.7 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 Hydro River 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
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 Hydro River to account for discharge to riparian zones that is lost to ET.	Outflow - positive (from baseflow reservoirs only)	yes

Table 6.7 SZ - Saturated Zone - Linear Reservoir all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
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)	

6.3.6 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.

6.4 Standard Water Balance Types

Table 6.8 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 6.8 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

Table 6.8 Water balance types available in the default configuration files

Water balance type	Description
Irrigation component	Irrigation component water balance
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)

6.5 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.

Table 6.9 MSHE_Wbl_Config.pfs - example section to edit to customize the water balances

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
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 6.1 to Table 6.7. Map items can only have one group (NoGroups = 1)

Table 6.9 MSHE_Wbl_Config.pfs - example section to edit to customize the water balances

Line item	Comment
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.

6.5.1 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 6.1.

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.

.

Table 6.10 MSHE_Wbl_Config.pfs - example section to edit to customize the chart water balance

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

Table 6.10 MSHE_Wbl_Config.pfs - example section to edit to customize the chart water balance

Line item	Comment
[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

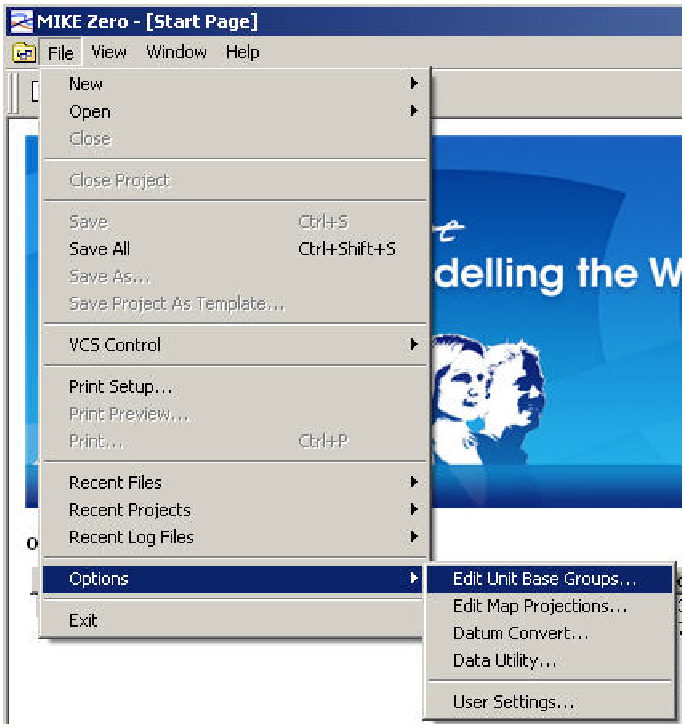
MIKE ZERO OPTIONS

7 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.

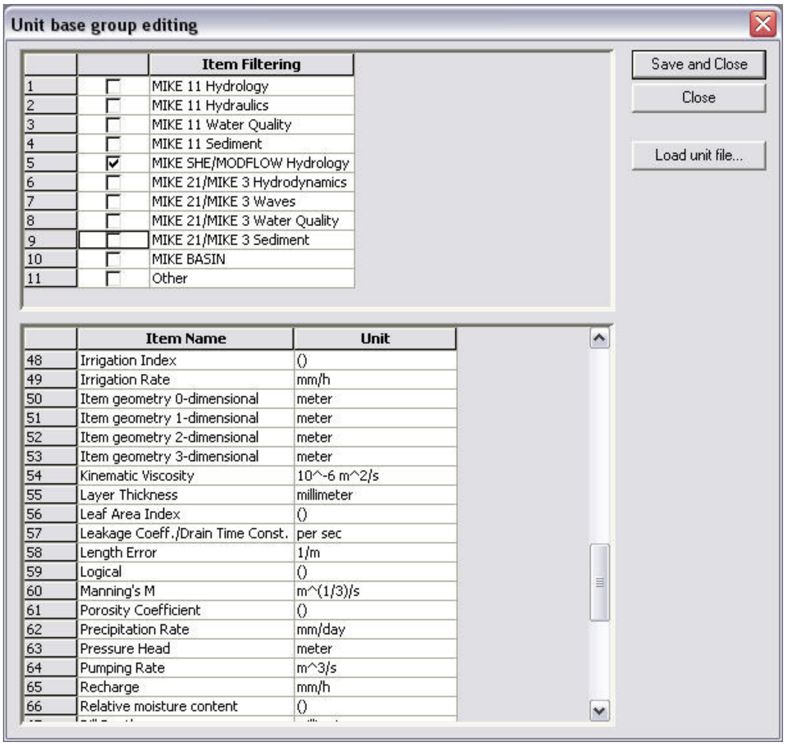
After you have changed the data units, click 'Save and Close'. This saves your changes to the default Unit Base Groups (.ubg) file:

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 text box, 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 Centre, who should forward your request to the developer for inclusion in the next release.

7.1 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.

7.2 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.

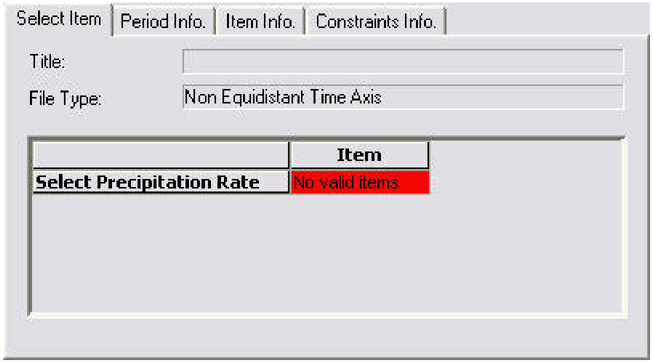
Note! If you want to save any of your model specific changes, then you should first save the MIKEZero.ubg to a new name.

7.3 Changing the EUM data type of a Parameter

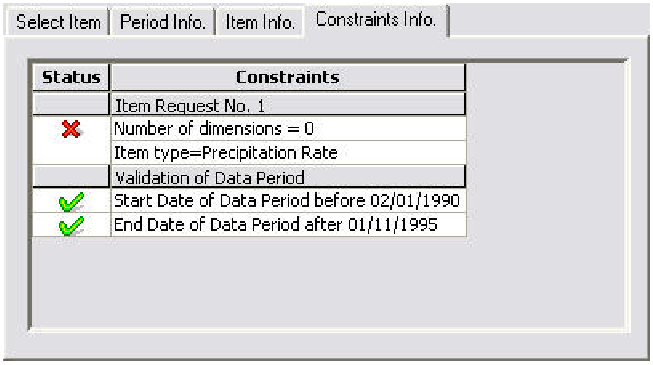
When you create a .dfs0 or .dfs2 parameter file, you must also define the EUM data type for each parameter in the file. When you assign a .dfs0 or a .dfs2 file to a parameter value, then MIKE SHE automatically verifies that the correct EUM data type is being used. If the wrong data type is present then 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.

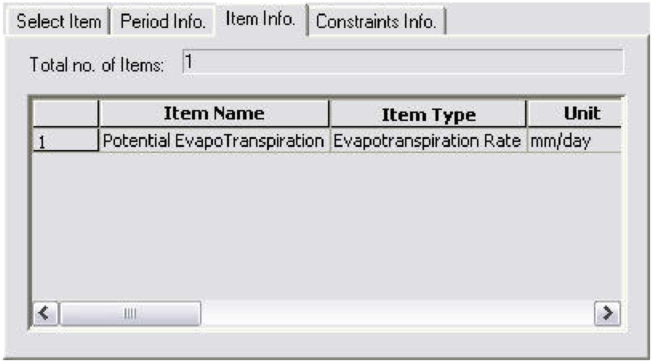


The find out why there is no valid items, you should look in the Constraints Info tab:



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:



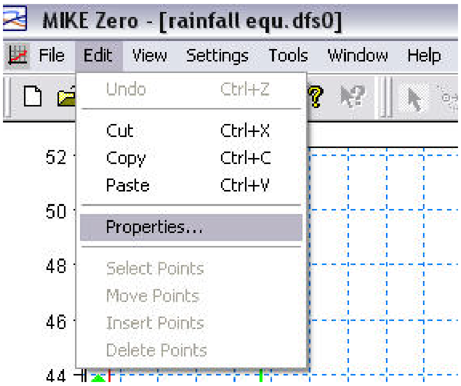
,

where you can see that the current Item Type is Evapotranspiration Rate.

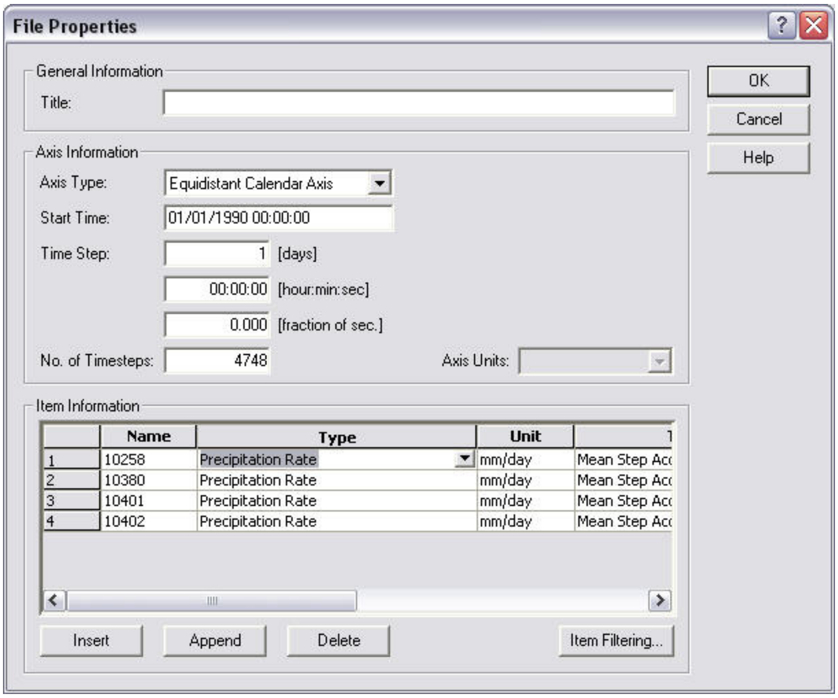
The next two sections outline how to change the EUM Type of an existing file.

7.3.1 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

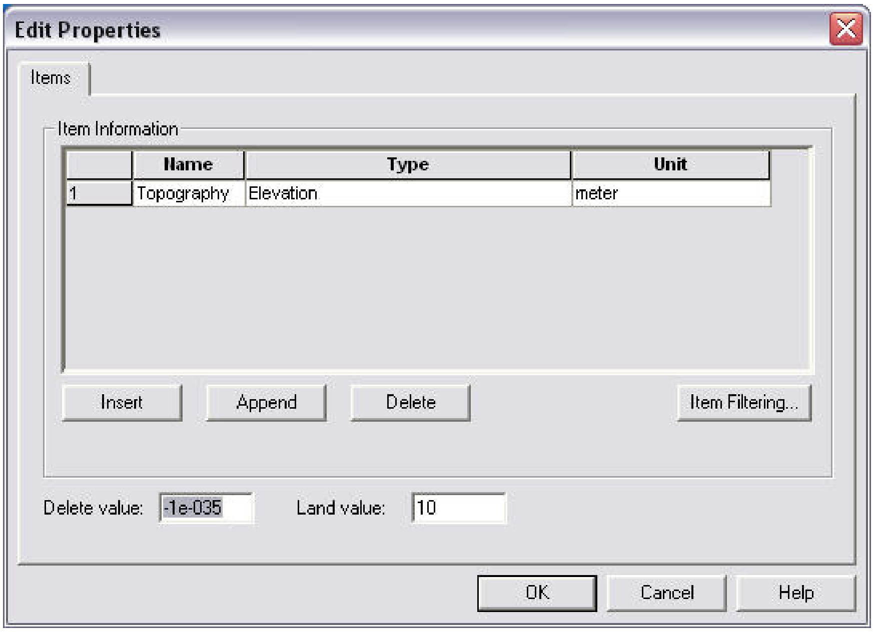


where you can change the EUM Type and the EUM Unit that is assigned for each time series in the file.

7.3.2 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

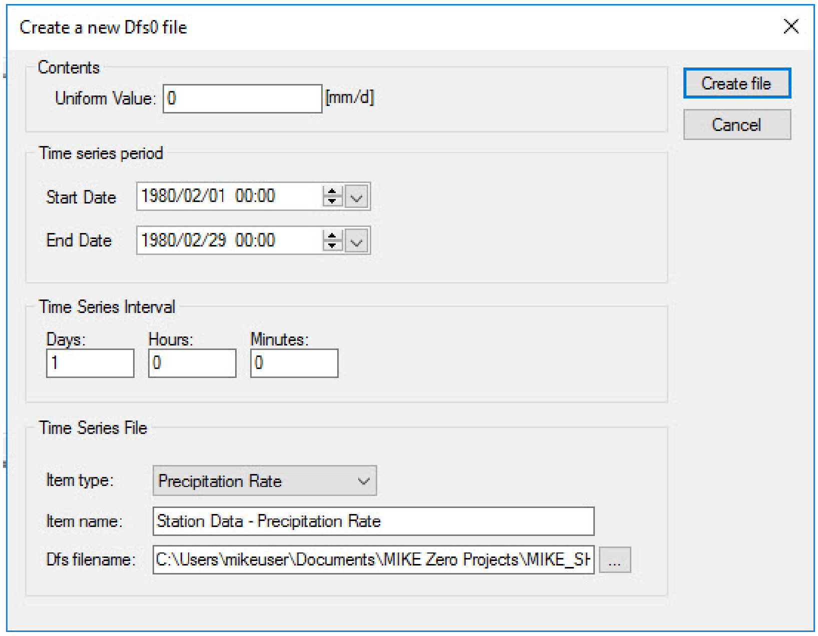
8 Time Series Data

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.

8.1 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:



Uniform time series

Every time step will have the same value. You can create new dfs0 files from Excel and ASCII data using the Time Series Batch Conversion tool in the MIKE Zero Toolbox.

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 *(V1 p. 176)*. 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 *(V1 p. 131)*) 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.

8.1.1 Import from ASCII or Excel

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.

You can create new dfs0 files from Excel and ASCII data using the Time Series Batch Conversion tool in the MIKE Zero Toolbox.

8.2 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.

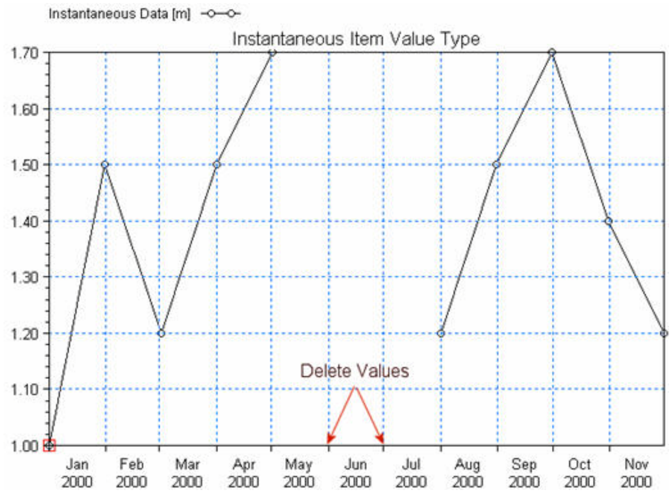
The result is a dfs2 file with one grid for each time step and the grid values are the time series values.

8.3 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:

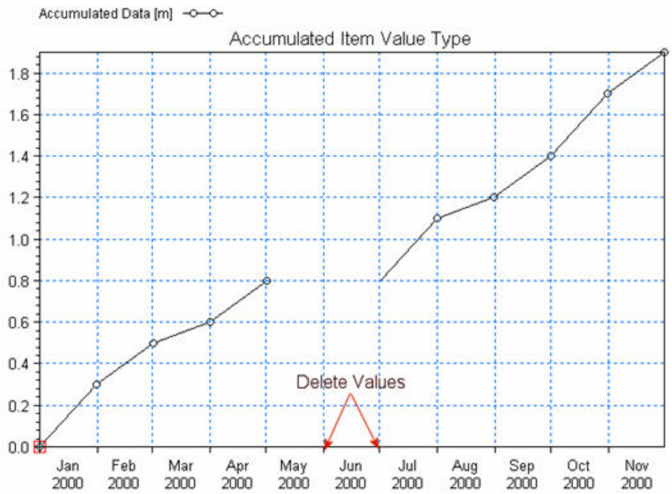
Instantaneous

The values are measured at a precise instant. For example, the air temperature at a particular time is an instantaneous value.


Instantaneous Data [m]
Instantaneous Item Value Type
1.70
1.60
1.50
1.40
1.30
1.20
1.10
1.00
Delete Values
Jan 2000
Feb 2000
Mar 2000
Apr 2000
May 2000
Jun 2000
Jul 2000
Aug 2000
Sep 2000
Oct 2000
Nov 2000

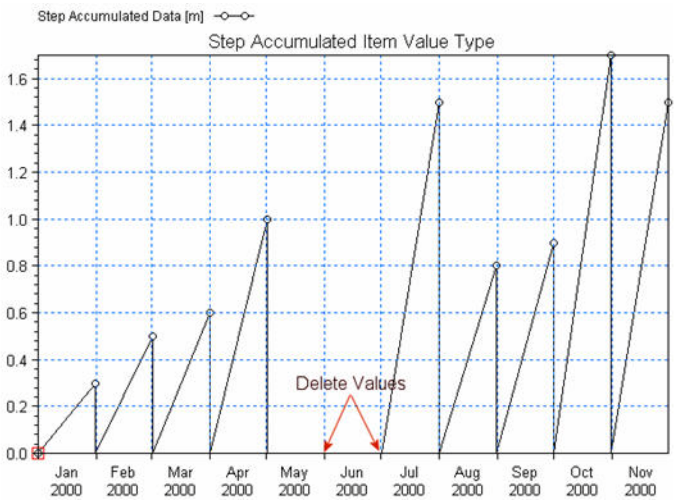
Accumulated

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.



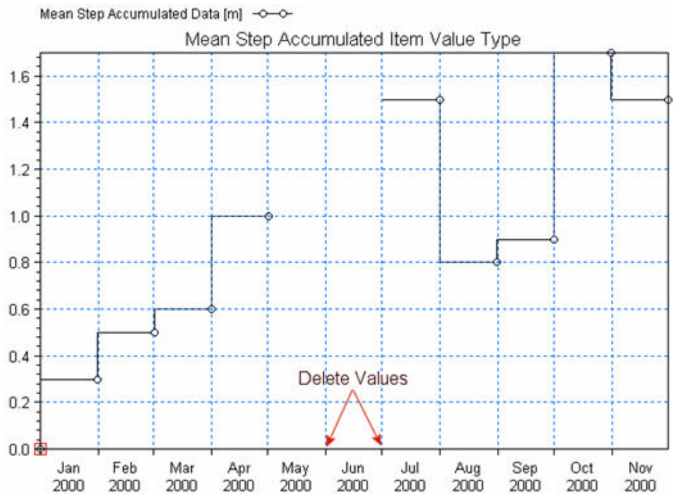
Step Accumulated

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.



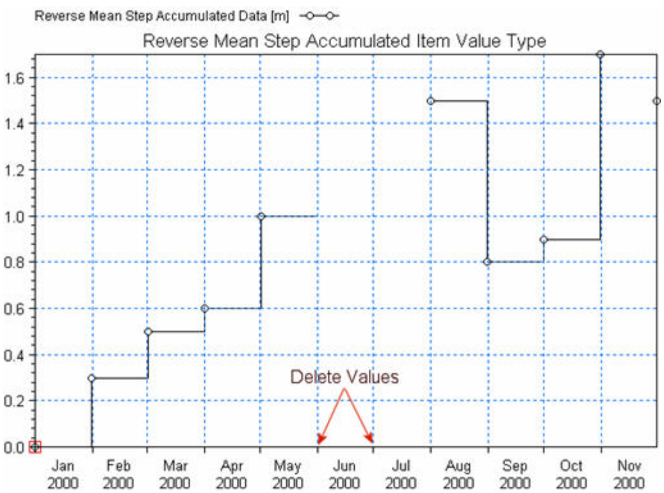
Mean Step Accumulated

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.



9 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.

9.1 Converting ArcGIS ASCII Grids to dfs

ArcGIS grids cannot be added directly in the MIKE SHE Setup Editor, but they can be converted to the dfs2 file format using the MIKE Zero Toolbox.

1. Select New from the top menu,
2. then select the MIKE Zero Tool box and
3. then choose GIS in the list.
4. finally, chose the Grid2Mike tool to convert your ArcGIS grid files to the dfs2 file format.

9.2 Converting dfs files to ASCII Grids and shape files

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 right click on any map view in the MIKE SHE Setup Editor 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.

10 Spatial Data

Spatial data includes all model data that can be location dependent, for example precipitation rates and soil parameters.

10.1 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-centred finite difference scheme. Whereas, MIKE SHE is based on a block-centred 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.

10.2 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

10.3 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

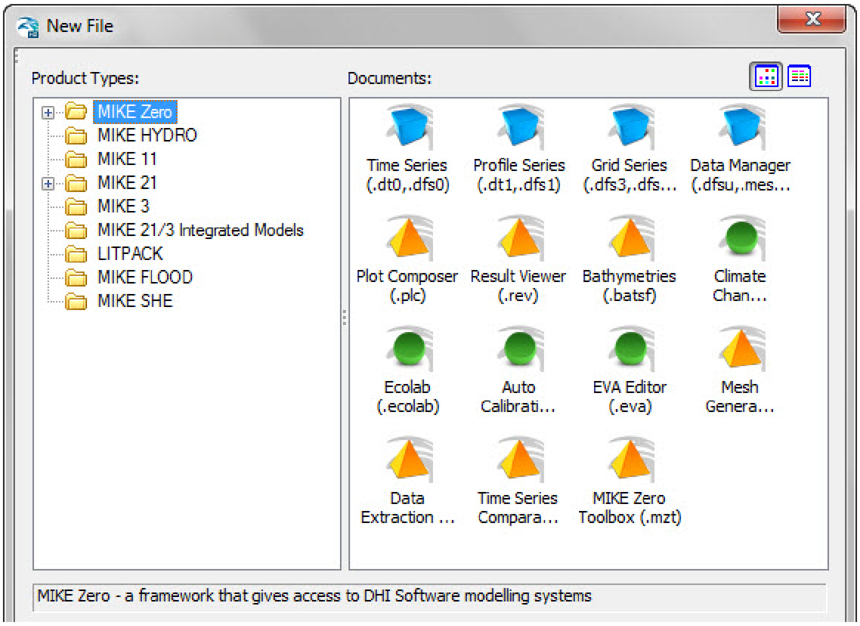
Grid Codes and Integer values do not normally vary with time. If such parameters do vary in time, then you must divide the simulation into time periods and run each time period as a separate simulation, starting each simulation from the end of the previous simulation using the Hot Start options (see Simulation Period).

10.4 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.

10.4.1 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.

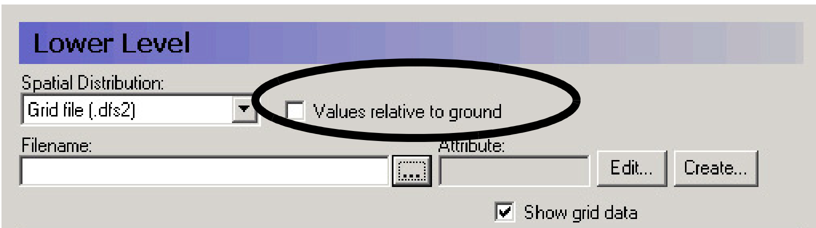
The following interpolation methods are included:

- Bilinear Interpolation *(V1 p. 157)*, or
- Triangular Interpolation *(V1 p. 160)*
- Inverse Distance Interpolation *(V1 p. 161)*.

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 Manning's 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

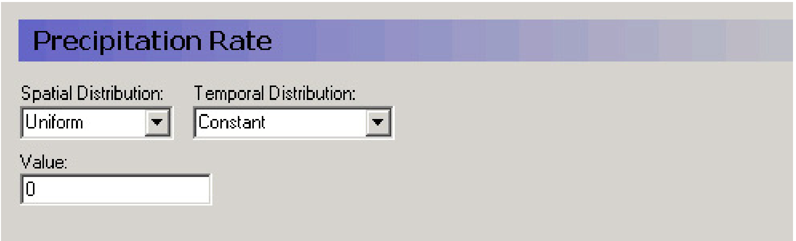
6. Save the file and then load it into the Data item for which you wanted it.

10.4.2 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).

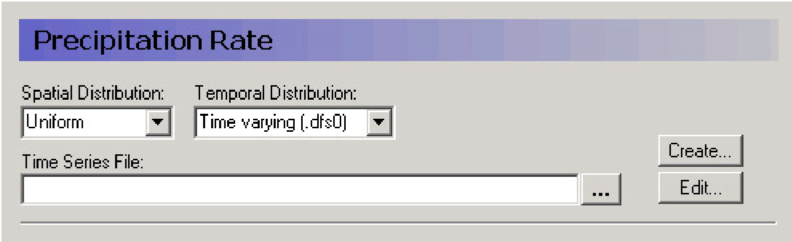
Often, time-varying data, such as precipitation rate, are spatially distributed using measurement stations, which in the model are translated into model zones using, for example, Thiessen polygons. In this case, each station is associated with a .dfs0 time series file that contains the time series of 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.

10.4.3 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 *(V1 p. 215)*
- Precipitation Rate *(V1 p. 221)*
- Vegetation *(V1 p. 236)*,
- Reference Evapotranspiration *(V1 p. 225)*
- UZ Soil Profile Definitions *(V1 p. 278)*,
- SZ Internal boundary conditions *(V1 p. 311)*, and
- Horizontal Extent *(V1 p. 336)* of SZ Lenses.

Note: The Horizontal Extent *(V1 p. 336)* 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.

10.5 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.

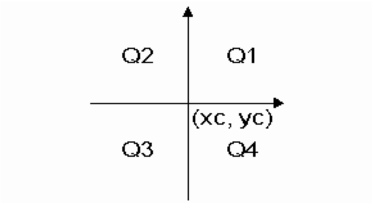


Figure 10.1 Definition of quadrants

10.5.1 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.

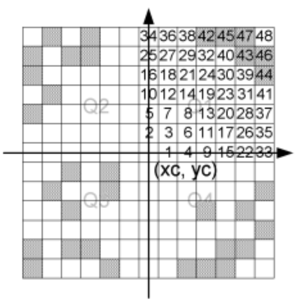


Figure 10.2 Illustration of the neighbouring grid cells being sorted

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 10.3.

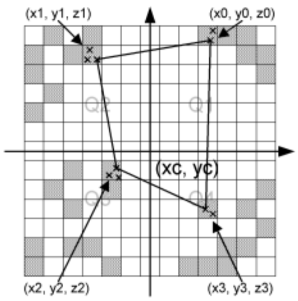


Figure 10.3 Illustration of the closest raw data points in each quadrant.

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 10.4.

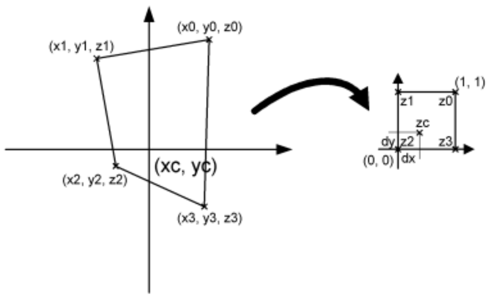


Figure 10.4 Illustration of bilinear interpolation.

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:

$$\begin{aligned} A_1 &= x_0 \\ A_2 &= y_0 \\ B_1 &= x_1 - x_0 \\ B_2 &= y_1 - y_0 \\ C_1 &= x_3 - x_0 \\ C_2 &= y_3 - y_0 \\ D_1 &= x_2 - x_1 + x_0 - x_3 \\ D_2 &= y_2 - y_1 + y_0 - y_3 \end{aligned} \tag{10.1}$$

Mapping the coordinates (x_c, y_c) to the normalized square (dx, dy) is done by solving equation (10.2).

$$ax^2 + bx + c = 0 \tag{10.2}$$

where the coefficients are

$$\begin{aligned} a &= D_1 B_2 - D_2 B_1 \\ b &= D_2 x_c - D_1 y_c - D_2 A_1 + D_1 A_2 + C_1 B_2 - C_2 B_1 \\ b &= C_2 x_c - C_1 y_c + C_1 A_2 - C_2 A_1 \end{aligned} \tag{10.3}$$

Solving equation (10.2) gives us *dx*.

$$dx = \frac{-b \pm \sqrt{b^2 - 4ac}}{2a} \tag{10.4}$$

where $0 \le dx \le 1$ is used to choose the correct root. *dy* can now be computed in two ways:

$$dy = \frac{x_c - A_1 - B_1 dx}{C_1 - D_1 dx} \tag{10.5}$$

or

$$dy = \frac{x_c - A_2 - B_2 dx}{C_2 - D_2 dx} \tag{10.6}$$

Choosing between (10.5) and (10.6) 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:

$$z_c = (1 - dx)(1 - dy)\, z_2 + dx(1 - dy)\, z_3 + (1 - dx) dy\, z_1 + dx dy\ z_0 \tag{10.7}$$

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:

$$w_i = \frac{1}{\sqrt{(x_i - x_c)^2 + (y_i - y_c)^2}} \tag{10.8}$$

$$w_s = \sum_{i=1}^{N} w_i \tag{10.9}$$

$$z_c = \frac{1}{w_s} \sum_{i=1}^{N} w_i z_i \tag{10.10}$$

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.

10.5.2 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:

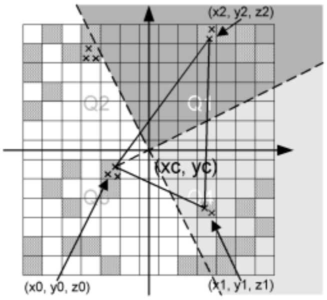


Figure 10.5 Illustration of triangular interpolation

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:

$$A = \frac{-(y_1 - y_0)(z_2 - z_0) + (y_2 - y_0)(z_1 - z_0)}{(x_1 - x_0)(y_2 - y_0) - (x_2 - x_0)(y_1 - y_0)}$$
$$B = \frac{(x_1 - x_0)(z_2 - z_0) + (x_2 - x_0)(z_1 - z_0)}{(x_1 - x_0)(y_2 - y_0) - (x_2 - x_0)(y_1 - y_0)} \qquad (10.11)$$
$$C = z_0 - Ax_0 - By_0$$

And secondly the actual interpolation is done:

$$z_c = Ax_c + By_c + C \qquad (10.12)$$

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.

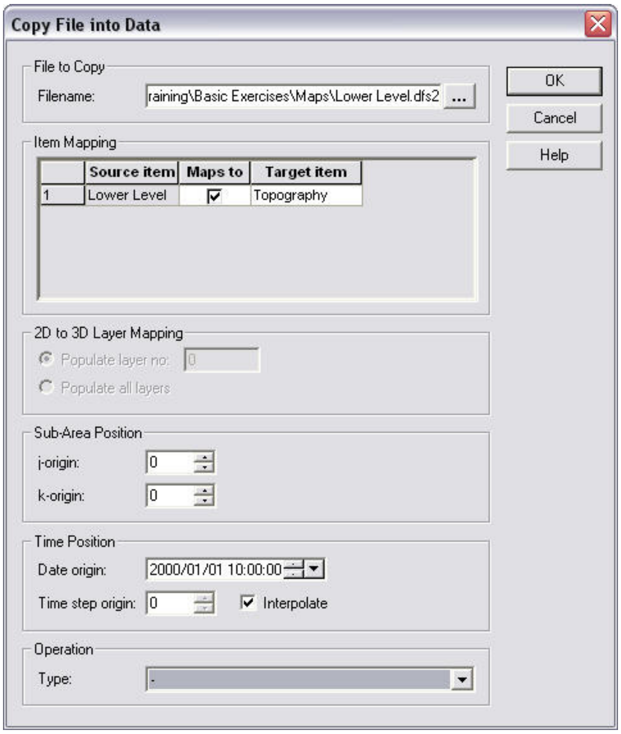
10.5.3 Inverse Distance Interpolation

The two inverse distance interpolation methods both use the nearest point in each quandrant to calculate the interpolated value, weighted by the distance or the distance squared respectively.

10.6 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

10.7 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. The only real limitation is that the grid files must have the same grid dimensions. Thus, this tool is much more flexible than the grid operations available in the Grid Editor. With this tool, you can make complex chains of operations, save the setup, and create batch files. You can also run it from a command line, which can save you a lot of time if you are doing the same operation many times.or after each simulation

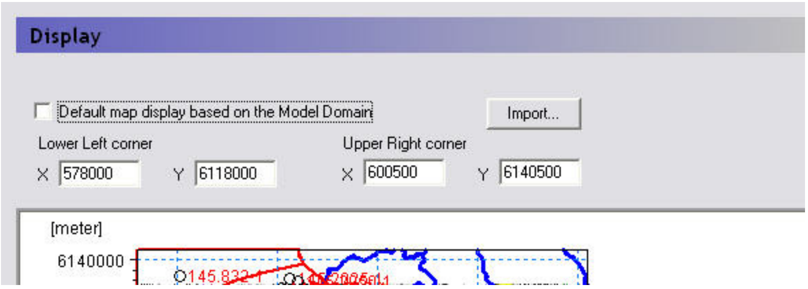
The Grid Calculator works like a wizard, with Next and Back buttons to move between dialogues.

11 Setup Data Tab

- This chapter is organized around the Setup Data Tree. For each branch in the data tree there is a corresponding subsection. The main sections include:
- **Display** *(V1 p. 168)* - display of map overlays
- **Simulation Specification** *(V1 p. 174)* - control and selection of water movement engines
- **Water Quality Simulation Specification** *(V1 p. 194)* - control and selection of water quality engines
- **Species** *(V1 p. 210)* - specification of species for water quality simulations
- **Water Quality Sorption and Decay** *(V1 p. 212)*
- **Model Domain and Grid** *(V1 p. 215)* - definition of model extent and grid
- **Subcatchments** *(V1 p. 217)* - definition of catchment boundaries for lumped parameter water movement engines
- **Topography** *(V1 p. 219)* - specification of land surface elevation
- **Climate** *(V1 p. 220)* - specification and extent of climate measurements, such as precipitation and evapotranspiration
- **Land Use** *(V1 p. 235)* - specification of vegetation and irrigation
- **Rivers and Lakes** *(V1 p. 253)* - link to MIKE Hydro River channel flow model
- **Overland Flow** *(V1 p. 256)* - specification of 2D overland sheet flow parameters for both water movement and water quality
- **Unsaturated Zone** *(V1 p. 273)* - specification of 1D unsaturated zone columns
- **Groundwater table for lower UZ boundary** *(V1 p. 289)* - specification of static lower boundary condition for unsaturated flow, if saturated zone not included
- **Saturated Zone** *(V1 p. 289)* - specification of 3D saturated zone parameters for both water movement and water quality
- **Sources** *(V1 p. 323)* - location and extent of solute sources for water quality simulation

- **Storing of Results** *(V1 p. 325)* - output selection for calibration time series and gridded data
- **Extra Parameters** *(V1 p. 334)* - extra input data for model options not yet available in the data tree

11.1 Display



From this dialogue and data tree branch, you can control the map overlays and size of the map view in the rest of the dialogues.

In any map view in the Setup Data tab, you can right click and chose Zoom In from the pop-up menu. The enlarged view of your map is persistent across all of the map views in the Setup Data tab, as well as to the Processed Data tab.

Also in the right click pop-up menu is a Zoom Extents function, which zooms the map view out to the full extents. By default, the maximum extents of the map view in the MIKE SHE dialogues is set to the size of the model, as defined in the Model Domain and Grid dialogue. However, un-checking the checkbox,

you can chose to set the lower left and upper right coordinates of the maximum extents of the map view.

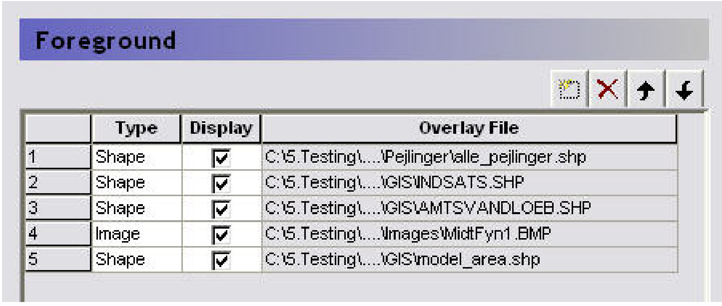
Import - The Import button allows you to read the coordinate extents from a map file, such as a .dfs2 file or a .shp file.

Note: The correct display of overlays requires that your Color Settings (found in your computer's Display Properties dialog) are set to Highest (32-bit). If your computer is set to 16-bit colours, a warning message will appear. This is not normally a problem, except that some display devices, such as older data projectors, will automatically reset your colour settings.

Related Items:

- Model Domain and Grid *(V1 p. 215)*

11.1.1 Foreground/Background



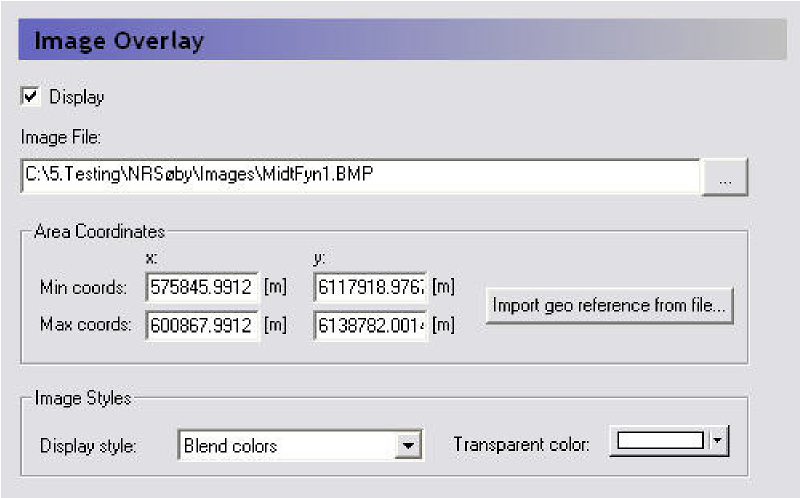
The Foreground and Background items are used to add map overlays to the map view. The table gives you an overview of the defined overlays and allows you to add, delete and hide overlays.

The order of the overlays in the list controls to some extent the way the overlays are displayed. Furthermore, the Foreground/Background choice determines whether the overlays are displayed in front of or behind the current grid, which in turn controls the way the colours etc. are displayed. The best way to understand the way the overlays are displayed, is to simply play around with a model and some maps to see how the display changes when the map is placed in the foreground vs background, or the order is changed.

The available map types includes,

– ESRI shape (.shp) files,
– Grid (.dfs2) files,
– Image (.bmp, .gif and .jpg) files,
– MIKE 11 river network (.nwk11) files,
– MIKE Hydro River (.mhydro), and
– MIKE SHE well database (.wel) files.

11.1.2 Image Overlays



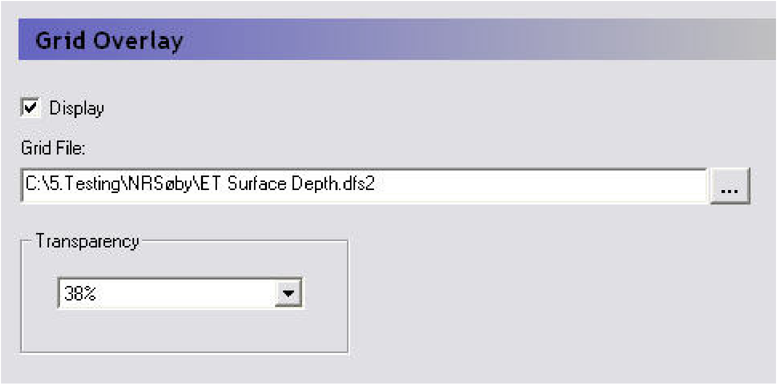
If you want to display a background image in your map view, then you should add an Image overlay. The available image formats include: .bmp, .gif and .jpg.

Area Coordinates - Since image files do not contain geographic information, you must specify the spatial location of the image. This is done by specifying the map coordinates of the lower left corner of the image (Minimum X and Y) and the upper right corner of the image (Maximum X and Y).

Import geo reference from file... - Some DHI programs allow you to geo reference an image file, in which case either a .bpw or a .bmpw file is created that contains the origin coordinates and some scaling information.

Image Styles - The Image Style is related to the way the pixel colours are averaged (or not averaged) in overlying grids and images. The Image Style variables have little influence on the image display when the image is displayed in the Background. However, in the Foreground, the Image style is very important. The best results are obtained with the Blend Colours selection., in which case the Transparent colour option is not used.

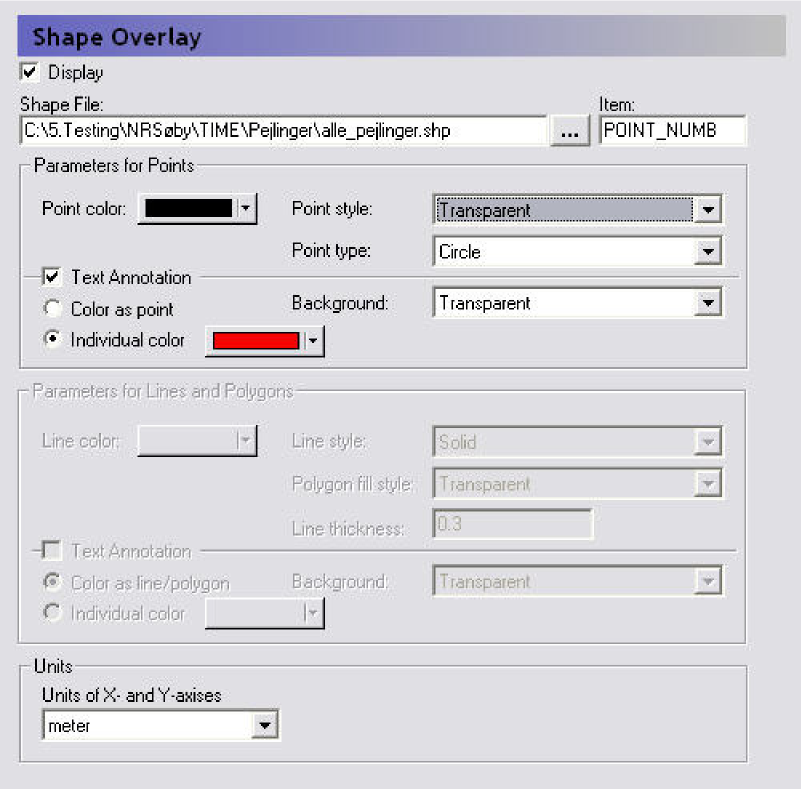
11.1.3 Grid Overlays



If you want to add a static .dfs2 grid on the map view, then you should add a Grid Overlay. If you add a time varying .dfs2 file the program will not object, but only the first time value will be displayed. If you add a .dfs2 file containing multiple items, such as a results file, you can select the grid item to display from within the file browser.

Transparency - This is used to control the way pixel colours are averaged for displaying images that overlay one another.

11.1.4 Shape Overlays



If you want to add an ESRI shape (.shp) file to the map view, then you must select a Shape overlay.

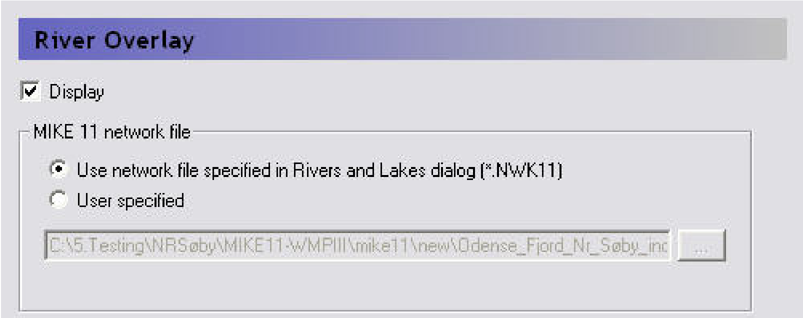
Shape File and Item - In the file browser dialogue you can select from the available items in the .shp file.

Parameters for Points - This section allows you to customize the way point .shp themes are displayed.

Parameters for Lines and Polygons - This section allows you to customize the way line and polygon themes are displayed.

Units - The ESRI .shp format does not include information on whether the length units are SI or Imperial. So, this combobox allows you to select the length units from a range of SI and Imperial length units.

11.1.5 River Overlays

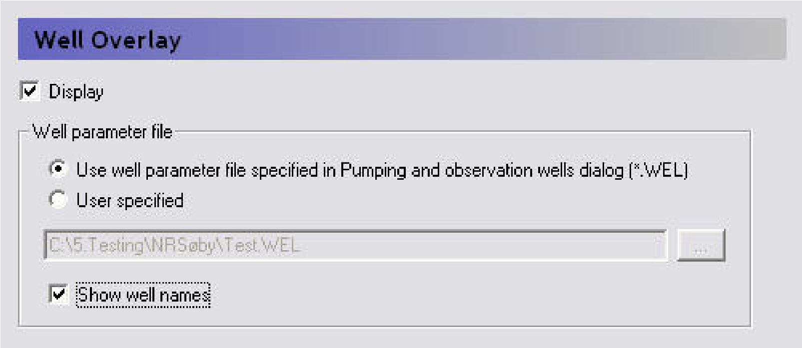


If you want to display a MIKE 11 or MIKE Hydro River network in your map views, then you can add a River Overlay. By default, the river network defined in the Rivers and Lakes dialogue is displayed. If you would rather display a different river network, for example, an overview network with fewer branches, then you can chose **User Specified** and specify the river network file.

Related Items:

- **Rivers and Lakes** *(V1 p. 253)*

11.1.6 MIKE SHE Well Database Overlays



If you want to display a MIKE SHE Well database in your map views, then you can add a Well Overlay. By default, the well database defined in the Pumping Wells dialogue is displayed. If you would rather display a different well database, for example, an overview database with fewer wells, then you can chose **User Specified** and specify the well database file.

Show well names This checkbox turns the well names on and off in the map view.

Related Items:

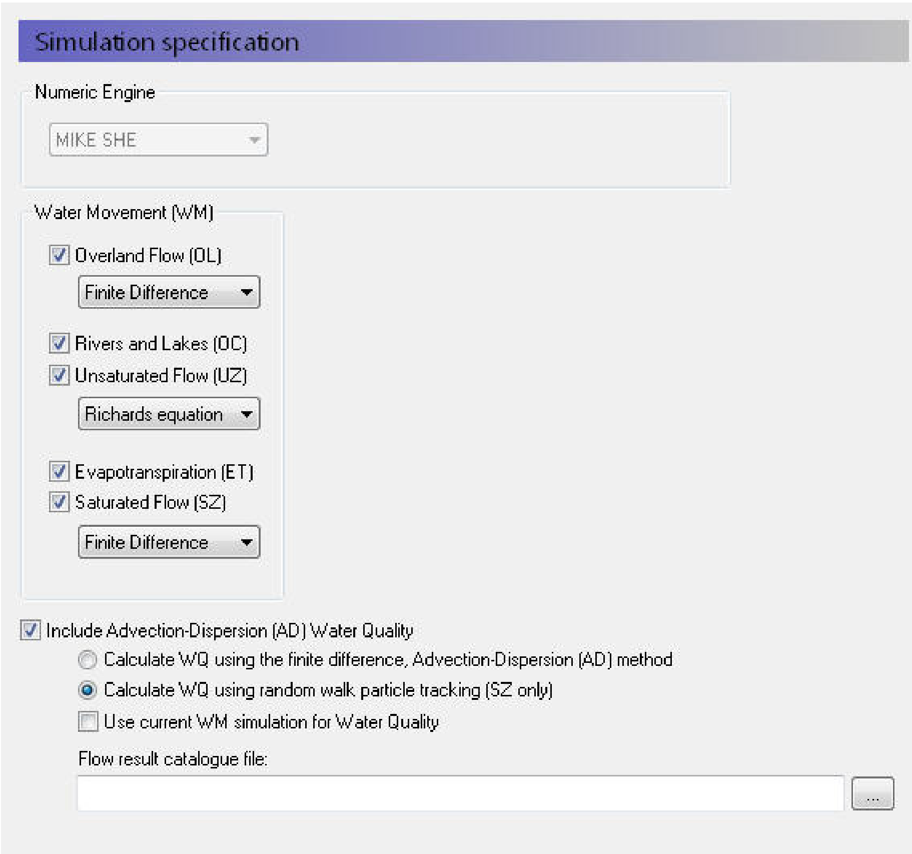
- Pumping Wells *(V1 p. 322)*

11.1.7 Current Layer

The Current Layer item refers to the grid item currently being displayed in the current map view.

Transparency - This is used to control the way pixel colours are averaged for displaying images that overlay one another.

11.2 Simulation Specification



The Simulation Specification dialogue is the key dialogue in the program. In this dialogue, you can select the key options for each of the components included in the simulation, including:

- Overland Flow - see Overland Flow - Technical Reference *(V2 p. 51)*)
  - Finite Difference Method *(V2 p. 51)*
  - Simplified Overland Flow Routing *(V2 p. 68)*
- Rivers and Lakes - see Channel Flow - Technical Reference *(V2 p. 109)*
- Unsaturated Flow - see Unsaturated Zone - Technical Reference *(V2 p. 141)*
  - 1D Richards Equation *(V2 p. 142)*
  - simplified 1D Gravity Flow *(V2 p. 153)*
  - Two-Layer Water Balance *(V2 p. 155)* for shallow water tables
- Evapotranspiration - see Evapotranspiration - Technical Reference *(V2 p. 17)*)
- Saturated Flow - see Saturated Flow - Technical Reference *(V2 p. 189)*)
  - 3D Finite Difference Method *(V2 p. 189)*
  - Linear Reservoir Method *(V2 p. 205)*

These choices are immediately reflected in the data tree, where the appropriate parameters are added or removed.

There is only one calculation option in this dialogue for Rivers and Lakes because the calculation methods are defined in the MIKE Hydro River User Interface. Likewise, the use of the simple or advanced Evapotranspiration methods are defined by the unsaturated flow method selected.

Water Quality options

Include Advection Dispersion (AD) Water Quality

At the bottom of this dialogue is a checkbox, where you can specify whether or not to include water quality in the simulation. If checked, the data tree will expand to include water quality data items.

PT versus AD

Below the initial checkbox to include water quality, there is a radio button to select the water quality method. There are two methods for calculating water quality in MIKE SHE.

The traditional Advection dispersion method can be used to calculate the movement of multiple solutes throughout the hydrologic system, including sorption and degradation.

The particle tracking (PT) method, tracks the movement of thousands of particles that move with the water. The PT method is currently only available in the saturated zone. The PT method allows you to calcuate well field capture zones and areas of influence.

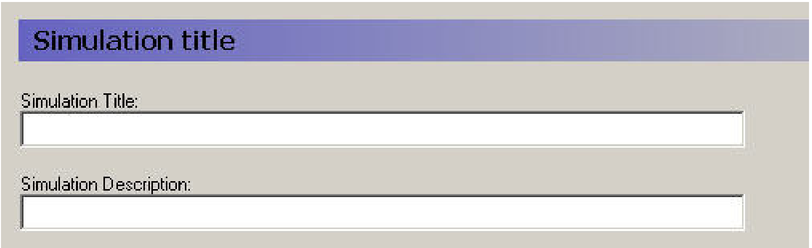
Use Current WM simulation for Water Quality

If you uncheck this check box, then you will be able to specify a different different water movement simulation as the source of the cell-by-cell flows for the water quality simulation. This allows you to use one water quality setup and calculate water quality based on several water movement scenarios. You must be careful though to not overwrite your results files from the previous water quality simulations.

Related Items:

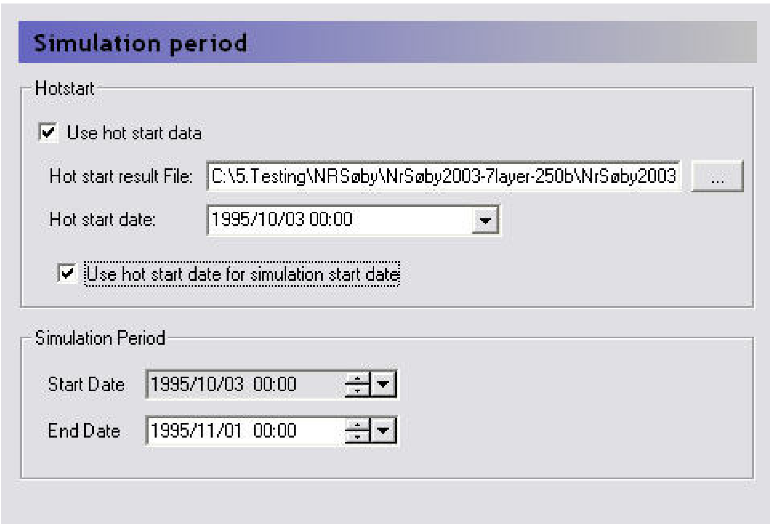
- Working with Solute Transport - User Guide *(V2 p. 297)*
- Particle Tracking-Reference *(V2 p. 319)*
- Advection Dispersion - Reference *(V2 p. 255)*

11.2.1 Simulation Title



Title and Description - The Title and Description will be written to output files and appear on plots of the simulation results.

11.2.2 Simulation Period



In the MIKE SHE GUI, all of the simulation 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).

The Simulation Period dialogue is primarily used to define the beginning and end of a transient simulation or the beginning and end of the averaging period for a steady state simulation.

However, the 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. To start a model from a previous model run, you must first save the hot start data, in the Storing of Results *(V1 p. 325)* dialogue.

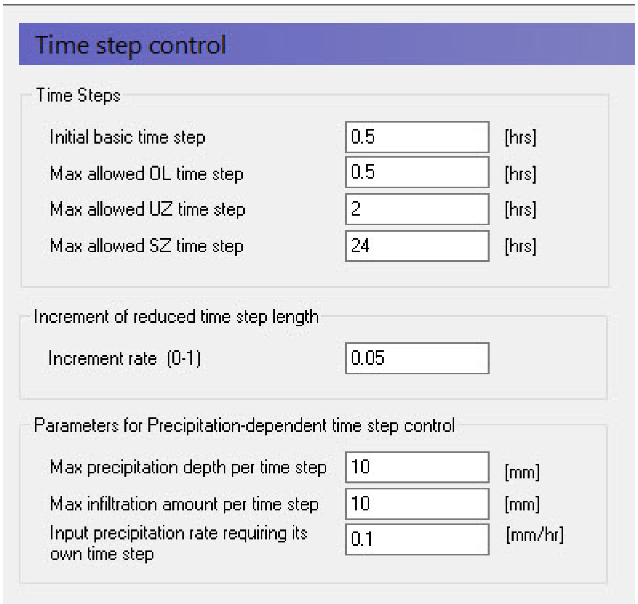
Hot start date - The hot start information is saved at specified intervals and the list of hot start dates is automatically filled in from the hot start file.

Use Hot start date for simulation start date - if you select this option, the simulation start date is greyed out and the simulation starts from the selected hot start date. Otherwise, you are free to chose an independent starting date and only the hot start data is simply used as initial conditions.

Related Items:

- Storing of Results *(V1 p. 325)*

11.2.3 Time Step Control



Time Step Control	
Conditions:	Maximum OL, UZ, and SZ time steps are only active when the component is selected

Table 11.1

Variable	Dimensions
Initial basic time step	[hrs]
Max allowed OL time step	[hrs]
Max allowed UZ time step	[hrs]
Max allowed SZ time step	[hrs]
Increment rate	-
Max precipitation depth per time step	[mm]
Max infiltration amount per time step	[mm]
Input precipitation rate requiring its own time step	[mm/hr]

Time Steps

Initial basic time step This is the initial time step for the Basic time step, unless the component's maximum allowed time step is less than the initial basic time step.

The Basic time step is the "heart beat" so to speak of the simulation. If all the components are included, this is the UZ time step. If UZ is omitted, then it is the SZ time step. If only OL is included, then it is the OL time step.

Assuming all components are included, then one SZ time step can have several Basic time steps, and one Basic time step can include several OL time steps.

If the Basic time step changes, for example due to precipitation (see below), then the OL, UZ and SZ time steps change together to maintain their relative sizes.

Max allowed 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 Hydro River is running with a constant time step, then the Max allowed Overland (OL) time step must be a multiple of the MIKE Hydro River constant time step. If MIKE Hydro River is running with a variable time step, then the actual OL time step will be truncated to match up with the nearest MIKE Hydro River 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.

Increment of reduced time step length

Increment rate This is a factor for both decreasing the time step length and increasing the time step length back up to the maximum time step, after the time step has been reduced. See Parameters for Precipitation-dependent time step control *(V1 p. 180)* for more details on when and how the time step is changed. A typical increment rate is about 0.05.

Parameters for 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. 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.

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 re-sampled 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.

This calculation depends on the amount of ponded water available for infiltration. This is the sum of the ponded water from the previous time step, the rainfall in the current time step and the total irrigation added in the current time step - plus the potential snow melt. The potential snow melt is more difficult because it depends on the air temperature and the wet snow storage fraction. If the air temperature is above the threshold melting temperature and the wet snow storage is greater than 95% of the maximum wet snow storage fraction, then the potential snow release equals the total snow storage.

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 measurment 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 timestep.

If the precipitation is corrected for elevation, then the calculation first loops over all the cells to estimate the maximum time step length. Then it checks the maximum precipitation criteria and adjusts the time step if necessary.

However, normally, the actual time series is read sequentially until the criteria is met or the end of the time step is reached. However, if the precipitation is corrected for elevation, the time series are not analysed on a cell-by-cell basis. So, there are time series conditions when the maximum precipitation criteria could still be exceeded.

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

$$timestep = timestep \times (1 + IncrementRate) \tag{11.1}$$

until the maximum allowed time step is reached.

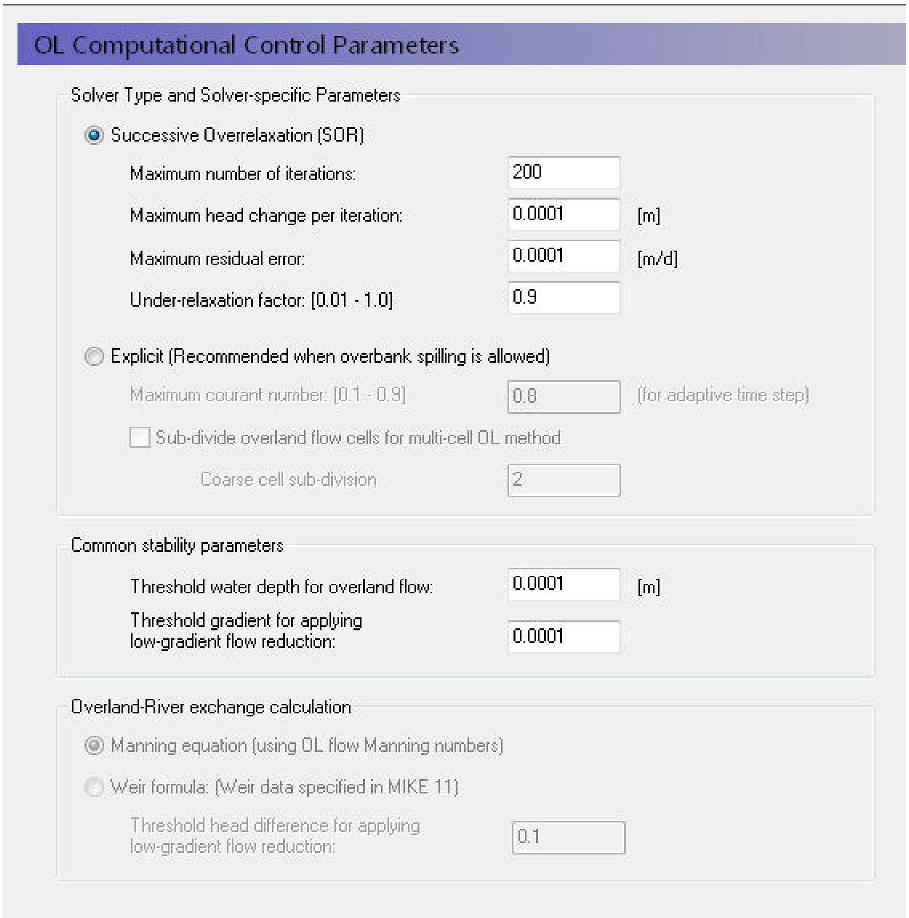
Relationship to Storing Time Steps

The Storing Time Step specified in the Detailed time series output *(V1 p. 328)* 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.)

11.2.4 OL Computational Control Parameters



Overland Computational Control Parameters	
Conditions:	if Overland Flow specified in Model Components

Table 11.2

Variable	Units
Maximum number of iterations	-
Maximum head change per iteration	[m]
Maximum residual error	[m/d]

Table 11.2

Variable	Units
Under-relaxation factor	-
Maximum courant number	-
Threshold water depth for overland flow	[m]
Threshold gradient for applying low gradient flow reduction	-
Threshold head difference for applying low gradient flow reduction	[m]

Solver Type for Overland Flow

Overland flow can be solved using either using the implicit Successive Over-Relaxation (SOR) Numerical Solution *(V2 p. 55)* or an Explict Numerical Solution *(V2 p. 56)*).

In the SOR method, the depth of overland flow is solved iteratively (implicitly) using a Gauss-Seidel matrix solution. The SOR method is a means of speeding up convergence in the Gauss-Seidel method. The iteration procedure is identical to that used in the saturated zone, except that no over-relaxation is allowed. The SOR method is faster but not as accurate compared to the Explicit method. However, when calculating overland flow to generate runoff, the SOR method is typically accurate enough.

The Explicit method is typically much slower than the implicit SOR method because it usually requires much smaller time steps. However, it is generally more accurate than the SOR method and is often used to calculate surface water flows during flooding. Thus, we recommend that you use the explicit method when overbank spilling from MIKE Hydro River to overland flow is allowed.

SOR parameters

Maximum number of iterations If the maximum number of iterations is reached, then simulation will go onto the next time step and a warning will be written to the simulation log file. The default value is 200 iterations, which is normally reasonable. You may want to increase this if you are consistently exceeding the maximum number of iterations, and the residuals are slowly decreasing.

Increasing the following two residual values will make the solution converge sooner, but the solution may not be accurate. However, it may be such that only a couple of points outside your area of interest are dominating the convergence and inaccuracies in these points may be tolerable. Decreasing these values will make the solution more accurate, but the solution may not converge at very small values and it may take a long time to reach the solution. If you decrease these values then you may also have to increase the maximum number of iterations.

Maximum head change per iteration If the difference in water level between iterations in any grid cell is greater than this amount, then a new iteration will be started. This will continue until the maximum number of iterations is reached. The default value is 0.0001 m, which is normally reasonable.

Maximum residual error If the difference in water level between iterations divided by the time step length in any grid cell is greater than this amount, then a new iteration will be started. This will continue until the maximum number of iterations is reached. The default value is 0.0001m/d, which is normally reasonable.

Under-relaxation factor The change in head for the next iteration is multiplied by the under-relaxation factor to help prevent numerical oscilations. Thus, lowering the under-relaxation factor is useful when your solution is failing to converge due to oscillations. This will have the affect of reducing the actual head change used in the next iteration. However, often it is more effective to reduce the time step. The under-relaxation factor must be between 0.01 and 1.0. The default value is 0.9, which is normally reasonable.

Explicit parameters

Maximum courant number The courant number represents the ratio of the speed of wave propagation to the grid spacing. In other words, a courant number greater than one would imply that a wave would pass through a grid cell in less than one time step. This would lead to severe numerical instabilities in an explicit solution. The courant number must be greater than 0.1 and less then 1.0. For a detailed discussion of the courant criteria see the Courant criteria *(V2 p. 57)*.

Subdivide overland flow cells for multi-cell flow method The explicit overland flow solver includes an additional option for solving the water levels on a finer grid scale than the flow velocities. This method also modifies the intercell conductance to account for average flow area across the cell. See Multi-cell Overland Flow Method *(V2 p. 64)* for more information.

Common stability parameters

The common stability parameters are used by both the implicit SOR solver and the explicit solver.

Threshold water depth for overland flow This is the minimum depth of water on the ground surface before overland flow is calculated. Very shallow depths of water will normally lead to numerical instabilities. The default value is 0.0001 m.

The threshold depth for overland flow should not be confused with the Detention Storage *(V1 p. 259)*. The detention storage is related to the amount of water stored in local depression on the ground surface, which must be filled before water can flow laterally to an adjacent cell.

Threshold gradient for applying low gradient flow reduction In flat areas with ponded water, the head gradient between grid cells will be zero or nearly zero and numerical instabilities will be likely. To dampen these numerical instabilities in areas with low lateral gradients a damping function has been implemented. The damping function essentially increases the resistance to flow between cells. 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 can become significant.

The damping function is controlled by a minimum gradient below which the damping function becomes active. Experience suggests that you can get reasonable results with a minimum gradient between 0.0001 and 0.001. The default minimum gradient is 0.0001. Higher values may lead to a divergence from the Mannings solution. Lower values may lead to more accurate solutions, but at the expense of smaller time steps and longer simulation times.

For a detailed discussion of the damping function see the Low gradient damping function *(V2 p. 59)* or Threshold gradient for overland flow *(V2 p. 80)*.

Overland River Exchange Calculation

Overland flow will discharge into the River Link if the water elevation in the cell is higher than the bank elevation. The rate of discharge to the river is dictated by the Mannings calculation for overland flow. However, this flow is only one way, that is from overland flow to the river.

If you want to include overbank spilling from the river to the overland grid cells, then you must use the weir formula, which provides a mechanism for water to flow back and forth across the river bank.

Note Whether or not to allow overbank spilling from the river to overland flow is made in MIKE Hydro River for each coupling reach. If you do not allow overbank spilling in MIKE Hydro River, then the overland river exchange is only one way, but uses the weir formula instead of the Mannings formula for calculating the amount of exchange flow.

If you do not use the overbank spilling option, then you can still use the flood inundation option to “flood” a flood plain. In this case, though, the flooding is not calculated as part of the overland flow, but remains part of the water balance of MIKE Hydro River. For more information on the flood inundation method see the section on Flooding from MIKE Hydro River to MIKE SHE using Flood Codes *(V2 p. 129)*.

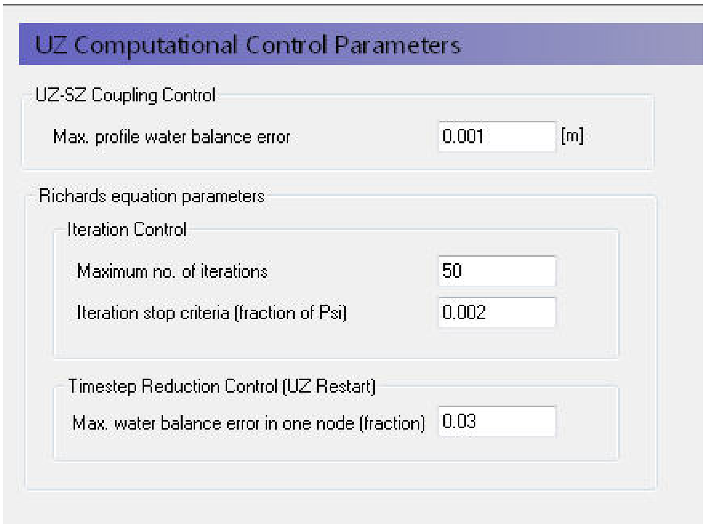
Threshold head difference for applying low gradient flow reduction If the difference in water level between the river and the overland flow cell is less than this threshold, then the flow over the weir is reduced to dampen numerical instabilities. In this case, the same damping function is used as in low gradient areas. The damping function essentially increases the resistance to flow between the cell and the river link. 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 can become significant.

The damping function is controlled by a minimum head difference between the river and cell below which the damping function become active. Experience suggests that you can get reasonable results with a minimum head difference between 0.05 and 0.1 metres. The default minimum head difference is 0.1. Higher values may lead to a divergence from the Mannings solution. Lower values may lead to more accurate solutions, but at the expense of numerical instabilities, smaller time steps and longer simulation times.

Related Items:

- Low gradient damping function *(V2 p. 59)*
- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

11.2.5 UZ Computational Control Parameters



UZ Computational Control Parameters	
Conditions:	if Unsaturated Flow specified in Model Components

Table 11.3

Variable	Units
Maximum profile water balance error	EUM [water level]
Maximum number of iterations	-
Iteration Stop criteria	-
Maximum water balance error in one node	m

The unsaturated flow is solved iteratively when the Richards Equation method is chosen, but is solved directly for both the Gravity Flow module and the Two-Layer Water Balance methods.

UZ-SZ Coupling Control

Since the Two-Layer Water Balance method does not calculate a water table, the UZ-SZ coupling control is only used in the Richards Equation and Gravity Flow modules.

Maximum profile water balance error The tolerance criteria in the UZ-SZ coupling procedure is the maximum allowed accumulated water balance error in one UZ column. If this value is exceeded the location of the groundwater table will be adjusted and additional computations in the UZ component will be done until this criteria is met. The recommended value is 1e-5 m or less. Details on this coupling procedure can be found in Coupling the Unsaturated Zone to the Saturated Zone *(V2 p. 167)*.

Additional Richards Equation Control Parameters

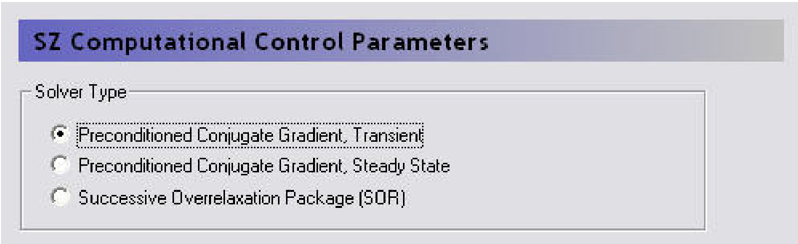
When the Richards Equation method is used, you can specify:

Maximum number of interations This is the maximum number of iterations that the Richards Equation implicit solver will do before it goes onto the next time step. If the solver has not converged, a message will be written to the log file.

Iteration Stop Criteria The solution is deemed to have converged when the difference in pressure head between iterations for all nodes is less than or equal to the iteration stop criteria. The recommended value is 0.02m or less.

Maximum water balance error in one node This is defined as the fraction of the total saturated volume in the node. The time step will automatically be reduced if the error is exceeded. The recommended value is 0.03 or less. However, smaller values may lead to very short UZ time steps and long run times.

11.2.6 SZ Computational Control Parameters



SZ Computational Control Parameters	
Conditions:	if Saturated Flow specified in Model Components

For the saturated zone in MIKE SHE there are two solvers to choose from:

- the pre-conditioned conjugate gradient method, and
- the successive over-relaxation method.

The Successive Over-relaxation solver is the original solver in MIKE SHE and the Pre-conditioned Conjugate Gradient Solver is based on the USGS's PCG solver for MODFLOW (Hill, 1990).

Steady-state vs Transient Simulations

The Solver type controls whether or not the simulation is run as a Steady-state model or not - if you chose the Pre-conditioned Conjugate Gradient-Steady-State option then the simulation will be run in steady-state. Otherwise, the simulation will be run as a transient simulation.

If the SZ simulation is steady-state, then the PCG solver is the only solver available. Although the same options are available for both the steady-state and the transient PCG solvers the optimal parameters or combination of parameters and options is most likely different in the two cases. Thus, the recommended settings are different in both cases.

Iteration Control

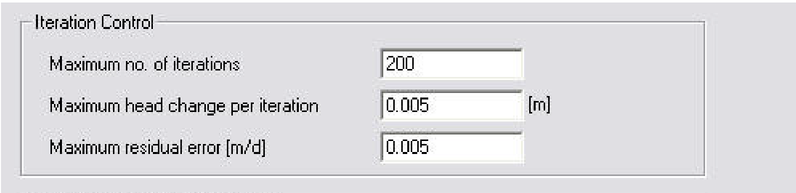


Table 11.4

Variable	Units
Maximum number of iterations	-
Maximum head changed per iteration	The same unit as specified by the EUM [elevation]
Maximum residual error	[m/day]

The iteration procedure can be stopped when either the iteration stop criteria are reached or when the maximum number of iterations is reached. The iteration stop criteria consist of a mass balance criteria and a head criteria. Both of these criteria must be chosen carefully to ensure that the solution has converged to the correct solution.

The default option settings normally perform well in most applications. Usually there is no need for changes. Changes to the default options should not be done unless the solution does not converge or convergence is extremely slow.

Maximum number of iterations - The maximum number of iterations should be sufficiently large to avoid water balance errors due to non-convergence.

Maximum head change per iteration - The head criteria determines the accuracy of the solution. The computational time is very dependent on the value used. A value of 0.01m (0.025ft) is usually sufficient. During the initial model calibration a higher stop criteria can be used. The sensitivity of the head stop criteria should always be examined.

Maximum residual error - The maximum residual error is the tolerable mass balance error, which should be low but sufficiently high that the number of iterations is not excessive. A value of 0.001m/d is usually good for regional groundwater studies. In smaller scale applications, where solute transport will be investigated the mass balance criteria should be reduced, for example, to 0.0001 or 0.00001m/d. In general, a larger mass balance criteria should be used during model calibration to keep the initial simulation times shorter. For scenario calculations, the mass balance criteria can be reduced to ensure more accurate simulations and smaller mass balance errors. The SZ water balance should always be checked at the end of the simulation to ensure that the mass balance criteria used was reasonable.

Sink de-activation in drying cells

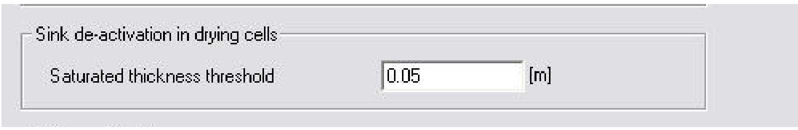


Table 11.5

Variable	Units
Saturated thickness threshold	The same unit as specified by the EUM [water level] (eg [ft] or [m])

Saturated thickness threshold - To avoid numerical stability problems the minimum depth of water in a cell should always be greater than zero. However, if the water depth is close to zero, then sinks in the cell, such as wells, should be turned off, since the cell is effectively ‘dry’. This value is the minimum depth of water in the cell and the depth at which the sinks are deactivated.

Maximum exchange from river during one time step

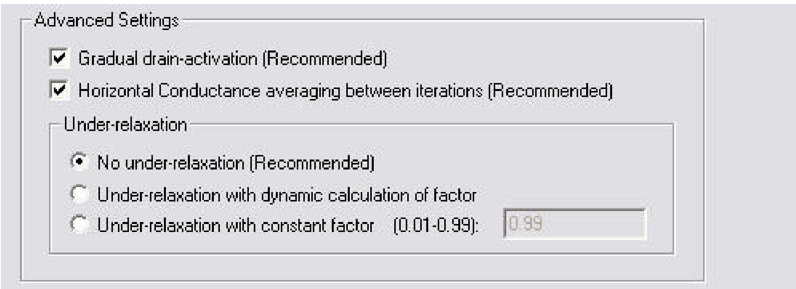
Table 11.6

Variable	Units
Maximum fraction of H point volume	-

Max. fraction of H-point volume - If you are simulating rivers with MIKE Hydro River, then this represents the maximum water that can be removed from the river during one SZ time step. Removing larger amounts of water could effectively dry out the river. If this occurs, then the SZ solver will issue a warning and only this fraction of water will be removed, which prevents rapid drying out of the river during a single time step.

Pre-conditioned Conjugate Gradient

Advanced Settings



Gradual activation of SZ drainage - To prevent numerical oscillations the drainage constant may be adjusted between 0 and the actual drainage time constant defined in the input for SZ drainage. The option has been found to have a dampening effect when the groundwater table fluctuates around the drainage level between iterations (and does not entail reductions in the drain flow in the final solution). For the steady-state solver and the transient solver the option is by default turned ON.

Horizontal conductance averaging between iterations - To prevent potential oscillations of the numerical scheme when rapid changes between dry and wet conditions occur a mean conductance is applied by taking the conductance of the previous (outer) iteration into account. By default this option is enabled for both steady-state and transient simulations.

Under-relaxation - Under-relaxation factors can be calculated automatically as part of the outer iteration loop. The algorithm determines the factors based on the minimum residual-2-norm value found for 4 different factors.To avoid numerical oscillations the factor is determined as 90% of the factor used in the previous iteration and 10% of the current optimal factor.

The second option is to define a constant relaxation factor between 0 and 1. In general a low value will provide convergence, but at a low convergence rate - i.e. with many SZ iterations. Higher values increases the convergence rates, but also the risk of non-convergence. As a general rule a value of 0.2 has been found suitable for most setups.

The time used for automatic estimation of relaxation factors may be significant compared to subsequently solving the equations and the option is only recommended in steady-state cases. In transient simulations, 'No under-relaxation' is recommended.

Successive Over-relaxation

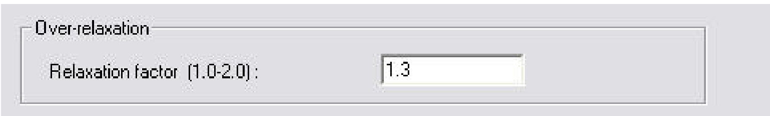


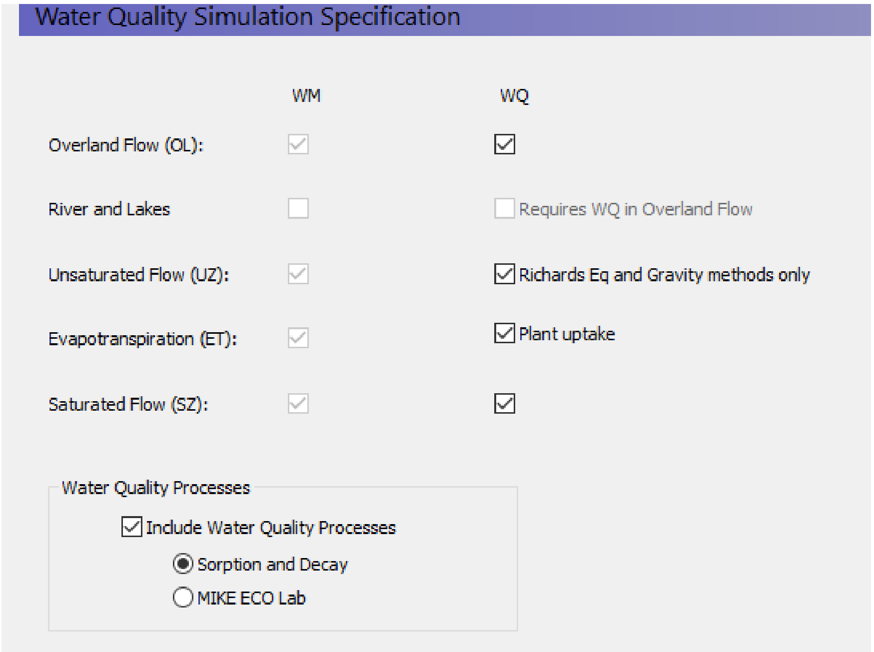
Table 11.7

Variable	Dimensions
Relaxation Factor	-

Over-relaxation

Relaxation factor - The speed of convergence also depends on the relaxation coefficient. Before you set up your model for a long simulation, you should test the iteration procedure by running a few short simulations with different relaxation coefficients. This coefficient must be between 1.0 and 2.0, with a typical value between 1.3 and 1.6.

11.3 Water Quality Simulation Specification



WQ Simulation Specification	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

In the WQ Simulation Specification dialogue, you can select which components of the hydrologic cycle will be included in the water quality simulation. The advection dispersion method calculates solute movements based on the intercell flows calculated in a water movement simulation. Therefore, only those components that are included in the water movement solution can be selected.

Solute transport in surface water bodies is specified in and calculated by MIKE Hydro River. If solute exchange to MIKE Hydro River is to be simulated, then the water quality must also be simulated in overland flow.

In the unsaturated zone, water quality cannot be calculated in the 2-layer water balance method.

If selected, plant uptake by roots is treated as a solute sink in the unsaturated zone.

Include Water Quality Processes - Sorption and first-order decay of solutes can be calculated by the water quality module. Turning on this option will allow you to specify decay in overland flow, and both sorption and decay in the UZ (Gravity and Richards methods) and SZ (Finite Difference method) modules.

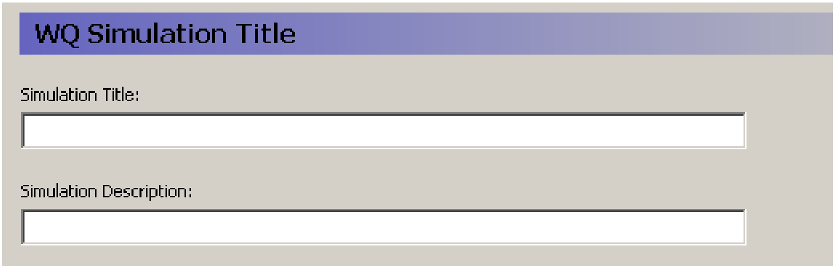
Sorption and Decay -

MIKE ECO Lab - MIKE ECO Lab options (see Working with MIKE ECO Lab in MIKE SHE - User Guide *(V2 p. 307)*)

Related Items:

- Working with Solute Transport - User Guide *(V2 p. 297)*
- Advection Dispersion - Reference *(V2 p. 255)*
- Reactive Transport - Reference *(V2 p. 281)*

11.3.1 WQ Simulation Title



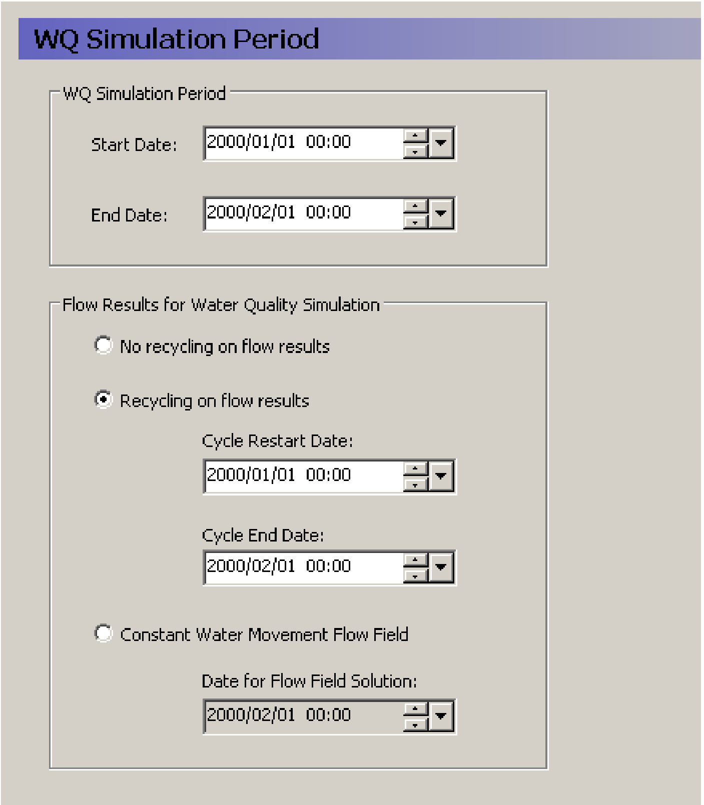
WQ Simulation Title	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

Title and Description - The Title and Description will be written to output files and appear on plots of the simulation results.

Related Items:

- Working with Solute Transport - User Guide *(V2 p. 297)*
- Advection Dispersion - Reference *(V2 p. 255)*

11.3.2 WQ Simulation Period



WQ Simulation Period	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

WQ Simulation Period

The water quality simulation does not have to be the same length as the water movement simulation. The only restriction is that the start date for the water quality simulation must be within the water movement simulation.

Flow Results for Water Quality Simulation

A water quality simulation requires the cell-by-cell water fluxes calculated by the water movement simulation. However, the water quality simulation does not have to be the same period as the water movement simulation. Therefore, the user interface is flexible in how it will use water movement cell-by-cell flow data.

No recycling on results - In this case, the water quality simulation end date must also be within the water movement simulation period, which means that the water quality simulation cannot extend beyond the water movement simulation.

Recycling on flow results - In this case, the water quality simulation can be much longer than the water movement simulation, based on a repeated set of water movement results. The water quality simulation starts on the Start Date with the flow results from the Cycle Restart Date. When the water quality simulation period reaches the Cycle End Date, the WQ simulation will continue but the flow results will be restarted at the Cycle Restart Date.

If the recycle dates do not match one of the saved time steps, then the nearest saved time step is used.

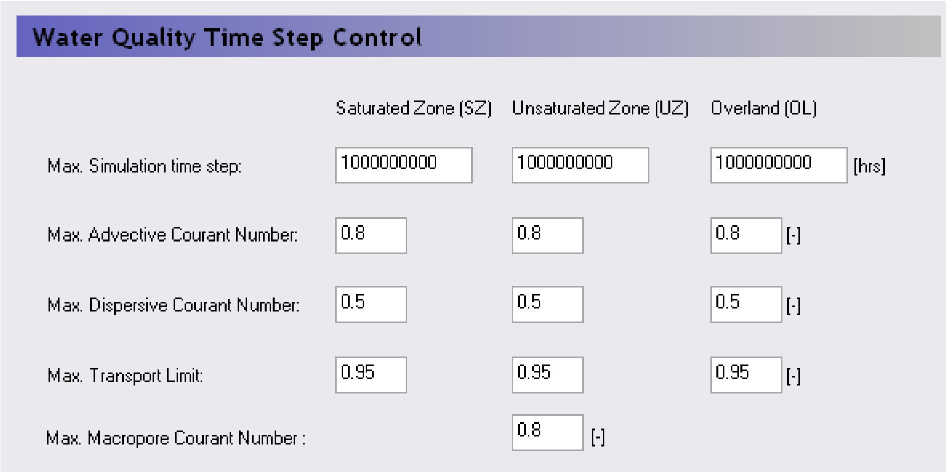
For example, you may have a two-year water movement simulation but you may want to simulate water quality for 10 years. To do this, you would specify the start and stop dates of the part of the water movement simulation that you want repeated. If you want to repeat the whole water movement simulation, then you would specify the beginning and end of the water movement simulation.

Constant water movement flow field - In this case, the nearest saved time step to this date will be used as a steady-state flow field for the transient water quality simulation.

Related Items:

- Working with Solute Transport - User Guide *(V2 p. 297)*
- Advection Dispersion - Reference *(V2 p. 255)*

11.3.3 Water Quality Time Step Control



Water Quality Time Step Control	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

The water quality simulation is completely decoupled from the water movement simulation and like the water movement itself, the water quality time steps can be different in each of the overland flow, unsaturated flow and saturated flow.

Maximum Simulation Time Step - This is the maximum user-specified time step allowed. The default value is very high so that the simulation runs by default with the highest possible time step. You might want to set this value to a short time interval, if you want the WQ time step to be uniform during the WQ simulation.

Stability Criteria

The courant number is a measure of the ratio of flow rate to grid size. For numerical stability, it is important that a dissolved solute does not travel too far in one time step. A courant number greater than 1.0 implies that a particle (solute) would move completely across or through a cell in a time step. The time step is reduced until all the time step criteria below are met.

Max. Advective Courant Number - The advective courant number represents the ratio of cell size to the time it would take a particle to move across a cell. This criteria is likely to be controlling if your flow velocities are high, or your dispersivity values are very low or zero. The default value is 0.8. If your actual time step is being controlled by this criteria, then you could increase it to make the simulation run faster. However, you will need to check to make sure the simulation has converged properly and that the mass balance is reasonable.

Max. Dispersive Courant Number - The dispersive courant number represents the ratio of the cell size to time it would take a particle to move across a cell due to dispersion. This criteria is likely to be controlling when the velocities are very slow and the dispersivity is non-zero. The default value is 0.5. If your actual time step is being controlled by this criteria, then you could increase it to make the simulation run faster. However, you will need to check to make sure the simulation has converged properly and that the mass balance is reasonable.

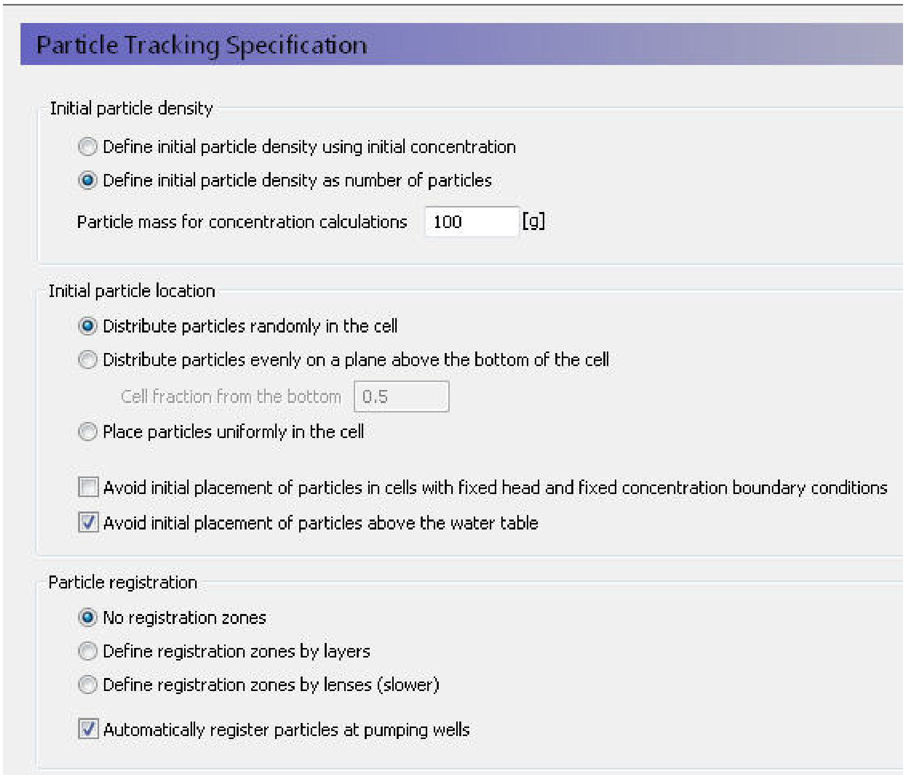
Max. Transport Limit - The transport limit restricts the fraction of the total amount of mass that can leave the cell in one time step. The default value 0.95, which usually does not limit the time step.

Max. Macropore Courant Number - The macropore courant number represents the ratio of UZ cell thickness to the time it would take a particle to travel vertically across a UZ cell due to macropore flow. This criteria is likely to be controlling if your macropore velocities are high. The default value is 0.8. If your actual time step is being controlled by this criteria, then you could increase it to make the simulation run faster. However, you will need to check to make sure the simulation has converged properly and that the mass balance is reasonable.

Related Items:

- Working with Solute Transport - User Guide *(V2 p. 297)*
- Advection Dispersion - Reference *(V2 p. 255)*

11.4 Particle Tracking Specification



Particle Tracking Specification	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue and the Random Walk Particle Tracking sub-option is specified

In the Particle Tracking Specification dialogue, you can select the initial density of particles, the initial location of the particles in the cells, and how the particles should be registered as the move through the saturated zone.

Initial particle density

The particle density relates the number of particles to a solute concentration. However, the initial number of particles can have a significant affect on simulation run time and output file size.

Define initial particle density as initial concentration - The initial concentration is defined for each SZ layer. See Initial (secondary) concentration *(V1 p. 313)*. The number of particles in each cell is calculated by dividing the mass of solute in the cell by the particle mass specified in the text box below.

Define initial particle density as number of particles - If this option is selected, then an Initial number of particles per cell *(V1 p. 314)* item is added to the data tree for each SZ layer.

Particle mass for concentration calculations - This mass is used for determining the intial number of particles if the first option above is selected. It is also used for converting the number of particles in a cell into a concentration during the simulation. A low mass per particle will yield more particles for the same concentration.

Initial particle location

The options for initial particle location determine the placement of the particles in each cell. If only a few particles are involved, the initial particle location can influence the results - especially if the cells are large.

A random distribution means that the particles may not be evenly distributed in every cell, but on average will be.

If the particles are evenly disributed on a plane, the particles are uniformly distributed on a plane parallel to the bottom of the cell. In this case, the optional fraction defines the location of the plane in the cell as a fraction of the cell thickness.

If the particles are evenly distributed in the cell, then the algorithm tries to distribute the particles evenly. This is relatively straight forward for regular numbers of particles, such as 4 or 5 particles, but more difficult for irregular numbers of particles, such as 11 particles.

Avoid initial placement in fixed head and concentration cells - Avoiding the initial placement in fixed head and fixed concentration cells, prevents some particles from being immediately extracted. (Default off)

Avoid initial placement above the water table - Avoiding the initial placement above the water table prevents particles being stuck right from the beginning of the simulation. If a particle is above the water table, then it will be immobile until the water table re-wets the location of the particle. This can happen when water tables are rising and falling. But, an initial particle above the water table may never become active and you may have a situation were nearly all of the initial particles in the upper SZ layer remain immobile for the entire simulation. (Default on)

Particle registration

The random walk method can track thousands of particles during a simulation. Unlike pathline analysis, the random walk method does not usually store all the resultant pathlines. Instead it stores the starting point and the end location or exit location of the particle. If you want to know if the particle has passed through a particle zone or layer, then you can create a registration zone. If the particle passes through the registration zone, then this will be recorded.

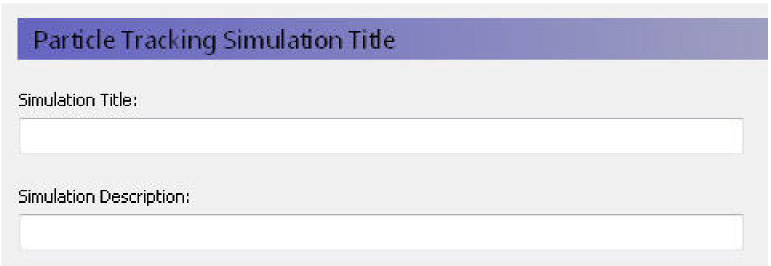
You can define the registration zones by numerical layer, or you can define them be lenses similar to the way you define geologic lenses. In both cases additional items are added to the SZ data tree. Defining the registration codes by lense is slower, because the PT engine has to continually check to see if the cell is located in one or more registration zone.

Particle registration is by default automatically done when particles are removed at wells to make it easier to calcuate well capture zones.

Related Items:

- PT Registration Codes/Lenses *(V1 p. 314)*
- Particle Tracking-Reference *(V2 p. 319)*

11.4.1 Particle Tracking Simulation Title



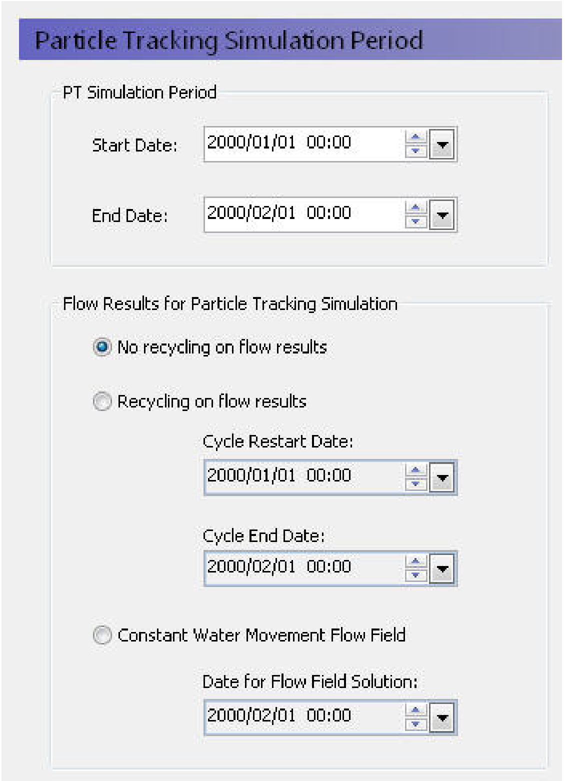
Particle Tracking Simulation Title	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue and the Random Walk Particle Tracking suboption is selected.

Title and Description - The Title and Description will be written to output files and appear on plots of the simulation results

Related Items:

- Working with Solute Transport - User Guide *(V2 p. 297)*
- Particle Tracking-Reference *(V2 p. 319)*

11.4.2 Particle Tracking Simulation Period



Particle Tracking Simulation Period	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue and the Random Walk particle tracking option is selected

PT Simulation Period

The particle tracking simulation does not have to be the same length as the water movement simulation. The only restriction is that the start date for the particle tracking simulation must be within the water movement simulation.

Flow Results for Particle Tracking Simulation

A particle tracking simulation requires the cell-by-cell water fluxes calculated by the water movement simulation. However, the particle tracking simulation does not have to be the same period as the water movement simulation. Therefore, the user interface is flexible in how it will use water movement cell-by-cell flow data.

No recycling on results - In this case, the water quality simulation end date must also be within the water movement simulation period, which means that the water quality simulation cannot extend beyond the water movement simulation.

Recycling on flow results - In this case, the water quality simulation can be much longer than the water movement simulation, based on a repeated set of water movement results. The water quality simulation starts on the Start Date with the flow results from the Cycle Restart Date. When the water quality simulation period reaches the Cycle End Date, the WQ simulation will continue but the flow results will be restarted at the Cycle Restart Date.

If the recycle dates do not match one of the saved time steps, then the nearest saved time step is used.

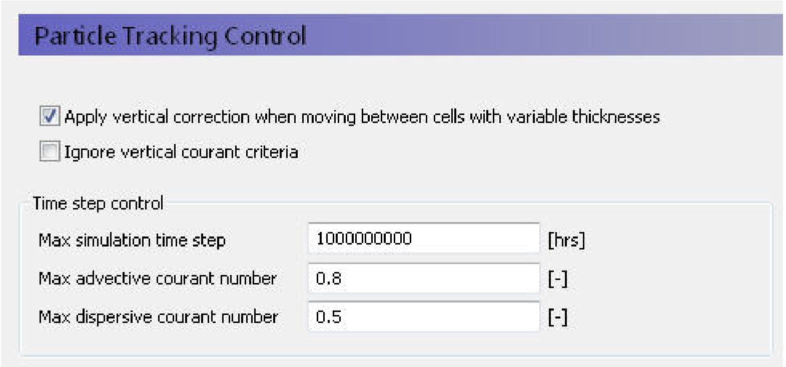
For example, you may have a two-year water movement simulation but you may want to simulate water quality for 10 years. To do this, you would specify the start and stop dates of the part of the water movement simulation that you want repeated. If you want to repeat the whole water movement simulation, then you would specify the beginning and end of the water movement simulation.

Constant water movement flow field - In this case, the nearest saved time step to this date will be used as a steady-state flow field for the transient water quality simulation.

Related Items:

- Working with Solute Transport - User Guide *(V2 p. 297)*
- Particle Tracking-Reference *(V2 p. 319)*

11.4.3 Particle Tracking Control



Particle Tracking Control	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue and the Random Walk Particle Tracking sub-option is selected.

The particle tracking simulation is completely decoupled from the water movement simulation and like the water movement itself, the water quality time steps can be different in each of the overland flow, unsaturated flow and saturated flow.

Apply vertical correction when moving between cells with variable thicknesses - This correction adjusts the vertical location of the particle, if the particle moves into a neighbouring cell with a different cell thickness. This prevents particles from being discharged to the ground surface when they move into an adjacent cell with a lower topography. By default, this option is on.

Ignore vertical courant criteria - With this option on, the calculation of the maximum time step length based on the courant criteria ignores the vertical velocities. In some cases, very small time steps may be required if the layers are thin and the vertical velocities are high. However, such conditions often occur near particle sinks (e.g. wells and seepag faces) where detailed particle pathlines are not important.

Time step control

Maximum Simulation Time Step - This is the maximum user-specified time step allowed. The default value is very high so that the simulation runs by default with the highest possible time step. You might want to set this value to a short time interval, if you want the WQ time step to be uniform during the WQ simulation.

The courant number is a measure of the ratio of flow rate to grid size. For numerical stability, it is important that a dissolved solute does not travel too far in one time step. A courant number greater than 1.0 implies that a particle (solute) would move completely across or through a cell in a time step. The time step is reduced until all the time step criteria below are met.

Max. Advective Courant Number - The advective courant number represents the ratio of cell size to the time it would take a particle to move across a cell. This criteria is likely to be controlling if your flow velocities are high, or your dispersivity values are very low or zero. The default value is 0.8. If your actual time step is being controlled by this criteria, then you could increase it to make the simulation run faster. However, you will need to check to make sure the simulation has converged properly and that the mass balance is reasonable.

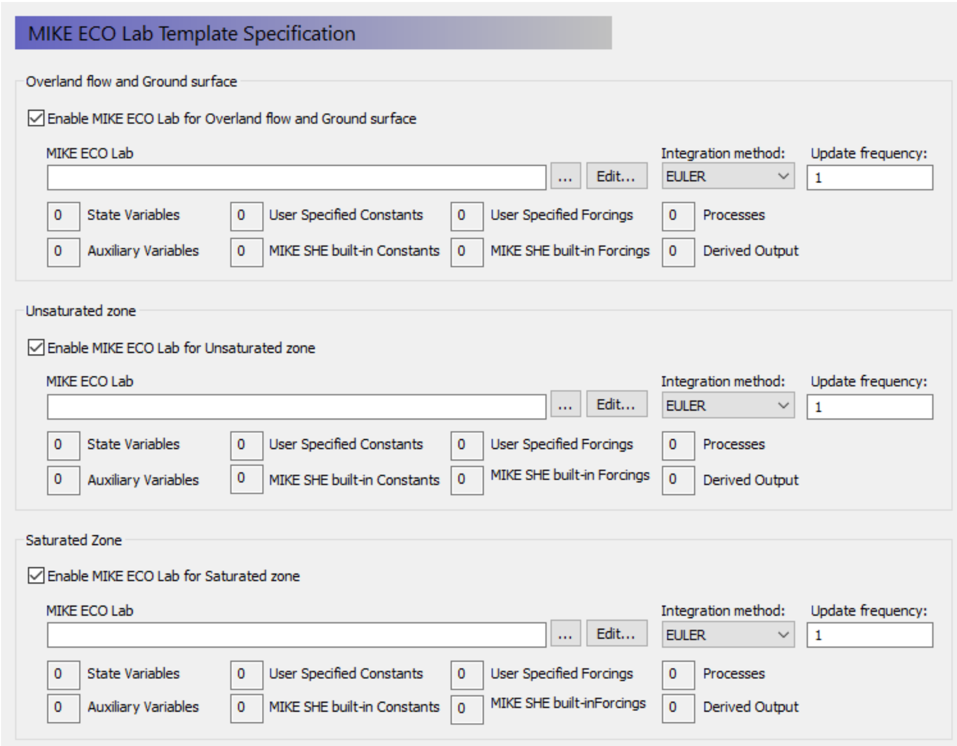
Max. Dispersive Courant Number - The dispersive courant number represents the ratio of the cell size to time it would take a particle to move across a cell due to dispersion. This criteria is likely to be controlling when the velocities are very slow and the dispersivity is non-zero. The default value is 0.5. If your actual time step is being controlled by this criteria, then you could increase it to make the simulation run faster. However, you will need to check to make sure the simulation has converged properly and that the mass balance is reasonable.

Related Items:

- Working with Solute Transport - User Guide *(V2 p. 297)*
- Particle Tracking-Reference *(V2 p. 319)*

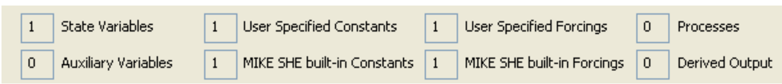
11.5 MIKE ECO Lab Template Specification

.



In the MIKE ECO Lab Template Specification dialog, 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 dialog. When you enable MIKE ECO Lab for the specific process, you will be able to browse to the required MIKE 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.



Interpolation Method

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).

Update Frequency

For each template you can specify an update frequency. The update frequency tells MIKE SHE how frequently to call MIKE ECO Lab. If the water quality processes are slow relative to the simulation time step, you can save considerable simulation time by calling MIKE ECO Lab less frequently. For example, to call MIKE ECO Lab every second or third simulation time step, you would specify an Update frequency of 2 or 3.

Important Note: Units

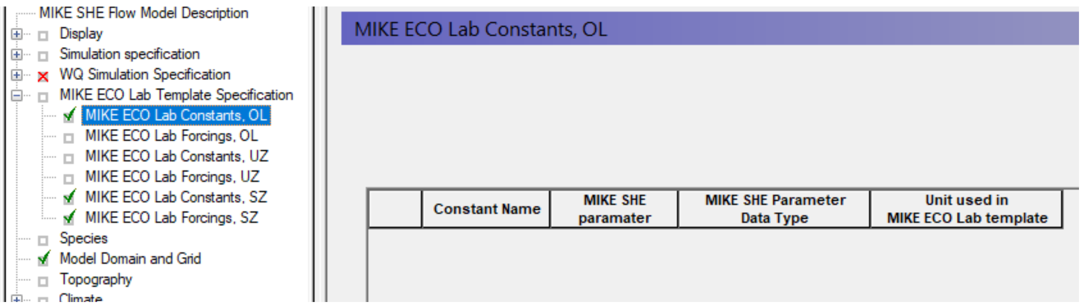
All concentrations passed from MIKE SHE to MIKE ECO Lab are in units of [g/m^3], which is equivalent to [mg/L].

Thus all parameters and equations defined in the MIKE 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].

Related Items:

- Working with MIKE ECO Lab in MIKE SHE - User Guide *(V2 p. 307)*
- MIKE ECO Lab Overview Tables *(V1 p. 208)*
- User Specified MIKE ECO Lab Constants and Forcings *(V1 p. 209)*
- Advection Dispersion - Reference *(V2 p. 255)*

11.5.1 MIKE ECO Lab Overview Tables



In the MIKE ECO Lab Template Specification data tree branch, there is a sub-branch for each of the active MIKE ECO Lab domains - Overland Flow, the Unsaturated Zone, and the Saturated Zone.

The tables are derived from the list of specified Constants and Forcings in the templates.

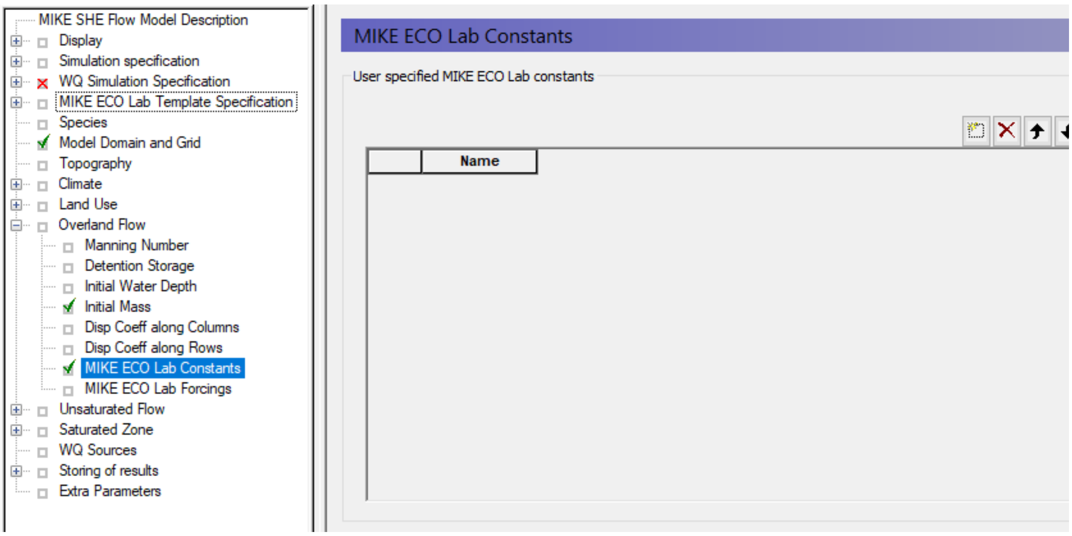
If the Forcing or Constant is not user defined (User Defined = “NO”), then the Forcing or Constant is an internal value in MIKE SHE and will be passed automatically to MIKE ECO Lab. The list of available parameters is quite short - primarily geometry related (e.g. cell volume), plus a few domain specific constants (e.g. porosity). You must specify the MIKE SHE parameter to use and the EUM unit used in the template.

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.

Related Items:

- Working with MIKE ECO Lab in MIKE SHE - User Guide *(V2 p. 307)*
- MIKE ECO Lab Template Specification *(V1 p. 207)*
- User Specified MIKE ECO Lab Constants and Forcings *(V1 p. 209)*
- Advection Dispersion - Reference *(V2 p. 255)*

11.6 User Specified MIKE ECO Lab Constants and Forcings



If the Forcing or Constant is user defined (User Defined = “YES”), then the Forcing or Constant must be specified explicitly in MIKE SHE.

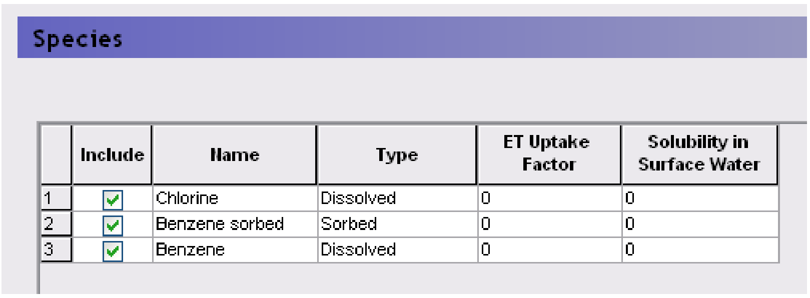
In this case, a Forcing or Constant item is added to the data tree under the appropriate branch in the data tree. In this branch, you will find a table of user specified Forcings and Constants, plus individual sections for each item. Each item can be spacially defined similarly to other constants or 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.

Related Items:

- Working with MIKE ECO Lab in MIKE SHE - User Guide *(V2 p. 307)*
- MIKE ECO Lab Template Specification *(V1 p. 207)*
- MIKE ECO Lab Overview Tables *(V1 p. 208)*
- Advection Dispersion - Reference *(V2 p. 255)*

11.7 Species



Species

	Include	Name	Type	ET Uptake Factor	Solubility in Surface Water
1	☑	Chlorine	Dissolved	0	0
2	☑	Benzene sorbed	Sorbed	0	0
3	☑	Benzene	Dissolved	0	0

Species	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue ET Uptake Factor appears when ET water quality is included. Solubility in Surface Water appears when overland flow water quality is included.

In this dialogue, you add species by clicking on the Insert icon. You delete species by selecting the species from the table and clicking on the Delete icon.

The table includes the list of species for the WQ simulation and the physical properties of the chemical species.

- **Include** - Turning off the include checkbox allows you to exclude a species from the simulation without having to remove it and all of its accompanying sources, etc.
- **Name** - This is the displayed identifier in all subsequent dialogues and in the data tree.

  MIKE ECO Lab note: The Name must be exactly the same as the State Variable Name used in the MIKE ECO Lab Template (except in the case of dual domain mass transfer, which uses the reserved suffix “_2”).
- **Type** - There are four species types. 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 MIKE ECO Lab. This allows MIKE ECO Lab to read and write arbitrary values to MIKE SHE during the simulation. Fixed species have an undefined unit.
- **ET uptake transpiration factor** - This is the factor that determines the rate at which plants will remove the mobile solute from the water.
- **Solubility in Surface Water** - Surface water (overland flow) sources are specified as a mass. Thus, the source will be active until all of the solute has been dissolved. This is important because the life of the source depends on the amount of surface water flow.

  Since evaporation can cause the overland concentration to increase, solubility needs to be specified to avoid unrealistic high concentrations. The species precipitates if the concentration exceeds the solubility. The precipitate dissolves again if the concentration falls below the solubility.

  The solubility is a uniform concentration value per species with the dimension [M L-3], that is, [mu-g/m^3], [g/m^3], [mg/l], etc, depending on the chosen user unit for Concentration. This means that, for example, a solubility of 100 [g/m^3] implies that 100g of mass will dissolve per m^2 of cell area per m depth of ponded water.

Related Items:

- Working with Solute Transport - User Guide *(V2 p. 297)*

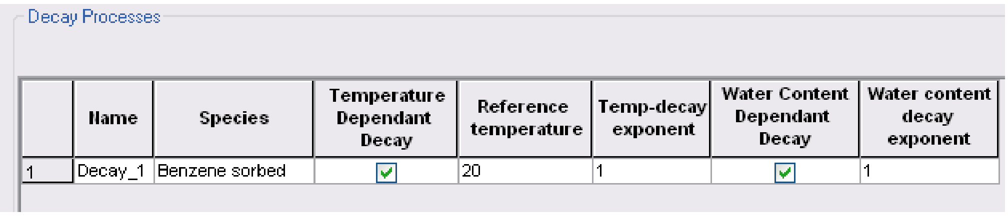
- Working with MIKE ECO Lab in MIKE SHE - User Guide *(V2 p. 307)*
- Advection Dispersion - Reference *(V2 p. 255)*

11.8 Water Quality Sorption and Decay

WQ Sorption and Decay	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue and the Include Water Quality Processes is turned on in the Water Quality Simulation Specification dialogue.

The Sorption and Decay dialogue appears when you have turned on the Water Quality Processes option.

Decay Processes


Decay Processes

	Name	Species	Temperature Dependant Decay	Reference temperature	Temp-decay exponent	Water Content Dependant Decay	Water content decay exponent
1	Decay_1	Benzene sorbed	☑	20	1	☑	1

Temperature dependent decay means that the rate of decay increases when the temperature is above a reference temperature and decreases when below. For overland flow, the ponded water is assumed to have the same temperature as the air. For the unsaturated and saturated zones the soil temperature is dynamically calculated based on the air temperature.

Reference Temperature - This is the reference temperture for the decay function in Equation (17.15).

Temp Decay Exponent - This is the decay exponent, α, in Equation (17.15).

Water content decay means that the actual rate of decay changes depending on the actual water content in the unsaturated zone relative to the saturated water content.

Water Content Decay Exponent - This is the decay exponent, β, in Equation (17.14).

Sorption Processes

Sorption Processes

	Name	Sorption Type	Species	Sorbed Species	Equilibrium Isotherm	Kinetic Type	Equilibrium Sorption Fraction	Sorption Bias Factor for Dual Porosity
1	Sorp_1	Equilibrium	Benzene	Benzene sorbed	Linear	No hysterisis	0.7	0

Sorption type - The sorption type can be Equilibrium or Equilibrium-Kinetic. In the first case, the sorption is assumed to be instantaneous. In the second case, the sorption is rate dependent. However, the solute is divided into an equilibrium fraction and a kinetic fraction, which defined by the value specified in the Equilibrium Sorption Fraction column. If the kinetic option is selected, then a kinetic rate constant will be added in the relevant Sorption process data tree items in the Unsaturated and Saturated sub-trees.

Species - This is the name of the species that will be sorbed. The combo box includes only of the **dissolved** species listed in the main **Species** *(V1 p. 210)* dialog.

Sorbed Species - This is the name of the sorbed version of the dissolved species. This combo box includes only the **sorbed** species from the **Species** *(V1 p. 210)* dialog.

Equilibrium Isotherm - The equilibrium isotherm can be either a Linear isotherm, or one of two non-linear isotherms: Freundlich or Langmuir. The selection of the particular sorption isotherm will create different data tree sub-items for the isotherm parameters, as described below.

The **Linear sorption isotherm** is mathematically the simplest isotherm and can be described as a linear relationship between the amount of solute sorbed onto the soil material and the aqueous concentration of the solute, where K_d is known as the distribution coefficient.

The **Freundlich sorption isotherm** is a more general equilibrium isotherm, which can describe a non-linear relationship between the amount of solute sorbed onto the soil material and the aqueous concentration of the solute,where K_f and N are constants.

Both the linear and the Freundlich isotherm suffer from the same fundamental problem. That is, there is no upper limit to the amount of solute that can be sorbed. In natural systems, there is a finite number of sorption sites on the soil material and, consequently, an upper limit on the amount of solute that can be sorbed.

The **Langmuir sorption isotherm** takes this into account. When all sorption sites are filled, sorption ceases. For the Langmuir sorption isotherm α is a sorption constant related to the binding energy and β is the maximum amount of solute that can be absorbed by the soil material.

For more details on the sorption isotherms, see Equilibrium Sorption Isotherms *(V2 p. 282)*.

Kinetic Type - The three equilibrium sorption isotherms described above can be extended to include kinetically controlled sorption. In the MIKE SHE AD module, a two-domain approach is used, where the sorption is assumed to be instantaneous for a fraction of the sorbed solute and rate-controlled for the remainder. (see below). If Equilibrium-Kinetic is selected in this combo box, then an additional data tree sub-item will be added in the UZ and SZ WQ sections for the kinetic rate constant specified in Equation (17.9).

Equilibrium Sorption Fraction - This is the fraction of solute that is sorbed instantaneously using the equlibrium isotherm. The remainder is sorbed more slowly based on the kinetic rate constant.

Sorption Bias Factor for Dual Porosity - Sorption depends on the porosity and the bulk density of the soil. In dual porosity systems this is rather complicated. The distribution of sorption between the matrix and the fractures should be calculated based on the bulk density and different porosities. However, this is not always practically possible, so MIKE SHE includes a sorption bias factor that allows you to explicitly control the sorption distribution between the fractures and the matrix. The sorption bias factor function is shown in Figure 17.3.

Sorption bias factor = - 1 Sorption is only in the matrix region.

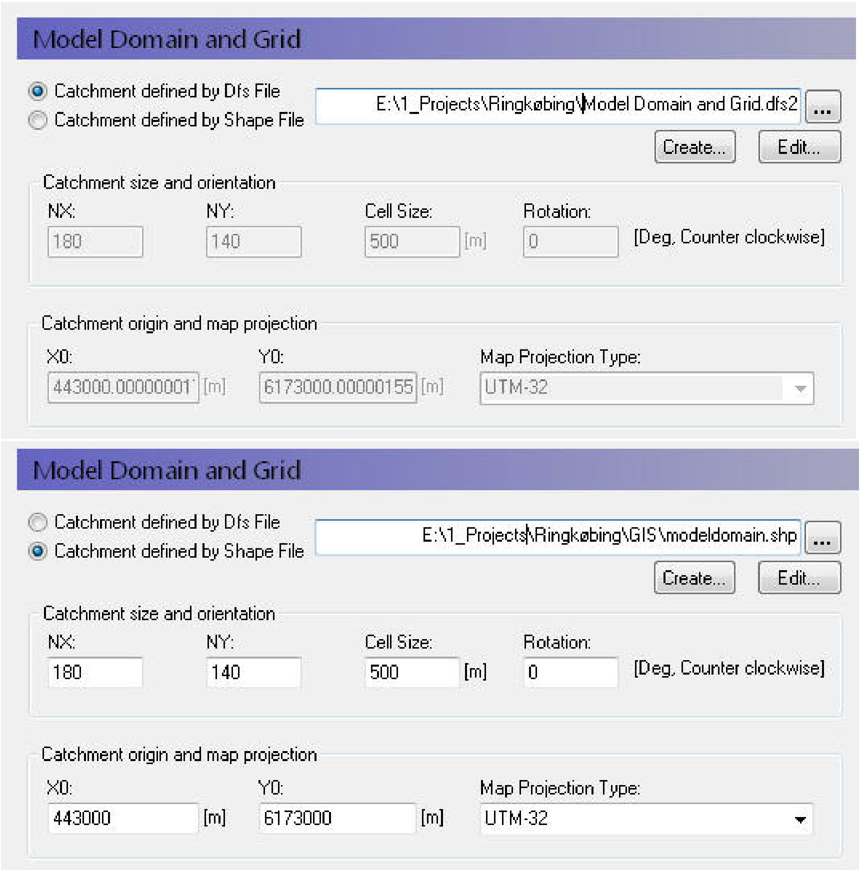
Sorption bias factor = 0 The distribution of sorption sites between macro pores and matrix is assumed to be proportional to the distribution of porosities.

Sorption bias factor = 1 Sorption is only in the macro pores.

Related Items:

- Working with Solute Transport - User Guide *(V2 p. 297)*
- Advection Dispersion - Reference *(V2 p. 255)*
- Reactive Transport - Reference *(V2 p. 281)*

11.9 Model Domain and Grid



Model Domain and Grid	
dialogue Type:	Special; Integer Grid Codes
EUM Data Units	Integer Grid Code
Valid Values	-1e-35, 1, 2

One of the first steps 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.

Any non-gridded data, such as point .shp files and xyz ASCII files are interpolated to the grid defined in this dialogue.

The grid defined in this dialogue is the primary grid. Other .dfs2 gridded data does not have to use the same grid. However, if another .dfs2 file uses a different grid then Real data is interpolated. If the two grids are coincident, that is the cells are the same size and the grids line up, then the data is bilinearly interpolated to the Model Grid. If the grids are not coincident, then the data is treated as if it were point values (i.e. the same as XYZ or .shp data). Integer Grid Code must use coincident grids, as it is impossible to interpolated integer values.

The model domain and grid can be rotated any angle measured clockwise from the x-axis.

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 - usually -1e-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.

The catchment definition is displayed in the greyed out text boxes but is not editable, since the catchment definition is part of the .dfs2 file format. If you want to change the cell size, origin, number of cells etc., you must change the .dfs2 file itself. For more information on editing and setting up the Model Domain and Grid see Conceptual geologic model for the finite difference approach *(V1 p. 55)*.

Using a shp file

It is much easier to define your Model Domain and Grid via an ArcView .shp file (i.e a grid independent polygon). In this case, the definition of integer code values is taken care of automatically. Further, the definition of the grid (number of rows and columns, cell size and origin) can be easily adjusted.

Map projections

A projection is the way the earth’s curved surface is mapped onto a flat plane. The projection causes distortions as you move away from the center of the projection. One of the most common projection systems is the UTM system, which divides the earth into zones to minimize the distortion within a particular zone. Every place on earth has a UTM zone.

The earth is not a sphere and the actual curvature varies locally across the globe. Thus, the other part of the projection is the asssumed shape of the earth, which is defined as the Ellipsoid type. The ellipsoid used for the UTM system is WGS 84.

Thus, unless the projection and the ellipsoid are consistent between two maps, they will not overlap correctly. This is typically the reason if your model results do not appear to be located in the right place when you try to map them to Google Earth.

Map projections in MIKE SHE

Up until the 2009 Release, the only map projection allowed in MIKE SHE was NON-UTM. This is essentially the same as local grid coordinates. From the 2009 Release onwards, MIKE SHE uses the same map projection utilities as the rest of the MIKE Zero products, which is based on a installed library of standard world map projections.

You must define the map projection when you create a dfs2 file and this projection becomes part of the properties of the file. It can be changed in the Properties dialogue in the Grid Editor. Likewise, the map projection is a property of all shape files and can be changed in your GIS program.

Note: In the Model Domain and Grid, you must define the map projection. All of the rest of the dfs2 and shape files used in the model setup must use the same projection.

11.10 Subcatchments

Subcatchments	
Conditions:	when either of the subcatchment-based methods for Overland Flow or Saturated Flow are selected in the Simulation Specification dialogue
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The Subcatchment item appears whenever you select one of the sub-catchment based methods - Simplified Overland Flow Routing or the Linear Reservoir Method for groundwater flow.

The subcatchment items are used to identify the hydrologic subwatersheds in your model domain. For the Simplified Overland Flow Routing, the calculated overland flow in the Overland Flow Zones flows from one zone to the next within the Subcatchment. For the Linear Reservoir Method for groundwater flow, the calculated interflow is routed from one zone to the next within the Subcatchment.

For each unique integer code in the main Subcatchment map view, an additional data item is added to the data tree. In each of these sub-items, there is only one additional variable - a checkbox for using the default river links.

Use default river links - in most cases you will link the simplified overland flow and the groundwater interflow to all of the river links found in the lowest topographic zone, or the lowest interflow zone in the subcatchment. However, in some cases you may want to link the flow to particular river links. For example, if your river network does not extend into the subcatchment, you can specify that the interflow discharges to a particular node or set of nodes in a nearby river network.

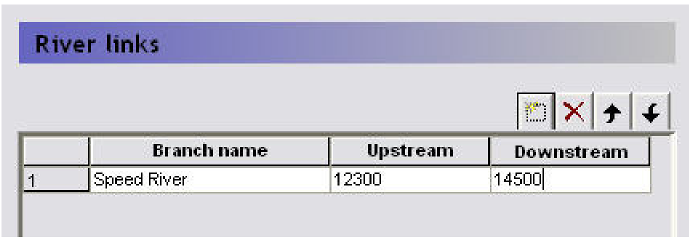
If you uncheck this checkbox, a sub-item will appear where you can specify the river branch and chainage to link the subcatchment to.

The river links for the baseflow zones are specified separately in the baseflow zone dialogues.

Related Items

- Simplified Overland Flow Routing *(V2 p. 68)*
- Overland Flow Zones *(V1 p. 271)*
- Linear Reservoir Method *(V2 p. 205)*
- Interflow Reservoirs *(V1 p. 293)*

11.10.1 River Links



If you have unselected the Use default river links option in the Subcatchments dialogue or in the Baseflow Reservoirs dialogue, then this dialogue will be added to the data tree. In this dialogue, you can specify the branch name to connect a subcatchment to, as well as the upstream and downstream chainage of the branch.

This dialogue is not intelligent, in the sense that it does not read the river network. You must type the branch name exactly as it appears in the river network file and specify valid chainages. If either the name or the chainages are invalid, then you will get an error during the model pre-processing stage.

Related Items

- Coupling of MIKE SHE and MIKE Hydro River *(V2 p. 115)*

11.11 Topography

Topography	
dialogue Type:	Stationary Real Data
EUM Data Units	Elevation

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 the defines the drainage surface for overland flow.

Many of the elevation parameters can be defined relative to the topography, such as

- Lower Level *(V1 p. 336)*,
- Upper Level *(V1 p. 335)*,
- Initial Potential Head *(V1 p. 308)*, and
- Drain Level *(V1 p. 318)*.

Depth parameters, such as ET Surface Depth *(V1 p. 287)*, are also measured from the topography.

Topography is typically defined from a DEM, defined from either a point theme shape file, or an ASCII file. If you have an ArcGIS Grid DEM, this can be converted to a dfs2 file using the MIKE Zero Toolbox. Surfer Grid files can be saved as an ASCII xyz files and then interpolated in MIKE SHE.

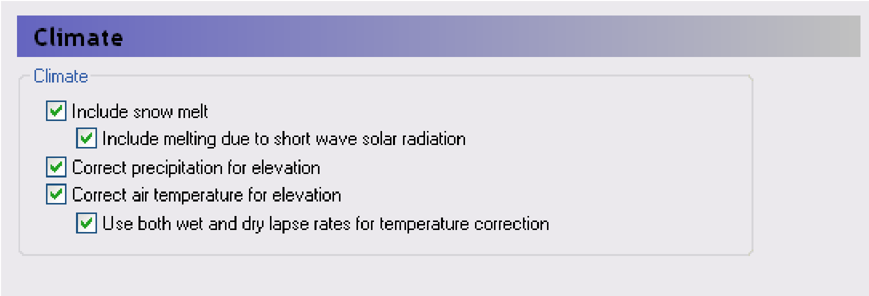
Non-dfs2 files or dfs2 files that have a different grid definition than the model grid are all interpolated to the grid defined in the Model Domain and Grid.

The bilinear interpolation method is useful for interpolating previously gridded DEM data. Whereas, the triangularisation method is useful for contour data digitized from a DEM.

Related Items

- Interpolation Methods *(V1 p. 157)*

11.12 Climate



Precipitation is always included in the data tree.

Include snow melt

If snowmelt is included, then Evapotranspiration must also be specified in the Simulation Specification *(V1 p. 174)* dialog. When snow melt is included, then a new Snow Melt *(V1 p. 230)* section is added to the data tree where the various snow melt parameters can be specified.

Include melting due to short wave radiation - When the Snow melt option is selected, then the check box for radiation melting will become editable. This option will add a new section to the climate data for incoming short wave radiation. Also added will be an item in the Snow Melt *(V1 p. 230)* section for the Melting Coefficient for Solar Radiation *(V1 p. 232)*, which defines the rate at which the snow melts per unit of incoming radiation.

Elevation corrections

Correct precipitation for elevation - In mountainous areas, precipitation can vary significantly with elevation - especially on the windward side of the mountains. However, there are rarely enough rain gauging stations to measure this spatial variability of rainfall. Choosing to correct the precipitation for elevation allows you to define a spatially variable correction factor, if you are using station-based rainfall data. See Precipitation Lapse Rate *(V1 p. 223)* for the correction equations.

Correct temperature for elevation - In mountainous areas, temperature varies significantly with elevation. Choosing to correct the temperature for elevation allows you to define a spatially variable correction factor, if you are using station-based temperature data. See Temperature Lapse Rate *(V1 p. 228)* for the correction equations.

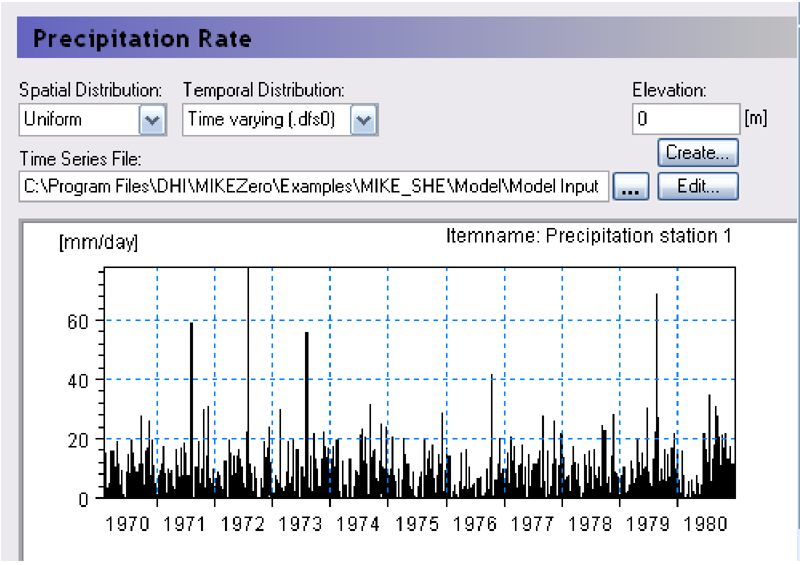
Use both wet and dry lapse rates for temperature correction - The moisture content of the air is a significant factor for the elevation correction of temperture. If this option is chosen, then you can specify a different correction factor for the time steps when it is raining.

Related Items

- Time Series Data *(V1 p. 141)*

- Air Temperature *(V1 p. 227)*
- Precipitation Rate *(V1 p. 221)*
- Reference Evapotranspiration *(V1 p. 225)*
- Short Wave Solar Radiation *(V1 p. 226)*
- Snow Melt *(V1 p. 230)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.1 Precipitation Rate



Precipitation Rate	
dialogue Type:	Time-varying Real Data
EUM Data Units	Grid Code
Time Series EUM Data Units	Precipitation Rate [e.g. mm/hr], or Rainfall [e.g. mm]

The precipitation rate is the measured rainfall, or snowfall in rainfall equivalent units.

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

If you use a rate, then the EUM Data Units must be Precipitation and the time series must be Mean Step Accumulated *(V1 p. 146)*.

If you use an amount, then the EUM Data Units must be Rainfall and the time series must be Step Accumulated *(V1 p. 145)*.

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.

If Include snow melt is included and the Air Temperature *(V1 p. 227)* is below the Threshold Melting Temperature *(V1 p. 231)*, The threshold melting temperature is the temperature at which the snow starts to melt - usually 0 C. If the air temperature is above this threshold, then the snow will melt at the rate specified in the Degree-day Melting or Freezing Coefficient (V1 p. 232) item. If the air temperature falls belowthen the precipitation will accumulate as snow.

Elevation - This is the station elevation. It is used as the reference elevation if the cell-by-cell precipitation is to be corrected for elevation differences (See Elevation corrections *(V1 p. 220)*). The actual precipitation in each cell will then be calculated based on the elevation of the cell relative to the station elevation and the Precipitation Lapse Rate *(V1 p. 223)*. Elevation correction is not available for fully-distributed precipitation data.

Related Items

- Precipitation Lapse Rate *(V1 p. 223)*
- Creating Time Series in MIKE SHE *(V1 p. 141)*
- Working with Spatial Time Series *(V1 p. 143)*
- Time Series Types *(V1 p. 143)*

11.12.2 Precipitation Lapse Rate

Precipitation Lapse Rate	
Conditions	if Correct precipitation for elevation checkbox is on in the Climate *(V1 p. 220)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Correction of precipitation (e.g. % per 100m)

Precipitation is typically measured at only a few locations within a watershed, but in mountainous areas the actual precipitation in an area can be strongly influenced by elevation - especially on the windward side of the mountains. The Precipitation Lapse Rate is used to estimate local preciptation based on the relevative elevation difference to the weather station. The precipitation in the cell, P_{cell}, is calculated as

$$P_{cell} = P_{station} \cdot (1 + \beta_{cell} \cdot (H_{cell} - H_{station})) \tag{11.2}$$

where $P_{station}$ is the measured precipitation at the weather station, $H_{station}$ is the elevation of the station, H_{cell} is the elevation of the cell, and β_{cell} is the Precipitation Lapse Rate for the cell in units of, for example, [% change per 100m of elevation difference].

Related Items

- Precipitation Rate *(V1 p. 221)*

11.12.3 Net Rainfall Fraction

Net Rainfall Fraction	
Conditions	If Evapotranspiration is NOT selected
dialogue Type	Stationary Real Data
EUM Data Units	Fraction

The Net Rainfall Fraction is the fraction of rainfall that is available for infiltration and overland flow. It is used to account for leaf interception and evapotranspiration when ET is not explicitly simulated.

The net recharge to the groundwater table, R_{net}, is

$$R_{net} = Prec \cdot Rainfall_{net} \cdot Infil_{frac} \tag{11.3}$$

where *Prec* is the actual precipitation, $Rainfall_{net}$ is the Net Rainfall Fraction, and $Infil_{frac}$ is the Infiltration fraction.

Related Items

- Precipitation Rate *(V1 p. 221)*
- Infiltration Fraction *(V1 p. 224)*

11.12.4 Infiltration Fraction

Infiltration Fraction	
Conditions	If Overland flow is simulated but unsaturated flow is NOT simulated.
dialogue Type	Stationary Real Data
EUM Data Units	Fraction

The Infiltration Fraction is the fraction of ponded water that infiltrates. It is used when the unsaturated zone is not explicitly simulated.

Normally the unsaturated zone simulation calculates the amount of infiltration from overland flow, since the amount of infiltration depends on the water content of the upper most soil horizon. If the soil is saturated, then the infiltration will be low. If the soil is very dry, then the infiltration could be very high.

However, the Net Infiltration Fraction is a stationary variable. The only way to simulate the dynamic changes in the amount of infiltration is to simulate the unsaturated zone.

Note When MIKE Hydro River is used in MIKE SHE, overland flow must always be included. If you want to simulate strictly saturated flow coupled to MIKE Hydro River, then you will need to use the Infiltration Fraction instead of the unsaturated flow.

The net recharge to the groundwater table, R_{net}, is

$$R_{net} = Prec \cdot Rainfall_{net} \cdot Infil_{frac} \tag{11.4}$$

where *Prec* is the actual precipitation, $Rainfall_{net}$ is the Net Rainfall Fraction, and $Infil_{frac}$ is the Infiltration fraction.

Related Items

- Precipitation Rate *(V1 p. 221)*
- Net Rainfall Fraction *(V1 p. 223)*

11.12.5 Reference Evapotranspiration

Reference Evapotranspiration	
dialogue Type	Time-varying Real Data
EUM Data Units	Grid Code
Time Series EUM Data Units	Evapotranspiration Rate

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.

Related Items

- Creating Time Series in MIKE SHE *(V1 p. 141)*
- Working with Spatial Time Series *(V1 p. 143)*
- Time Series Types *(V1 p. 143)*
- Vegetation Properties Editor *(V1 p. 373)*
- Evapotranspiration - Technical Reference *(V2 p. 17)*
- Working with Evapotranspiration - User Guide *(V2 p. 37)*

11.12.6 Short Wave Solar Radiation

Short Wave Solar Radiation	
Conditions	if melting due to short wave radiation is checked on in Climate *(V1 p. 220)* dialog
dialogue Type	Time-varying Real Data
EUM Data Units	Grid Code
Time Series EUM Data Units	Radiation intensity; Instantaneous

This is the intensity of the incoming shortwave solar radiation. This value is typically measured at a weather station. It is used to calculate the amount of additional snowmelt due to adsorption of the radiation.

Solar radiation varies with latitude and time of day, as well as the degree of cloud cover. It also depends on the pitch of the slope and its angle towards the south. Thus, the peak radiation will be experienced at midday on moderately sloped, south facing slopes.

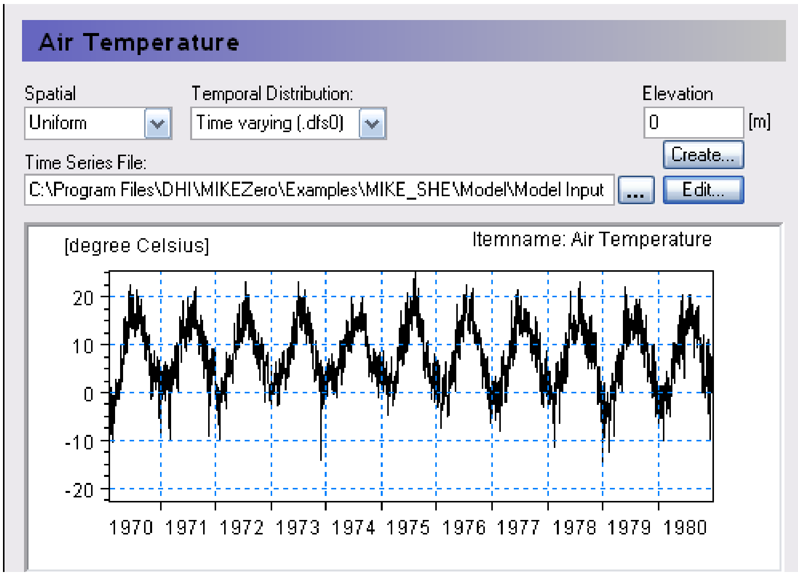
Daily average values will typically underestimate the amount of snow melt. So model time steps and input data that account for the diurnal fluctations in radiation will give better results.

Spatially constant values should be reasonable for large areas, but you may need to correct radiation data in mountainous catchments when the station-based data has been measured in the valley.

Related Items

- Creating Time Series in MIKE SHE *(V1 p. 141)*
- Working with Spatial Time Series *(V1 p. 143)*
- Time Series Types *(V1 p. 143)*
- Climate *(V1 p. 220)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.7 Air Temperature



Air Temperature	
Conditions	if Snow melt selected, or if Temperature Dependent Decay is selected
dialogue Type	Time-varying Real Data
EUM Data Units	Grid Code
Time Series EUM Data Units	Temperature; Instantaneous

Elevation - This is the station elevation. It is used as the reference elevation if the cell-by-cell temperatures are to be corrected for elevation differences (See Elevation corrections *(V1 p. 220)*). The elevation correction is specified by the Temperature Lapse Rate *(V1 p. 228)*, or optionally by the Wet Lapse Rate *(V1 p. 229)* if it is raining. Elevation correction is not available for fully distributed temperature data.

Snowmelt - This is the temperature in Celsius that is used to calculate snow melt and accumulation:

- If the Air temperature is below the Threshold Melting Temperature *(V1 p. 231)* value, then the precipitation accumulates as snow; above it is treated as rain.

- If the Air temperature is above the Threshold Melting Temperature *(V1 p. 231)* value, then dry snow is converted to wet snow using the Degree-day Melting or Freezing Coefficient *(V1 p. 232)*.
- If the Air temperature is below the Threshold Melting Temperature *(V1 p. 231)* value, then wet snow is converted to dry snow using the Degree-day Melting or Freezing Coefficient *(V1 p. 232)*.

Temperature Dependent Decay - This is the air temperature used for calculating the soil temperature for temperature dependent decay in the water quality module.

Related Items

- Creating Time Series in MIKE SHE *(V1 p. 141)*
- Working with Spatial Time Series *(V1 p. 143)*
- Time Series Types *(V1 p. 143)*
- Snow Melt *(V1 p. 230)*
- Threshold Melting Temperature *(V1 p. 231)*
- Degree-day Melting or Freezing Coefficient *(V1 p. 232)*
- Decay *(V2 p. 286)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.8 Temperature Lapse Rate

Temperature Lapse Rate	
Conditions	if Correct temperature for elevation checkbox is on in the Climate *(V1 p. 220)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Temperature Lapse Rate (e.g. C per 100m)

Temperature is typically measured at only a few locations within a watershed, but in mountainous areas the actual temperature at a location is strongly influenced by elevation. The Temperature Lapse Rate is used to estimate local temperature based on the relevative elevation difference to the weather station. The temperature in the cell, T_{cell}, is calculated as

$$T_{cell} = T_{station} + \beta_{cell} \cdot (H_{cell} - H_{station}) \tag{11.5}$$

where $T_{station}$ is the measured temperature at the weather station, $H_{station}$ is the elevation of the station, H_{cell} is the elevation of the cell, and β_{cell} is the Temperature Lapse Rate for the cell in units of, for example, [degrees C per 100m of elevation difference]. β_{cell} is a negative value since temperature decreases with increasing elevation.

However, the moisture content of the air is a significant factor for the elevation correction of temperture. Thus, you have the option in the Climate *(V1 p. 220)* dialog to specify a different correction factor for the time steps when it is raining.

Related Items

- Climate *(V1 p. 220)*
- Air Temperature *(V1 p. 227)*
- Wet Lapse Rate *(V1 p. 229)*

11.12.9 Wet Lapse Rate

Wet Lapse Rate	
Conditions	if Correct temperature for elevation checkbox is on in the Climate *(V1 p. 220)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Temperature Lapse Rate (e.g. C per 100m)

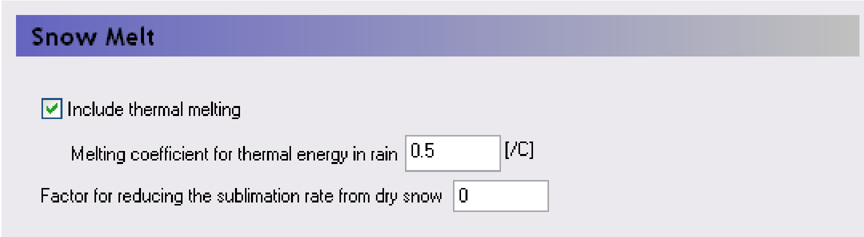
The moisture content of the air is a significant factor for the elevation correction of temperture. Thus, you have the option in the Climate *(V1 p. 220)* dialog to specify a different correction factor for the time steps when it is raining.

If there is precipitation during the time step, then the Wet Lapse Rate will be used in Equation (11.5).

Related Items

- Climate *(V1 p. 220)*
- Air Temperature *(V1 p. 227)*
- Temperature Lapse Rate *(V1 p. 228)*

11.12.10Snow Melt



The Snow Melt data sub-tree includes all of the specific parameters related to the snow melt module.

Snow melts by converting from dry snow to wet snow. Wet snow is converted to overland flow when the Maximum Wet Snow Fraction in Snow Storage *(V1 p. 234)* is exceeded.

Include thermal melting - Thermal melting of snow in response to the heat content of rain can be an important factor during spring thaw events, where rain on saturated snow can trigger a significant flood. The rain is assumed to be at the same temperature as the air.

Melting coefficient for thermal energy in rain - The melting coefficient accounts for the energy content or the rain.

$$M_{rain} = C_{rain} \cdot P \cdot (T_{air} - T_0) \tag{11.6}$$

where M_{rain} is the melting rate, C_{rain} is the melting coefficient due to the energy content of the rain, P is the precipitation rate, T_{air} is the current air temperature and T_0 is the Threshold Melting Temperature *(V1 p. 231)*.

Factor for reducing the sublimation rate from dry snow -

The ET module will remove water from snow storage before any other ET is removed.

1. ET is removed first from wet snow as evaporation because the energy requirements for evaporation are lower than sublimation. The ET is removed from wet snow at the full rate, assuming that wet snow can be treated the same as ponded water.
2. If there is no wet snow (either because it is too cold or all the wet snow as been evaporated) then ET will be removed from dry snow as sublimation.

However, sublimation has a higher energy requirement than evaporation, so MIKE SHE includes a user defined factor for controlling sublimation. The sublimation factor is a multiplier that reduces the actual ET rate from the snow. If the sublimation factor = 0, then the ET rate is 0. If the sublimation factor = 1.0, then the ET rate is the specified Reference ET rate. Thus, *Maximum ET = (Reference ET) * (Crop Coefficient) * (Sublimation Factor)*

Related Items

- Climate *(V1 p. 220)*
- Air Temperature *(V1 p. 227)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.11Threshold Melting Temperature

Threshold Melting Temperature	
Conditions	if Snow melt included in n the Climate *(V1 p. 220)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Temperature

The threshold melting temperature is the temperature at which the snow starts to melt - usually 0 C. If the air temperature is above this threshold, then the snow will melt at the rate specified in the Degree-day Melting or Freezing Coefficient *(V1 p. 232)* item. If the air temperature falls below

Related Items

- Degree-day Melting or Freezing Coefficient *(V1 p. 232)*
- Air Temperature *(V1 p. 227)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.12Degree-day Melting or Freezing Coefficient

Degree-day Melting or Freezing Coefficient	
Conditions	if Snow melt included in the Climate *(V1 p. 220)* dialog
dialogue Type	Time-varying Real Data
EUM Data Units	Grid code
Time Series EUM Data Units	Melting coefficient [e.g. mm snow/day/degree C]

The degree day factor is the amount of snow that melts per day for every degree the Air Temperature *(V1 p. 227)* is above the Threshold Melting Temperature *(V1 p. 231)*.

The degree day factor is a time varying coefficient because the rate of melting varies as the snow pack changes over the winter. The melting coefficient is often used as a calibration parameter to calibrate the volume of snow melt to the observed runoff.

Related Items

- Threshold Melting Temperature *(V1 p. 231)*
- Air Temperature *(V1 p. 227)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.13Melting Coefficient for Solar Radiation

Melting Coefficient for Solar Radiation	
Conditions	if radiation melting is included in the Climate *(V1 p. 220)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Radiation melting coefficient [e.g. mm/(KJ/m$^{2)}$]

The melting coefficient for solar radiation represents that depth of dry snow converted to wet snow per unit of incoming short wave radiation.

In principle, radiation melting coefficient is a time varying parameter that varies as the snow darkens with age. However, the degree-day melting coefficient is the primary calibration parameter for calibrating runoff.

Related Items

- Short Wave Solar Radiation *(V1 p. 226)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.14 Minimum Snow Storage for Full Area Coverage

Minimum Snow Storage for Full Area Coverage	
Conditions	if Snow melt included in the Climate *(V1 p. 220)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Storage depth [e.g. mm]

Snow tends not to be uniformly distributed. It tends to be deeper in sheltered areas, such as along fence lines and in wooded areas.

The Minimum storage depth allows you to assign a minimum snow thickness that covers the entire cell with snow. Snow depths below this value will linearly reduce the snow cover area. The Minimum storage depth reduces the melting rate by an area fraction, thereby preventing small amounts of snow from instantly melting.

The default value of zero implies that no area reduction occurs and that all snow is deposited evenly across the cell.

Related Items

- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.15Maximum Wet Snow Fraction in Snow Storage

Maximum Wet Snow Fraction in Snow Storage	
Conditions	if Snow melt included in the Climate *(V1 p. 220)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Fraction [-]

Melting snow does not instantly lead to runoff. When snow melts, the liquid water content of the snow first increases until a critical water content is reached. When this water content is reached additional melted snow will form runoff. If the air temperature drops below freezing the liquid water stored in the snow will re-freeze. This repeated thawing and freezing leads to snow compaction and changes in the snow structure over the winter. This in turn justifies the time dependancy of the Degree-day Melting or Freezing Coefficient *(V1 p. 232)*.

The snow storage is divided into Dry snow storage and Wet snow storage, where the wet snow is actually absorbed liquid water in the snow structure. The wet snow fraction is the amount of wet snow divided by the total amount of snow storage. When the Maximum wet snow fraction is exceeded, any excess melted snow will be converted to ponded water. The ponded water is then available for UZ infiltration, evapotranspiration, or overland runoff.

Related Items

- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.16Initial Total Snow Storage

Initial Total Snow Storage	
Conditions	if Snow melt included in the Climate *(V1 p. 220)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Storage depth [e.g. mm]

This is the initial total storage depth of snow in the cell. This value is used to also determine the initial snow coverage on the cell. Thus, if this value is less than the Minimum Snow Storage for Full Area Coverage *(V1 p. 233)*, then the area coverage fraction will be less than one.

Related Items

- Minimum Snow Storage for Full Area Coverage *(V1 p. 233)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.12.17Initial Wet Snow Fraction

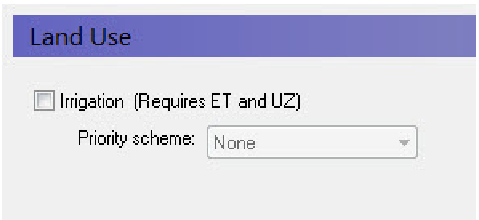
Initial Wet Snow Fraction	
Conditions	if Snow melt included in the Climate *(V1 p. 220)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Fraction [-]

The Initial Wet Snow Fraction determines the initial liquid water content of the snow storage. If this value is greater than the Maximum Wet Snow Fraction in Snow Storage *(V1 p. 234)*, then runoff will be generated in the first time step. If this value is zero, then the initial snow is dry.

Related Items

- Maximum Wet Snow Fraction in Snow Storage *(V1 p. 234)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Working with Freezing and Melting - User Guide *(V2 p. 47)*

11.13 Land Use



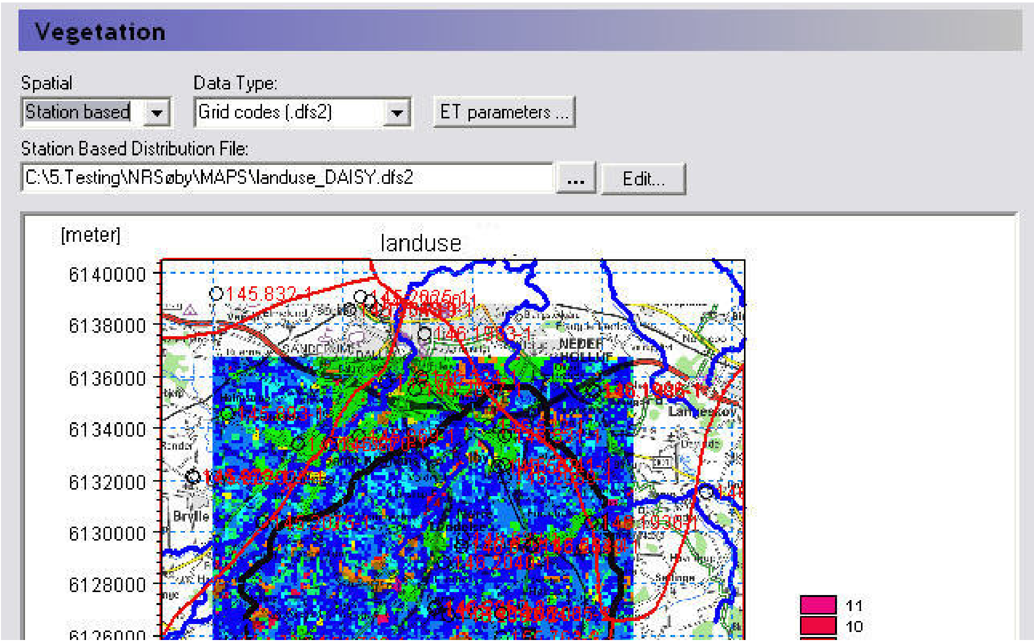
The Land Use item in the data tree is used to define the items that are on the land surface that affect the hydrology in your model area, including

- Vegetation distribution, and
- Irrigation.

Irrigation - The Irrigation option allows you to specify a demand driven irrigation scheme with priorities. Activating the Irrigation option creates several sub-items in the data tree for the irrigation parameters. The Irrigation option requires that both Evapotranspiration and Unsaturated Flow be simulated. For more information see Irrigation Command Areas *(V1 p. 243)*

Priority Scheme - The priority scheme is used by the Irrigation module to rank the model areas in terms of priority for irrigation. Two options are allowed: Equal Volume or Equal Shortage. If the water is to be distributed based on equal volume, then all cells with the same priority number will receive an equal amount of water, regardless of their actual demand. If the water is to be distributed based on equal shortage, then all cells with the same priority number will receive an amount of water that satisfies an equal percentage of their actual demand. For more information see Irrigation Priorities *(V1 p. 252)*

11.13.1 Vegetation



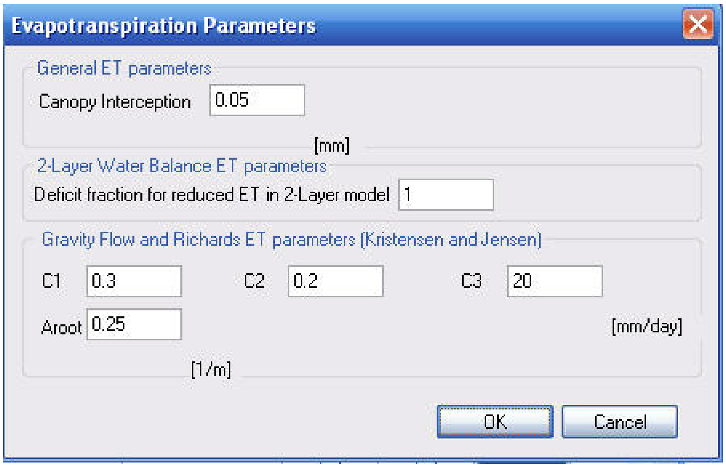
Vegetation	
Conditions:	If Evapotranspiration selected in the Simulation Specification dialogue
EUM Data Units	Grid Code

Vegetation	
Time Series EUM Data Units	Leaf Area Index and Root Depth, or Vegetation Property File (see Vegetation Properties Editor *(V1 p. 373)*)
dialogue Type:	Special version of Time-varying Real Data

The main Vegetation dialogue is used to define the distribution of vegetation across your model area. It works the same as any other dialogue for Grid Codes with associated time series data. In this case, however, there are two relevant time series parameters: the Leaf Area Index and the Root Depth. Both of these parameters can be defined as constants, via .dfs0 files, or they can be defined from a Vegetation Properties file.

Note. The crop coefficient, K_c, is only available in the Vegetation Properties file.

Evapotranspiration Parameters



The Vegetation dialogue also includes a button labelled **ET Parameters...** Clicking on this button pops up the dialogue for specifying the default ET parameters. The Evapotranspiration parameters in this dialogue do not vary in time and are global for the model. However, if the Vegetation Properties file option is used for the Leaf Area Index and Root Depth parameters, these values can be overridden by crop specific values specified in the vegetation properties file.

The following sections are condensed from the Evapotranspiration - Technical Reference *(V2 p. 17)* chapter, which should be consulted for more detailed information.

Canopy Interception

The interception process is modelled as an interception storage, which must be filled before stem flow to the ground surface takes place. The coefficient C_{int} defines the interception storage capacity of the vegetation per unit of LAI. A typical value is about 0.05 mm but a more exact value may be determined through calibration.

Note The interception storage is calculated each time step and the rate of evaporation is usually high enough to remove all the interception storage in each time step. Thus, the total amount of water removed from interception storage depends on the length of the time step. This can lead to confusion when comparing water balances, if the time steps are different. For example, assuming it rains a little in each time step and all the interception storage is removed by ET in every time step, if the time step is halfed, then the total volume of interception storage removed will double.

Deficit Fraction for reduced ET

In nature, plant roots will remove water from the soil until the water content of the soil reaches a critical level. Once this level is reached, transpiration will decrease with decreasing water content until the wilting point is reached, where the amount of transpiration drops to zero.

In the 2-layer model, the maximum deficit is (θmax - θmin), which if the water table is below the extinction depth is field capacity minus the wilting point water contents.

The deficit fraction is the fraction of this maximum deficit before ET begins to be reduced. Thus, transpiration will occur at the maximum rate until this deficit fraction is reached. Then the transpiration will decrease linearly until θmin is reached, at which point transpiration becomes zero.

A typical value for this deficit fraction is about 0.5, which implies that the ET from UZ will be start to be reduced when the deficit is half the maximum allowed. A deficit fraction of 0 implies that no ET will be reduced linearly starting at θmax. A deficit fraction of 1 implies that no ET reduction will be applied and ET will be removed at the maximum rate until the wilting point is reached.

For more information on this variable, see the Section The 2-Layer Water Balance Method *(V2 p. 27)* in the Technical documentation.

C_1, C_2 and C_3

The Kristensen and Jensen equations for actual transpiration and soil evaporation contain three empirical coefficients, C_1, C_2, and C_3. The coefficients C_1 and C_2 are used in the transpiration function, $f_1(LAI)$. C_3 is the only variable found in the soil moisture function.

C1 - C_1 is plant dependent. For agricultural crops and grass, C_1 has been estimated to be about 0.3. C_1 influences the ratio soil evaporation to transpiration. This is illustrated in Figure 2.7. For smaller C_1 values the soil evaporation becomes larger relative to transpiration. For higher C_1 values, the ratio approaches the basic ratio determined by C_2 and the input value of LAI.

C2 - For agricultural crops and grass, grown on clayey-loamy soils, C_2 has been estimated to be about 0.2. Similar to C_1, C_2 influences the distribution between soil evaporation and transpiration, as shown in Figure 2.8. For higher values of C_2, a larger percentage of the actual ET will be soil evaporation. Since soil evaporation only occurs from the upper most node (closest to the ground surface) in the UZ soil profile, water extraction from the top node is weighted higher. This is illustrated in Figure 2.8, where 23 per cent and 61 per cent of the total extraction takes place in the top node for C_2 values of 0 and 0.5 respectively.

Thus, changing C_2 will influence the ratio of soil evaporation to transpiration, which in turn will influence the total actual evapotranspiration possible under dry conditions. Higher values of C_2 will lead to smaller values of total actual evapotranspiration because more water will be extracted from the top node, which subsequently dries out faster. Therefore, the total actual evapotranspiration will become sensitive to the ability of the soil to draw water upwards via capillary action.

C3 - C_3 has not been evaluated experimentally. Typically, a value for C_3 of 20 mm/day is used, which is somewhat higher than the value of 10 mm/day proposed by Kristensen and Jensen (1975). C_3 may depend on soil type and root density. The more water released at low matrix potential and the greater the root density, the higher should the value of C_3 be. Further discussion is given in Kristensen and Jensen (1975).

Root Mass Distribution Parameter, AROOT

In the Kristensen and Jensen model, water extraction by the roots for transpiration varies over the growing season. In nature, the exact root development is a complex process, which depends on the climatic conditions and the moisture conditions in the soil. Thus, MIKE SHE allows for a root distribution determined by the root depth (time varying) and a general, vertical root-density distribution, defined by *AROOT*, see Figure 2.3. In the above dialogue, AROOT is not time varying, but can be specified as a time series using the Vegetation Properties Editor *(V1 p. 373)*.

How the water extraction is distributed with depth depends on the *AROOT* parameter. Figure 2.4 shows the distribution of transpiration for different values of *AROOT*, assuming that the transpiration is at the reference rate with no interception loss ($C_{int}=0$) and no soil evaporation loss ($C_2=0$). *The figure shows* that the root distribution, and the subsequent transpiration, becomes more uniformly distributed as *AROOT* approaches *0*. During simulations, the total actual transpiration tends to become smaller for higher values of *AROOT* because most of the water is drawn from the upper layer, which subsequently dries out faster. The actual transpiration, therefore, becomes more dependent on the ability of the soil to conduct water upwards (capillary rise) to the layers with high root density.

Figure 2.5 shows the effect of the root depth, given the same value of *AROOT*. A shallower root depth will lead to more transpiration from the upper unsaturated zone layers because a larger proportion of the roots will be located in the upper part of the profile. However, again, this may lead to smaller actual transpiration, if the ability of the soil to conduct water upwards is limited.

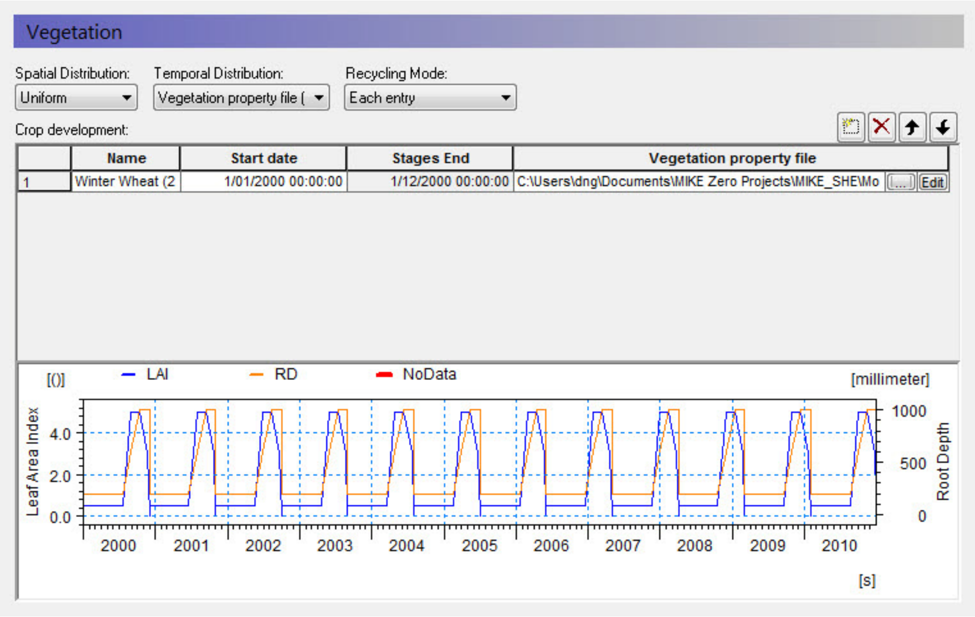
The root distribution is also important for distributing the ET extraction between the SZ and UZ models. If the water table is above the bottom of the roots then ET will be extracted from SZ. The relative amount of ET removed from SZ will depend on the fraction of roots below the water table.

Thus, *AROOT* is an important parameter for estimating how much water can be drawn from the soil profile under dry conditions.

Related Items

- Kristensen and Jensen method *(V2 p. 19)*
- The 2-Layer Water Balance Method *(V2 p. 27)*
- Vegetation Properties *(V1 p. 241)*
- Vegetation Properties Editor *(V1 p. 373)*

11.13.2 Vegetation Properties



Vegetation	
Conditions:	If Evapotranspiration selected in the Simulation Specification dialogue
EUM Data Units	Grid Code
Time Series EUM Data Units	Leaf Area Index and Root Depth, or Vegetation Property File (see Vegetation Properties Editor *(V1 p. 373)*)
dialogue Type:	Special version of Time-varying Real Data

There are two relevant time series parameters: the Leaf Area Index and the Root Depth. Both of these parameters can be defined as constants, via .dfs0 files, or they can be defined from a Vegetation Properties file.

Note. The crop coefficient, K_c, is only available in the Vegetation Properties file.

Using a Vegetation Properties file

The Vegetation Properties file typically contains a time series of the root depth and leaf area index for either one year or for the growing season. If you are using a properties file, then you have to specify the crop development schedule.

Name - This is the name of the crop or vegetation type in the properties file. This name must match exactly the name in the properties file. When you select the file name using the browse button, **[...]**, you can select the vegetation item from a list of available vegetation types found in the file.

Start Date - The vegetation properties file typically contains information on a year or growing season basis - starting from Day 0. Thus, the Start Date is the calendar date for the beginning of the growing year or growing season. If data is missing in the time series, then the Leaf Area Index and the Root Depth are both assumed to equal zero. If the start dates overlap with the growing season information in the vegetation database, a warning will be issued in the log file that says the crop development was not over yet before the new crop was started. MIKE SHE will then start a new crop cycle at the new start date.

Tip: The times in the .etv file are instantaneous, which implies that the values between the entries are linearly interpolated. So, it is critical that you start your rotation scheme at Day 0. Otherwise, the subsequent values will all be interpolated from zero to the first rotation entry.

If you are using a Vegetation Property file, the Recycling Mode allows you to define a seasonal rotation to cover your simulation period - without having to specify a new line for each season. There are four recycling modes:

- **None** - if no recycling mode is defined, then you have to either specify enough lines to cover your simulation period, or that your development period in the .etv file is long enough to cover your simulation period.
- **Each entry** - each line will be repeated until the next start date. The last entry will be repeated until the end of the simulation
- **Entire scheme** - when the end of the scheme is reached, the entire scheme will start over. The scheme will repeat until the end of the simulation period. This mode requires that there are no gaps in the scheme.
- **Entries, then scheme** - each entry will be repeated until the start of the next entry. At the end of the scheme, the entire scheme will start over.

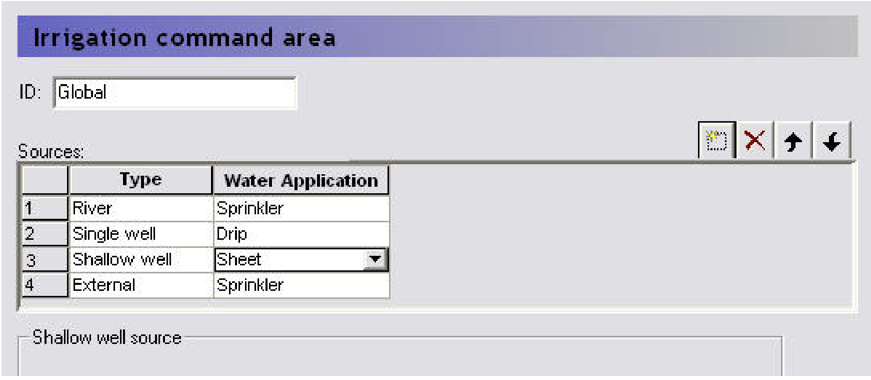
The Vegetation dialogue also includes a button for the Evapotranspiration Parameters, which are global stationary parameters.

Related Items

- Kristensen and Jensen method *(V2 p. 19)*
- The 2-Layer Water Balance Method *(V2 p. 27)*

- Vegetation Properties *(V1 p. 241)*
- Vegetation Properties Editor *(V1 p. 373)*

11.13.3 Irrigation Command Areas



Irrigation Command Areas	
Conditions:	Irrigation selected in Land Use and UZ and ET simu-lated
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The Irrigation Command Areas are used to describe where the water comes from and how the irrigation water is applied to the model.

The Irrigation Command Area data item is divided into two dialogues. The first is the distribution dialogue for the Command Areas and the second is the Water source and Application method for each of the command areas.

Each source can also be limited by a licensed maximum amount of water in any period - License Limited Irrigation *(V1 p. 248)*.

Using shp files

If you specify a shp file for the distribution of the command areas, then an extra dialogue will appear where you can specify a .dbf file to import all of the command area information. If you want to use this option, please email your local support center to obtain an example .shp and .dbf file combination.

Calculation sequence and shortage handling for SZ Linear Reservoirs

For each rank (no. of ranks = max no. of sources specified for any command area) and each Priority (usually only 1) do the following:

1. Calculate the total demand of remote- and shallow well sources of the actual Rank and Priority from all baseflow reservoirs (1 & 2).
2. Calculate a “supply factor” for each baseflow reservoir 1 & 2: If the demand is less than the storage of the actual reservoir, the supply factor is 1.0. Otherwise calculate a value between 0 and 1 = available storage / demand.
3. Calculate the final irrigation from each source of the actual Rank and Priority = calculated demand from actual baseflow reservoir 1 and/or 2 X corresponding supply factor.
4. Subtract the final irrigation volumes from the available volume of each baseflow reservoir 1 & 2 and go to next rank and priority.

In the next SZ Linear Reservoir time step, the calculated irrigation volumes are subtracted from the baseflow reservoir storages (and depths), and the irrigation pumping is stored together with the other SZLR results for water balance calculation, etc.

The available volume of water in each baseflow reservoir is calculated as

$$V_1 = (D_r - D_w) \cdot S_y \cdot A_T \cdot F_1 \tag{11.7}$$

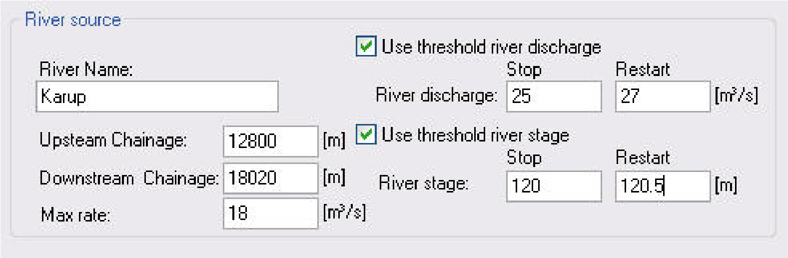
$$V_2 = (D_r - D_w) \cdot S_y \cdot A_T \cdot (1 - F_1) \tag{11.8}$$

where V is the available volume of water, D_r is the depth of the reservoir, D_w is the depth to the water surface in the reservoir, S_y is the specific yield of the reservoir, A_T is the total surface area of the reservoir and F_1 is the fraction of infiltration that is added to the first Baseflow reservoir.

Water Source Types

The Sources table specifies the different sources available for irrigation for a single command area. The order of the sources in the table is also their priority. For example, in the figure above, as long as there is sufficient water in the river, the irrigation water will be removed from the river. If the River falls below a specified level and/or discharge, then the irrigation water will be taken from the single well, and so on.

River Sources



To use a river as a source of water, you must specify the MIKE Hydro River location to be used followed by the permitted river conditions that allow water to be removed. The River source actually has two conditions that can be used alone or combined.

River Name - The Branch name of the river source. This name must exist in the MIKE Hydro River model and it must be spelled correctly.

Upstream/Downstream Chainage - The upstream and downstream chainage locations to use for the river source. MIKE SHE will use the combined volume of all the included river links as the storage volume. The abstraction from each river link is volume weighted based on the total volume of the contained and partial river links.

Max rate - This is the maximum extraction rate for the river. If more water is required for irrigation, then the next source will be activated.

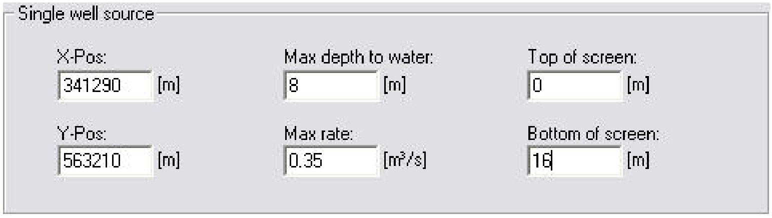
Use threshold river discharge - If the flow rate in the river falls below the **Stop** value, then water will no longer be taken from the River. However, if the flow rate in the river increases again and reaches the **Restart** value, the river source will be reactivated. The discharge threshold is applied at the upstream chainage location to ensure that the inflow to the river source area meets the minimum flow rate.

Use threshold river stage - If the water level in the river falls below the **Stop** value, then water will no longer be extracted from the River. However, if the water level in the river increases again and reaches the **Restart** value, the river source will be reactivated. The river stage threshold is applied at the downstream chainage to ensure that the minimum water level in the river source area is maintained.

If both threshold values are specified, then the most critical one is used, and the source will not restart until both are satisfied.

Note: There is no restriction on the number of river sources at a location. However, if the sources are located in the same model grid then a warning message will be printed to the *projectname*_preprocesssor_messages.log file. The sources will be merged, retaining the maximum threshold stages and the sum of the capacities. The preprocessor also checks the license application volume to make sure these are the same. If not, the preprocessor will stop with an error.

Single Well Sources



To use a well source in the model, you must specify the location and filter depth of the well. In a future release, this dialogue will be connected to the well database, but at the moment it is not.

X, Y -Pos - This is the X and Y map coordinates of the source well.

Max depth to water - this is the threshold value for the water depth in the well. If the water level in the well falls below this depth (as measured from the topography), the extraction will stop until the water level rises above the threshold.

Max rate - This is the maximum extraction rate for the well. If more water is required for irrigation, then the next source will be activated.

Top/Bottom of Screen - The depth of the top and bottom of the screen is used to define from which numerical layers water can be extracted. Pumping will stop if the water table falls below the bottom of the layer that contains the filter bottom.

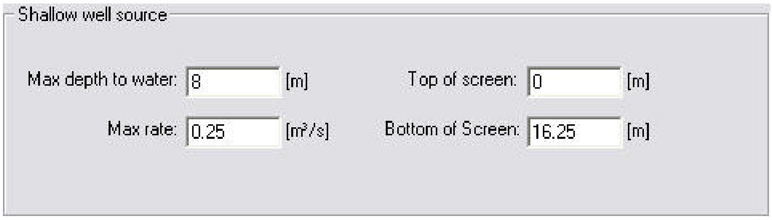
There is no restriction on the number of wells at a location. However, if the wells are located in the same model grid, and have overlapping screen intervals, then a warning message will be printed to the *projectname*_preprocessor_messages.log file. The sources will be merged, retaining the maximum threshold depth, the sum of the capacities and the joint screening interval. The preprocessor also checks the license application volume to make sure these are the same. If not, the preprocessor will stop with an error.

In the linear reservoir groundwater method, multiple single wells are allowed in each baseflow reservoir. No warnings are given.

When the linear reservoir method is used, the screen interval is ignored and the water is pumped from the two baseflow reservoirs. The distribution between the two reservoirs is determined by the faction give in the Baseflow Reservoirs *(V1 p. 294)* dialogue. If the demand from one of the reservoirs exceeds the available water, the pumping will be reduced. The pumping rate at the other reservoir will not be increased to compensate.

Also in the Linear Reservoir method, the specified “max depth to water” for the actual command area and source is used. In other words, it is not using the “threshold depth for pumping” in the baseflow reservoir menu. Pumping is allowed when the depth to the water table is less than the specified threshold value at the start of the time step.

Shallow Well Sources



In many cases, farmers have several shallow wells for irrigation, most of which may not be mapped exactly. Especially in regional scale models, each grid cell could thus contain many shallow groundwater wells. In such cases, the Shallow Well source can be used to simply extract water for irrigation from the same cell where it is used, without having to know the exact coordinates of the wells. By specifying this option, one well is placed in each cell of the command area.

Note: A cell (i, j, layer) containing a shallow well cannot also have a single well specified in the same cell (i.e. the same cell and layer).

Max depth to water - this is the threshold value for the water depth in the well. If the water level in the well falls below this depth (as measured from the topography), the pumping will stop until the water rises above the threshold depth again.

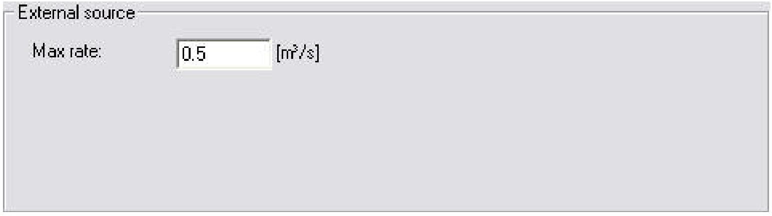
Max rate - This is the maximum extraction rate for the shallow well **in each cell**. If more water is required for irrigation, then the next source will be activated.

Top/Bottom of Screen - The depth of the top and bottom of the screen is used to define from which numerical layers water can be extracted. Pumping will stop if the water table falls below the bottom of the layer that contains the filter bottom.

Shallow well sources are removed from baseflow Reservoir 1 if the Linear Reservoir groundwater method is used. The screen interval is ignored.

Note: Shallow wells can be located in cells containing single sources. The preprocessor will give a warning for such violations. Multiple shallow wells are not allowed in the same command area.

External Sources



In some case, the irrigation water can be from outside of the watershed being modelled. In this case, the only constraint is the maximum amount of water than can be extracted from the source.

Max rate - This is the maximum extraction rate for the source. If more water is required for irrigation, then the next source will be activated.

Water Application methods

There are three ways to apply the irrigation water in the model.

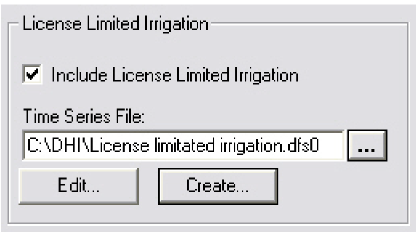
Sprinkler - If the water is applied as sprinkler irrigation, it is added to the precipitation component.

Drip - If the water is applied as Drip irrigation, it is added directly to the ground surface as ponded water.

Sheet - If the water is applied as Sheet irrigation, then an additional data tree item is required to define where the water is to be added within the command area. The idea behind this option is that water is flooded onto one or more cells of the command area and then distributed to the adjoining cells as overland flow. The sheet irrigation is applied directly to the cells as ponded water.

All three methods are allowed in the Simple, sub-catchment based overland flow method. However, the sheet method does not really make sense if the subcatchment overland flow method is used.

License Limited Irrigation



Sometimes, the total amount of irrigation water that a user can apply is limited by a license over a certain period (e.g. 10000 m^3 / year). The license limited option, allows you to specify a dfs0 time series file with a time series of maximum amounts. If the maximum amount is reached within the license period, then the irrigation will be stopped until the next license period, when it will be started again. The license period length is defined by the time steps in the specified dfs0 file.

During the simulation the license data is included in the calculation of the available water volume of each source. The module keeps track of the “actual available license volume”. Whenever this is reached or exceeded, the source will be closed until a new license period starts (or the end of the simulation). When a source is closed for this reason, a message is printed in the wm_print.log file.

Notes

- The dfs0 file EUM Data Units *(V1 p. 131)* must be Water volume (m^3 or other volume unit) and the time series-type must be Step Accumulated *(V1 p. 145)*.
- The specified volumes cover the period from the previous value (or start of simulation) until the date of the actual value.
- The files may contain delete values. These are simply ignored. This makes it possible to include licenses for several sources in one file, even when the dates of the different source licenses differ.
- An irrigation log file is included in the results output: *projectname*_IrrigationLicenseLog.dfs0.
- This log file contains the “actual available license volume” of each source with license included, stored as instantaneous values at the end of every time step. This makes it easy to identify the periods where sources have been closed due to “license shortage”.

Note: Unused license volumes are NOT carried over to the next license period (use it or loose it !).

11.13.4 Sheet Application Area

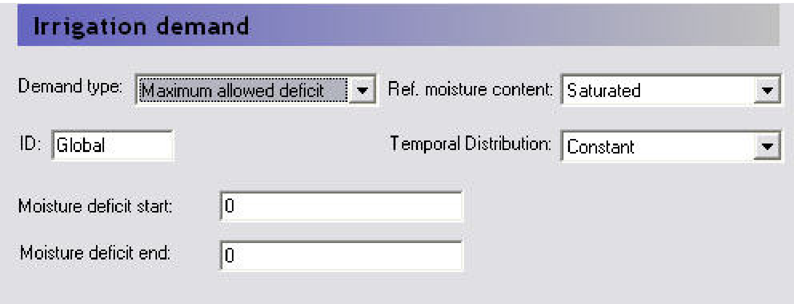
Sheet Application Area	
Conditions:	Irrigation selected in Land Use and UZ and ET simulated and Sheet selected as an application method
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

The sheet application area is used to define where the irrigation will be applied. The program does not make any distinction between sheet application areas. The sheet irrigation will be distributed on every cell with a non-delete value or non-zero integer code within the command area.

Example If the calculated demand in the command area is 100 mm/day and the area of the command area is 5000 m^2, then the total amount of irrigation water required will be 100 mm x 5000 m^2 per day.

The total amount of irrigation water will be divided equally among the cells in the sheet application area. If each cell is 100 m^2 and the irrigation is applied to 10 cells, then each of the 10 cells will receive 500 mm/day of irrigation water as ponded water.

11.13.5 Irrigation Demand



Irrigation Demand	
Conditions:	Irrigation selected in Land Use and UZ and ET simulated
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The Irrigation Demand is used to describe when the water will be applied in the model.

The Irrigation Demand data item is divided into two dialogues. The first is the distribution dialogue for the Demand Areas and the second contains the information on when the water will be applied in each Demand area.

Demand Type

User Specified - For the User Specified demand type, the demand is not calculated. Rather it is simply specified as a constant value or as a time series.

Crop stress factor - The crop stress factor is the minimum allowed fraction of the crop specific reference ET that the actual ET is allowed to drop to before irrigation starts . That is, the minimum allowed *(Actual ET)/(Reference ET x* K_c*)* relationship. This should be a value between 1 and 0. If the actual Crop Stress factor falls below the given value, irrigation will be added.

Ponding depth - When using this option, the demand will be equal to the difference between the actual ponding depth and specified ponding depth. The option is typically used for modelling irrigation of paddy rice. If the ponding depth falls below the specified value then more irrigation water is added.

Max allowed deficit - The available water for crop transpiration (AW) is the difference between the actual water content and the water content at the wilting point for the root zone. The maximum available water for crop transpiration (MAW) is then the value of AW for the reference moisture content, where either the saturated water content or the field capacity can be specified for the reference moisture content. The deficit can then be defined as the fraction of the MAW that is missing and is a value between 0 and 1, where 0 is the deficit when the actual moisture content is equal to the reference moisture content, and 1 is the deficit when the available water for crop transpiration is zero, which is when the water content drops below the water content at the wilting point. When using the Max Allowed Deficit method, irrigation is started when the deficit exceeds the moisture deficit start value and stops at the moisture deficit end value.

If, for example, the reference moisture content is the field capacity and irrigation should start when 60 % of the maximum available water in the root zone is used and cease when field capacity is reached again, the value for the Start should be 0.6, and the value for the Stop value should be 0

Reference Moisture Content

If the Maximum allowed deficit is used, then the reference moisture content can be either for calculating the deficit is either the saturated water content or the field capacity water content.

Temporal Distribution

In each of the demand types the demand factor can be specified as a constant value or as a time series. However, an additional option in this combobox is to use the Vegetation Properties file. To use this option, you must include a vegetation properties file when you specify the vegetation. Further, you must specify the irrigation properties in the vegetation properties file itself. In this case, the Vegetation properties file, will contain all of the values needed by each of the different methods and the demand values cannot be input in this dialogue.

11.13.6 Irrigation Priorities

Irrigation Priorities	
Conditions:	Irrigation selected in Land Use and UZ and ET simulated and Priorities option selected in Land Use dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

If there is insufficient water available to satisfy all of the irrigation demand, then the irrigation areas can be prioritised. In this case, each area of the model can be assigned a priority number (1 = highest priority). All the areas with the highest priority will be irrigated first. If there is sufficient water after the first areas have been irrigated, then the areas with the next highest priority will be irrigated.

However, if there is insufficient water to completely satisfy the demand of a particular priority region (all of the cells with priority value = 1, for example), then the water will be distributed to each of the cells based on either Equal Volume or Equal Shortage. The choice of the two priority schemes is assigned in the main **Land Use** *(V1 p. 235)* dialogue.

Equal Volume - If the water is to be distributed based on equal volume, then all cells with the same priority number will receive an equal amount of water, regardless of their actual demand. For example, if there is a demand for 100 m^3 of water in 10 cells, but only 50 m^3 is available, then each cell will receive 5 m^3 of water, regardless of the actual demand.

Equal Shortage - If the water is to be distributed based on equal shortage, then all cells with the same priority number will receive an amount of water that satisfies an equal percentage of their actual demand. For example, if there is a demand for 100 m^3 of water in 10 cells, but only 50 m^3 is available, and 5 of the cells have a demand of 15 m^3, while 5 have a demand of only 5 m^3, then the first cells will receive 7.5 m^3, while the latter will receive only 2.5 m^3.

11.14 Rivers and Lakes

In this dialogue, you can link MIKE SHE to a MIKE 11 or MIKE Hydro River simulation. The text box will accept a MIKE 11 setup (.sim11) file or the newer MIKE Hydro River setup (.hydro) file.

In principle, there are three basic steps for developing an integrated MIKE SHE/MIKE Hydro River model:

1. Establish a MIKE Hydro River hydraulic model as a stand-alone model, make a performance test and, if possible, a rough calibration using prescribed inflow and stage 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 Hydro River by defining branches (reaches) where MIKE Hydro should interact with MIKE SHE.

The chapter Coupling of MIKE SHE and MIKE Hydro River *(V2 p. 115)* describes in more detail the three steps above.

Flooding and Inundation

Flow on the flood plain can be simulated by overbank spilling. However, this can be numerically intensive.

In many cases, you do not need to calculate the actual flow but only the water levels on the ground surface, for example in wetland areas on the flood plain. For such cases, you can use Flood Codes to define the inundated areas. See Flooding from MIKE Hydro River to MIKE SHE using Flood Codes *(V2 p. 129)*.

Flood codes - With this option, you can specify the Flood Code map that shows which cells flood during a storm event, or are otherwise covered with water (e.g. permanent wetlands).

Bathymetry - The bathymetry is used to more accurately simulate the topography of the flood code cells, when the MIKE SHE topography is specified on a larger grid.

Water Quality with MIKE 11

Finally, if you have selected Water Quality calculation for Rivers and Lakes in the **Water Quality Simulation Specification** *(V1 p. 194)* dialog, then an extra .sim11 specification will be displayed. In this text box, you must specify the .sim11 file that contains the water quality model for MIKE 11. This may or may not be the same as the one specified for the water movement calculation.

The extra .sim11 specification is required because the water quality model is run independently of the water movement model. In other words, the water movement model is run first, which calculates the exchange between the groundwater and surface flow, with the MIKE 11 river model. Then the water quality model uses the water movement results to calculate the solute fluxes between the different water components. Thus, the water quality .sim11 file could be different than the water movement .sim11 file if you are running several solute transport scenarios. However, the .sim11 files must be consistent. This consistancy will be checked at run time.

The MIKE 11 water movement results cannot be recycled in the same way that the rest of the MIKE SHE results can be. Your MIKE 11 simulation must cover the entire period of you water quality simulation.

Related Items:

- **Water Quality Simulation Specification** *(V1 p. 194)*
- Channel Flow - Technical Reference *(V2 p. 109)*
- Coupling of MIKE SHE and MIKE Hydro River *(V2 p. 115)*

11.14.1 Flood codes

Flood Codes	
Conditions:	If Rivers and Lakes selected in the Simulation Specification dialogue and Flood Codes selected in the River and Lakes dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

Flood codes are required when cusing the Flooding from MIKE Hydro River to MIKE SHE using Flood Codes *(V2 p. 129)*. This requires a .dfs2 file with Integer Grid Codes which are then used for making the flood mapping for the coupling reaches.

Related Items:

- Coupling of MIKE SHE and MIKE Hydro River *(V2 p. 115)*
- Flooding from MIKE Hydro River to MIKE SHE using Flood Codes *(V2 p. 129)*

11.14.2 Bathymetry

Bathymetry	
Conditions	If Rivers and Lakes selected in the Simulation Specification dialogue and Bathymetry selected in the Rivers and Lakes dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

The bathymetry option allows you to specify a detailed flood plain and river bottom topography, which can then be used for more accurate definition of the topography in the flood code cells.

Note: This file is only used if the Bed Topography Option is set in MIKE Hydro River.

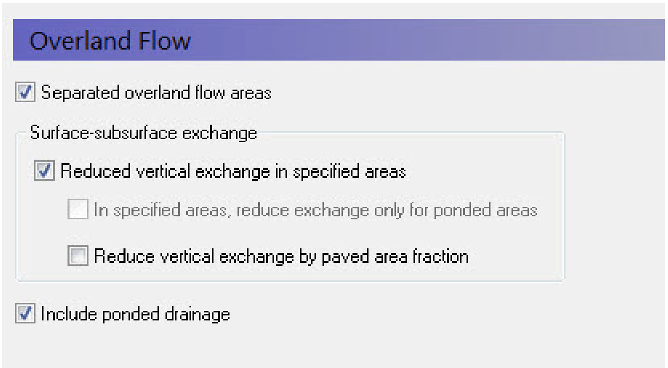
Related Items:

- Coupling of MIKE SHE and MIKE Hydro River *(V2 p. 115)*

- Flooding from MIKE Hydro River to MIKE SHE using Flood Codes *(V2 p. 129)*

11.15 Overland Flow

The main dialogue for overland flow includes several options when the Finite difference method is selected. It does not contain any options when the Sub-catchment based method is selected.



If the Finite Difference method is selected in the Simulation Specification dialogue, the basic items required for the calculation of Overland Flow are

- the Manning number, which is equivalent to the Stickler roughness coefficient,
- the Detention Storage, and
- the Initial Water Depth on the ground surface (ponded water).

There are three main options available in the Overland Flow dialogue.

Separated overland flow areas

The first allows you to divide the model area into overland flow zones, which are conceptually areas separated by dikes or embankments. With this option, overland flow will not be allowed to flow between zones. If this is checked, then an additional item, Separated Flow Areas *(V1 p. 264)*, will be added to the data tree.

Surface-subsurface exchange

If the soil profile becomes completely saturated, then the unsaturated flow component, which normally controls surface-subsurface exchange, is no longer active. In this case, the overland flow module must exchange water with the saturated zone component directly. The surface-subsurface exchange option allows you to specify an exchange coefficient to reduce the exchange of water between the overland flow and the saturated zone. However, this coefficient is also taken into account when calculating vertical infiltration from ponded water to the unsaturated zone.

If the reduced vertical exchange option is chosen then a new item, the Surface-Subsurface Leakage Coefficient *(V1 p. 260)*, must be specified.

In specified areas, reduce exchange only for ponded areas - An additional sub-item is also available that allows you to apply the exchange coefficient in sub-areas where it is only used when water is ponded. During rainfall, this will not have an effect, as the rainfall is always added to ponded storage first. However, after the rainfall, this will allow water to flow downhill along a drainage pathway with a reduced infiltration rate.

Note: This option is only available if you are using Multi-cell Overland Flow *(V2 p. 82)*

Reduced vertical exchange by paved area fraction - Paving is common in urban areas and has a significant impact on infiltration and runoff. This option allows you to additionally specify a paved area fraction that further reduces the infiltration in the cell by the fraction of paving.

When a Paved Area Fraction is specified, it is used as a linear scaling fraction for the Surface-Subsurface Leakage Coefficient. That is, the effective leakage coefficient is reduced by the Paved Area Fraction.

EffLeakCoef = (1-PAreaFrac) x SurfSubSurfLeakCoef

Ponded Drainage

In natural systems, runoff does not travel far as sheet flow. Rather it drains into natural and man-made drainage features in the landscape, such as creeks and ditches. Then, it generally discharges into streams, rivers or other surface water features. In urban areas, it may discharge into storm water retention basins designed to capture runoff.

The Ponded Drainage (OL Drainage) option was introduced in the 2017 Release to support conceptual drainage networks. Conceptually, the OL Drainage is the similar to the SZ Drainage in that a drainage network is calculated based on a downhill flow path from each node until it reaches a stream, a boundary, or a local depression.

Selecting this option creates additional sub-items in the data tree for the drainage options.

Related Items:

- Separated Flow Areas *(V1 p. 264)*
- Surface-Subsurface Leakage Coefficient *(V1 p. 260)*
- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

- Overland and Ponded Drainage *(V2 p. 93)*
- Coupling of MIKE SHE and MIKE Hydro River *(V2 p. 115)*

11.15.1 Manning number

Manning number	
Condition	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	ManningsM

The Manning M is equivalent to the Stickler roughness coefficient, the use of which is described in Overland Flow - Technical Reference *(V2 p. 51)*.

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, be aware that this will also prevent overland flow from entering into the cell.

Note: You can specify a time varying Mannings M using an Extra Parameter option.

Related Items:

- Overland Flow - Technical Reference *(V2 p. 51)*
- Time varying surface roughness (Manning's M) *(V2 p. 335)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

11.15.2 Detention Storage

Detention Storage	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Storage Depth

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.

The detention storage also limits the inflow into the river. Thus, overland flow into the river will only occur if the ponded depth is above the detention storage. If you are using Flood codes, then MIKE Hydro River will only control the water level in the cell if the water level in the river link is above the topography + detention storage.

Water trapped in detention storage continues to be available for infiltration to the unsaturated zone and to evapotranspiration. Using detention storage, you can simulate water that is trapped in depressions that are smaller than a grid cell.

Detention storage can be used effectively to improve the Overland simulation performance. In areas of permanently ponded water, you can set the Detention Storage equal to depth of ponding. This will remove these areas from the Overland Flow calculation and prevent the time step from be reduced to accurately calculate lateral circulation in the ponded areas.

Related Items:

- Overland Flow - Technical Reference *(V2 p. 51)*
- Overland Flow Performance *(V2 p. 78)*
- OL Drainage Options *(V2 p. 338)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

11.15.3 Initial Water Depth

Initial Water Depth	
Condition	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Water Depth

This is the initial condition for the overland flow calculations, that is the initial depth of water on the ground surface. The initial water depth is usually zero.

The initial water depth for overland flow is also the boundary condition for overland flow. In other words, if you specify an initial depth of 2mm, then the boundary will always have 2mm of water and there will be an inflow of water to the model whenever the internal cell has less than 2mm of ponded water.

Note: You can specify a time varying OL boundary condition using an Extra Parameter option.

Related Items:

- Overland Flow - Technical Reference *(V2 p. 51)*
- Time-varying Overland Flow Boundary Conditions *(V2 p. 332)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

11.15.4 Surface-Subsurface Leakage Coefficient

Surface-Subsurface Leakage Coefficient	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue AND the Reduced vertical exchange in specified areas option selected in the main Overland Flow dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Leakage Coeff./Drain Time Const.

The Surface-subsurface leakage coefficient 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. It is also used to simulate soil compaction and paved areas.

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 leakage coefficient is also applied to the vertical leakage from ponded water to the UZ model. In this case, the vertical leakage is a series calculation based on the harmonic mean of the UZ hydraulic conductivity and the leakage coefficient.

The specified leakage coefficient is used wherever it is specified. In areas where a delete value is specified, it is ignored.

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.

Note: A time-varying leakage coefficient can be specified as an Extra Parameter to support reduced infiltration due to frozen soil in the winter.

Related Items:

- Flooding from MIKE Hydro River to MIKE SHE using Flood Codes *(V2 p. 129)*
- Time varying surface infiltration (frozen soils) *(V2 p. 334)*
- Groundwater exchange with MIKE Hydro River *(V2 p. 123)*
- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

11.15.5 Sub-areas for reduced ponded exchange

Sub-areas for reduced ponded exchange	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue AND Surface-subsurface leakage coeffient AND In specified areas, reduce exchange only for ponded areas are both selected in the main Overland Flow dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

An additional sub-item is also available that allows you to apply the exchange coefficient in sub-areas where it is only used when water is ponded. During rainfall, this will not have an effect, as the rainfall is always added to ponded storage first. However, after the rainfall, this will allow water to flow downhill along a drainage pathway with a reduced infiltration rate.

Related Items:

- Flooding from MIKE Hydro River to MIKE SHE using Flood Codes *(V2 p. 129)*
- Groundwater exchange with MIKE Hydro River *(V2 p. 123)*
- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

11.15.6 Paved Area Fraction

Paved Area Fraction	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue and Reduced vertical exchange by paved areas is selected in the main Overland Flow dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Fraction

Paving is common in urban areas and has a significant impact on infiltration and runoff. This option allows you to additionally specify a paved area fraction that further reduces the infiltration in the cell by the fraction of paving.

When a Paved Area Fraction is specified, it is used as a linear scaling fraction for the Surface-Subsurface Leakage Coefficient. That is, the effective leakage coefficient is reduced by the Paved Area Fraction.

EffLeakCoef = (1-PAreaFrac) x SurfSubSurfLeakCoef

Note: A time-varying paved area fraction can be specified as an Extra Parameter to support changing paving conditions over time.

Related Items:

- Overland Flow - Technical Reference *(V2 p. 51)*
- Time varying OL drainage parameters *(V2 p. 341)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

11.15.7 Separated Flow Areas

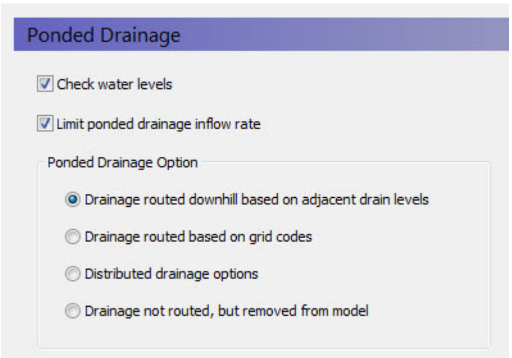
Separated Flow Areas	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue and Separated Flow Areas selected in the main Overland Flow dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

By specifying separated overland flow areas you can simulate areas that are separated by dikes or embankments. Separated overland flow areas are defined by specifying a .dfs2 Integer Grid Code file, containing a unique code value for each flow area. The model will then disable overland flow between grids with different flow codes. Thus, embankments can be simulated by defining different flow codes on each side of the embankment. Legal code values are 1 and higher. Delete values are not allowed inside the model area.

Related Items:

- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

11.15.8 Ponded Drainage



The OL Drainage function requires a reference system for linking the drainage to a recipient node or cell. The recipient can be a MIKE Hydro River node, another OL grid cell, or a model boundary.

The Ponded Drainage module is complex. The section Overland and Ponded Drainage *(V2 p. 93)* includes a lot of helpful information on using the this module

Check water levels - This means that the destination water level must be lower than the ponded water level in the cell.

Limit ponded drainage inflow rate - This creates an new data tree item for the maximum inflow rate to the drain. This supports culverts, etc that restric drain inflow

There are four different options for setting up the drainage source-recipient reference system

- **Routed downhill based on drain levels** - routed to the nearest downhill river, boundary or internal depression
- **Routed based on grid codes** - routed to the nearest river or boundary with the same grid code.
- **Distributed drainage options** - routing to specified River Links or MIKE Urban manholes
- **Drainage not routed, but removed**

Note: A time-varying ponded drainage parameters can be specified using an Extra Parameter option.

Related Items:

- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*
- Overland and Ponded Drainage *(V2 p. 93)*
- Time varying OL drainage parameters *(V2 p. 341)*

11.15.9 Runoff Coefficient

Runoff Coefficient	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Ponded Drainage selected in Overland Flow dialogue
dialogue Type:	Stationary Real Data
EUM Data Units	Fraction

The Runoff Coefficient defines the fraction of **ponded water - not precipitation** - that drains to storm sewers and other surface drainage features in paved areas. However, since rainfall is added to the ponded depth after leaf interception and before evapotranspiration and infiltration is calculated, it effectively acts on net rainfall.

The Ponded Drainage module is complex. The section Overland and Ponded Drainage *(V2 p. 93)* includes a lot of helpful information on using the this module

The Runoff Coefficient acts in two ways:

1. it tells MIKE SHE where it should be applied, and
2. the value specifies how much of the overland flow is allowed to infiltrate and how much should be 'drained away'.

Thus, in non-drainage cells, you should use a value of zero or an Undefined Value (-1e-35). Whereas, in drainage cells, you must specify a value between 0 and 1.

In most cases, the coefficient value defines the fraction of the ponded water that will be discharged to the ponded drainage network in the current time step. However, the discharge may be limited by the inflow time constant, the gradient between the ponded depth and the drain storage, and the specified maximum drain inflow rate.

Note: A time-varying value can be specified as an Extra Parameter to support changing paving conditions over time.

Related Items

- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*
- Overland and Ponded Drainage *(V2 p. 93)*
- Time varying OL drainage parameters *(V2 p. 341)*

11.15.10Maximum Storage Change Rate

Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Ponded Drainage selected in Overland Flow dialogue
dialogue Type:	Stationary Real Data
EUM Data Units	Storage change rate

If the **Specify max rate of change in storage depth** option is checked, then MIKE SHE will restrict the rate at which ponded water is drained to the paved drainage function. This allows you to retain ponded water on a cell and drain it away at a specified rate to the river.

Note: A time-varying value can be specified as an Extra Parameter to support changing paving conditions over time.

Related Items

- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*
- Overland and Ponded Drainage *(V2 p. 93)*
- Time varying OL drainage parameters *(V2 p. 341)*

11.15.11Inflow Time Constant

Inflow Time Constant	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Ponded Drainage selected in Overland Flow dialogue
dialogue Type:	Stationary Real Data
EUM Data Units	Leakage coefficient/Drain time constant

This is the time constant that controls the inflow rate to the Overland drainage storage.

Note: A time-varying value can be specified as an Extra Parameter to support changing paving conditions over time.

Related Items

- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*
- Overland and Ponded Drainage *(V2 p. 93)*
- Time varying OL drainage parameters *(V2 p. 341)*

11.15.12Outflow Time Constant

Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Ponded Drainage selected in Overland Flow dialogue
dialogue Type:	Stationary Real Data
EUM Data Units	Leakage coefficient/Drain time constant

This is the time constant that controls the outflow rate from the Overland Drainage storage to the recipient.

Note: A time-varying value can be specified as an Extra Parameter to support changing paving conditions over time.

Related Items

- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*
- Overland and Ponded Drainage *(V2 p. 93)*
- Time varying OL drainage parameters *(V2 p. 341)*

11.15.13Drain Level

Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Ponded Drainage selected in Overland Flow dialogue, and the Drain Level option selected
dialogue Type:	Stationary Real Data
EUM Data Units	Height above ground

This is the level that controls the source - destination reference system.

Note: A time-varying value can be specified as an Extra Parameter to support changing paving conditions over time. However, this will exclude using any of the drainage distribution methods that rely on the Drain Level.

Related Items

- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*
- Overland and Ponded Drainage *(V2 p. 93)*
- Time varying OL drainage parameters *(V2 p. 341)*

11.15.14Drain Codes

Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Ponded Drainage selected in Overland Flow dialogue, and the Drain code option selected
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

The drainage reference system will be created only within a common drain code.

Related Items

- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

- Overland and Ponded Drainage *(V2 p. 93)*
- Time varying OL drainage parameters *(V2 p. 341)*

11.15.15Drainage Option Distribution

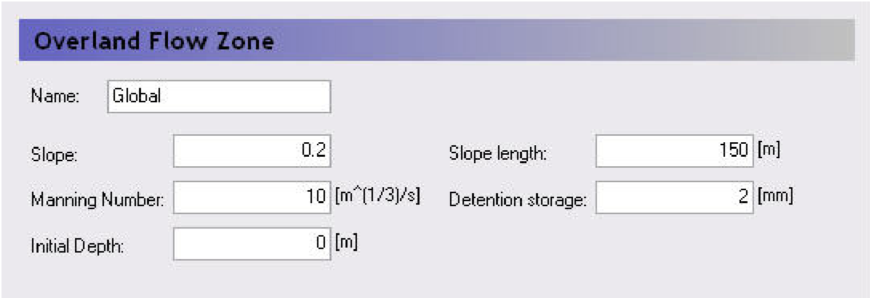
Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Ponded Drainage selected in Overland Flow dialogue, and the Distributed option selected
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

This allows you to defined drainage to specified River Links or MIKE Urban manholes, via an Extra Parameter option.

Related Items

- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*
- Overland and Ponded Drainage *(V2 p. 93)*
- OL Drainage to Specified MIKE Hydro River H-points *(V2 p. 339)*
- Time varying OL drainage parameters *(V2 p. 341)*

11.15.16Overland Flow Zones



Overland Flow Zones	
Conditions:	If Overland Flow + the Subcatchment-based method is chosen in the Simulation Specification dialogue
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The Overland flow zones are used to defined the topographic zones for the simple, catchment-based overland flow solution. The overland flow zones are typically defined as areas with similar topography. For example, the river flood plain would be a typical topographic zone, although it might be included in many subcatchments.

Each unique grid code in the Overland Flow Zones map will generate a sub-item in the data tree where the following parameters must be specified.

Slope - The Slope variable is a representative slope in the topographic area. It can be thought of as the average slope, but it is really a calibration parameter as it can't really be calibrated to a true average.

Slope Length - Like the Slope itself, the Slope Length is a representative distance that water must travel as overland flow before reaching a ditch or stream. It can be thought of as the average travel distance, but, like the Slope, it is really a calibration parameter as it can't really be calibrated to a true average.

Manning Number - The Manning M is equivalent to the Stickler roughness coefficient. 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.

Detention Storage - Detention Storage is used to limit the amount of water that can flow over the ground surface. 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. Water trapped in detention storage continues to be available for infiltration to the unsaturated zone and to evapotranspiration. Using detention storage, you can simulate water that is trapped in depressions that are smaller in area than a grid cell.

Initial Depth - This is the initial condition for the overland flow calculations, that is the initial depth of water on the ground surface.

Related Items:

- Simplified Overland Flow Routing *(V2 p. 68)*
- Overland Flow - Technical Reference *(V2 p. 51)*
- Working with Overland Flow and Ponding- User Guide *(V2 p. 75)*

11.15.17 Initial Mass (per Species)

Initial Mass	
Condition	when water quality for Overland Flow is selected in the Water Quality Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Mass per unit area [e.g. g/m^2]

The initial mass for overland transport is given as a surface concentration [e.g kg/m^2]. This makes it easier to control the mass of solute introduced because you do not have to consider surface water depth.

Related Items:

- Solute Transport in Overland Flow *(V2 p. 275)*
- Initial Conditions *(V2 p. 278)* in Overland Transport

11.15.18Dispersion coefficient along columns/rows

Dispersion coefficient along columns/rows	
Condition	when water quality for Overland Flow is selected in the Water Quality Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Dispersion coefficient

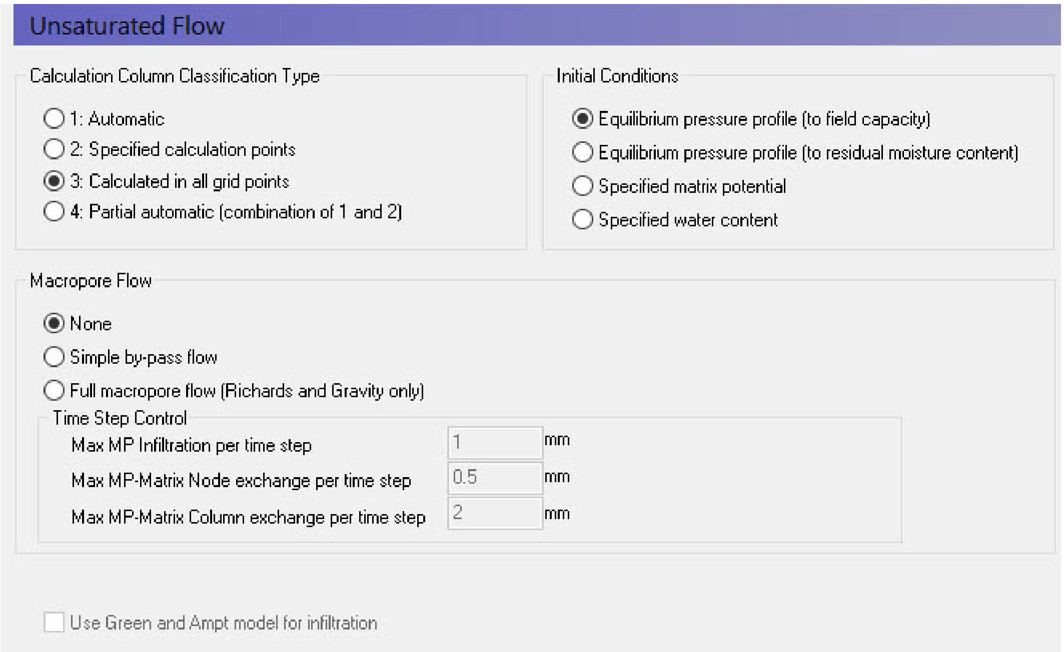
For the overland flow transport, two dispersion coefficients are required - one along the rows and the other along the columns.

Note Unlike for the unsaturated and saturated flow, the overland transport module requires the actual dispersion coefficient [m^2/s] - not the dispersivity [m].

Related Items:

- Solute Transport in Overland Flow *(V2 p. 275)*

11.16 Unsaturated Zone



There are three methods in MIKE SHE to calculate Unsaturated Flow:

- Richards Equation *(V2 p. 142)*,
- the Gravity Flow *(V2 p. 153)*, and
- the Two-Layer Water Balance *(V2 p. 155)*.

The available options in this dialog and the items in the data tree depend on which UZ calculation method chosen in the Simulation Specification *(V1 p. 174)* dialog.

Column Classification Classification Type

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

If the either Richards Equation or the Gravity Flow Module are chosen for calculating the unsaturated zone flow then the top-level Unsaturated Zone dialogue contains a section for the column classification. However, if the Two Layer Method is chosen then the Column Classification section is hidden.

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 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

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.

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.

- **Calculated in all Grid points (default)**

  For smaller scale studies, or studies where the classification system becomes intractable, you can 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.

Macropore Flow

Flow through macropores in unsaturated soil is important for many soil types.

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. The actual bypass fraction is a function of a user-specified maximum fraction and the actual water content of the unsaturated zone, assuming that macropore flow occurs primarily in wet conditions.

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 described in the Reference section under Simplified Macropore Flow (bypass flow) *(V2 p. 156)*.

Full Macropore Flow - Macropores are defined as a secondary, additional continuous pore domain in the unsaturated zone, besides the matrix pore domain representing the microporous bulk soil. Macropore flow is initiated when the capillary head in the micropore domain is higher than a threshold matrix pressure head, corresponding to the minimum pore size that is considered as belonging to the macropore domain. Water flow in the macropores is assumed to be laminar and not influenced by capillarity, thus corresponding to gravitational flow.

Full Macropore flow is described in the Reference section under Full Macropore Flow *(V2 p. 157)*.

Max macropore infiltration per time step - This is a stability criteria that prevents too much water from entering the macropores in one time step.

Max macropore matrix exchange per time step - This is a stability criteria that prevents too much water from exchanging between the macropores and the bulk matrix in one node in one time step.

Max macropore-matrix column exchange per time step - This is a stability criteria that prevents too much water from exchanging between the macropores and the bulk matrix in the entire column in one time step.

Initial Conditions

The initial conditions section defines the initial moisture content for the soil profile.

- **Equilibrium pressure profile (to field capacity)** This is the default option. In this case, the initial soil moisture distribution follows the water content-pressure relationship from the soil database with a minimum water content equal to the calculated field capacity. In other words, the profile will be saturated at the water table and decrease above the water table until field capacity is reached. All cells above this will have a water content equal to field capacity. This is a reasonable initial condition for most temperate climates, where the water content is regularly recharged by rainfall. Generally, this will lead to an initial drainage from the UZ as the water content equilibrates with the rainfall rate.
- **Equilibrium pressure profile (to residual moisture content)** This option is useful in arid and semi-arid conditions where the natural soil moisture below the root zone is low. When the rate of rainfall is low, then the natural soil moisture distribution will approximate the soil-moisture pressure curve. This option is useful in dry conditions because drainage to dry soil moisture with depth may require a long simulation to reach equilibrium.

The two options for specifying the water content or matrix potential with depth are typically only used in special conditions where the models are very detailed and accurate soil moisture or pressures have been measured. For example, in column experiments. Specifying these values requires you to define either a uniform value, or values and specified depths.

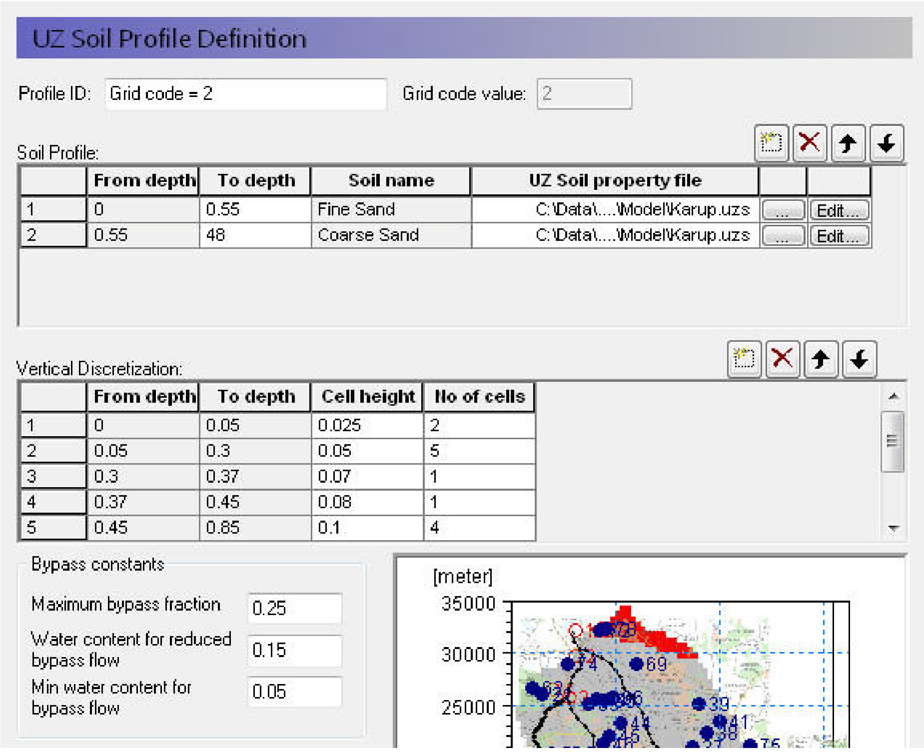
Green and Ampt Infiltration

The Green and Ampt infiltation is an analytical solution to the increased infiltration experienced in dry soils due to capillarity. It is available for the 2-Layer WB and the Gravity Flow UZ solution methods. The Richards equation method already includes capillarity so the Green and Ampt method is not applicable.

Related Items:

- Unsaturated Zone - Technical Reference *(V2 p. 141)*
- Richards Equation *(V2 p. 142)*
- Gravity Flow *(V2 p. 153)*
- Two-Layer Water Balance *(V2 p. 155)*
- Green and Ampt Infiltration *(V2 p. 162)*
- Simplified Macropore Flow (bypass flow) *(V2 p. 156)*
- Full Macropore Flow *(V2 p. 157)*
- Coupling the Unsaturated Zone to the Saturated Zone *(V2 p. 167)*

11.16.1 Soil Profile Definitions



Soil Profile Definitions	
Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue and Richards Equation or the Gravity Flow module selected for the numeric engine
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The first part of the soil profile definition is to define the areas with the same soil profiles. This can be done using either a uniform value or distributed using a dfs2 integer grid code file or a polygon .shp file. For each unique grid code or polygon in the distribution file, a separate sub-item will appear in the data tree, in which you can define the soil profile data.

In the sub-item dialogues, the soil type and distribution of the soil layers (i.e. depth and thickness of each soil type) in the individual profiles is specified, as well as the vertical discretisation of the soil profile.

Definition of soil properties

Profile ID - This is the editable name displayed in the data tree for this profile.

Grid code value - This is the current integer grid code or polygon value read from the soil distribution.

Soil Profile - The soil profile section allows you to define the vertical soil profile. Soil layers can be added, deleted and moved up and down using the icons.

From and To Depths refers to the distances to the top and bottom of the soil layer, below the ground surface. Only the To Depth item is editable, as the From Depth item is equal to the bottom of the previous layer.

Soil name is the name of the soil selected in the UZ Soil Property file. It is not directly editable, but must be chosen from the list of available soil names when you assign the UZ Soil property file using the file browser.

UZ Soil property file is the file name of the soil database, in which the soil definition is available. The Edit button opens the specified Soil property database file, whereas the Browse button [...] opens the file browser to select a file.

Vertical discretisation - In this section you specify the vertical discretisation of the soil profile, which 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.

From and To Depths refers to the distances to the top and bottom of the soil layer, below the ground surface. Neither are directly editable since they are calculated from the number of cells and their thicknesses.

Cell Height is the thickness of the numerical cells in the soil profile.

No. of Cells is the number of cells with the specified cell height. Together these two values define the total thickness of the current section.

Dispersivity is the longitutinal dispersivity in the column. This column only appears if WQ is simulated in the unsaturated zone.

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 during the pre-processing to give a smoother change in cell heights.

Bypass Constants The bypass constants are editable when Simple Bypass flow is checked on in the main Unsaturated Zone *(V1 p. 273)* dialogue. The availabe bypass parameters include:

Maximum bypass fraction - This is the maximum fraction of net rainfall that will infiltrate via simple macropore flow. Valid values are between 0 and 1.0.

Water content for reduced bypass flow - This is the threshold water content below which the bypass fraction is reduced. If the water content 10cm or 50cm below the ground surface is less than this water content, then the soil is dry and the bypass flow will be reduced.

Limit on water content for bypass flow - This is the minimum water content for bypass flow. If the water content 10cm or 50cm below the ground surface is less than this limit, then the soil is very dry and the bypass flow will be zero.

The actual relationship between the bypass constants and the calculation of the bypass flow is described in Simplified Macropore Flow (bypass flow) *(V2 p. 156)*.

Related Items:

- Unsaturated Zone - Technical Reference *(V2 p. 141)*
- Richards Equation *(V2 p. 142)*
- Gravity Flow *(V2 p. 153)*

11.16.2 Mass Transfer Coefficient to Unsaturated Macropores

Mass transfer coefficient to unsaturated macropores	
Condition	when water quality for Unsaturated Flow is selected in the Water Quality Simulation Specification dialogue and Full Macropore Flow is selected in the main Unsatured Flow dialog
dialogue Type	Stationary Real Data
EUM Data Units	1st-order rate WQ model [e.g. 1/day]

Macropores are defined as a secondary, additional continuous pore domain in the unsaturated zone, besides the matrix pore domain representing the microporous bulk soil.

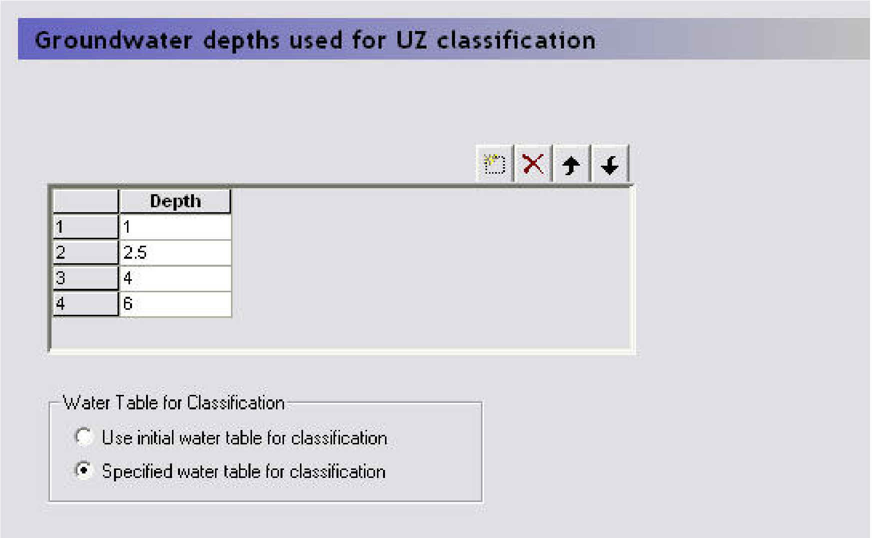
In other words, tthe macropores represent a continous secondary porous media that is “parallel” to the soil matrix porosity. Mass transfer to macropores occurs mostly by means of diffusion from the unsaturated zone soil matrix.

This coefficient represents the rate of transfer from the soil matrix to the macropores. It is defined per species.

Related Items:

Full Macropore Flow *(V2 p. 157)*

11.16.3 Groundwater Depths used for UZ Classification



Groundwater Depths used for UZ Classification	
Conditions:	if either the Automatic or Partially automatic column classifications selected in the main Unsaturated Zone dialogue

The automatic or partially automatic classification requires a distribution of groundwater elevations (see the main Unsaturated Zone dialogue). This can be either the initial depth to the groundwater based on the initial heads (default option), or you can supply a .dfs2 map of the groundwater elevations (second option). If you chose the second option, then a new item, Ground Water Table for UZ classification, will appear in the data tree.

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.

Related Items:

- Lumped UZ Calculations (Column Classification) *(V2 p. 186)*

11.16.4 Ground Water Table for UZ classification

Groundwater table for UZ classification	
Conditions:	if Specified water table for classification selected in the Groundwater Depths used for UZ Classification dia-logue
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

If the Specified water table for classification is selected in the Groundwater Depths used for UZ Classification dialogue, then this is the ground water table used for the classification.

Related Items:

- Lumped UZ Calculations (Column Classification) *(V2 p. 186)*

11.16.5 Partial automatic classification

Partial Automatic Classification	
Conditions:	if the Partially automatic column classifications selected in the main Unsaturated Zone dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code
Valid Values	1 or 2

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 with the following grid codes:

- In grid points where automatic classification should be used the grid code must be 1.
- In grid points where computation should be performed for all cells the grid code must be 2.

Related Items:

- Lumped UZ Calculations (Column Classification) *(V2 p. 186)*

11.16.6 Specified classification

Specified Classification	
Conditions:	if Specified classification selected in the main Unsaturated Zone dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code
Valid values	+/- 2 to the number of SZ cells (-1, 0, 1 not valid)

This is a data file specifying Integer Grid Codes, where UZ computations are to be carried out. Grid codes range from 2, up to the number of UZ columns. 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.

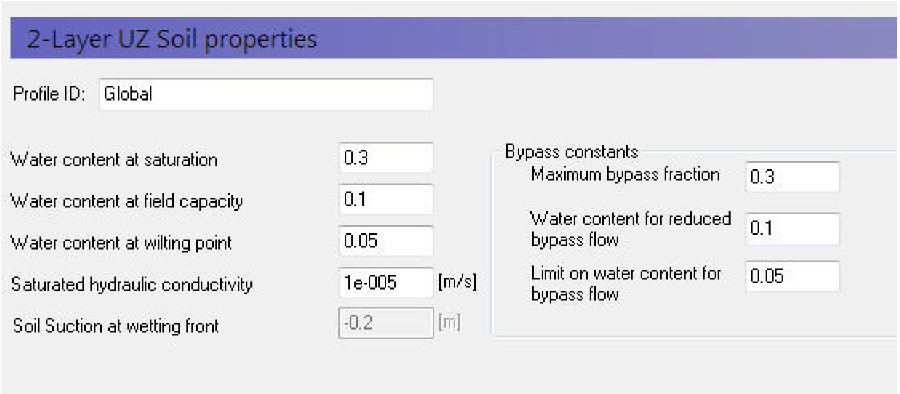
For example, if a grid code holds the value -2, a UZ computation will be carried out for the profile located in that grid. Simulation results will subsequently be transferred to all grid codes with code value +2.

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.

Related Items:

- Lumped UZ Calculations (Column Classification) *(V2 p. 186)*

11.16.7 2-Layer UZ soil properties



2-Layer UZ Soil Properties	
Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue and the Two-Layer Water Balance method selected for the numeric engine
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The first part of the soil definition is to define the areas with the same soil profiles. This can be done using either a uniform value or distributed using a dfs2 integer grid code file or a polygon .shp file. For each unique grid code or polygon in the distribution file, a separate sub-item will appear in the data tree, in which you can define the soil profile data.

Definition of soil properties

Profile ID - This is the editable name displayed in the data tree for this profile.

Grid code value - This is the current integer grid code or polygon value read from the soil distribution.

Water content at saturation - This is the maximum water content of the soil, which is usually approximately equal to the porosity,

Water content at field capacity - This is the water content at which vertical flow becomes negligible. It practice, this is the water content that is reached when the soil can freely drain. Although, it is usually higher than the residual saturation, which is usually defined as the minimum saturation that can be achieved in a laboratory test.

Water content at wilting point - This is the lowest water content that plants can extract water from the soil.

Saturated hydraulic conductivity - The saturated hydraulic conductivity of the soil is equal to the maximum infiltration rate of the soil.

Soil suction at wetting front - The soil suction at the wetting front is used in the Green and Ampt infiltration model to increase the infiltration rate beyond the saturated hydraulic conductivity when the soil is dry. This is a soil specific value that accounts for the capillarity of the soil.

Bypass Constants The bypass constants are editable when Simple Bypass flow is checked on in the main Unsaturated Zone *(V1 p. 273)* dialogue. The availabe bypass parameters include:

Maximum bypass fraction - This is the maximum fraction of net rainfall that will infiltrate via simple macropore flow. Valid values are between 0 and 1.0.

Water content for reduced bypass flow - This is the threshold water content below which the bypass fraction is reduced. If the water content 10cm or 50cm below the ground surface is less than this water content, then the soil is dry and the bypass flow will be reduced.

Limit on water content for bypass flow - This is the minimum water content for bypass flow. If the water content 10cm or 50cm below the ground surface is less than this limit, then the soil is very dry and the bypass flow will be zero.

The actual relationship between the bypass constants and the calculation of the bypass flow is described in Simplified Macropore Flow (bypass flow) *(V2 p. 156)*.

Related Items:

- Unsaturated Zone - Technical Reference *(V2 p. 141)*
- Two-Layer Water Balance *(V2 p. 155)*
- Simplified Macropore Flow (bypass flow) *(V2 p. 156)*

11.16.8 ET Surface Depth

ET Surface Depth	
Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue and the Two-Layer Water Balance method selected for the numeric engine
dialogue Type	Stationary Real Data
EUM Data Units	Depth below ground

In unsaturated soils, capillary action can lead to saturated conditions existing some distance above the water table. If the water table is close to the ground surface, ET will continue to occur at the maximum rate, so long as this capillary zone reaches the ground surface. That is, evapotranspiration will not decrease the saturation, but draw water directly from the water table due to capillary action. Similarly, when the water table is deeper, plant roots will draw water directly from the saturated zone as long as the roots reach the capillary zone.

In MIKE SHE, the ET surface depth is the thickness of the capillary fringe. This thickness is added to the root depth to define the thickness of Layer 1 in the Two-Layer Water Balance ET/UZ method.

In the absence of vegetation, if the water table is above this depth then soil evaporation will draw water from the water table (SZ). However, the rate that it can remove water from the water table decreases linearly with depth. That is, if the water table is at the ground surface it will remove water at the full rate and if the water table is at the ET Surface Depth then the rate will go to zero.

This is exactly the same function as MODFLOW, except MODFLOW does not describe it like this. In MODFLOW, because there is no vegetation, MODFLOW users typically use the linear function to also “simulate” root zone ET.

In MIKE SHE, if vegetation is present then the exact same function is used, but instead of being relative to the soil surface, it is relative to the bottom of the root zone. If the water table is at or above the bottom of the root zone then ET will be remove from SZ at the full rate. As the water table falls below the root zone, ET from SZ will be reduced linearly until the water table is at distance below the root zone equal to the ET Surface depth – at which point the ET from SZ becomes zero.

Note, that Layer 1 in the 2-Layer Water Balance UZ/ET method is the thickness of the root zone plus the ET Surface depth. If the water table falls below the bottom of the Layer 1, then ET from the SZ will stop. This does not mean that ET stops. ET will continue to remove water from the available water in

Layer 1. However, Layer 1 is greater than the root thickness, which means that the available storage is increased by the thickness of the ET Surface Depth. This is probably not significant when the roots are thick relative to the thickness of the capillary fringe. However, if the capillary fringe is thick, say more than 1m in a silty-clay, then the amount of water available in Layer 1 may be artificially high.

Richards equation, there is no ET Surface depth, because the capillary thickness is defined by the Sat-Pressure curve. In the Gravity method, the capillarity is ignored.

In coarse to medium sands, the ET surface depth is typically less than 10cm. In fine sands and silts, the ET surface depth could be a half a metre or more.

Note The ET surface depth must be greater than or equal to zero.

Related Items:

- Unsaturated Zone - Technical Reference *(V2 p. 141)*
- Two-Layer Water Balance *(V2 p. 155)*

11.16.9 Initial Conditions (UZ)

Initial conditions for the unsaturated zone	
Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue AND either The Water Quality is turned on for the unsaturated zone, OR An initial pressure or saturation has been selected for the Richards/Gravity flow solution method.
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Concentration or Temperature

There are two different initial conditions for the unsaturated zone.

Water movement - For water flow in the unsaturated zone (Richards/Gravity methods only), the initial conditions can be defined to be in equilibrium, which means the the initial pressure head and water content are derived from the soil pressure-moisture content relationship, or you can specify a water content or water pressure directly. The choice of initial condition is defined in the main Unsaturated Zone *(V1 p. 273)* dialogue.

Water quality - For the water quality module, an initial concentration is required for each mobile and sorbed species. Furthermore, if temperature dependent decay is specified, then an initial soil temperature must also be specified.

The initial conditions for the unsatured zone can be defined as a constant value, or by layer, or from a fully 3D dfs3 file.

Related Items:

- Unsaturated Zone - Technical Reference *(V2 p. 141)*

11.17 Groundwater table for lower UZ boundary

Groundwater table for lower UZ boundary	
Conditions	If Unsaturated Flow selected in the Simulation Specification dialogue without selecting Saturated Flow, OR If the Unsaturated Flow is selected and the Linear Reservoir Method for groundwater is selected.
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The groundwater table must be explicitly defined if unsaturated flow is simulated without explicitly simulating groundwater flow. The specified groundwater table is used as the lower boundary condition for the unsaturated model.

If the Linear Reservoir Method is used for the groundwater simulation, the water table is not calculated, thus requiring the water table to be explicitly defined.

However, the specified groundwater table is a static variable. If you need to relate your unsaturated zone model to a dynamic water table, you must include the saturated zone in your model based on the 3D Finite Difference Method.

Related Items:

- Saturated Flow - Technical Reference *(V2 p. 189)*
- Linear Reservoir Method *(V2 p. 205)*

11.18 Saturated Zone

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 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 Finite Difference method and MODFLOW. The differences that are present are limited to differences in the discretisation and to some differences in the way some of the boundary conditions are defined.

Main saturated zone dialogue

The top dialog for the saturated zone section is slightly different depending on whether or not the Linear Reservoir Method *(V1 p. 290)* or the 3D Finite Difference Method *(V1 p. 291)* is chosen.

11.18.1 Linear Reservoir Method

Setting up a saturated groundwater model using the Linear Reservoir Method involves defining the Interflow and Baseflow Reservoirs, as well as their respective properties.

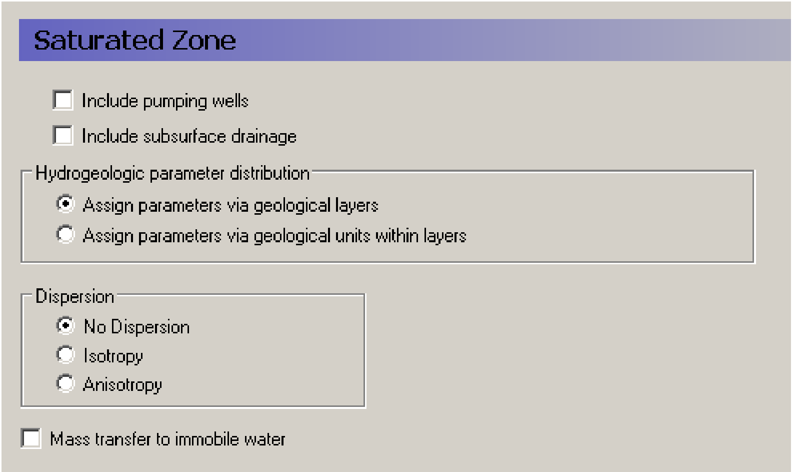
Pumping wells

By default, wells are not included, but in most applications pumping wells play a major role in the hydrology of the area. If wells are included in the model, then this must be checked and a new item in the data tree appears where the well database can be defined. Pumping wells extract water only from the baseflow reservoirs.

Related Items:

- Saturated Flow - Technical Reference *(V2 p. 189)*
- Linear Reservoir Method *(V2 p. 205)*

11.18.2 3D Finite Difference Method



Setting up a saturated zone hydraulic model based on the 3D Finite Difference Method 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 Hydro River) 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.

Pumping wells
By default, wells are not included, but in most applications pumping wells play a major role in the hydrology of the area. If wells are included in the model, then this must be checked and a new item in the data tree appears where the well database can be defined.

Subsurface drainage
Subsurface drainage is used to limit the amount of groundwater that reaches the ground surface and to route near surface groundwater to local streams

and rivers. There are a number of drainage options for specifying surface drains in MIKE SHE, which are described in more detail in the section Drainage.

Hydrogeologic parameter definition
The first option allows you to specify the hydrologic parameters of the geologic layers and lenses directly by means of .dfs2 grid files, point/line theme .shp files, or irregular xyz point values. The second option allows you to assign the hydrologic parameters to the geologic layers by means of zones with uniform properties, whereby the zones are defined by integer grid codes. Dispersion

If your simulation includes water quality modelling in the saturated zone, then you must also define the type of dispersion you want to simulate. Dispersion is the physical process that causes solutes to spread longitudinally, vertically and horizontally as they move through the soil. The dispersion essentially represents the natural, microscopic variations in pore geometry that cause small scale variations in flow velocity.

No dispersion - Dispersion is ignored and no dispersivities need to be specified.

Isotropic - The transverse horizontal and transverse vertical dispersivities are assumed to be the same. Only two dispersivities need to be specified - the longitudinal and the transverse dispersivities.

Anisotropic - The horizontal and vertical transverse dispersivities are different, which requires the specification of five different dispersivities.

Transfer to Immobile Water

If many cases, immobile water trapped in the bulk soil matrix is both a source and sink of solutes. In other words, as solutes move in the bulk water media, some of the solutes will diffuse into water trapped in the pores of the soil grains.

Furthermore, solutes in fractured media will be transported by diffusion in and out of the soil matrix. The porosity of the soil matrix is often at least ten times less than the fractures. The diffusion of solutes into the soil matrix will retard the initial breakthrough of the solute and the matrix will act as a long term sink as the solute slowly diffuses out of the matrix long after the main solute plume has passed by.

If you turn on this option, then four additional data items may be required, namely

- the Secondary Porosity *(V1 p. 302)* will be added as a geologic property,
- a Dual Porosity transfer coefficient *(V1 p. 306)* per species will be required for each Water Quality layer,

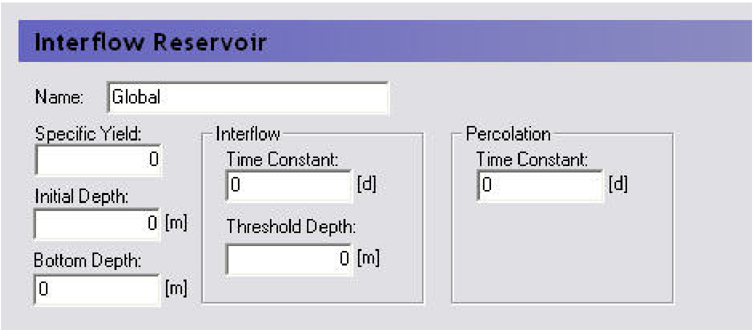
- a Macropore/Secondary Half Life *(V1 p. 338)* coefficient will be required for each species to calculate the solute decay in the matrix, and
- an initial concentration in the matrix for each species.

Note: there is no seperate sorption process for the immobile water.

Related Items:

- Saturated Flow - Technical Reference *(V2 p. 189)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*

11.18.3 Interflow Reservoirs



The interflow reservoirs are used to route near-surface groundwater to local streams. In the Linear Reservoir Method, each reservoir is assumed to be like a bathtub, with an inflow from infiltration and the upstream reservoir, as well as an outflow flowing into the next downstream reservoir and down into the baseflow reservoir beneath. Each linear reservoir flows only into the next downstream interflow reservoir, or into a stream if it is the lowest reservoir.

Note Polygon shape files are currently not allowed for defining interflow reservoirs. The flow reference between interflow reservoirs depends precisely on the integer code numbers assigned to the reservoirs. Within a subcatchment, the interflow reservoir with the higher number always flows into the reservoir with the next lowest number.

Each Interflow reservoir requires a value for:

Specific Yield - to account for the fact that the reservoir contains a porous media, and is not an actual bathtub.

Initial depth - the initial depth of water in the reservoir, measured from the ground surface.

Bottom depth - the depth below the ground surface of the bottom of the reservoir. If the water level drops to the bottom of the reservoir, percolation stops.

Interflow time constant - a calibration parameter that represents the time it takes for water to flow through the reservoir to the next reservoir.

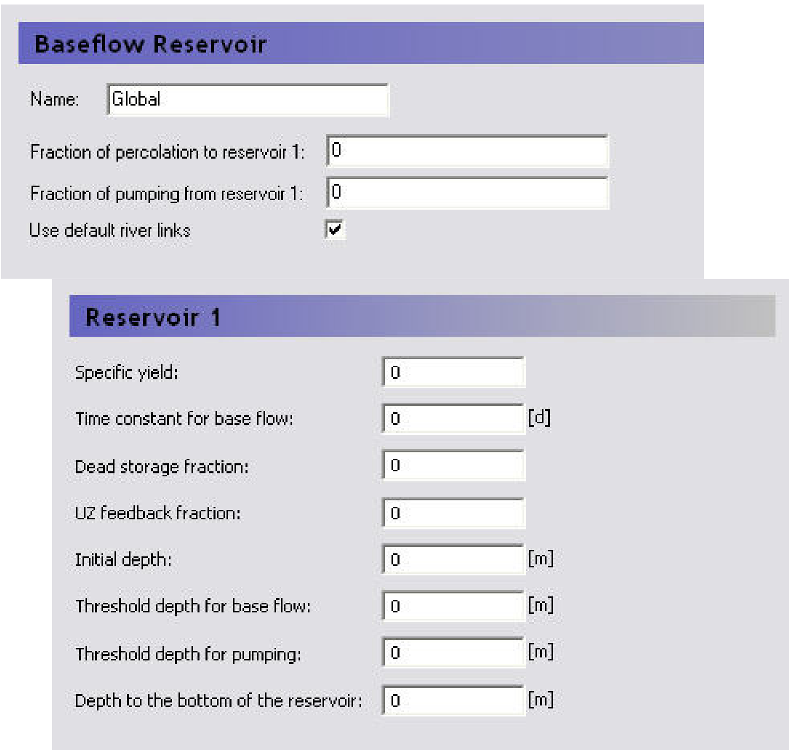
Percolation time constant - a calibration parameter that represents the time it takes for water to seep down into the baseflow reservoir

Interflow threshold depth - the depth below the ground surface when interflow stops. If interflow stops, percolation will continue until the reservoir is empty (i.e the water level reaches the bottom depth). The threshold depth must be less than or equal to the depth to the bottom of the reservoir.

Related Items:

- Saturated Flow - Technical Reference *(V2 p. 189)*
- Linear Reservoir Method *(V2 p. 205)*
- Calculation of Interflow *(V2 p. 212)*

11.18.4 Baseflow Reservoirs



In the Linear Reservoir Method in MIKE SHE, each baseflow reservoir is divided into two parallel baseflow reservoirs. The two parallel baseflow reservoirs each receive a fraction of the percolation water from the interflow reservoirs as their only source of inflow. Each baseflow reservoir can discharge to pumping wells, to the unsaturated zone adjacent to streams and rivers (i.e. the zone beneath the lowest interflow reservoir), as well as directly to the river network.

In the primary baseflow reservoir map view, you can define the number of baseflow reservoirs in your system. You can define any number of baseflow reservoirs, but typically, there are only one or two.

For each baseflow reservoir pair, there are three items to define:

Fraction of percolation to reservoir 1 - this is used to divide the percolation between each of the two parallel baseflow reservoirs.

Fraction of pumping from reservoir 1 - this is used to divide the pumping (if it exists) between each of the two parallel baseflow reservoirs.

Use default river links - in most cases you will link the simplified overland flow and the groundwater interflow to all of the river links found in the lowest interflow reservoir in each subcatchment. However, in some cases you may want to link the flow to particular river links. For example, if your river network does not extend into the subcatchment, you can specify that the interflow discharges to a particular node or set of nodes in a nearby river network.

If you uncheck this checkbox, a River Links *(V1 p. 218)* sub-item will appear where you can specify the river branch and chainage to link the subcatchment to.

In the sub-dialogue for each of the parallel baseflow reservoirs, you must define the following:

Specific Yield - to account for the fact that the reservoir contains a porous media and is not an actual bathtub.

Time constant for base flow - a calibration parameter that represents the time it takes for water to flow through the reservoir

Dead storage fraction - the fraction of the received percolation that is not added to the reservoir volume but is removed from the available storage in the reservoir.

UZ feedback fraction - the fraction of base flow to the river that is available to replenish the water deficit in the unsaturated zone adjacent to the river (i.e. the lowest interflow reservoir in the subcatchment).

Initial depth - the initial depth to the water in the reservoir measured from the ground surface

Threshold depth for base flow - the depth below the ground surface when base flow stops. The threshold depth must be less than or equal to the depth to the bottom of the reservoir.

Threshold depth for pumping -the depth below the ground surface when pumping is shut off. The threshold depth must be less than or equal to the depth to the bottom of the reservoir.

Depth of the bottom of the reservoir - the depth below the ground surface of the bottom of the reservoir.

Related Items:

- Saturated Flow - Technical Reference *(V2 p. 189)*
- Linear Reservoir Method *(V2 p. 205)*
- Calculation of Baseflow *(V2 p. 214)*

11.18.5 Geological Units

Geological units

	Soil name	Soil code	Horizontal conductivity	Vertical conductivity	Specific yield	Storage coefficient	Porosity
1	DL	1	4e-006	2e-006	0.1	3e-005	0.6
2	DS+TS	10	5e-005	1e-005	0.15	3e-005	0.6
3	ES	3	5e-005	1e-005	0.15	3e-005	0.6
4	FT	4	4e-007	2e-007	0.15	3e-005	0.6
5	FP,FS,ML	8	4e-008	2e-008	0.1	3e-005	0.6
6	HG	6	2e-005	6e-006	0.15	3e-005	0.6
7	HS	7	1e-005	2e-006	0.15	3e-005	0.6
8	MS	9	2e-005	4e-006	0.15	3e-005	0.6

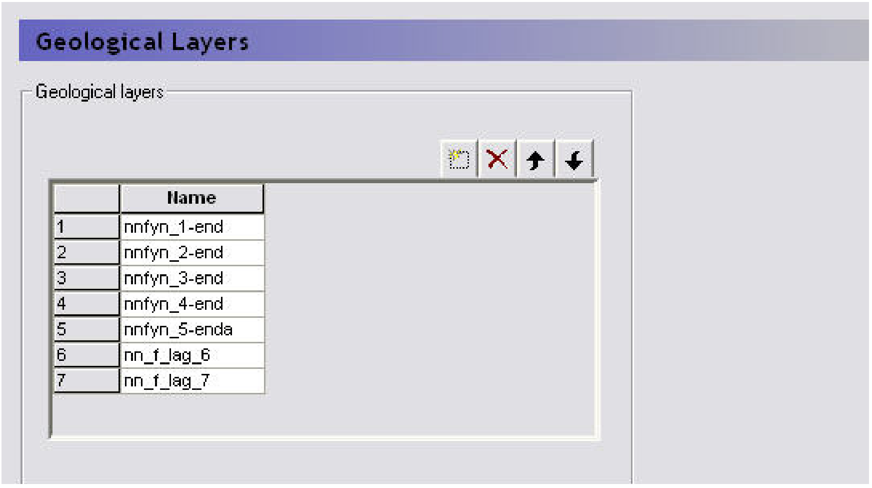
If you specify your geologic conceptual model via geological units, you can add each of your geologic units and its associated hydrogeologic properties to the table. Then, instead of specifying the hydrogeologic properties for each geological layer, you only need to specify the distribution of the units within the geologic layer or lense.

Related Items:

- Horizontal Hydraulic Conductivity *(V1 p. 299)*
- Vertical Hydraulic Conductivity *(V1 p. 300)*
- Specific Yield *(V1 p. 300)*
- Specific Storage *(V1 p. 301)*

- Porosity *(V1 p. 302)*
- Dispersion Coefficients LHH, THH, TVH, LVV, THV *(V1 p. 304)*

11.18.6 Geological Layers



For each geologic layer, you must specify the hydrogeologic parameters of the layer including

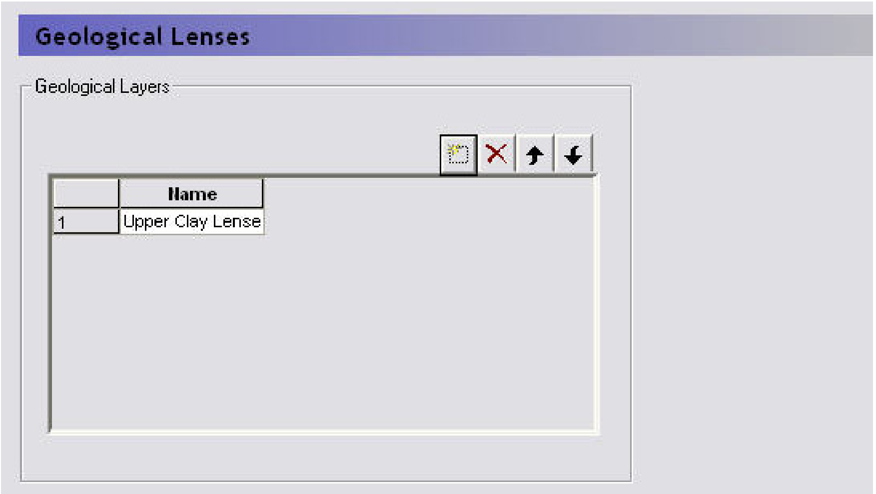
- Lower Level *(V1 p. 336)*,
- Horizontal Hydraulic Conductivity *(V1 p. 299)*,
- Vertical Hydraulic Conductivity *(V1 p. 300)*,
- Specific Yield *(V1 p. 300)*
- Specific Storage *(V1 p. 301)*,

If you define your hydrogeology by , then most of the physical properties will be defined as properties of the Geological Unit and there will be an additional item, the Geological Unit Distribution, in the data tree.

Related Items:

- Saturated Flow - Technical Reference *(V2 p. 189)*

11.18.7 Geological Lenses



For each geologic layer, you must specify the hydrogeologic parameters of the layer including

- Lower Level *(V1 p. 336)*,
- Upper Level *(V1 p. 335)*,
- Horizontal Extent *(V1 p. 336)*,
- Horizontal Hydraulic Conductivity *(V1 p. 299)*,
- Vertical Hydraulic Conductivity *(V1 p. 300)*,
- Specific Yield *(V1 p. 300)*,
- Specific Storage *(V1 p. 301)*,

If you define your hydrogeology by , then most of the physical properties will be defined as properties of the Geological Unit and there will be an additional item, the Geological Unit Distribution, in the data tree.

Related Items:

- Lenses *(V1 p. 55)*

11.18.8 Geological Unit Distribution

Geological Unit Distribution	
Conditions:	If geology defined by Geologic Units
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code
Valid Values	each value must be in the Geological Units table

The Geological Unit Distribution references the geological units defined in the table. Each Integer Code must refer to one of the geological units in the table.

Related Items:

- Geological Units *(V1 p. 296)*
- Lenses *(V1 p. 55)*

11.18.9 Horizontal Hydraulic Conductivity

Horizontal Hydraulic Conductivity	
dialogue Type	Stationary Real Data
EUM Data Units	HydrConductivity

The hydraulic conductivity is a function of the soil texture and is related to the ease with which water can flow through the soil. Loose, coarse uniform soils have a higher conductivity than compacted soils with a range of particle sizes. Thus, a loose, uniform coarse sand can have a horizontal hydraulic conductivity as high as 0.001 m/s. Whereas, a tight, compacted clay can have a have a horizontal hydraulic conductivity as low as $1x10^{-8}$ m/s - which is 5 orders of magnitude.

The horizontal hydraulic conductivity is typically 5 to 10 times higher than the vertical hydraulic conductivity.

MIKE SHE assumes that the horizontal conductivity is isotropic in the x and y directions

Related Items:

- Geological Units *(V1 p. 296)*
- Vertical Hydraulic Conductivity *(V1 p. 300)*

- Lenses *(V1 p. 55)*

11.18.10Vertical Hydraulic Conductivity

Vertical Hydraulic Conductivity	
dialogue Type	Stationary Real Data
EUM Data Units	HydrConductivity

The hydraulic conductivity is a function of the soil texture and is related to the ease with which water can flow through the soil. Loose, coarse uniform soils have a higher conductivity than compacted soils with a range of particle sizes. Thus, a loose, uniform coarse sand can have a horizontal hydraulic conductivity as high as 0.001 m/s. Whereas, a tight, compacted clay can have a have a horizontal hydraulic conductivity as low as $1x10^{-8}$ m/s - which is 5 orders of magnitude.

The vertical hydraulic conductivity is typically 5 to 10 times lower than the horizontal hydraulic conductivity.

Related Items:

- Geological Units *(V1 p. 296)*
- Horizontal Hydraulic Conductivity *(V1 p. 299)*
- Lenses *(V1 p. 55)*

11.18.11Specific Yield

Specific Yield	
dialogue Type	Stationary Real Data
EUM Data Units	Specific Yield

In an unconfined aquifer, the Specific Yield is defined as the volume of water released per unit surface area of aquifer per unit decline in head. The specific yield is much higher than the Specific Storage because the water that is released is primarily from the dewatering of the pores at the water table. This results in a unit of $L^3/L^2/L$, which is dimensionless.

The Specific Yield is only used in transient simulations, but must always be input. Furthermore, the specific yield is only used in the cells that contain the water table. In the cells below the water table, the Specific Storage is used.

Specific Yield of upper SZ layer

The specified value for specific yield is not used for the specific yield of the upper most SZ 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.

Related Items:

- Specific Yield of the upper SZ numerical layer *(V1 p. 222)*
- Geological Units *(V1 p. 296)*
- Lenses *(V1 p. 55)*

11.18.12Specific Storage

Specific Storage	
dialogue Type	Stationary Real Data
EUM Data Units	Elastic Storage

In a confined aquifer, the specific storage is defined as the volume of water released per volume of aquifer per unit decline in head. This is slightly different than the specific yield because the water released from storage comes primarily from the expansion of the water and aquifer compression due to the reduction in water pressure (increase in effective stress). Thus, the water released from storage is released from the entire column of water in the aquifer, not just at the phreatic surface. This results in a unit of $L^3/L^3/L$, or 1/L.

The Specific Storage Coefficient is only used in transient simulations, but must always be input. Furthermore, the specific storage coefficient is only used in the cells below the water table. In the cells containing the water table, the Specific Yield *(V1 p. 300)* is used.

11.18.13Porosity

Porosity	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue
dialogue Type:	Stationary Real Data
EUM Data Unitse:	Porosity

In a porous media, most of the volume is taken up by soil particles and the actual area available for flow is much less than the nominal area. This distinction is important when calculating flow velocities for solute transport. The porosity is the cross-sectional area available for flow divided by the nominal cross-sectional area. This is often referred to as the effective porosity, since it discounts the dead end pore spaces that are not available for flow. In the absences of dead end pores, the porosity is equal to the specific yield.

The Porosity must be greater than 0 and less than 1. For unconsolidated porous media, the porosity is usually from 0.15 to 0.3 depending of the grain size distribution (the more uniform the higher the effective porosity). For fractured media the porosity is usually much lower, in the from 0.01 to 0.05.

.Related Items:

- Secondary Porosity *(V1 p. 302)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*

11.18.14Secondary Porosity

Secondary Porosity	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialog AND if the Transfer to Immobile Water *(V1 p. 292)* option selected in the Saturated Zone dialog
Dialog Type	Stationary Real Data
EUM Data Units	Porosity

Solutes travelling in a fractured media will be transported by diffusion into and out of the soil matrix of the surrounding media. The velocity in fractured media due to the lower effective porosity will cause very fast break-through down gradient. However, the solute mass that diffuses into the surrounding media will act like a long term solute source after the main plume has passed, as the solute diffuses back into the fractures.

The matrix source/sink process can be included in MIKE SHE by activating the Transfer to Immobile Water *(V1 p. 292)* option in the Saturated Zone dialog.

Matrix porosity must be between 0 and 1. Matrix porosities are generally very difficult to measure and are usually calibrated against breakthrough curves to estimate a realistic value. Furthermore, the value needs to be the "effective" matrix porosity - that is, the matrix porosity that is "actively" involved in solute diffusion. This can be significantly lower than the matrix porosity measured by core analysis. For a limestone aquifer the matrix porosity has been calibrated to be as small as 4 per cent (core samples indicated 20 to 35%) whereas for a clay till sample it was calibrated to be 20 to 30% - a little less than the total matrix porosity.

Related Items:

- Dual Porosity transfer coefficient *(V1 p. 306)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Transport in Fractured Media *(V2 p. 269)*

11.18.15Bulk Density

Bulk Density	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue AND the SZ solute transport option is turned on AND at least one sorption process is active
dialogue Type:	Stationary Real Data
EUM Data Unitse:	Density

The bulk density is the average density of the in-place soil material, including void space. In other words, if you were to remove one cubic meter of soil, how much would it weigh? A typical bulk density is about 1700 kg/m^3, but representative local values are usually available.

The bulk density is used in the calculation of sorption of solutes, so you need to specify a bulk density whenever any sorption processes are active in SZ.

Related Items:

- Solute Transport in the Saturated Zone *(V2 p. 259)*

11.18.16Dispersion Coefficients LHH, THH, TVH, LVV, THV

Dispersion Coefficients LHH, THH, TVH, LVV, THV	
dialogue Type	Stationary Real Data
EUM Data Units	Dispersion Coefficient

If dispersion is included, then the two different dispersion options differ in the number of dispersion parameters required. If you assume isotropic conditions you need to specify the longitudinal dispersivity, α_L, and the transversal dispersivity, α_T . If you assume anisotropic conditions you need to specify five dispersivities.

The magnitude of the dispersivity coefficient depends on the degree of heterogeneity in your geology and the degree to which these heterogeneities have been described in the model. The more heterogeneous your geology is, the larger the dispersivities should be. On the other hand, the more detailed you have described the heterogeneities with your model geometry, the smaller dispersivities should be.

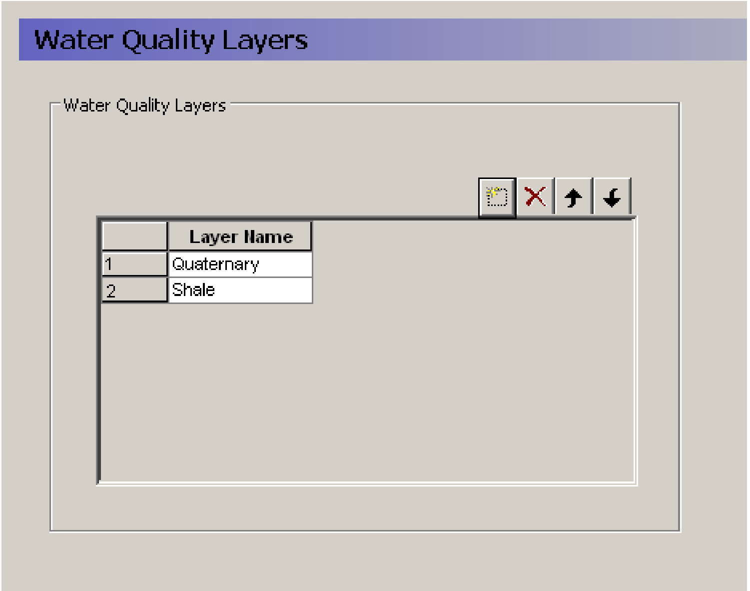
Further, the magnitude of the dispersivities depends on the size of the model and on the model grid size. The larger the model, the larger the dispersivities should be. Whereas, the larger the grid size, the smaller the dispersivities should be due to numerical dispersion.

Thus, it is difficult to give a rule of thumb for the values of dispersivity. Recent field experiments on solute transport, though, indicate that the longitudinal dispersivity should be in the range of 1% or less of the travel distance, the transverse-horizontal dispersivity should be at least 50 times less than this and the transverse-vertical dispersivity should be 2 or more times less than the transverse-horizontal.

Related Items:

- Solute Transport in the Saturated Zone *(V2 p. 259)*

11.18.17Water Quality Layers



Water Quality Layers	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue AND the SZ solute transport option is turned on AND at least one sorption or decay process is active

The water quality layers present a conceptual layer system like the geologic layers that applies to chemistry parameters. In some cases, the water quality layers will mimic the geologic layers, but in other case they will not. For example, redox potential will not generally follow the geologic layers and water quality parameters related reaction processes will be quite different in oxidizing and reducing envrironments. Likewise, you may have several geologic layers in the upper quaternary deposits that affect the flow hydraulics, but the water chemistry can be divided into a quaternary sand and silt zone overlain on an iron rich shale layer.

Unfortunately, the different water quality parameters may not all require identical water quality layers. However, in the current version, only one set of water quality layers is available. If two or more parameters require different distributions, then you will have to divide the overlapping layers into seperate units.

When you add a Water Quality Layer to the table, an item is added to the data tree containing all of the relevant data for the Water Quality Decay Processes *(V1 p. 337)* and Water Quality Sorption processes *(V1 p. 338)* that are active. The location of the WQ Layer is specified by a Lower Level *(V1 p. 336)*.

Related Items:

- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Water Quality Decay Processes *(V1 p. 337)*
- Water Quality Sorption processes *(V1 p. 338)*
- Lower Level *(V1 p. 336)*

11.18.18 Dual Porosity transfer coefficient

Mass transfer coefficient to immobile water	
Dialog Type	Stationary Real Data
EUM Data Units	

Solutes travelling in a fractured media will be transported by diffusion into and out of the soil matrix of the surrounding media. The velocity in fractured media due to the lower effective porosity will cause very fast break-through down gradient. However, the solute mass that diffuses into the surrounding media will act like a long term solute source after the main plume has passed, as the solute diffuses back into the fractures.

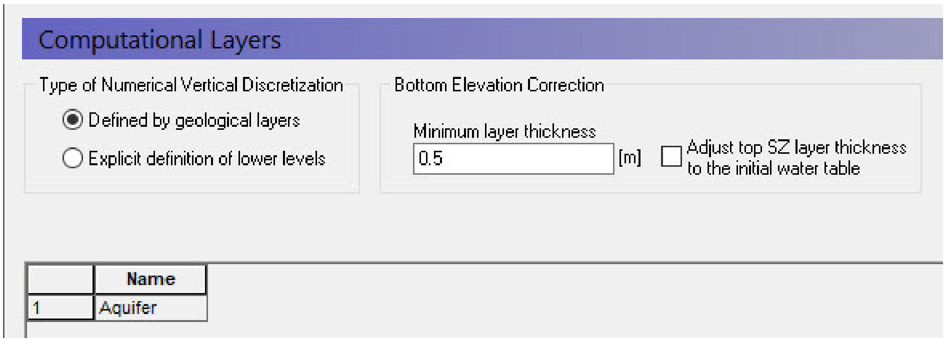
The matrix source/sink process can be included in MIKE SHE by activating the Transfer to Immobile Water *(V1 p. 292)* option in the Saturated Zone dialog.

The mass transfer coefficient represents the rate at which mass is transfered between the fractures and the matrix. It depends on the geology and chemistry. It is defined per species.

Related Items:

- Secondary Porosity *(V1 p. 302)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Transport in Fractured Media *(V2 p. 269)*

11.18.19Computational Layers



The vertical discretisation in the saturated zone can be defined in two ways:

- by the geological layers, in which case there will be one calculation node in each geological layer,
- by explicitly defining the lower level of each calculation layer.

Vertical discretisation

Defined by the geological layers - Groundwater flow in a multi-layer aquifer can be described by a model in which the computational layers follow the interpreted geological layers. Each layer is characterised only by its base level specified either by a constant level or by a distributed file. The number of numerical layers will be identical with the number of geologic layers.

Explicit definition of lower levels - If you define the vertical discretisation explicitly each computational layer is defined by its lower elevation.

Bottom elevation correction

There are two corrections that may be made to the layer elevations.

Minimum layer thickness - The minimum thickness of the calculation layers must be specified to adjust the geological model or the specified levels to prevent layers with zero thickness or very thin layers. Otherwise, very thin layers may cause numerical difficulties. The default value is 0.5, which is usually sufficient. This means that if the two geological layers or specified layer elevations approach one another then the bottom layer will be pushed down to maintain the minimum layer thickness.

Adjust top SZ layer thickness to the initial water table - In principle, the UZ calculates vertical infiltration in the unsaturated zone and the SZ calculates 3D saturated flow below the water table. However, the UZ and SZ layer elevations are defined independently of one another. In MIKE SHE, the UZ flow is only calculated to the water table, or to the bottom of top SZ layer. Any UZ cells below this are ignored and outflow from the UZ model is added to the top layer of the SZ model. This option allows you to more easily keep the UZ and SZ models consistent based on the depth of the initial SZ water table.

Related Items:

- Coupling the Unsaturated Zone to the Saturated Zone *(V2 p. 167)*
- Definition of vertical UZ numerical grid *(V2 p. 179)*
- Saturated Flow - Technical Reference *(V2 p. 189)*

11.18.20Initial Potential Head

Initial Potential Head	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

The Initial Potential Head is the starting head for transient simulations and the initial guess for steady-state simulations. The choice of initial head for steady state simulations may affect the rate of numerical convergence depending on the solver used.

Related Items:

- Saturated Flow - Technical Reference *(V2 p. 189)*

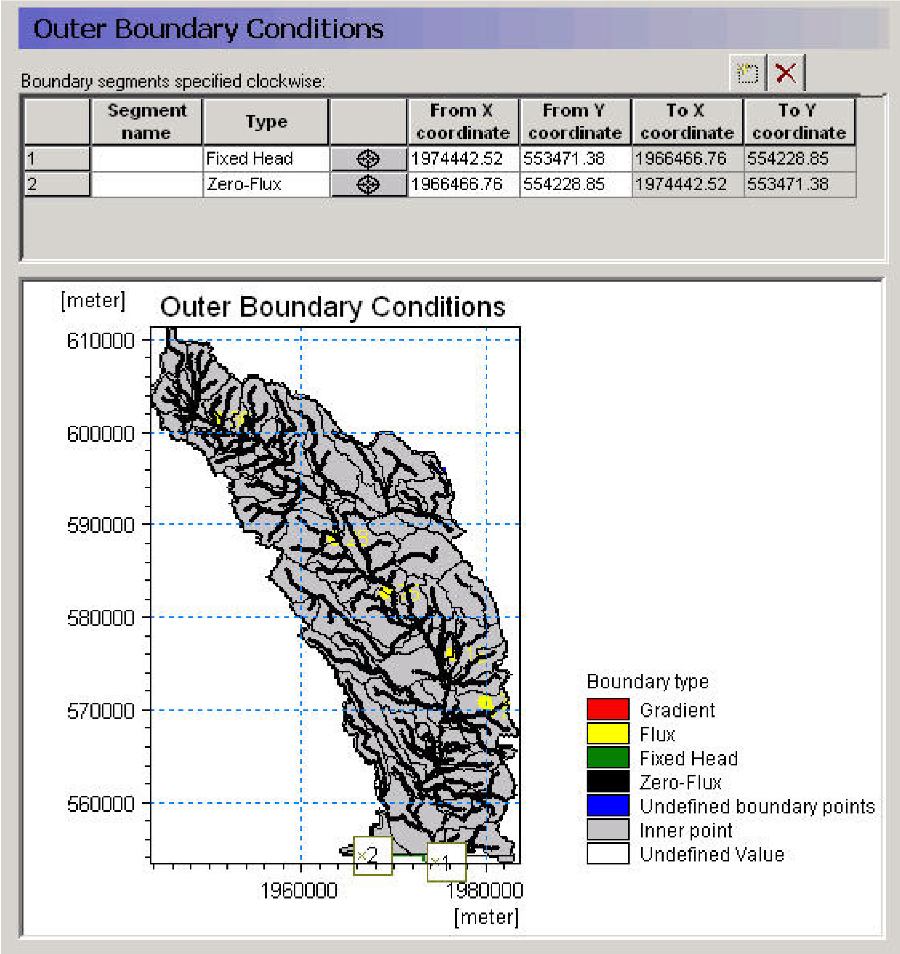
11.18.21Initial Soil Temperature

The initial soil temperature is required if temperature dependent decay is specified. The current soil temperature is calculated at every time step based on an empirical formula using the air temperature.

Related Items:

- Decay *(V2 p. 286)*

11.18.22Outer boundary conditions



	Segment name	Type		From X coordinate	From Y coordinate	To X coordinate	To Y coordinate
1		Fixed Head		1974442.52	553471.38	1966466.76	554228.85
2		Zero-Flux		1966466.76	554228.85	1974442.52	553471.38

The outer boundary conditions are defined as line segments between two boundary points. The boundary points are, in principle, independent of the model domain because they do not need to lie on the model boundary. Rather they are projected onto the nearest model boundary cell. Thus, the model boundary can be modified slightly without having to modify the boundary locations. However, if the model boundary is moved significantly or if the boundary is convoluted then the calculated 'nearest' node might not be the one expected.

Specifying a boundary condition

To specify an outer boundary segment,

1. add a new line to the outer boundary points table
2. click on the target icon,

3. click on one end of your boundary segment,
4. add a second line to the outer boundary points table
5. click on the new target icon,
6. click on the other end of your boundary segment
7. Change the name of the top line of the points table,
8. Select the appropriate boundary condition for the boundary segment.

Available boundary conditions

Fixed Head - This boundary prescribes a head in the boundary cell. The head can be fixed at a prescribed value, fixed at the initial value from the initial conditions, or assigned to a .dfs0 time series file. If a time series input is used, then the actual value used in the model at the current time step is linearly interpolated from the available values.

The last option is a time varying dfs2 file, which is typically extracted from a regional results file. This can be done using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. MIKE SHE then interpolates in both time and space from the .dfs2 file to the local head boundary at each local time step.

Zero flux - This is a no-flow boundary, which is the default.

Flux - This boundary describes a constant or time varying flux across the outer boundary of the model. A time varying flux can be specified as a mean step-accumulated discharge (e.g. m^3/s) or as a step-accumulated volume (e.g. m^3). A positive value implies an inflow to the model cells.

Gradient - This boundary describes a constant or time varying gradient between the node on the outer boundary and the first internal node. A time varying gradient can be specified as an instantaneous dimensionless or percent value. A positive gradient implies a flux into the model.

Notes

1. The head is calculated in a No Flow outer boundary cells, whereas the head is specified in the Fixed Head outer boundary cells, but in both cases all properties must be assigned to all outer boundary cells.
2. An internal model cell in contact with multiple boundary cells will not receive multiple quantities of water.
3. Additional detailed information can be found under Boundary Conditions *(V2 p. 199)* in the Saturated Flow - Technical Reference *(V2 p. 189)*.
4. An error will be generated if the flux/gradient input cell

- has zero thickness,
- has a horizontal hydraulic conductivity of zero,
- is an inactive internal boundary cell, or

- is a fixed head internal boundary cell.

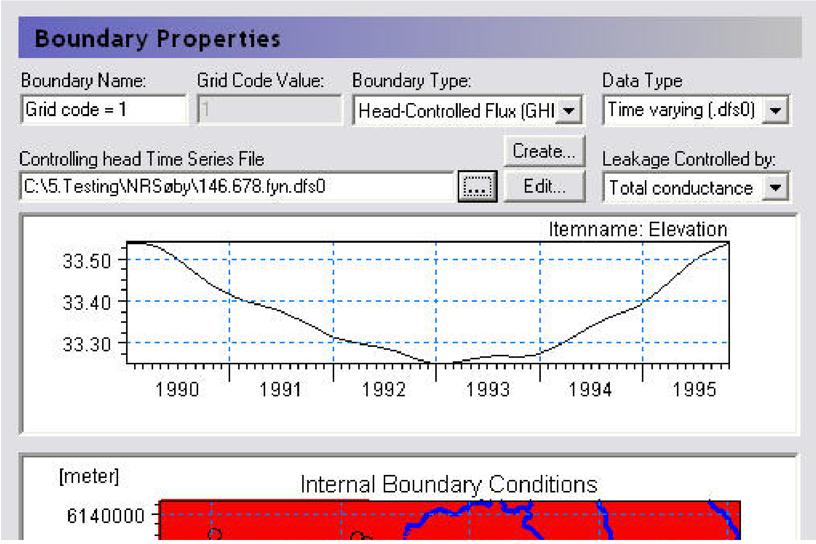
5. A warning will be issued if the flux/gradient boundary

- is a head controlled flux (GHB) internal boundary cell, or
- is a fixed head drain internal boundary cell.

Related Items:

- Saturated Flow - Technical Reference *(V2 p. 189)*
- Boundary Conditions *(V2 p. 199)*

11.18.23Internal boundary conditions



Internal Boundary Conditions	
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The map dialogue for the internal boundary conditions allows you to specify the locations of the various boundary conditions. Any Integer Code value is permissible and a separate item will be added to the data tree below this level with one item for each unique integer code in the domain.

Only one integer code is allowed per cell, which means that no cell can have more than one boundary condition.

If you use a polygon .shp file, each unique polygon will have a separate entry in the data tree.

Available boundary conditions

The following boundary conditions can be defined for each integer code:

Fixed Head For a fixed head boundary, you specify the head in the cell. The model will not calculate the head in the cell. Care should be taken when specifying fixed head boundary conditions, as the cell becomes an infinite source or sink of water. The fixed head can be a prescribed value, fixed at the initial value from the initial conditions, assigned to a .dfs0 time series file, or assigned to a time varying dfs2 file. The last option is typically from a results file. It could be from a regional results file, which can be extracted using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. Or, it could be from a previous run of the same model.

Fixed Head Drain - For a fixed head drain boundary, you specify a reference head. If the cell water level is above the reference level, then the boundary acts like a normal fixed head boundary conditions. If the head in the cell falls below the reference level, then the boundary condition is turned off. That is, if the simulated head drops below the head reference level the flux is set to zero. Thus, the fixed head drain allows only water extraction.

The reference head can be a prescribed value, fixed at the initial value from the initial conditions, assigned to a .dfs0 time series file, or assigned to a time varying dfs2 file. The last option is typically from a results file. It could be from a regional results file, which can be extracted using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. Or, it could be from a previous run of the same model.

Note: This boundary condition was previously called the Head controlled abstraction boundary condition in early versions of MIKE SHE.

Head Controlled Flux (GHB) - The head controlled flux, or General Internal Head Boundary is similar to the fixed head. However, a flow resistance is incorporated via a user specified leakage coefficient.

The head can be a prescribed value, assigned to a .dfs0 time series file, or assigned to a time varying dfs2 file. The last option is typically from a results file. It could be from a regional results file, which can be extracted using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. Or, it could be from a previous run of the same model.

If the GHB is selected, an extra item is added to the data tree below the boundary condition for the leakage coefficient. The leakage coefficient can be specified as either a simple leakage coefficient [1/time] or as a total conductance [length^2/time].

Note: This boundary condition was previously called the General Internal Head Boundary condition in early versions of MIKE SHE

Inactive Cells - This boundary condition is used to make interior cells of the model inactive. It works by assigning a hydraulic conductivity of zero to the cells if the simulation is transient, or a value of 10^{-15} if the simulation is steady-state. Note, though, that this method means that in the pre-processed data, you will see the inactive cells show up in the maps of hydraulic conductivity rather than in the maps of boundary conditions. Also note that since the inactive cells are actually active cells with zero conductivity, the results will also include head values in these points.

Related Items:

- Saturated Flow - Technical Reference *(V2 p. 189)*
- Boundary Conditions *(V2 p. 199)*

11.18.24 Initial (secondary) concentration

Initial (secondary) concentration	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialog
Dialog Type	Stationary Real Data
EUM Data Units	Concentration

There is an initial concentration item for each computational layer. Under each initial concentration item, there is one sub-item for each active species.

The initial concentration is used by the MIKE SHE Water AD engine as the starting concentration for the Water quality simulation.

If you have specified Transfer to Immobile Water *(V1 p. 292)* option selected in the Saturated Zonedialog, then you will also have to specify an initial concentration in the immobile water located in the secondary (matrix) porosity.

Related Items:

- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Initial Conditions *(V2 p. 267)*

11.18.25 Initial number of particles per cell

Initial number of particles per cell	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialog and the Random walk particle tracking sub-option is specified and the initial particle density is specified as a number of particles
Dialog Type	Stationary Real Data
EUM Data Units	Integer

There is an initial number of particles item for each computational layer. The initial number of particles is used by the MIKE SHE Random Walk particle tracking engine as the starting concentration.

Related Items:

- Particle Tracking-Reference *(V2 p. 319)*

11.18.26 PT Registration Codes/Lenses

PT Registration Codes/Lenses	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialog and the Random walk particle tracking sub-option is specified and PT Registration codes are defined by layer or lense
Dialog Type	Stationary Real Data
EUM Data Units	Integer

When you run the random walk particle tracking option, registration areas can be defined. All particles passing through a registration area will be tagged. This allows you to create a point shape file that contains the starting location of all particles that passed through the registration area.

Registration codes by layer

If you specify the registration codes by layer, then you can specify a dfs2 integer grid code array for each numerical layer. Each unique integer codes will be used as a registration zone.

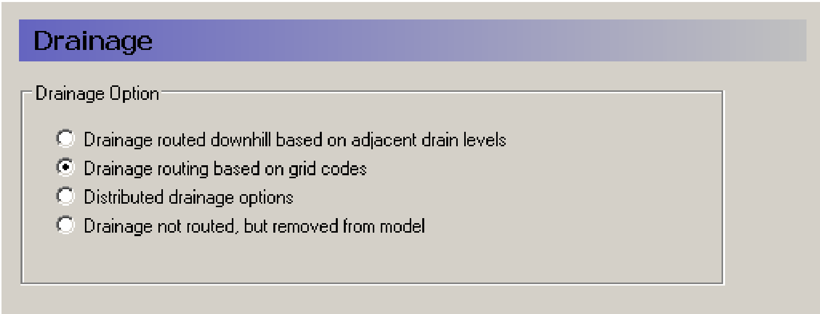
Registration codes by lenses

If you specify the registration zones by lenses, then you have to specify an Horizontal Extent *(V1 p. 336)*, as well as an Upper Level *(V1 p. 335)* and a Lower Level *(V1 p. 336)* for the lense. This is a more flexible way to define the registration zone, but is significantly slower because every particle has to be checked against the extent and the top and bottom elevations.

Related Items:

- PT Registration Extraction *(V1 p. 358)*
- Particle Tracking-Reference *(V2 p. 319)*

11.18.27Drainage



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 Hydro River. It can also be used to simulate simple overland flow in a lumped conceptual approach. Saturated zone drainage is applied to 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.

Drain flow is simulated using an empirical 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 1e-7 1/s.

MIKE SHE also requires a reference system for linking the drainage to a recipient node or cell. The recipient can be a river node, another SZ grid cell, or a model boundary. Whenever drain flow is produced during a simulation, the computed drain flow is routed to the recipient point using a linear reservoir routing technique.

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.

If the cell does not contain a boundary or river link, then the next search is repeated until either a local minimum is found or a boundary cell or river link is located.

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 drainage water will become part of the local overland flow system.

Note: Be aware that 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. In other words, any drainage that is generated at a node, is immediately moved to the recipient node. The assumption here, is that the time step length is longer than the time it takes for the drainage to reach its destination.

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.

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.

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 Branch and chainage. At the moment, this option requires the use of **Extra Parameters** *(V1 p. 334)* and is described in SZ Drainage to Specified MIKE Hydro River H-points *(V2 p. 350)*.

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** *(V1 p. 334)* and is described in the section Using MIKE SHE with MIKE URBAN *(V2 p. 239)*.

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.

Related Items:

- Groundwater drainage *(V1 p. 57)*
- Time varying SZ drainage parameters *(V2 p. 353)*
- Saturated Zone Drainage *(V2 p. 202)*
- SZ Drainage to Specified MIKE Hydro River H-points *(V2 p. 350)*
- Using MIKE SHE with MIKE URBAN *(V2 p. 239)*

11.18.28Drain Level

Drain Level	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

If surface drainage is routed by drain levels, the drainage routing reference system is created automatically using the slope of the drains calculated from the drainage levels in each cell.

The drain levels are only used for two things:

- to calculate the drain cell-drain recipient relations, and
- to calculate the amount of drain flow produced in each node when the water table is above the drain level.

The drain level also determines from which SZ layer the drain water will be extracted.

Important note: If the drain level is set equal to the topography in a cell, then the drainage will be turned off in the cell. Drain levels above the topography are not allowed. In this case, the drain level will be automatically adjusted to just below the topography.

Related Items:

- Drainage *(V1 p. 315)*
- Groundwater drainage *(V1 p. 57)*
- Time varying SZ drainage parameters *(V2 p. 353)*
- Saturated Zone Drainage *(V2 p. 202)*
- SZ Drainage to Specified MIKE Hydro River H-points *(V2 p. 350)*
- Using MIKE SHE with MIKE URBAN *(V2 p. 239)*

11.18.29Drain Time Constant

Drain Time Constant	
dialogue Type	Stationary Real Data
EUM Data Units	Leakage Coefficient./Drain Time Constant

Drainage flow is simulated using an empirical formula that requires a drainage level and a time constant (leakage factor) for each cell. Mathematically, the time constant is exactly the same as a leakage coefficient - it is simply a factor that is used to regulate how quickly the water can drain. A typical time constant is between 1e-6 and 1e-7 1/s.

Related Items:

- Drainage *(V1 p. 315)*
- Groundwater drainage *(V1 p. 57)*
- Time varying SZ drainage parameters *(V2 p. 353)*
- Saturated Zone Drainage *(V2 p. 202)*
- SZ Drainage to Specified MIKE Hydro River H-points *(V2 p. 350)*
- Using MIKE SHE with MIKE URBAN *(V2 p. 239)*

11.18.30Drain Codes

Drain Codes	
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

If the drainage routing is specified by Drain Codes, a grid code map is required that is used to link the drain flow producing cells to recipient grid cells. The drain levels are still used to calculate the amount of drain flow produced in each node, but the routing is based only on the code values in the drain code file.

The Drain Code can be any integer value, but the different values have the following special meanings:

Code = 0 - Grid cells with an Drain Code value of zero will not produce any drain flow and will not receive any drain flow.

Code > 0 - Grid cells with **positive** Drain Code values will drain to the nearest river, boundary or local depression in the drain level - in that priority - located next to a cell with the same Drain Code value. Thus, if a grid cell produces drainage,

1. If there are one or more cells with the same drain code next to a river link, then the drain flow will be routed to the nearest of these cells.

2. If there are no cells with the same Drain Code located next to a river link, then the drain flow will be routed to the nearest boundary cell with the same Drain Code value.

3. If there are no boundary cells with the same Drain Code value, the drain flow will be routed to the cell with the lowest drain level that has the same Drain Code value (which may create a lake).

Code < 0 - Grid cells with **negative** Drain Code values will drain to either a boundary or a local depression, in that order. Thus, if a grid cell produces drainage,

1. If there are no cells with the same Drain Code located next to a river link, then the drain flow will be routed to the nearest boundary cell 2ith the same Drain Code value.

2. If there are no boundary cells with the same Drain Code value, the drain flow will be routed to the cell with the lowest drain level that has the same Drain Code value (which may create a lake).

Related Items:

- Drainage *(V1 p. 315)*
- Groundwater drainage *(V1 p. 57)*
- SZ Drainage to Specified MIKE Hydro River H-points *(V2 p. 350)*
- Using MIKE SHE with MIKE URBAN *(V2 p. 239)*

11.18.31 Option Distribution

Option Distribution	
Conditions:	always when Surface Drainage active AND when the Distributed drainage option is used.
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code
Valid Values:	1, 2, 3 and 4 only

The drain type distribution is used to distinguish areas of the model where different drainage options are used.

Code = 1 - Drainage in grid cells with a value of 1 is routed downhill based on the value of the drain level specified in Drain Level data item.

Code =2 - Drainage in grid cells with a value of 2 is routed via Drain Codes as specified in the Drain Codes data item.

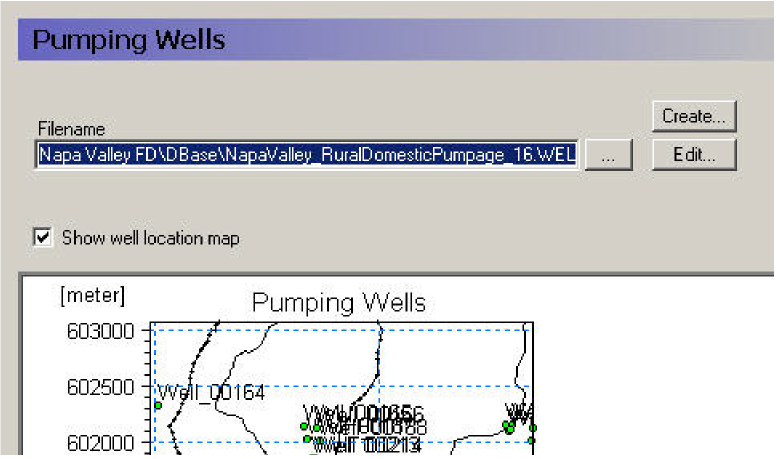
Code = 3 - Drainage in grid cells with a value of 3 is routed to a specified branch and chainage. At the moment, this options requires the use of Extra Parameters.

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.

Related Items:

- Saturated Zone Drainage *(V2 p. 202)*
- **Extra Parameters** *(V1 p. 334)*
- SZ Drainage to Specified MIKE Hydro River H-points *(V2 p. 350)*
- Using MIKE SHE with MIKE URBAN *(V2 p. 239)*

11.18.32Pumping Wells



If pumping wells are active in the model domain, then you must specify the name of the well database to include in the model setup.

Edit The Edit button will open the current Well Database with the current maps and overlays open in the well editor.

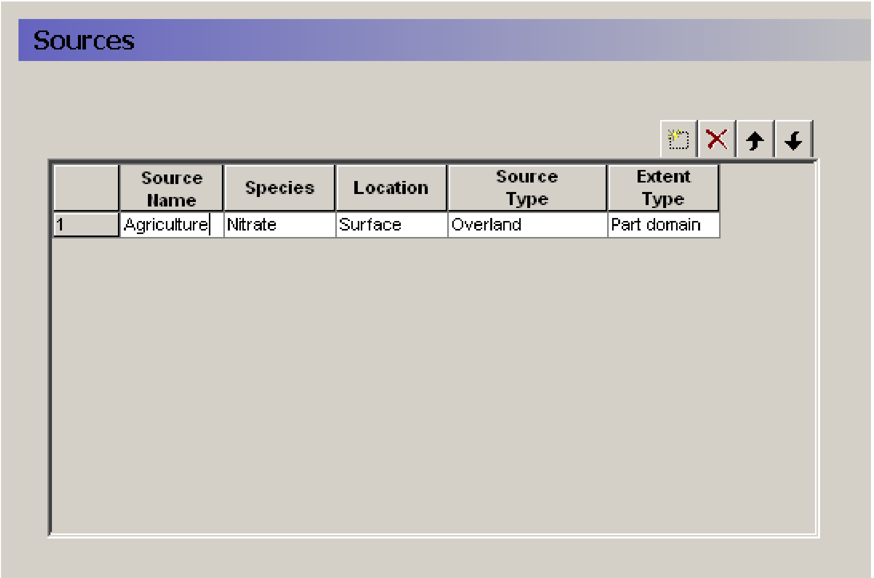
You may get an error in when you open the Well Database if the model has not yet been pre-processed or if the model data has changed without re-preprocessing the model. This error happens because MIKE SHE is trying to reconcile and plot both the geologic layers (input data) and the numerical layers (preprocessed data) for each cell containing a pumping well.

Create The Create button will create a new Well Database file

Related Items:

- Well editor *(V1 p. 367)*

11.19 Sources



The specification of water quality sources is very flexible. The Sources dialogue allows you to add and delete sources, as well as define the type and location of the source. The table provides an overview of all of your sources in your model.

An important feature of the source location definition is the partial extent distribution function. This allows you to define, for example, a distributed global source file - say of the field scale agricultural inputs in your catchment - and run individual water quality scenarios for each sub-catchment (modelled as an partial extent) to assess the subcatchment contributions to the global stream impact.

Source Name - The name appears in the data tree for reference.

Species - You can only choose from the list of available species that you have defined in the **Species** *(V1 p. 210)* dialogue.

Location - The location is defines whether the source is located on the ground surface (Surface) or in the saturated or unsaturated zone (Subsurface). The available source types depend on where the source is located.

Source Type - If the source is located on the ground surface, then it can be either a Precipitation source (concentration in precipitation water) or an Overland source (mass on the surface). In both cases, the solute can infiltrate or runoff as lateral overland flow.

Extent Type - The source can cover the entire domain (Full domain) or only part of the domain (Part domain). In both cases, the actual source strength can very spatially and temporally. The Extent is used simply to restrict the source data to a zone smaller than the model domain.

Related Items:

- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Solute Transport in the Unsaturated Zone *(V2 p. 270)*
- Solute Transport in Overland Flow *(V2 p. 275)*

11.19.1 Strength

Strength	
dialogue Type:	Time-varying Real Data
EUM Data Units	Grid Code
Time Series EUM Data Units	Concentration or Mass per time

Since the source is a common data item for overland, unsaturated and saturated flow, the units of the source strength depend on the type of source being simulated.

The source strength 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. If the data is fully distributed, then you can enter a time varying dfs2 file.

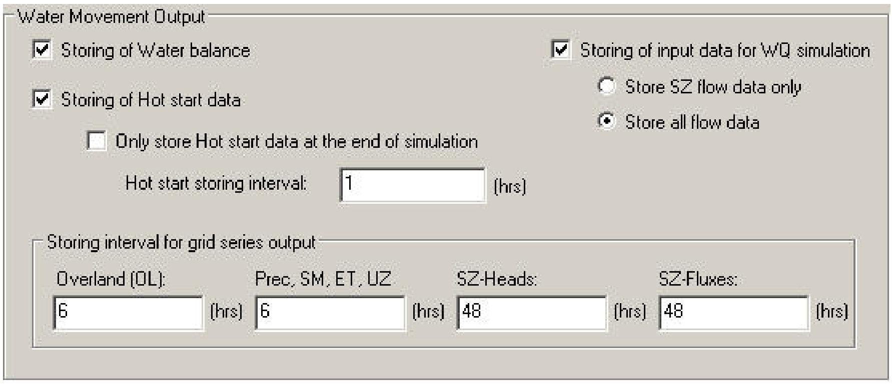
Related Items:

- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Source/Sinks, Boundary Conditions and other Exchanges *(V2 p. 267)* in the saturated zone
- Solute Transport in the Unsaturated Zone *(V2 p. 270)*
- Source/Sinks, Boundary Conditions and other Exchanges *(V2 p. 274)* in the unsaturated zone
- Solute Transport in Overland Flow *(V2 p. 275)*
- Source/Sinks, Boundary Conditions and other Exchanges *(V2 p. 278)* in overland flow

11.20 Storing of Results

Default output folder - If you unselect this option, then you can change the output folder for the results. If you change the output folder, then you must re-run the model for the Results tab to point to the correct folder.

Water Movement Output



Storing of water balance data - When this option is selected, MIKE SHE will store all of the relevant output data for the analysis of the water balance. This will automatically select the required items in the Grid Series Output section.

Storing of Hot start data - The option allows you to save a simulation that can be used as the start a new simulation. See Time Step Control for more information on using Hot Start data as the initial data for a simulation.

Only store hot start data at the end of the simulation - Typically, the follow on simulation starts at the end of the previous simulation. However, if you want to test the sensitivity of the results to the starting condition, for example, you may want to save hot start data more frequently. However, the hot start file can become very large if the hot start data is saved frequently.

Store AD input data during water movement simulation - A MIKE SHE water quality simulation is calculated based on the cell-by-cell water fluxes calculated by the water movement module. If water quality is included in the model setup, then the necessary data is automatically saved and this item is hidden. However, if water quality is not included in the simulation, you can optionally tell MIKE SHE to save the necessary data for the water quality model by checking this box on and selecting the save option. The first option only saves the saturated zone data, which is suitable if you are only going to calculate the water quality in the saturated zone. For example, the random walk particle tracking is only available in the saturated zone and there is, therefore, no need to save overland flow data for a particle tracking simulation. The second option saves all of the data necessary for a water quality simulation in the complete integrated model.

Storing interval for grid series output

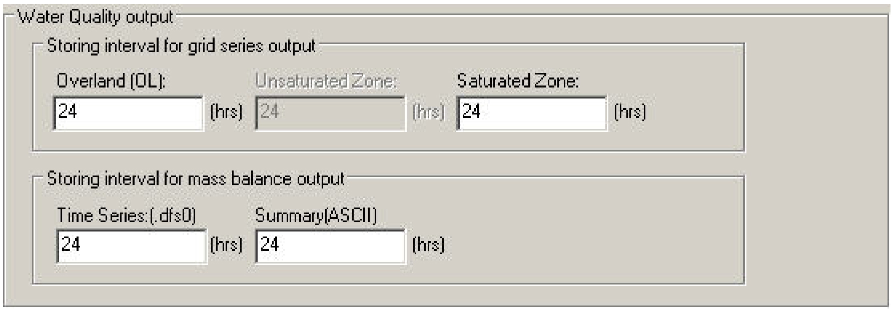
Gridded output can create very large output files if the data is stored at every time step. Gridded data is not usually compared to frequent measurements, such as daily groundwater levels, so the output frequency can be much less than the time step length. In fact, the output frequency of gridded output is often determined by visualization needs, such as to make smooth animations.

The gridded output for the different processes can be saved at different frequencies - the overland frequency is separate from unsaturated zone frequency is separated from the saturated zone frequency. However, you cannot save individual items at different frequencies. Thus, since precipitation, evapotranspiration and unsaturated flow output items are related, they are all saved at the same frequency. The saturated zone heads and fluxes are separated into different frequencies, however, because the gridded output files for a detailed 3D model can get very large.

Note: The Storing Time Step for SZ must be an integer multiple of the Maximum Allowed SZ Time Step that is specified in the Time Step Control dialogue. In other words, if the Maximum allowed SZ time step is 24 hrs, then the Storing Time Step for SZ can only be a multiple of 24 hours (e.g. 24, 48, 72 hours, etc.).

Similarly, the storing frequency for SZ Fluxes must be an integer multiple of the SZ heads frequency, and the storing frequency for overland flow and unsaturated flow items, must be an integer multiple of their respective maximum allowed time step length.

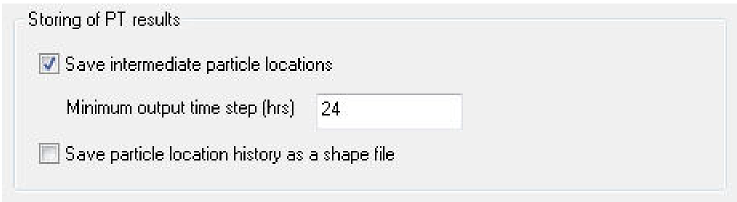
Water Quality Output



Storing interval for grid series output - similar to the storing interval for water movement data, the gridded output data files for a detailed transport simulation can get very large. These three frequencies allow you to save only the data you need.

Storing interval for mass balance output - similar to the storing interval for grid series output, these separate storing frequencies allow you to save only the data you need.

Particle Tracking Output



Save intermediate particle locations - If you want to plot pathlines, then you need to turn this option on. You should be careful, when using this option, as the file size can easily become very large.

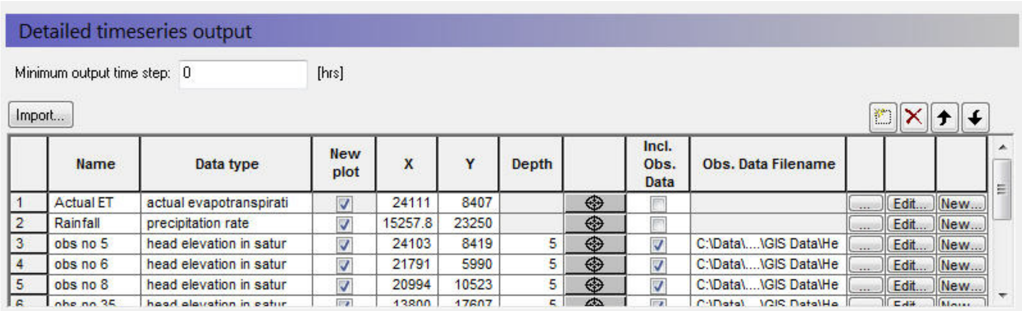
Minimum output time step - This value determines how often the particle locations will be saved. The more frequent the saving, the larger the output file.

Save particle location history as a shape file - The intermediate points can be saved directly into a shape file. However, since there can be hundreds or even thousands of particles in a PT simulation, this can generate very large shape files.

Related Items:

- Time Step Control *(V1 p. 178)*
- Detailed time series output *(V1 p. 328)*
- Grid series output *(V1 p. 333)*

11.20.1 Detailed time series output



The Detailed time series output dialogue allows you to specify the location at which you want detailed time series output and the item that you want output. For each specified point, the output variable is stored in a .dfs0 file with one value for every simulation time step. Finally, for each item in the detailed time series table, an HTML plot is created in the Result Tab.

Note All of the detailed time series items are stored in one .dfs0 file. This can lead to file size and disk space errors, if you have a long detailed simulation or more than ~200 detailed time series items. Also, the HTML output in the Results Tab will become very slow if you have a lot of items, since it has to read the entire .dfs0 file and generate all of the graphs every time you access the Detailed Time Series page in the Results Tab.

Minimum Output time step - In some simulations, the output time step can become very small due to the automatic time step adjustment. If the output is being saved every time step, then the output files can become very large. This value allows you to set a lower limit on the output time step.

Name - This is a text field that can be used to specify a reference name for the location, for example, a borehole name. This is also the name that will be used for the time series item in the Dfs0 file created during the simulation.

Data Type - This is the list of available output items is dynamic in the sense that the list changes in response to the processes included in the Simulation Specification dialogue. Further, additional items are available that are related to simulation variables, such as the number of iterations during each Saturated Zone time step. A list of available Data Types can be found in Output Items *(V1 p. 88)*.

New plot - If this is checked, then the a new detailed time series HTML-plot will be created on the Results Tab. If this is unchecked, then the output will be added to the previous plot. You can use the Up and Down arrows to arrange the output points so that relevant points are plotted together

X, Y - Often, detailed time series are associated with measurement stations. That is, locations at which a time series of measurements are available, for example, water levels in a well or water depths on a flood plain. 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).

Target Icon - You can use the target icon to locate the output point exactly. Alternatively, you can type the exact coordinates or import the items from an ASCII file.

Include observation data - If this is checked, then a .dfs0 file can be specified that includes observation points. The observation points are automatically plotted along with the results in the HTML plot on the Results tab. The .dfs0 item is selected in the file browser dialogue. The **Edit** button opens the specified .dfs0 file and the **New** button can be used to create a new .dfs0 file with the correct item type etc. and at the same time import data from an Excel spreadsheet.

Importing 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>DataTypeCode>NewPlot>X>Y>Depth>UseObsdata>dfs0file name>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.

DataTypeCode - This is a numeric code used to identify the output data type. A list of available Data Type Codes can be found in Output Items *(V1 p. 88)*.

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 sub-surface 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.

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

dfs0file name - 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,

.\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.

The following is a simple example with three MIKE SHE observation points, where the file name is obsdata.dfs0

Obs_1 >20 >1 >234500. >456740. >0. >0 >.\time\obsdata >1

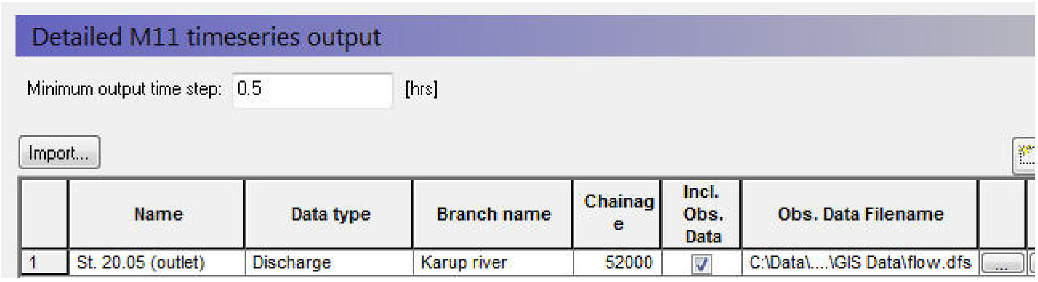
Obs_2 >15 >1 >239700. >458900. >10. >1 >.\time\obsdata >2

Obs_3 >16 >0 >241500. >459310. >20. >1 >.\time\obsdata >3

Related Items:

- Time Step Control *(V1 p. 178)*
- Detailed MIKE Hydro River Output *(V1 p. 331)*
- Grid series output *(V1 p. 333)*
- MIKE SHE Detailed Time Series *(V1 p. 350)*

11.20.2 Detailed MIKE Hydro River Output



The Detailed time series output for MIKE Hydro River allows you to specify the river chainage location at which you want detailed time series output and the item that you want output. For each specified point, the output variable is stored in a .dfs0 file with one value for every simulation time step. Finally, for each item in the detailed time series table, an HTML plot is created in the Result Tab - with or without observation data.

The principle advantage of this option is that you can now easily create calibration plots of calculated versus observed water levels without opening and having to create specific plots in MIKE View.

Minimum Output time step - In some simulations, the output time step can become very small due to the automatic time step adjustment. If the output is being saved every time step, then the output files can become very large. This value allows you to set a lower limit on the output time step.

Name - This is a text field that can be used to specify a reference name for the location, for example, a gage name.

Data Type - This is the list of available output items, which for MIKE Hydro River only contains two items - water level and flow rate.

Branch name - the Branch name must be a valid branch name in the MIKE Hydro River model. However, this is not checked until run time, at which point an error message will be generated if it is not valid and the simulation will be stopped.

Chainage - like the branch name, the chainage must be a valid MIKE Hydro River chainage.

Include observation data - If this is checked, then a .dfs0 file can be specified that includes observation points. The observation points are automatically plotted along with the results in the HTML plot on the Results tab. The .dfs0 item is selected in the file browser dialogue. The **Edit** button opens the specified .dfs0 file and the **New** button can be used to create a new .dfs0 file with the correct item type etc. and at the same time import data from an Excel spreadsheet.

Importing data

Detailed MIKE Hydro River Time Series data can be imported directly into the 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>DataTypeCode>BranchName>Chainage>UseObsdata>dfs0file name>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.

DataTypeCode - This is a numeric code used to identify the output data type.

Branch name - the Branch name must be a valid branch name in the MIKE Hydro River model. However, this is not checked until run time, at which point an error message will be generated if it is not valid and the simulation will be stopped.

Chainage - like the branch name, the chainage must be a valid MIKE Hydro River chainage.

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

dfs0file name - 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,

.\Time\Calibration\RiverstageObs

refers to the file *RiverStageObs.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.

The following is a simple example with three MIKE Hydro River observation points, where the file name is obsdata.dfs0

Obs_1 > ? > GrandRiver > 34500. >0 >.\time\obsdata >1

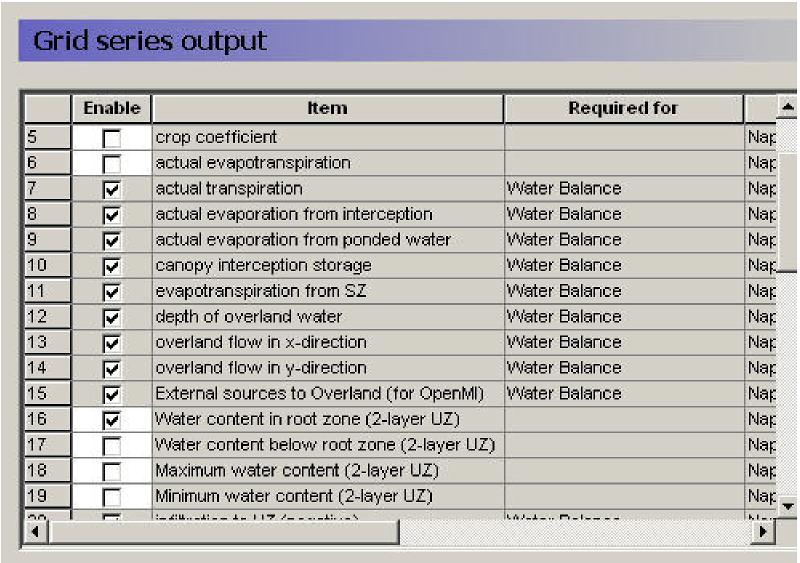
Obs_2 > ? > GrandRiver > 22500. >1 >.\time\obsdata >2

Obs_3 > ? > GrandRiver > 1500. >1 >.\time\obsdata >3

Related Items:

- Time Step Control *(V1 p. 178)*
- Detailed time series output *(V1 p. 328)*
- Grid series output *(V1 p. 333)*
- MIKE Hydro River Detailed Time Series *(V1 p. 353)*

11.20.3 Grid series output



The Grid series output dialogue allows you to specify the frequency at which you want detailed output of gridded data and the items that you want output. A list of available Data Types can be found in Output Items *(V1 p. 88)*. The list is dynamic in the sense that the list changes in response to the processes included in the Simulation Specification dialogue.

In some cases, such as when the Water Balance output has been specified (see Storing of Results), some of the items will be automatically selected and cannot be unselected. This will be noted in the Required for column of the dialogue.

Related Items:

- Time Step Control *(V1 p. 178)*
- Detailed time series output *(V1 p. 328)*
- Detailed MIKE Hydro River Output *(V1 p. 331)*

- Gridded Data Results Viewer *(V1 p. 351)*

11.21 Extra Parameters

The Extra Parameters Section is available to support new features in MIKE SHE that are not yet supported in the dialogues and data tree. Detailed descriptions of the features that use Extra Parameters are found in Extra Parameters *(V2 p. 325)*.

If you need to activate a feature that is only supported in the Extra Parameters section, you must first add the necessary number of lines to the Extra Parameters table. Then fill in the data that is required for the module.

Name - this is the name of the parameter that is required by the unsupported feature. It must be spelled exactly as specified in the documentation. In may be the actual name of the feature or the name of a parameter.

Type - The type is the type of parameter. The following types are available:

Float - Real, floating point number

Integer - Integer number

Boolean - an On/Off checkbox - typically used to turn a feature on or off

Text - a character string

file name - this is typically the file where more detailed input data is recorded

Value - this is the value associated with the Type above

11.22 Data description

11.22.1 Layer description

In some cases an input item may be defined as either a constant value or more detailed using a Layer based approach. In the latter case you need to create a layer for each input. Then for each specified layer a sub-item will appear where you can specify the input data.

11.22.2 Item description

For some items an input item comprises both a spatial distribution and a value that may be time varying.

The spatial distribution of the item can be defined in three ways:

- Uniform
- Station based
- Fully distributed

Uniform
The Uniform spatial distribution defines the same value in the entire area. The value can be defined by a constant or by a time-varying dfs0 file.

Station based
If the data is station-based the area is divided into a number of sub areas, one for each station. You define the spatial area related to each station by grid codes in a dfs2 file or by polygons in a shape file (.shp). Then for each station a sub-item will appear where you can enter the time series of values for the station.

Fully distributed
If the data is fully distributed you need to specify a dfs2 file with the values to be used.

11.23 Geometry

Many items in MIKE SHE are defined by a surface elevation. For example, numerical and geologic layers both require bottom surfaces. Some items, such as geologic lenses also require an upper surface. Finally, some items are only defined in parts of the domain, which requires a definition of the areal extent of the parameter.

11.23.1 Upper Level

Upper Level	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The Upper Level is the upper elevation of the lense or water quality source. It is used by the interpolation algorithm to assign geologic and water quality properties to the model cells.

Related Items:
- Geological Lenses *(V1 p. 298)*
- Lenses *(V1 p. 55)*
- Water Quality **Sources** *(V1 p. 323)*

11.23.2 Lower Level

Lower Level	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The Lower Level value is used to define the bottom of geologic layers and lenses, numerical layers, and water quality layers and sources.

The bottom of a layer always equal to the top of the layer underneath. However, in the case of lenses and sources, the lower level is used to interpolate geologic and source properties to the model cells. In the case of numerical layers, the lower level is only used when you are using the explict definition option.

Related Items:

- Initial Conditions (UZ) *(V1 p. 288)*
- Geological Layers *(V1 p. 297)*
- Geological Lenses *(V1 p. 298)*
- Lenses *(V1 p. 55)*
- Water Quality **Sources** *(V1 p. 323)*

11.23.3 Horizontal Extent

Horizontal Extent	
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

Lenses

The horizontal extent is used to define the lateral extents of geologic lenses. The horizontal extents is usually a .shp file polygon, or a dfs2 grid file. In either case, the polygon name or the .dfs2 codes are ignored. Any cell within a polygon or with a grid code different than 0 is treated as part of the lense.

Water Quality Sources

If the source extent is Part Domain, then you can define a local extent for which the larger source file is applied. This allows you to define, for example, a distributed global source file - say of the field scale agricultural inputs in your catchment - and run individual water quality scenarios for each sub-catchment (modelled as an partial extent) to assess the subcatchment contributions to the global stream impact.

Related Items:

- Geological Lenses *(V1 p. 298)*
- Lenses *(V1 p. 55)*
- Water Quality **Sources** *(V1 p. 323)*

11.24 Water Quality Decay Processes

If WQ in unsaturated flow is to be calculated, then each decay process specified in the **Water Quality Sorption and Decay** *(V1 p. 212)* dialog will add one item to this data tree branch, with the name equal to that specified in the dialog. Each branch will include the first-order decay half life for the species specified in the decay process.

11.24.1 Half Life

Half Life	
Condition	when Water quality processes are selected in the Water Quality Simulation Specification dialogue AND at least one decay process is active
dialogue Type	Stationary Real Data
EUM Data Units	Half-life

The half life equals the length of time for half of the current solute to disappear. For rapidly decaying solutes a typical half life could be a few days to a few months. For slowly decaying species this could be a few years to many centuries. But, for long-lived radioactive elements the half life can be 10000 years or more.

Related Items:

- **Water Quality Sorption and Decay** *(V1 p. 212)*
- Solute Transport in Overland Flow *(V2 p. 275)*
- Solute Transport in the Unsaturated Zone *(V2 p. 270)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*

11.24.2 Macropore/Secondary Half Life

Macropore/Secondary Half Life	
Condition	when Water Quality processes are selected in the Water Quality Simulation Specification dialogue AND at least one decay process is active AND Full Macropore Flow selected in the main Unsaturated dialog OR Transfer to immobile water selected for SZ
dialogue Type	Stationary Real Data
EUM Data Units	Half-life

The half life equals the length of time for half of the current solute to disappear. For rapidly decaying solutes a typical half life could be a few days to a few months. For slowly decaying species this could be a few years to many centuries. But, for long-lived radioactive elements the half life can be 10000 years or more.

The decay half life in macropores may or may not be the same as in the bulk media. For example, water in the macropores may be more oxygenated.

Related Items:

- **Water Quality Sorption and Decay** *(V1 p. 212)*
- Solute Transport in Overland Flow *(V2 p. 275)*
- Solute Transport in the Unsaturated Zone *(V2 p. 270)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*

11.25 Water Quality Sorption processes

If WQ in unsaturated flow is to be calculated, then each sorption process specified in the **Water Quality Sorption and Decay** *(V1 p. 212)* dialog will add one item to this data tree branch, with the name equal to that specified in the dialog. Each branch will include the appropriate sorption coefficients for the selected sorption isotherm for the species specified in the sorption process.

11.25.1 Linear Kd

Linear Kd	
Condition	when water quality for Unsaturated or Saturated flow is selected in the Water Quality Simulation Specification dialogue and the Linear Isotherm is selected for the sorption process
dialogue Type	Stationary Real Data
EUM Data Units	Kd

The linear sorption isotherm is mathematically the simplest isotherm and can be described as a linear relationship between the amount of solute sorbed onto the soil material and the aqueous concentration of the solute:

$$c^* = K_d c \tag{11.9}$$

where K_d is known as the distribution coefficient.

The distribution coefficient is often related to the organic matter content of the soil by an experimentally determined parameter (K_{oc}) which can be used to calculate the K_d values.

$$K_d = f_{oc} \cdot K_{oc} \tag{11.10}$$

where f_{oc} is the fraction of organic carbon.

Commonly referred to is the retardation factor (R), which is the ratio between the average water flow velocity (v) and the average velocity of the solute plume ($_v c$). The retardation factor is calculated as

$$R = \frac{v}{v_c} = 1 + \frac{\rho_b}{\theta} \cdot K_d \tag{11.11}$$

Related Items:

- **Water Quality Sorption and Decay** *(V1 p. 212)*
- Solute Transport in the Unsaturated Zone *(V2 p. 270)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Equilibrium Sorption Isotherms *(V2 p. 282)*

11.25.2 Freundlich Kf and N

Freundlich Kf and N	
Condition	when water quality for Unsaturated or Saturated flow is selected in the Water Quality Simulation Specification dialogue and the Freundlich Isotherm is selected for the sorption process
dialogue Type	Stationary Real Data
EUM Data Units	Kd and Dimensionless exponent

The Freundlich sorption isotherm is a more general equilibrium isotherm, which can describe a non-linear relationship between the amount of solute sorbed onto the soil material and the aqueous concentration of the solute

$$c* = K_f \cdot c^N \tag{11.12}$$

where K_f and N are constants. The relationship between K and N is shown in Figure 17.1.

Related Items:

- **Water Quality Sorption and Decay** *(V1 p. 212)*
- Solute Transport in the Unsaturated Zone *(V2 p. 270)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Equilibrium Sorption Isotherms *(V2 p. 282)*

11.25.3 Langmuir alpha and beta

Langmuir alpha and beta	
Condition	when water quality for Unsaturated or Saturated flow is selected in the Water Quality Simulation Specification dialogue and the Langmuir Isotherm is selected for the sorption process
dialogue Type	Stationary Real Data
EUM Data Units	Kd and Concentration_4

Both the linear and the Freundlich isotherm suffer from the same fundamental problem. That is, there is no upper limit to the amount of solute that can be sorbed. In natural systems, there is a finite number of sorption sites on the soil material and, consequently, an upper limit on the amount of solute that can be sorbed. The Langmuir sorption isotherm takes this into account. When all sorption sites are filled, sorption ceases. The Langmuir isotherm is often given as

$$\frac{c}{c^*} = \frac{1}{\alpha\beta} + \frac{c}{\beta} \tag{11.13}$$

or

$$c^* = \frac{c\alpha\beta}{1 + \alpha c} \tag{11.14}$$

where α is a sorption constant related to the binding energy and β is the maximum amount of solute that can be absorbed by the soil material. The relationship between α and β is shown in Figure 17.2.

Related Items:

- **Water Quality Sorption and Decay** *(V1 p. 212)*
- Solute Transport in the Unsaturated Zone *(V2 p. 270)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Equilibrium Sorption Isotherms *(V2 p. 282)*

11.25.4 Kinetic Sorption/Desorption Rate

Kinetic Sorption Rate	
Condition	when water quality for Unsaturated or Saturated flow is selected in the Water Quality Simulation Specification dialogue AND at least one active Sorption process is using the Kinetic/Equilibrium isotherm option
dialogue Type	Stationary Real Data
EUM Data Units	1st order rate WQ model

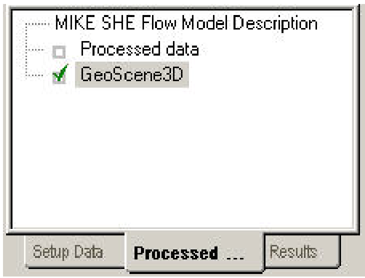
The kinetic sorption rate is the specified rate at which the solute is absorbed onto the soil matrix. The kinetic rate is used whenever a kinetic sorption isotherm is specified.

The kinetic de-sorption rate is the specified rate at which the solutes go back into solution from the soil matrix. The desorption rate is used whenever hysteresis is allowed for the kinetic sorption isotherm is used.

Related Items:

- **Water Quality Sorption and Decay** *(V1 p. 212)*
- Solute Transport in the Unsaturated Zone *(V2 p. 270)*
- Solute Transport in the Saturated Zone *(V2 p. 259)*
- Kinetic Sorption Isotherms *(V2 p. 284)*

12 Preprocessed Data Tab



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.

The Preprocessed Tab includes a data tree with two items:

- Processed Data *(V1 p. 343)*
- GeoScene3D *(V1 p. 347)*

12.1 Processed Data

On the main pre-processed dialog, there is a uneditable textbox 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 *(V1 p. 174)* 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.

12.1.1 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.

12.1.2 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.

12.1.3 MIKE Hydro River coupling

The coupling between MIKE Hydro River 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 Hydro River 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 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 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:

- **Rivers and Lakes** *(V1 p. 253)*
- Coupling of MIKE SHE and MIKE Hydro River *(V2 p. 115)*
- Channel Flow - Technical Reference *(V2 p. 109)*

12.1.4 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.

12.1.5 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 txt 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:

- Soil Profile Definitions *(V1 p. 278)*
- Unsaturated Zone *(V1 p. 273)*
- Column Classification Classification Type *(V1 p. 274)*
- Partial automatic classification *(V1 p. 283)*
- Specified classification *(V1 p. 284)*

12.1.6 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.

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 *(V1 p. 57)*.

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

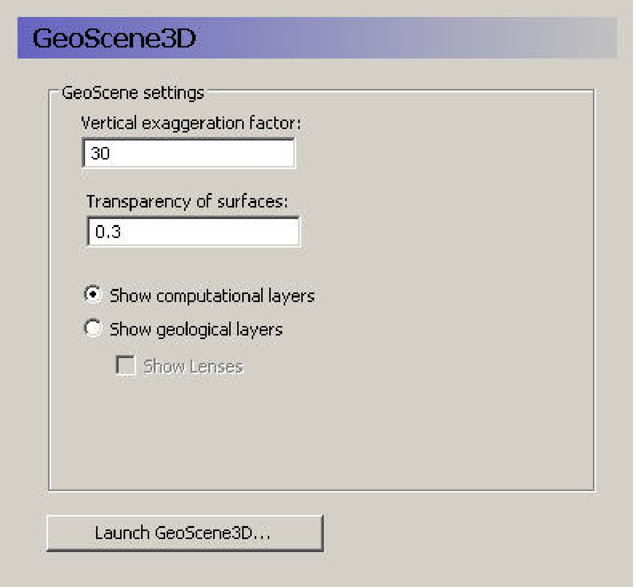
Drainage to local depressions and boundary - This grid displays all the cells that drain to local depressions or to the outer boundaries. All drainage from cells with the same negative value are drained to the cell with the corresponding positive code. If there is no corresponding positive code, then 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 *(V1 p. 57)*
- Drainage *(V1 p. 315)*
- Drain Level *(V1 p. 318)*
- Drain Time Constant *(V1 p. 319)*
- Drain Codes *(V1 p. 319)*
- Option Distribution *(V1 p. 321)*

12.2 GeoScene3D



GeoScene3D is a visualisation tool widely used in Denmark for displaying geologic data in a dynamic 3D environment. It has been adapted to read MKE SHE results. For more information on GeoScene3D, visit www.i-gis.dk (unfortunately mostly written in Danish).

The GeoScene3D dialogue appears when you have a licensed copy of GeoScene3D. The dialogue provides a link to the GeoScene3D program for viewing the preprocessed model setup. The preprocessed data does not include any transient or time data. So, GeoScene3D only shows stationary data.

Vertical exaggeration factor - In most models, the lateral extent is several orders of magnitude greater than the vertical extent. Thus, without a vertical exaggeration factor, the model view will be too thin to be useful. A vertical exaggeration of 10 to 30 will typically give you a good looking model that accentuates the vertical differences.

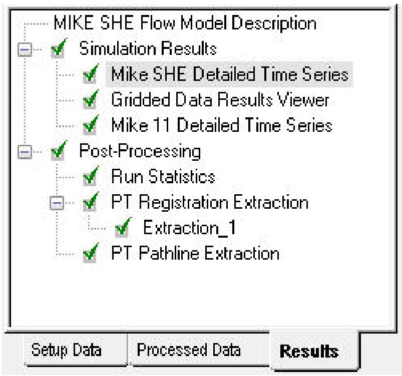
Transparency of surfaces - The transparency factor allows you to see through the model surfaces. This gives you a better feeling of what is happening below or above the surface that you are looking at.

Show computational layers - In this case, the model's computational layers will be visible and selectable in the GeoScene3D interface.

Show geological layers - In this case, the specified conceptual geologic layers will be visible and selected in the GeoScene3D interface.

Show lenses - If you select to show lenses, then they will be displayed on top of the geologic layers.

13 Results Tab



All the simulation results are collected in the Results tab.

The Simulation results are grouped under the Results of Simulation section. This includes Detailed time series output for both MIKE SHE and MIKE Hydro River, as well as Grid series output for MIKE SHE.

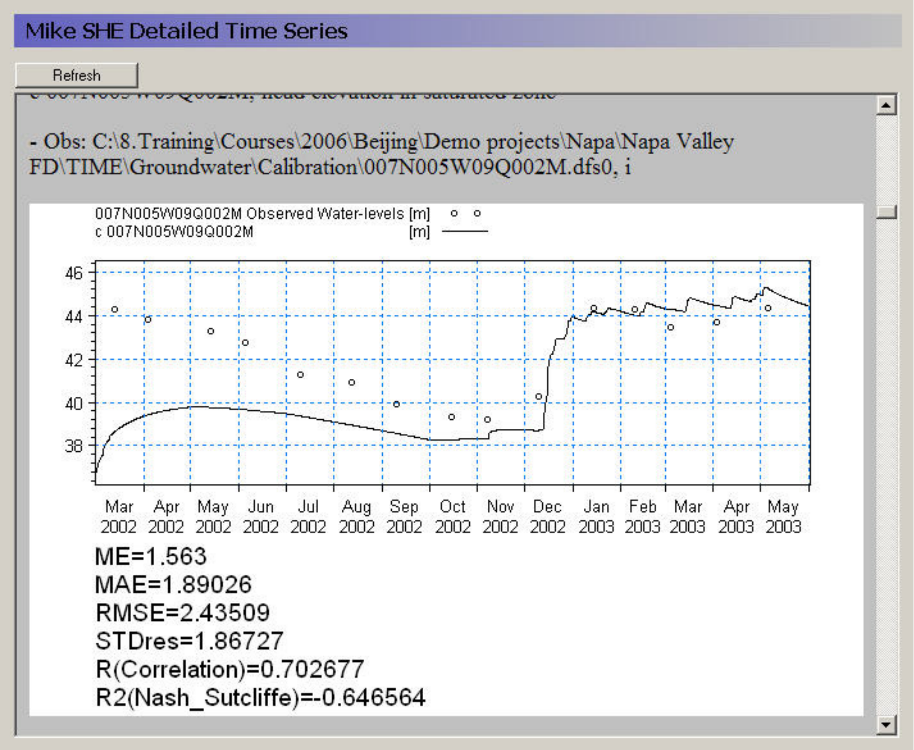
Under the Results Post-processing, you can find tools for extracting and manipulating the simulation results. This is where you will find the tools for extracting the shape file outputs for the Random walk particle tracking simulation. A Run Statistics tool is also available for helping you assimilate the calibration statistics for each of the detailed time series plots.

A link to the GeoScene3D program is available if you have a licensed installation for GeoScene3D. For more information on GeoScene3D, visit www.i-gis.dk (unfortunately mostly written in Danish).

13.1 Results of Simulation

The Results section includes the direct visualization and analysis of the simulation results

13.1.1 MIKE SHE Detailed Time Series



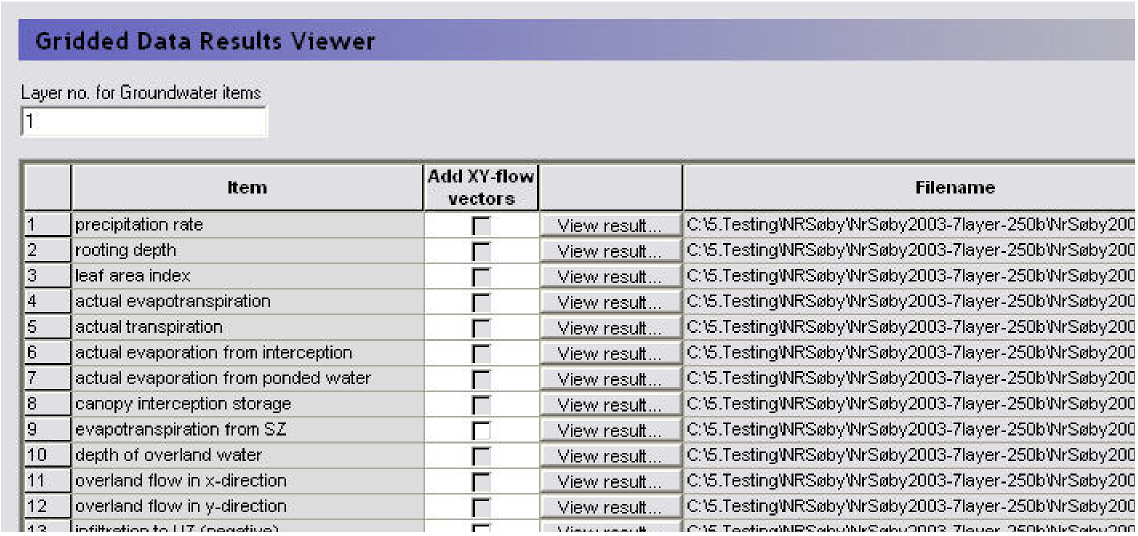
The MIKE SHE Detailed time series tab includes an HTML plot of each point selected in the Detailed time series output *(V1 p. 328)* dialogue. The HTML plots are updated during the simulation whenever you enter the plot, or click on the Refresh button.

When you have more than five detailed time series items, the plots are divided into seperate pages. In this case the main page is a list of the links to the individual plots and each html page includes five plots.

Related Items:

- Detailed time series output *(V1 p. 328)*
- Statistic Calculations *(V1 p. 355)*

13.1.2 Gridded Data Results Viewer



This table is a list of all gridded data saved during a MIKE SHE simulation. The items in the list originate from the list of items selected in the Grid series output *(V1 p. 333)* dialogue from the Setup tab.

View Result... - 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 river network is not transferred as an overlay.

Layer number for Groundwater Items - For 3D SZ data files, the layer number can be specified at the top of the table. However, the layer number can also be changed from within the Results Viewer. By default the top layer is displayed.

Add XY-flow 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.

file name - The file name column shows the name of the result file from which the gridded data will be extracted.

"The Result Viewer setup file already exists" 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 setup file is displayed in the title bar of the dialogue. Initially, the .rev file includes only the default view settings and the overlay information from MIKE

SHE. However, if you make changes to the view, such as changes the way contours are displayed, then when you close the view, you will be asked if you want to save your changes.

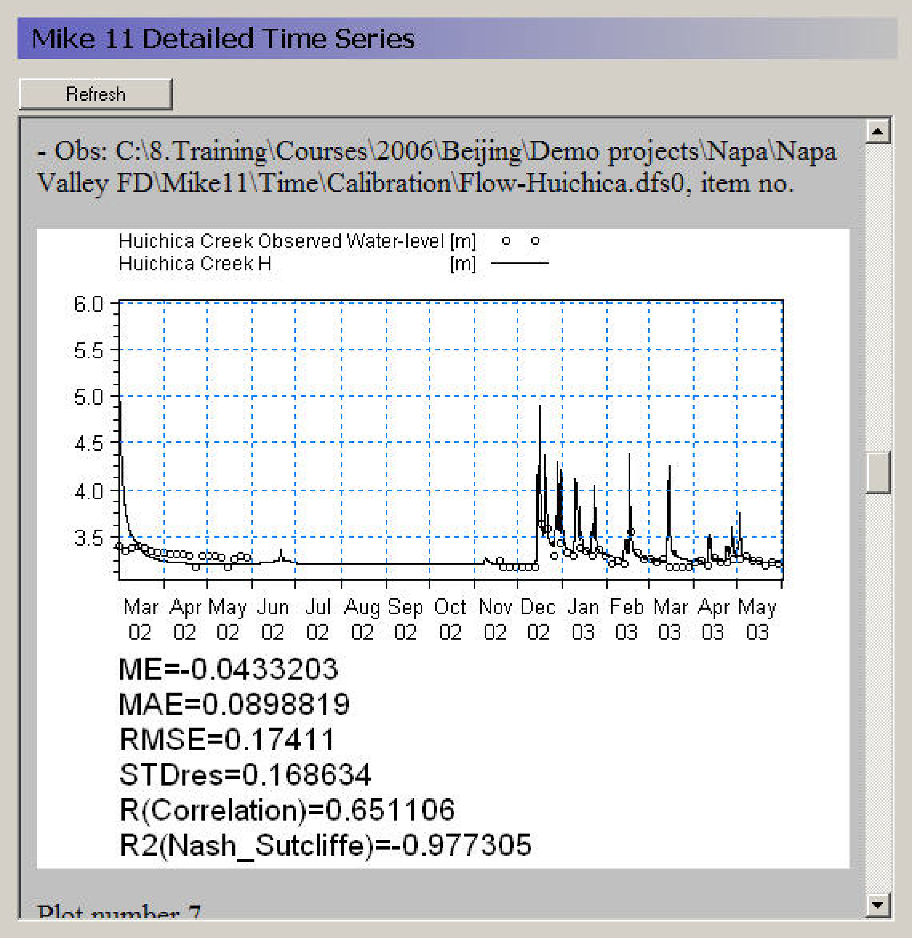
The next time you open the item in the table, you will be asked if you want to overwrite the existing .rev file. If you click on “Yes”, then a new .rev file will be created with the default values. If you click on “No”, then your previous settings will be re-loaded. This is a convenient way to set up the contouring, legend, etc., the way you want and then re-use the settings.

The .rev file can also be loaded directly in the Results Viewer by double clicking on the .rev file or loading the file into the MIKE Zero project explorer.

Related Items:

- Grid series output *(V1 p. 333)*
- The Results Viewer *(V1 p. 85)*

13.1.3 MIKE Hydro River Detailed Time Series



The MIKE Hydro River Detailed time series tab includes an HTML plot of each point selected in the Detailed MIKE Hydro River Output *(V1 p. 331)* dialogue. The HTML plots are updated during the simulation whenever you enter the view. Alternatively, you can click on the Refresh button to refresh the plot.

Note that the HTML plot is regenerated every time you enter the view. So, if you have a lot of plots and a long simulation, then the regeneration can take a long time.

When you have more than five detailed time series items, the plots are divided into seperate pages. In this case the main page is a list of the links to the individual plots and each html page includes five plots.

Related Items:

- Detailed MIKE Hydro River Output *(V1 p. 331)*

- Statistic Calculations *(V1 p. 355)*

13.2 Results Post-processing

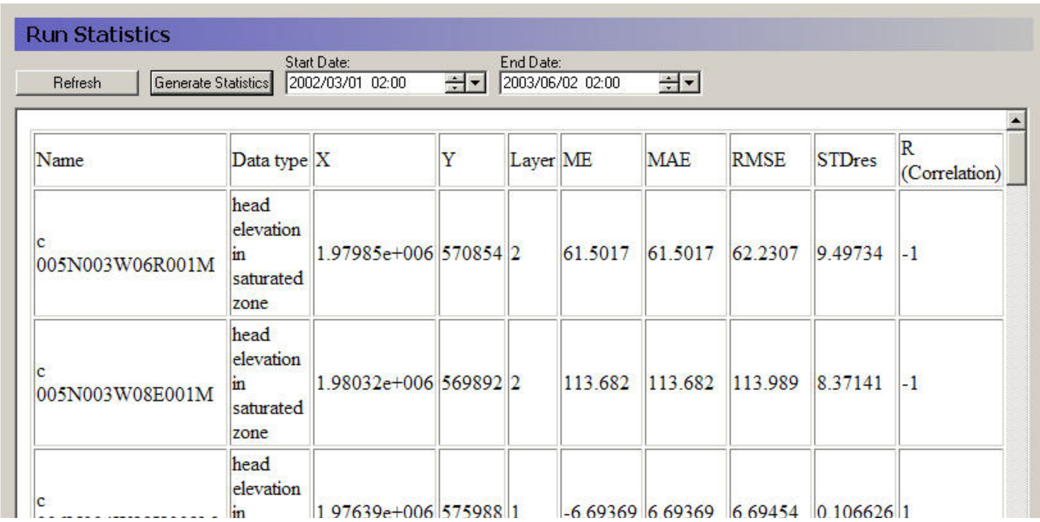
The post-processing section includes options for extracting the results from the various results files.

13.2.1 Run Statistics

Run statistics is a summary of the run statistics for the MIKE Hydro River and MIKE SHE detailed time series. The statistics are generated in HTML and shape file format. The calculations include all the detailed time series items that have observation data.

To calculate Run Statistics for a simulation, select the period you want to summarize, then click on the Generate Statistics button. For some simulations with long simulation periods and/or a lot of calibration data it can take several minutes to generate the run statistics.

The Run Statistics are displayed in HTML format on the Run Statistics page (see below).The HTML table can be copied and pasted directly into Microsoft Word or Excel.



Name	Data type	X	Y	Layer	ME	MAE	RMSE	STDres	R (Correlation)
c 005N003W06R001M	head elevation in saturated zone	1.97985e+006	570854	2	61.5017	61.5017	62.2307	9.49734	-1
c 005N003W08E001M	head elevation in saturated zone	1.98032e+006	569892	2	113.682	113.682	113.989	8.37141	-1
c [illegible]	head elevation in	1.97639e+006	575988	1	-6.69369	6.69369	6.69454	0.106626	1

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

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 SimulationName 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. This file must be created using standard ArcGIS methods.

Statistic Calculations

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 standard calibration statistics calculated based on the differences between the measured observations and the calculated values at the same location and time. Thus, the error, or residual, for an calculation - observation pair is

$$E_{i,t} = Obs_{i,t} - Calc_{i,t} \tag{13.1}$$

where $E_{i,t}$ is the difference between the observed and calculated values at location *i* and time *t*.

Mean (ME)

The mean error at location *i* where *n* observations exist is

$$ME_i = \overline{E}_i = \frac{\sum_t (E_{i,t})}{n} \tag{13.2}$$

Mean Absolute Error (MAE)

The mean of the absolute errors at location *i* where *n* observations exist is

$$MAE_i = \overline{|E_i|} = \frac{\sum_t |E_{i,t}|}{n} \tag{13.3}$$

Root Mean Square Error (RMSE)

The root mean square error at location *i* where *n* observations exist is

$$RMSE_i = \sqrt{\frac{\sum_t (E_{i,t})^2}{n}} \tag{13.4}$$

Standard Deviation of the Residuals (STDres)

The standard deviation of the residuals at location *i* where *n* observations exist is

$$STDres_i = \sqrt{\frac{\sum_t ((E_{i,t}) - \overline{E}_i)^2}{n}} \tag{13.5}$$

The standard deviation is a good measure to evaluate how well the dynamics of a certain observation are simulated.

Correlation Coefficient (R)

The correlation coefficient is a measure of the linear dependency between simulated and measured values. The closer the value is to 1.0, the better the match. The correlation coefficient at location *i* is

$$r_i = \frac{\sum_t (Calc_{i,t} - \overline{Calc}_{i,t}) \cdot (Obs_{i,t} - \overline{Obs}_{i,t})}{\sqrt{\sum_t (Calc_{i,t} - \overline{Calc}_{i,t})^2 \cdot \sum_t (Obs_{i,t} - \overline{Obs}_{i,t})^2}} \tag{13.6}$$

where $\overline{Obs_i}$ and $\overline{Calc_i}$ are the means of the observations and calculations at location *i* respectively.

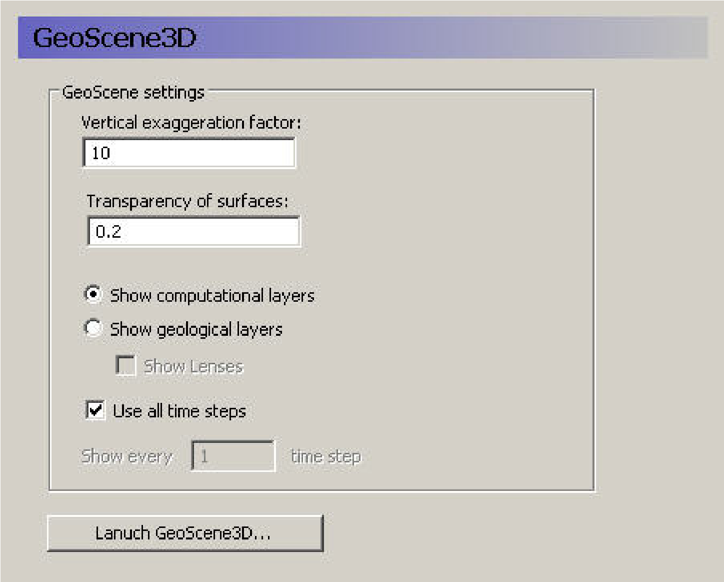
Nash Sutcliffe Correlation Coefficient (R2)

The Nash-Sutcliffe coefficient at location *i* where *n* observations exist is

$$R2 = 1 - \frac{\sum_t (E_{i,t})^2}{\sum_t (Obs_{i,t} - \overline{Obs}_i)^2} \tag{13.7}$$

where $\overline{Obs_i}$ is the mean of the observations at location *i*. R2 will be 1.0 if there is a perfect match.

13.2.2 GeoScene3D



GeoScene3D is a visualisation tool widely used in Denmark for displaying geologic data in a dynamic 3D environment. It has been adapted to read MKE SHE results. For more information on GeoScene3D, visit www.i-gis.dk (unfortunately mostly written in Danish).

The GeoScene3D dialogue appears when you have a licensed copy of GeoScene3D. The dialogue provides a link to the GeoScene3D program for viewing the model results.

Vertical exaggeration factor - In most models, the lateral extent is several orders of magnitude greater than the vertical extent. Thus, without a vertical exaggeration factor, the model view will be too thin to be useful. A vertical exaggeration of 10 to 30 will typically give you a good looking model that accentuates the vertical differences.

Transparency of surfaces - The transparency factor allows you to see through the model surfaces. This gives you a better feeling of what is happening below or above the surface that you are looking at.

Show computational layers - In this case, the model's computational layers will be visible and selectable in the GeoScene3D interface.

Show geological layers - In this case, the specified conceptual geologic layers will be visible and selected in the GeoScene3D interface.

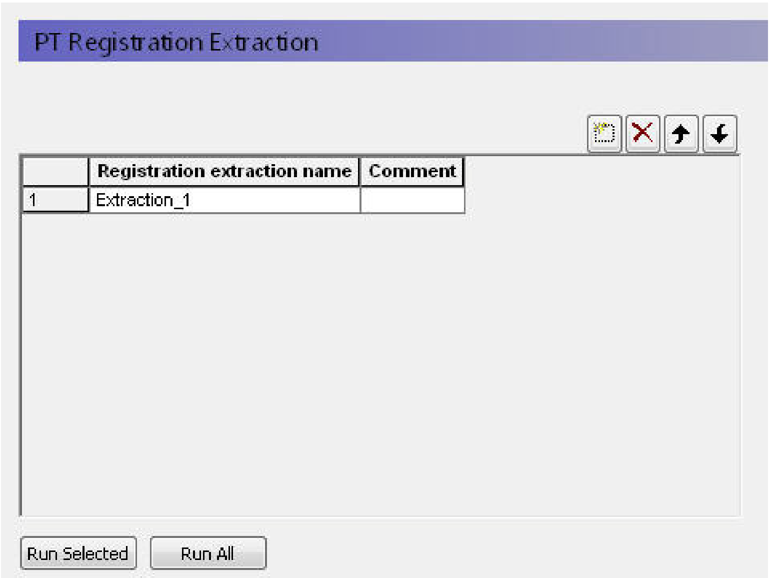
Show lenses - If you select to show lenses, then they will be displayed on top of the geologic layers.

Use all time steps - The default action is to display time varying data with all of the time steps

Show every __ time step - However, if your data files are visualized too slowly, then you can reduce the number of time steps being shown.

13.2.3 PT Registration Extraction

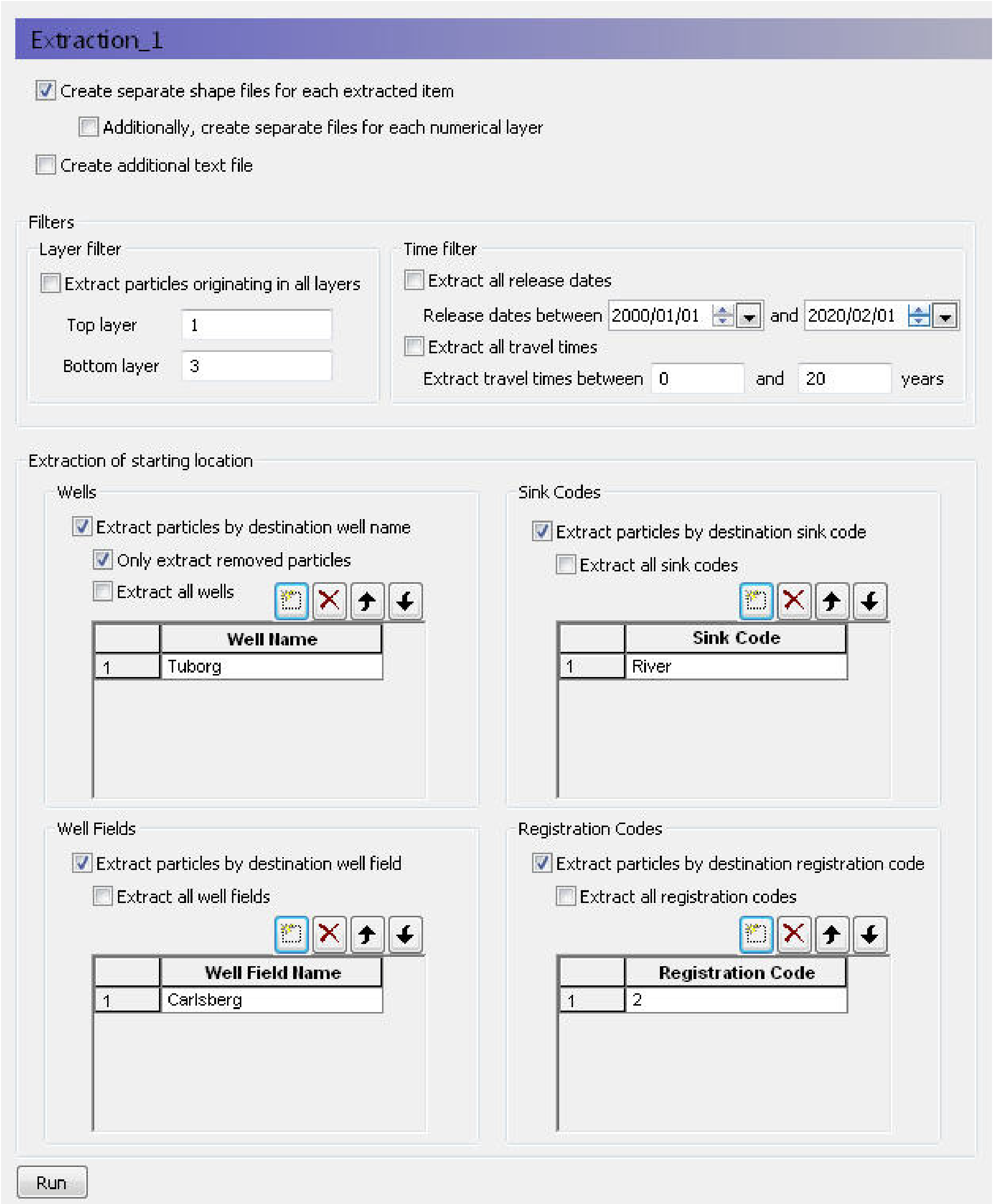
The SZ random walk particle tracking module is very useful for finding out the source of water to an area of interest, for example a well field or a wetland. A property of each particle is its starting location. If the particle is removed from the model or enters a specified area, then the particle is tagged, or 'registered'. At the end of the simulation, you can then sift through all of the particles and find the starting location for all the ones that you want. With this information, you can build up a dyanamic picture of the source of water entering an area or leaving the model through a specific boundary. The extraction itself will compile a point shape file with the starting locations of all the particles satisfying the specified criteria.



In the top level of this data tree item, you can specify any number of different particle extractions. These will be saved from simulation to simulation, allowing you to re-run the extraction for different scenarios.

The buttons at the bottom, allow you to run the selected extraction or all of the specfied extractions.

Extraction properties



The PT module can generate thousands of particles during a simulation. The extraction definition allows you to specify the criteria finding just the particles that you are interested in.

Create separate shape files for each extracted item - If this option is selected, then you will get a seperate shape file for Wells, Well Fields, Sink Codes and Registration codes.

Additionally, create separate files for each extracted layer - If you have selected the option above, then you also have the option to divide the shape files into one for each numerical layer.

Create additional text file -

Filters

For each extraction, the search for relevant particles can be restricted to specific layers and time periods.

Extraction of particle starting locations

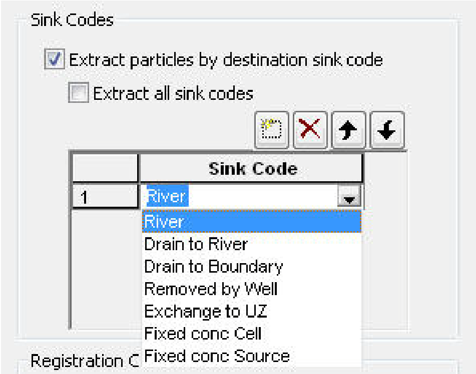
The extraction process finds the starting location of all particles that were registered based on the criteria you specify. By default, all the extraction options are turned off. When you active an extraction criteria, by default all particles that meet the criteria will be extracted. For example, particles for all of the sink codes will be extracted. Each different sink code will have a seperate attribute in the shape file. If you want to refine the search criteria, you can optionally specify the criteria in the table.

Note: an individual particle can be registered multiple times. For example, it might pass through a registration zone, then a well cell without being extracted, and then discharge into a river.

Wells - By default, all particles removed by a well are registered. However, particles that enter a cell containing a well are also registered, even if they are not actually removed by the well. For example, they could flow through the cell or not make it to the well before the simulation ends. There is an option in this section to separate out just the particles that were actually removed by the well. The table allows you to specify individual well names as specified in the Well editor *(V1 p. 367)*.

Well Fields - Each well in the Well Editor can be optionally assigned to a Well Field in the Well Locations *(V1 p. 368)* table. Using the Well Field option, you can group wells together and calculate the influence area of a the entire group of wells.

Sink Codes - Particles can be removed by various sinks. The sinks can be water sinks, such as a river, or they can be concentration sinks.



Drain to River/Boundary - Drain sinks refer to saturated zone drainage and are separated by the discharge point, that is SZ drainage flowing to a river or SZ drainage flowing to a boundary node.

Removed by Well - The particle is removed by a well. Note that, particles that may pass through a cell containing a well.

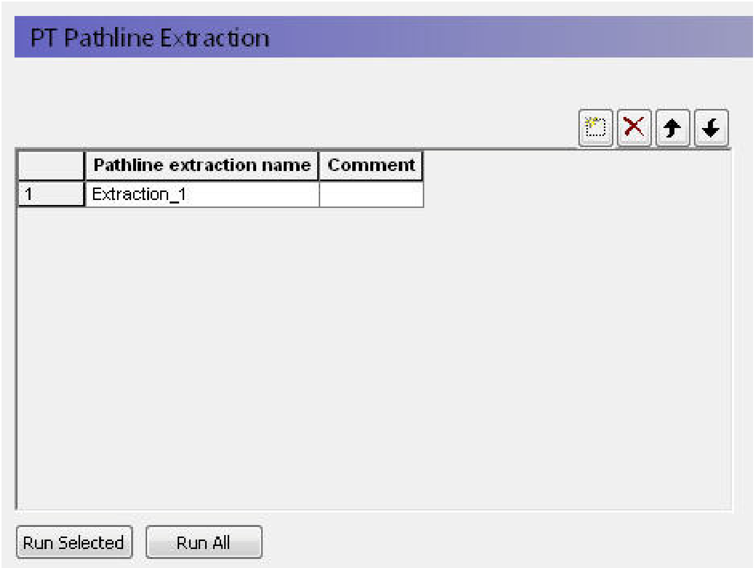
Exchange to UZ - This refers to particles that are removed by capillary flow from SZ into the UZ, or particles that become trapped in the UZ when the water table falls. However, a particle may become trapped and then later reactivated when the water table rises. Each time it is trapped it will be re-registered.

Fixed Conc Cell/Source - Fixed concentration is less than the incoming concentration. Note that the concentration is determined by the particle mass defined in the Particle Tracking Specification *(V1 p. 200)* dialog.

Registration Codes - The registration codes are user defined zones that create a particle registration whenever a particle passes **into** the zone. A particle can be registered multiple times, if it passes in and out of the zone. The registration zones are defined in the PT Registration Codes/Lenses *(V1 p. 314)* dialog. There is no cross-referencing to the list of Registration Codes defined in the simulation, so you have to be careful to specify the code that you want.

13.2.4 PT Pathline Extraction

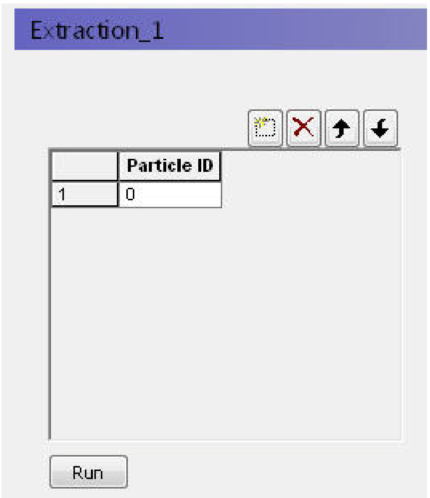
It is too cumbersome to extract and plot the pathlines for all of the thousands of particles that can be generated during a PT simulation. The PT Pathline Extraction utility allows you to extact the pathlines for specific particles if you have saved the intermediate locations in the Storing of Results *(V1 p. 325)* dialog.



In the top level of this data tree item, you can specify any number of different pathline extractions. These will be saved from simulation to simulation, allowing you to re-run the extraction for different scenarios.

The buttons at the bottom, allow you to run the selected extraction or all of the specfied extractions.

Extraction Properties



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.

MIKE SHE EDITORS

14 Well editor

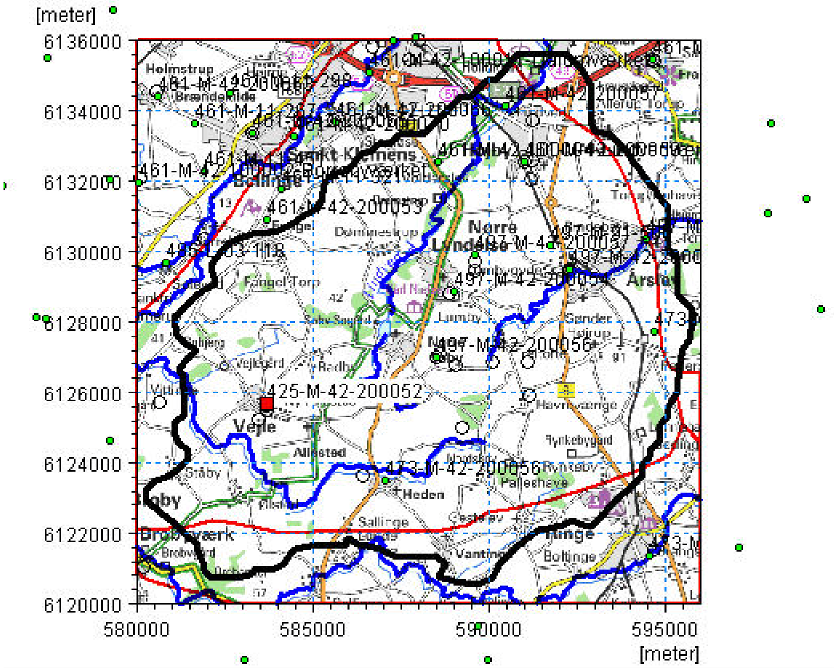
The Well Editor is the MIKE SHE tool for managing pumping wells. The data file for the well editor is independent of the numerical model. That is, the well file often contains all of the wells in the entire model region, including wells outside the model area. The preprocessing takes care of including only the relevant wells in the numerical model.

The dialogue for the Well Editor is divided into

- an Interactive Map *(V1 p. 367)* of well locations,
- a table of Well Locations *(V1 p. 368)*,
- a table of Well Filters *(V1 p. 369)* for the current well, and
- a schematic Layers Display *(V1 p. 370)* showing the relationship between the well screen, the current geologic model and the numerical layers.

The most common way to add data to the Well database is via ASCII import: Importing Data *(V1 p. 370)*.

14.1 Interactive Map



The interactive map displays all of the wells in the well file. Clicking on individual wells will select the corresponding item in the table of well locations. Similarly, selecting an item from the list will change the icon of the well on the map to a red square.

The overlays are automatically carried over from the model Setup Tab. You can't add or modify overlays directly in the Well Editor. This must be down from the Setup Tab

Right clicking on the map, allows you to control the zoom and a number of other functions:

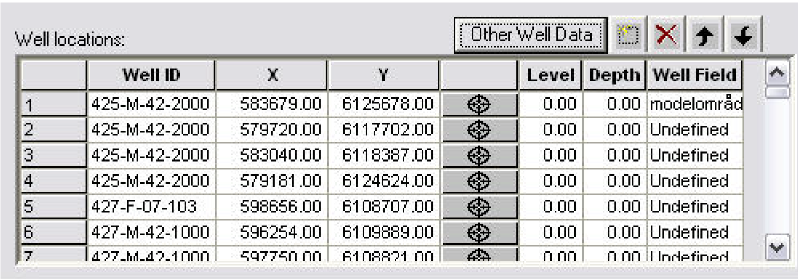
Grid - turns on/off a faint coordinate grid that changes with the zoom factor

Set new area coordinates - allows you to change the displayed area of the map

Text - turns on/off the display of the Well ID labels for the wells

Export Graphic - allows you to save the view to the clipboard, or a .bmp or wmf graphic file for importing into MSWord, for example.

14.2 Well Locations



	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

Well_ID - This is the user specified name of the well. The Well_ID cannot contain any spaces.

X, Y - These are the X and Y map coordinates of the well.

EUM Data Units: Item geometry 2-dimensional

Level - The Level defines the maximum elevation shown on the profile view of the geologic layers, calculation layers, and screened intervals for the well. The topography is shown if the Level is less than the topography.

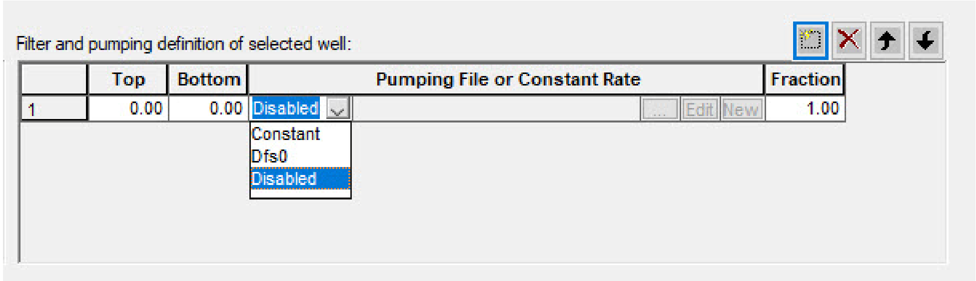
EUM Data Units: Elevation

Depth - The Depth is defined from the Level. It defines the maximum depth shown on the graphical view displaying the profile view of the geologic layers, calculation layers, and screened intervals for the well. The bottom of the geologic layers is shown if the Level minus the Depth is higher than the bottom of the geologic layers.

EUM Data Units: Depth below ground

Well Field - The Well Field item is used for filtering the displayed boreholes. The Mask item in the top menu bar uses the Well Field for it selection criteria. The Well Field item is also used to defined registration zones in the Random Walk Particle Tracking (PT) module. For more information, see Particle Tracking (PT) *(V1 p. 705)*, and Particle Tracking-Reference *(V2 p. 319)*.

14.3 Well Filters



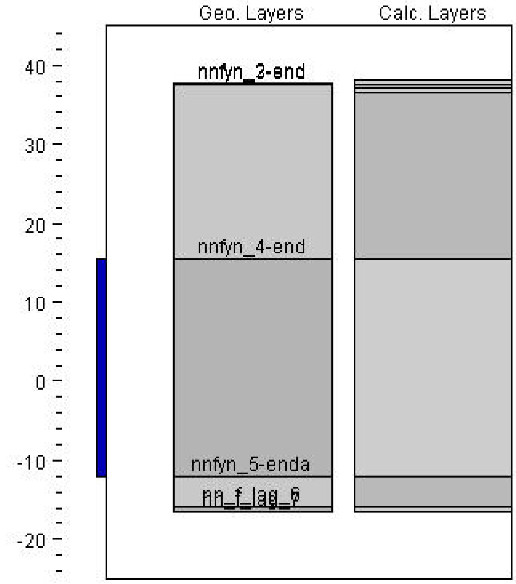
Top - This is the elevation of the top of the screen or open hole interval for the well (in the same units (ft, m, etc.) as specified in the EUM Database for item geometry 2-dimensional).

Bottom - The elevation of the bottom of the screen or open hole interval for the well (in same units (ft, m, etc.) as specified in the EUM Database for item geometry 2-dimensional).

Pumping File or Constant Rate - You have three choices here. Either a constant value for the pumping rate, name of the .dfs0 file with groundwater pumping data for the well (When using the Browse button, [...], to select the file, you will be given the option of specifying the Item number in the .dfs0 file.), or to disable pumping at this well.

Fraction - This is a multiplier for the groundwater pumping rate specified in the DFS0 File.

14.4 Layers Display



The Layer display section displays the location of all well screens assigned to the well. The Geo Layers column displays the geologic layers assigned in the Setup Tab for the well, and the Calc Layers is the numerical layers for the column of model cells in which the well is located.

Both the Geo Layers and the Calc Layers items require that the model has been successfully pre-processed. If you have not pre-processed the model yet, or if during the preprocessing an error occurred, then a warning message dialogue may appear saying that the model must be pre-processed first. If this happens, the Well Editor will function normally, but the Geo and Calc layers may not be shown.

14.5 Importing Data

In the top menu bar there is an Import menu, that allows you to import the following well data.

Zeus data (Adm and Lit) - The Zeus data is a specialized data format from the Geological Survey of Denmark and Greenland.

Tab delimited ASCII files - This is the most common way of imported well data.

14.5.1 Importing TAB delimited text file

The most common file format to import is a TAB delimited ASCII file, typically generated from Excel or a database program.

The only restriction on the import is that only one screened interval can be specified for each well. Addition screened intervals must be specified manually after the wells have been imported.

Below is the format that each line in the ASCII file must follow:

```
Well_ID>X>Y>Level>Depth>Well_Field>Top>Bottom>Frac-
tion>dfs0_File>dfs0_item
```

A simple example with three groundwater wells is given below.

```
CW1 7780.00 20331.00 0.00 0.00 CW   0. -60. 1.
.\Time\ClassPumpage.dfs0 1

CW2 8000.00 19000.00 0.00 0.00 CW -10. -50. 0.5
.\Time\ClassPumpage.dfs0 3

CW3 7600.00 21300.00 0.00 0.00 CW  10. -60. 1.
.\Time\ClassPumpage.dfs0 2
```

15 Vegetation Properties Editor

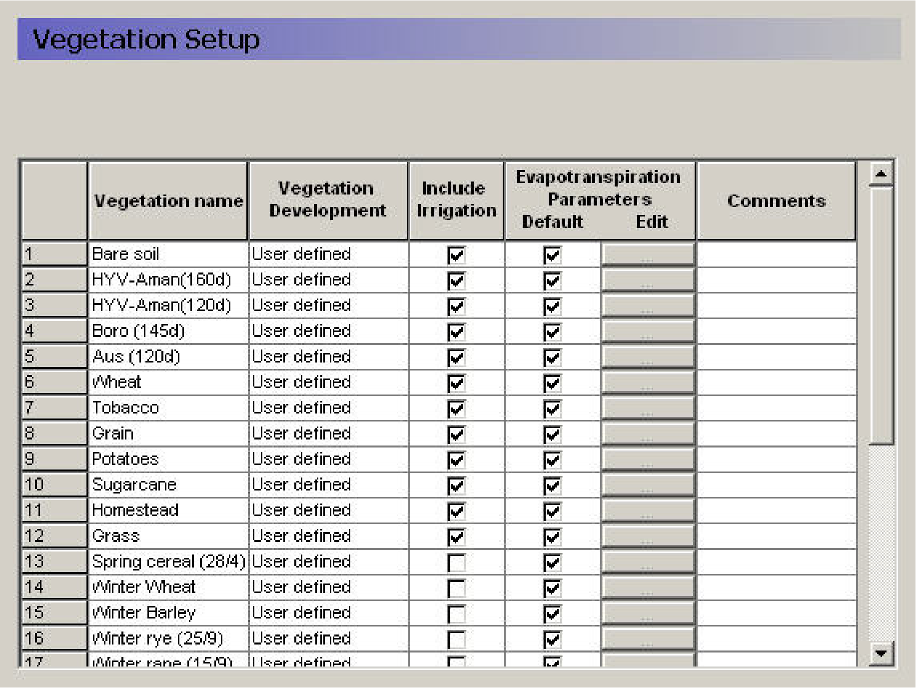
The vegetation editor is used to specify vegetation data for the evapotranspiration and irrigation management modules. The vegetation database contains the time varying vegetation characteristics for each type of vegetation that is specified in the model domain.

The vegetation database is optional and can be used only when the Evapotranspiration (ET) and Unsaturated Zone components are included in the model.

15.1 Vegetation Database Items

The vegetation database is organized around a data tree similar to the setup editor. To create a vegetation type in the database and populate with the corresponding data, simply add a vegetation item in the main dialogue and then fill out the tables in the dialogues that appear in the data tree under the new vegetation item.

15.1.1 Specifying Vegetation Properties in a Database



	Vegetation name	Vegetation Development	Include Irrigation	Evapotranspiration Parameters Default	Edit	Comments
1	Bare soil	User defined	☑	☑	...	
2	HYV-Aman(160d)	User defined	☑	☑	...	
3	HYV-Aman(120d)	User defined	☑	☑	...	
4	Boro (145d)	User defined	☑	☑	...	
5	Aus (120d)	User defined	☑	☑	...	
6	Wheat	User defined	☑	☑	...	
7	Tobacco	User defined	☑	☑	...	
8	Grain	User defined	☑	☑	...	
9	Potatoes	User defined	☑	☑	...	
10	Sugarcane	User defined	☑	☑	...	
11	Homestead	User defined	☑	☑	...	
12	Grass	User defined	☑	☑	...	
13	Spring cereal (28/4)	User defined	☐	☑	...	
14	Winter Wheat	User defined	☐	☑	...	
15	Winter Barley	User defined	☐	☑	...	
16	Winter rye (25/9)	User defined	☐	☑	...	
17	Winter rape (15/9)	User defined	☐	☑	...	

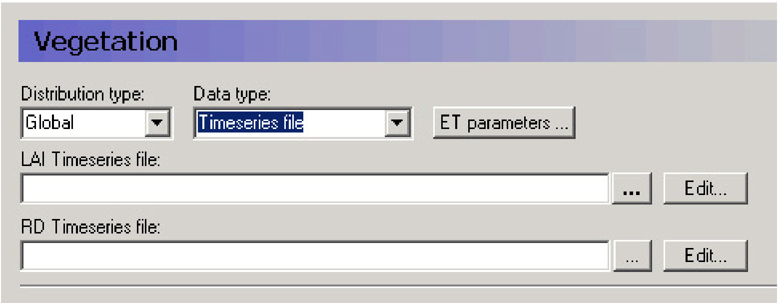
The vegetation database is populated with data in a number of steps. Firstly, all the vegetation types to be included in the database are entered in a table in the Vegetation Setup Menu. The data needed are:

- Vegetation name;

- Vegetation Development;
- Include Irrigation;
- Evaporation parameters.

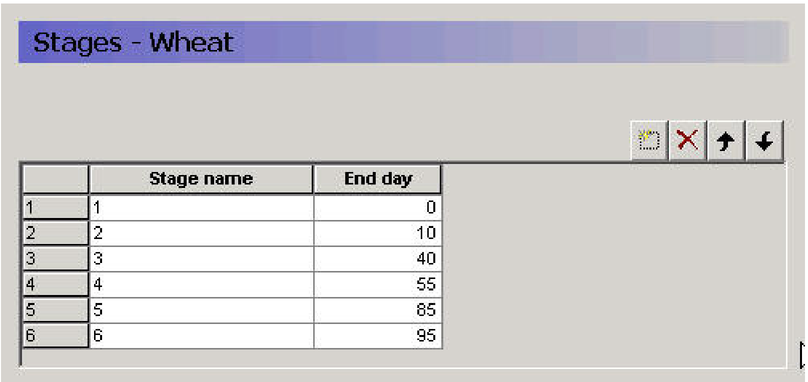
If the Irrigation module is used in the model you, may chose to use the irrigation demand values from the vegetation file. In this case, you should select the 'Include Irrigation' option and then specify the irrigation demand parameters by vegetation type.

You can also specify specific Evaporation Parameters for each vegetation type. If the 'Default' value is checked, then the global values defined in the main vegetation dialogue (shown below) will be used:



If the default value is unchecked, then you can specify the ET parameters by vegetation type.

15.1.2 Vegetation stages



The first thing to do, is to specify the standard vegetation stages for each crop or land use type in the database. These temporal variations in vegetation characteristics can normally be described by a number of characteristic stages of specific length. The changes are defined as a set of linear changes between two consecutive crop stages. Three parameters describe the stage, the leaf area index (LAI), the root depth (RD), and the crop coefficient K_c.

IMPORTANT NOTE: The values of LAI, RD and K_c are defined as Instantaneous parameters. Therefore, it is critical that you start with a Stage at 0 days. Otherwise, your initial value for LAI, RD or K_c will be zero.

The day number indicates the cumulative days from crop establishment (e.g. sowing) to the end of the specific crop stage.

If in the Vegetation *(V1 p. 236)* dialogue, subsequent start dates overlap with the development cycle, a warning will be issued in the log file that says the crop development was not over yet before the new crop was started. MIKE SHE will then start a new crop cycle at the new start date.

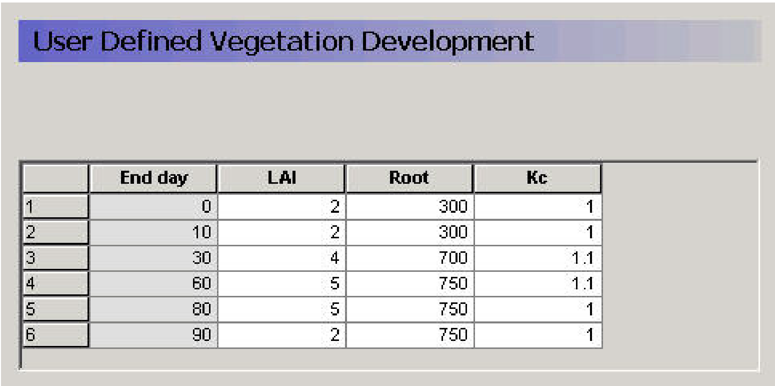
15.1.3 Evapotranspiration Parameters

The parameters used in the evapotranspiration calculations can be divided into three groups, which regulate interception, soil evaporation and plant transpiration, respectively.

The d

Note: The C1, C2, C3 ,and AROOT parameters are only used in the Richards Equation and Gravity Flow methods, and not in the 2-Layer UZ method.

15.1.4 Vegetation Development Table


User Defined Vegetation Development

	End day	LAI	Root	Kc
1	0	2	300	1
2	10	2	300	1
3	30	4	700	1.1
4	60	5	750	1.1
5	80	5	750	1
6	90	2	750	1

For each crop stage, three vegetation parameters need to be specified:

- **LAI** - The Leaf Area Index, which is the (Area of leaves)/(Area of the ground), can vary between 0 and 7 depending of the vegetation type;
- **Root** - The Rooting Depth of the crop. It will normally vary over the season. Consideration about the soil type should be taken because some crops may develop different root distribution depending on the soil characteristics;
- **Kc** - The crop coefficient.

The leaf area index and the root depth should be specified at the end of each crop stage. The development of LAI and root depth between the specified values are then interpolated linearly by the model. In addition to these parameters, it is often necessary to supply the crop coefficient (**Kc**), which is used to adjust the reference evapotranspiration relative to the actual evapotranspiration of the specific crop.

By the FAO definition, the reference evapotranspiration represents the potential evapotranspiration for a 8-15 cm high reference grass plane with ample water supply. Most farm crops may differ from this in two ways:

- In the early crop stages, where LAI of the farm crop is lower than the LAI of the reference grass crop, the evapotranspiration of the farm crop is less then the calculated reference evapotranspiration. This is accounted for in the Kristensen & Jensen ET calculation, since a crop LAI is used as input. **Therefore, for most field crops it is therefore not necessary to specify Kc values below 1 in the early crop stages.**
- In the crop mid-season the opposite situation may occur where crop potential evapotranspiration is larger than the calculated reference evapotranspiration of the reference grass crop. This is not handled in the ET calculations, and Kc values above 1 may therefore be relevant for some crops in the mid-season during the period where crop leaf area index is at its maximum.

A Kc value of 1 means that the maximum evapotranspiration rate will equal the reference evapotranspiration rate.

If pan evaporation data are used in place of reference evapotranspiration data in the model input, it often necessary to apply site specific pan coefficients to convert the pan evaporation to reference evapotranspiration. Pan coefficients are normally in the range 0.5 – 0.85.

15.1.5 Irrigation Parameters

In the irrigation module the amount of irrigation applied can be driven by the amount of water demanded by the crop. That is, in drier periods more irrigation water is required, so more irrigation is applied.

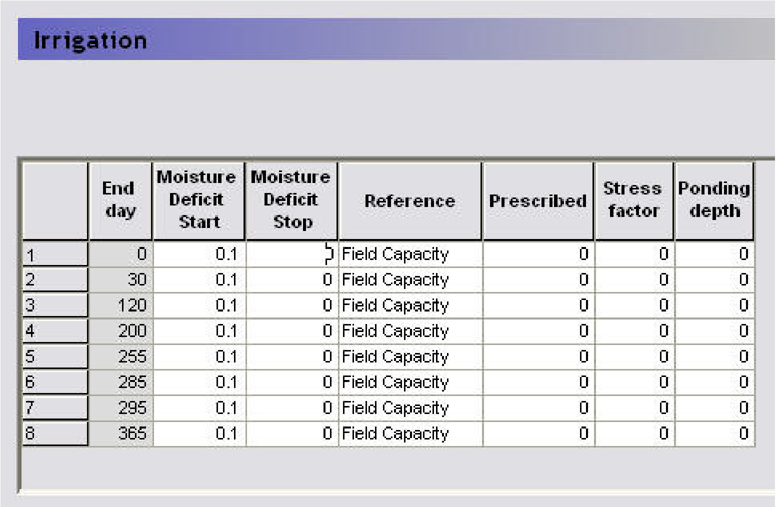
In the Irrigation demand dialogue, if you specify that the Vegetation Property file should be used for the demand calculations, then the demand values will be read from the vegetation file specified in the Vegetation *(V1 p. 236)* dialogue.

Further, in the irrigation module, you specify the type of demand calculation:

- User specified
- Maximum allowed deficit
- Crop stress factor

- Ponding depth

In the Irrigation dialogue of the Vegetation properties file,


Irrigation

	End day	Moisture Deficit Start	Moisture Deficit Stop	Reference	Prescribed	Stress factor	Ponding depth
1	0	0.1	0	Field Capacity	0	0	0
2	30	0.1	0	Field Capacity	0	0	0
3	120	0.1	0	Field Capacity	0	0	0
4	200	0.1	0	Field Capacity	0	0	0
5	255	0.1	0	Field Capacity	0	0	0
6	285	0.1	0	Field Capacity	0	0	0
7	295	0.1	0	Field Capacity	0	0	0
8	365	0.1	0	Field Capacity	0	0	0

you must specify a value for each of these demand types. Although, those that will not by used, may be left at the default values.

Maximum allowed deficit - If irrigation is handled automatically based on the actual moisture content in the soil, the soil moisture deficits are the deficits at which irrigation is going to start and stop. The soil moisture deficit is defined relative to the plant available water content in the root zone, which is the difference between a reference moisture content and the moisture content at wilting point. If, for example, the reference moisture content is the moisture content at field capacity and irrigation should start when 60 % of the available water in the root zone is used and cease when field capacity is reached, the value in the start column should be 0.6, the value in the stop column should be 0 and the reference input should be “field capacity”.

User specified - Alternatively, the irrigation amount applied in each crop stage can simply be prescribed.

Crop stress factor - The crop stress factor is the minimum allowed fraction of the reference ET that the actual ET is allowed to drop to before irrigation starts . That is, the minimum allowed (Actual ET)/(Reference ET) relationship This should be a value between 1 and 0.

Ponding depth - When using this option, the demand will be equal to the difference between the actual ponding depth and specified ponding depth. The option is typically used for modelling irrigation of paddy rice.

Thus:

- **Moisture Deficit Start** - the maximum allowable moisture deficit below the specified reference level. Irrigation will start at this level.
- **Moisture Deficit Stop** - This is the minimum allowable moisture deficit below the specified reference level. If irrigation takes place it will stop at this level;
- **Reference** - The **reference** moisture content. It can be chosen as either saturation or field capacity.
- **Prescribed** - This is the value used for the irrigation demand when 'User specified' is chosen in the Irrigation Demand *(V1 p. 250)* dialogue.
- **Stress Factor** - This is the minimum allowed (Actual ET)/(Reference ET) relationship This should be a value between 1 and 0.
- **Ponding Depth** - Irrigation will stop when the ponding depth reaches this value.

15.2 Example database

There is an example database (MIKE_SHE_vege.ETV) in the installed examples.

As the growing season of a given crop differs significantly depending on climatic region, you may have to adjust the vegetation development to local conditions.

16 UZ Soil Properties Editor

To solve Richards equation two important hydraulic functions are required for all soil types which characterise the individual soil profiles within the model area:

- the Soil Moisture Retention Curve and
- the Hydraulic Conductivity Function.

This information, along with the following parameters, is stored in the soil property database:

- soil moisture at saturation (θs) [-]
- soil moisture at effective saturation (θeff) [-]
- capillary pressure at field capacity (pF_{fc})
- capillary pressure at wilting point (pF_{w})
- residual soil moisture content (θr) [-]
- saturated hydraulic conductivity (K_s)

The soil moisture at effective saturation θeff is the maximum achievable soil moisture content.

16.1 Definitions of soil properties

pF scale

pF is a log scale for representing soil matric potential. Thus,

$$pF = \log10(-100\psi) \tag{16.1}$$

where ψ is the matric potential in metres of water. Notice that ψ is always negative under unsaturated conditions.

For example, if your measured wilting point is -15 bar (-152.96 m water), then the pF value is 4.2.

Saturated moisture content - θ_s

This is the maximum water content of the soil, which is equal to the porosity. A typical saturated moisture content is about 0.25 to 0.3.

Effective saturation

This is the maximum water content achieved under field conditions, which is usually slightly less than the saturated moisture content because of entraped air.

Field capacity - θ_{fc}

This is the water content of a free draining soil. In other words, it is the water content in the soil 2-3days after a heavy soaking rain. A typical field capacity is about 0.1 to 0.2.

pFfc - This is the suction pression of a soil when it is at field capacity. The pF_{fc} (field capacity) is used as the initial condition in the unsaturated flow module. A typical value is about 2.0.

Wilting point - θ_{wp}

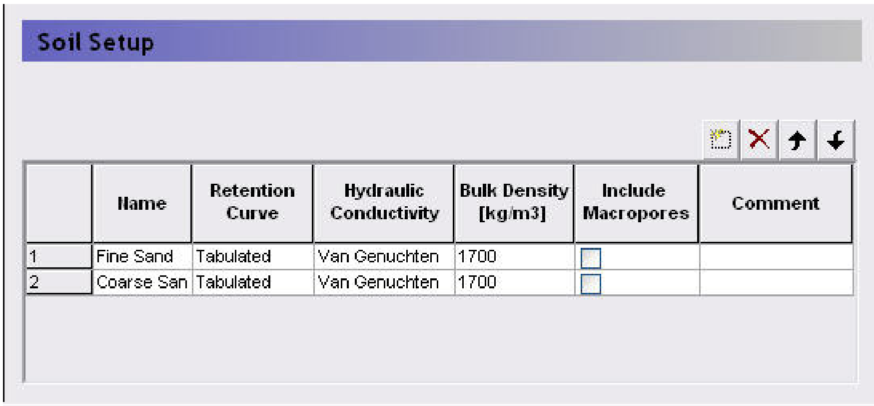
This is the water content when plants can no longer remove water from the soils and start to wilt. It is the minimum water content of the soil in the absence of evaporation. A typical value is about 0.01 to 0.05

pFw - This is the suction pression of the soil when it is at the wilting point. The pF_w (wilting) is typically about 4.2.

Residual moisture content - θ_r

This is the minimum water content at very high suction pressures. A typical residual moisture content is on the order of 10^{-3}.

16.2 Soils Database


Soil Setup

	Name	Retention Curve	Hydraulic Conductivity	Bulk Density [kg/m3]	Include Macropores	Comment
1	Fine Sand	Tabulated	Van Genuchten	1700	☐	
2	Coarse San	Tabulated	Van Genuchten	1700	☐	

In the main soils database dialogue, you can add any number of soils. MIKE SHE only reads information on the soils being used, so there is no penalty for having large databases that include soils used in different projects.

Name - The Name field is used to identify the soil when loading the soil information in MIKE SHE's UZ Soil Profile Definitions *(V1 p. 278)* dialogue.

Retention Curve and Hydraulic Conductivity - The Soil Moisture Retention Curve *(V1 p. 381)* and the Hydraulic Conductivity Function *(V1 p. 384)* are described in the next sections.

Bulk Density - The bulk density is used for calculating sorption during water quality simulations.

Include Macropores - If Macropore Flow *(V1 p. 275)* has been specified, then the macropore data will be required. If this item is checked, then an additional property page will be added to the data tree - Macropore Parameters *(V1 p. 387)*.

Comment - This field can be used to provide meta data for the soil, such as the literature source for the values, or the project that it was last used in.

Curves for the calculated Retention and Conductivity are shown for each defined soil.

16.3 Soil Moisture Retention Curve

The relationship between the water content, θ, and the matric potential, ψ, is known as the soil moisture retention curve, which is basically a function of the texture and structure of the soil. The amount and type of organic material may also have an influence on the relationship. Characteristically, the pressure head decreases rapidly as the moisture content decreases. Hysteresis is also common, that is the relationship between θ and ψ is not unique, but depends on whether the moisture content is increasing or decreasing. MIKE SHE allows for any shape of the soil moisture retention curve, but does not take hysteresis into account (i.e. a unique relation between θ and ψ is assumed).

Typically, the soil moisture curve is measured in a laboratory or assumed based on typical values for similar soils. If laboratory data is available, the measured θ-ψ values can be input directly into MIKE SHE as tabular data. Intermediate values are then calculated by MIKE SHE, using a cubic spline method, and stored internally in the code. Alternatively, the measured values can be fitted to commonly used functional relationships. The appropriate function parameters can be input directly or more refined tabular data may be generated externally to MIKE SHE (e.g. in MS Excel) and input as tabular data.

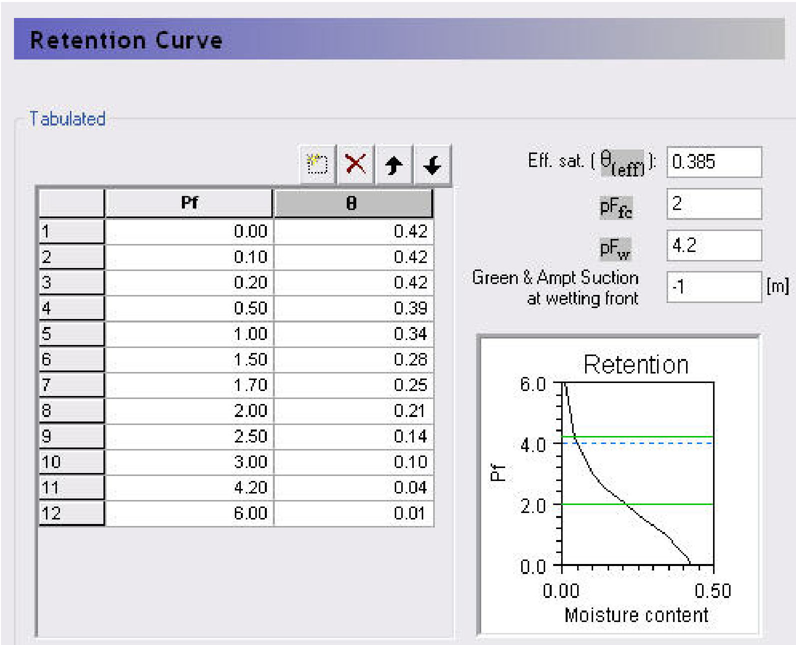
Several parametric forms of the soil moisture retention curve have been developed over the years. The MIKE SHE interface allows the user to specify two of the most common parametric forms.

Common data

The upper part of the dialogue contains the common parameters for describing the soil-moisture relationship, which are described in the Definitions of soil properties *(V1 p. 379)* section. The number of parameters in this section changes depending on the retention curve definition that was selected.

Green and Ampt suction - This is the empirical suction [metres of water] ahead of the wetting front in the Green and Ampt empirical relationship. It is essentially a calibration parameter, but should be about the same as the soil suction value just below field capacity. In other words, when the soil has been freely drained, plus had some water removed by evapotranspiration.

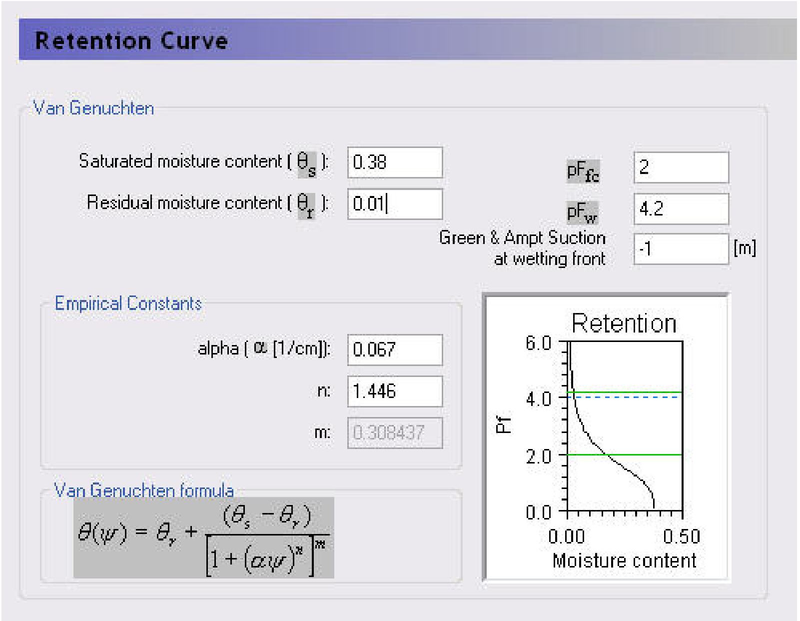
Tabulated



The data points describing the pressure conductivity curve can be given as a table of pF versus θ (moisture content) values. The table should be specified starting with the lowest value of pF (wettest condition) and given in increasing order of pF. (see pF scale *(V1 p. 379)*)

To get a smooth retention curve MIKE SHE adopts a cubic spline curve fitting procedure. As a minimum, you should specify the conductivity at saturation, the field capacity and the wilting point. However, we strongly advise against this because the cubic spline function is unlikely to be able to fit an appropriate function to only 3 points.

Van Genuchten Function



The Van Genuchten formula is the most widely used soil moisture-pressure relationship. The Van Genuchten formula was first published by M. Th. van Genuchten in 1980, in his classic paper. "A closed-form equation for predicting the hydraulic conductivity of unsaturated soils". Soil Science Society of America Journal 44 (5): 892–898. The Van Genuchten formula is

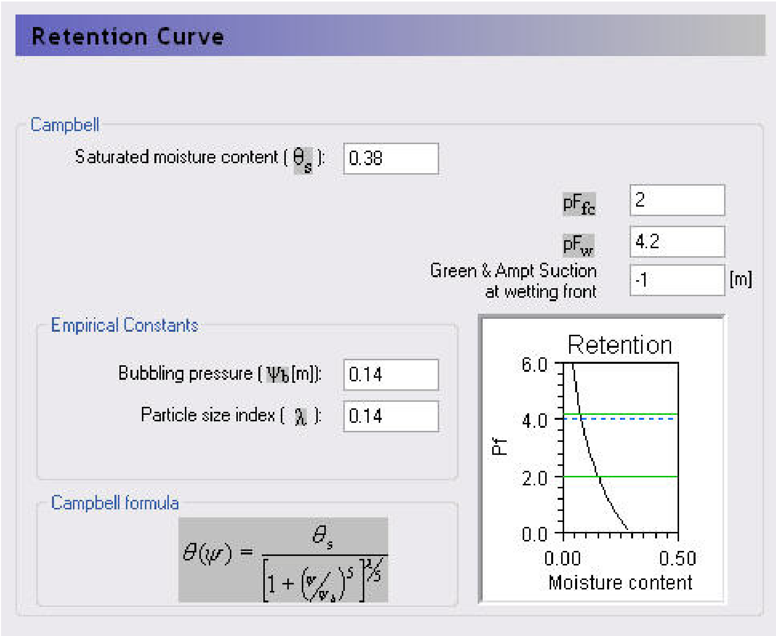
$$\theta(\psi) = \theta_r + \frac{(\theta_s - \theta_r)}{[1 + (\alpha \cdot \psi)^n]^m} \qquad (16.2)$$

where the exponent m is related to the empirical constant n by $m = 1 - \frac{1}{n}$. and α is the other Van Genuchten empircial constant.

Both *n* and *a* can be optimized using AUTOCAL program.

Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

Campbell Function



Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

16.4 Hydraulic Conductivity Function

The Governing Equation for the unsaturated flow requires information about two hydraulic functions: The hydraulic conductivity function, $K(\theta)$ and the soil moisture retention curve $\psi(\theta)$ are important.

The hydraulic conductivity decreases strongly as the moisture content θ decreases from saturation. This is not surprising since the total cross-sectional area for the flow decreases as the pores are getting filled with air. In addition, when a smaller part of the pore system is available to carry the flow, the flow paths will become more tortuous. Also, there is an increase of the viscosity of the water, when the short range adsorptive forces become dominant in relation to the capillary forces.

The experimental procedure for measuring the $K(\theta)$ function is rather difficult and not very reliable. Alternatively procedures have been suggested to derive the function from more easily measurable characterizing properties of the soil or simply to rely on empirical relationships.

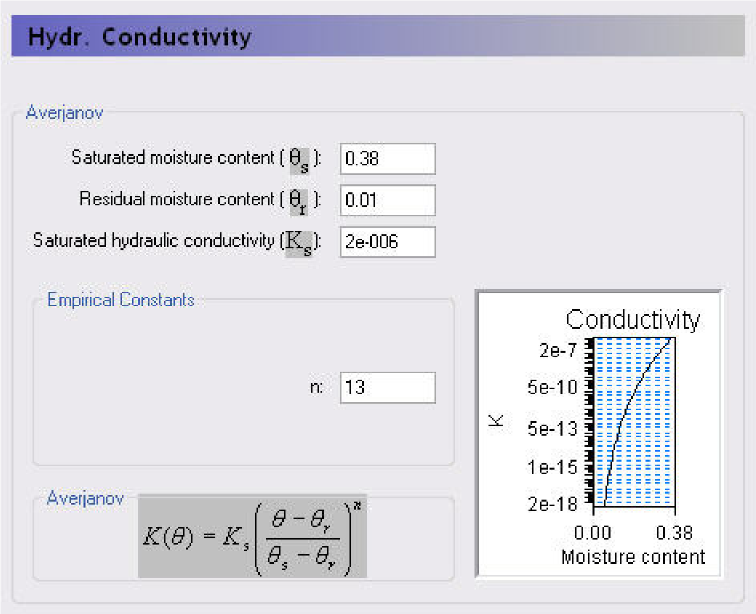
Reviews of various methods for predicting the conductivity function can be found in the literature.

Tabulated

The data points describing the pressure conductivity curve can be given as a table of pF versus K values. The table should be specified starting with the lowest value of pF (wettest condition) and given in increasing order of pF.

To get a smooth hydraulic conductivity curve, MIKE SHE adopts a cubic spline curve fitting procedure. As a minimum, you should specify the conductivity at saturation, the field capacity and the wilting point. However, we strongly advise against this because the cubic spline function is unlikely to be able to fit an appropriate function to only 3 points.

Averjanov Function



In the Averjanov method, the hydraulic conductivity, *K,* is described as a function of the effective saturation, *Se*:

$$K_{(E)} = K_{sat}(S_e)^n \tag{16.3}$$

where

$$S_e = (\theta - \theta_r)/(\theta_s - \theta_r) \tag{16.4}$$

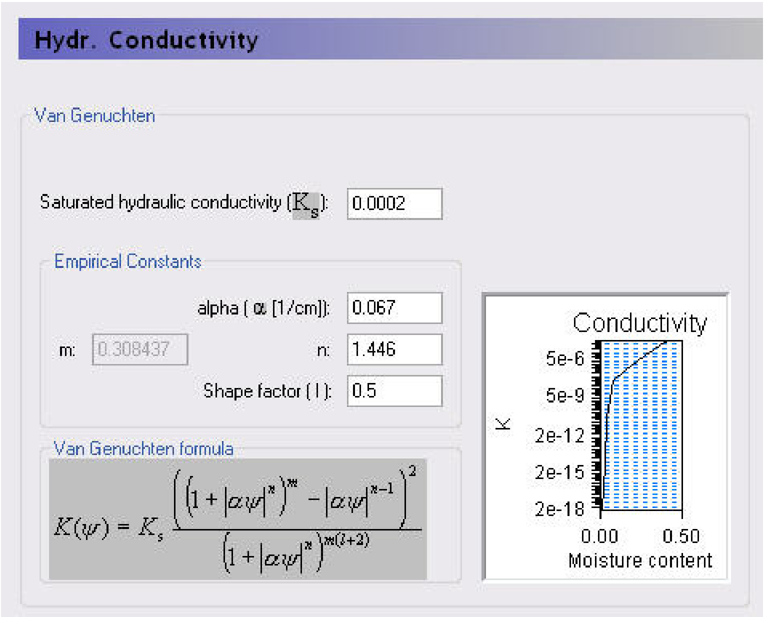
in which θs, θ and θr are saturated, actual and residual moisture contents, respectively.

The full knowledge about the hydraulic conductivity function is seldom available, and the parameter n has to be estimated by calibration.

As a guideline the exponent n is usually small for sandy soils (2-5) and large for clayey soils (10-20). It is important to note that the value of the exponent n will influence the percolation rate in the soil and thereby influence the actual evaporation rate.

Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

Van Genuchten Function

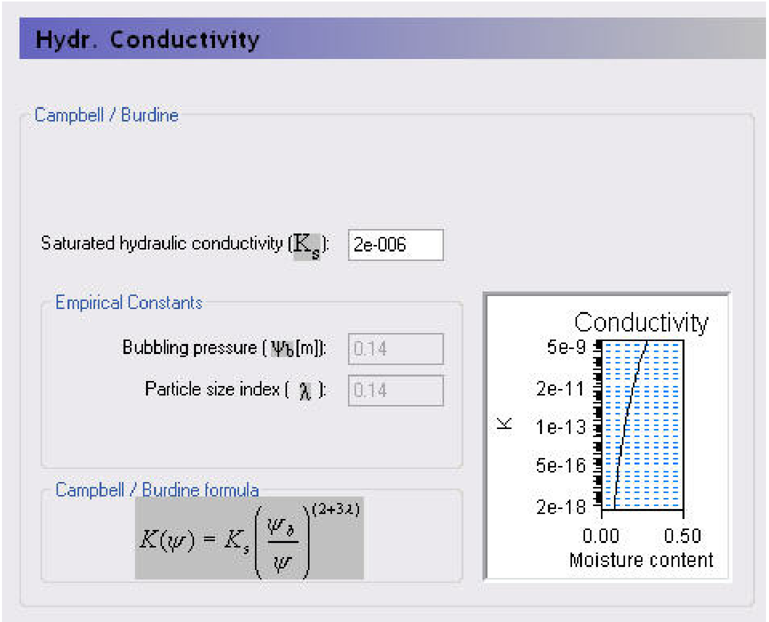


In addition to the tabulated values, parametric functions are available using the Van Genuchten and the Campbell/Burdine formulations. It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input.

The van Genuchten Shape Factor, *l*, is soil texture dependent, with a minimum allowed value of -4.

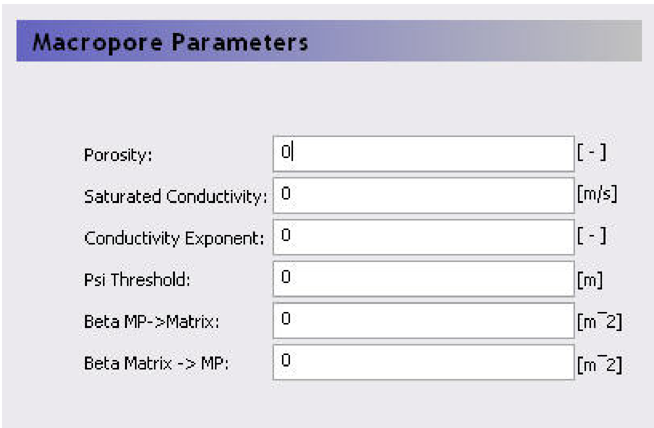
Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

Campbell/Burdine Function



Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

16.5 Macropore Parameters



The macropore parameters are used for the full macropore flow in the unsaturated zone.

Porosity - is the macropore porosity.

Saturated conductivity - is the saturated conductivity of the macropores.

Conductivity exponent - is the exponent, n, in Equation (10.35), which represents the simple power function assumed for the conductivity-water content relationship.

Psi threshold - is the capillary pressure in the matrix above which flow in the macropores can occur. Pressures below this value would result in water being absorbed into the matrix via capillarity.

Beta - is the mass transfer rate between the macropores and the matrix.

17 Water Balance Editor

The water balance utility is a flexible, post-processing tool for generating water balance data for MIKE SHE simulations. Output from the water balance utility can include area normalized flows (storage depths), storage changes, and model errors resulting from convergence problems. Water balance data can be generated at a variety of spatial and temporal scales and in a number of different formats.

To extract the water balance data, you must specify which simulation you are going, 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.

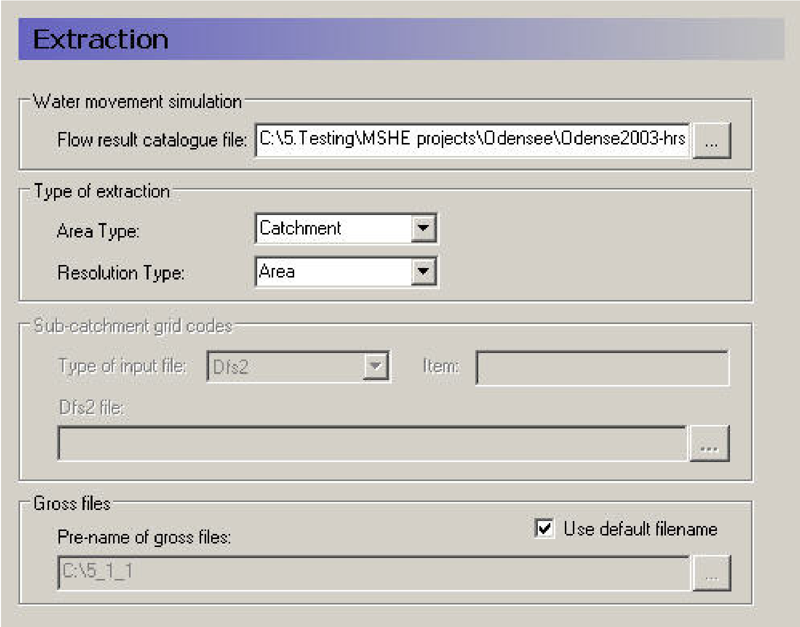
Once you have created a new water balance document, the following three tabs will be displayed

- Extraction *(V1 p. 390)*
- Postprocessings *(V1 p. 391)*
- Postprocessing Detail *(V1 p. 392)*
- Results *(V1 p. 394)*

Related items

- Using the Water Balance Tool *(V1 p. 91)*

17.1 Extraction



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 must match exactly the model 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

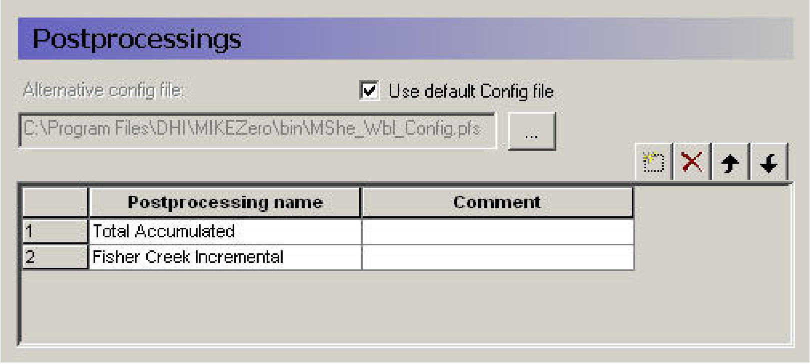
The pre-processor extracts the water balance data from the standard MIKE SHE output files and saves the data in a set of “gross” files. The file names of the gross files is built up from the project name and prefix specified here. The default value is normally fine.

Related items

- Using the Water Balance Tool *(V1 p. 91)*

17.2 Postprocessings

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

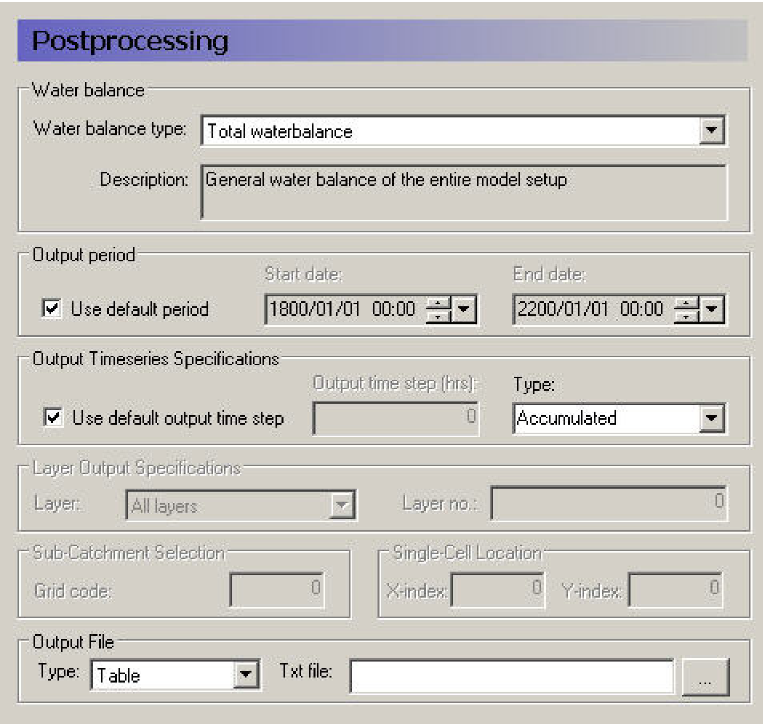
Unchecking the Use default Config file checkbox, allows you to specify the location of a custom water balance Config file. Development of custom water balance configuration files is described in detail in Making Custom Water Balances *(V1 p. 124)*.

Related items

- Using the Water Balance Tool *(V1 p. 91)*

17.2.1 Postprocessing Detail

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 Standard Water Balance Types *(V1 p. 123)* 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, then if you are not creating a map output, then 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 an 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 graphic.

Related items

- Using the Water Balance Tool *(V1 p. 91)*

17.3 Results

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.

Related items

- Using the Water Balance Tool *(V1 p. 91)*

APPENDIX A

A.1 MIKE SHE Output Items

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.

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 Hydro River 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 *(V1 p. 91)*. 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 instantaneous 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 the following tables, with each table comprising the potential output items in each potential output file. Not all files are output for each simulation and not all Items are in each file.

Note The **Code**, in the Tables is a **Data Type Code** that is needed when importing time series items into the Detailed time series output *(V1 p. 328)* dialogue.

List of Output Tables

Table A.1.1 Output items for the Simulation Status (Detailed Time Series)

Output Item	Item Type	Default unit	Time Series Type	Code
SimStatus: Basic time step length	TimeStep	hour	MeanStepAcc	145
SimStatus: SZ time step length	TimeStep	hour	MeanStepAcc	146
SimStatus: No. of SZ iterations / time step	ItemUndefined	IntCode	MeanStepAcc	147
SimStatus: Avg. no. UZ iterations / column / time step	ItemUndefined	IntCode	MeanStepAcc	148
SimStatus: No. of Overland iterations per time step	ItemUndefined	IntCode	MeanStepAcc	149

Table A.1.2 Gridded output items in projectname_ET_AllCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
precipitation rate	PrecipitationRate	millimeter/Day	MeanStepAcc	10
Air temperature	Temperature	degreeCelsius	MeanStepAcc	303
average water content in the rootzone	VolumetricWaterContent	One/One	Instantaneous	128
rooting depth	RootDepth	meter	Instantaneous	11
leaf area index	LAI	One/One	Instantaneous	12
crop coefficient	Fraction	One/One	Instantaneous	182

Table A.1.3 Gridded output items in projectname_ET_UzCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
reference evapotranspiration	ETRate	millimeter/Day	MeanStepAcc	355
ETref x Kc	ETRate	millimeter/Day	MeanStepAcc	340
actual evapotranspiration	ETRate	millimeter/Day	MeanStepAcc	15
actual transpiration	ETRate	millimeter/Day	MeanStepAcc	16
actual soil evaporation	ETRate	millimeter/Day	MeanStepAcc	13
actual evaporation from interception	ETRate	millimeter/Day	MeanStepAcc	17
actual evaporation from ponded water	ETRate	millimeter/Day	MeanStepAcc	18
canopy interception storage	Interception	millimeter	Instantaneous	19
evapotranspiration from SZ	ETRate	millimeter/Day	MeanStepAcc	14

Table A.1.4 Gridded output items in projectname_SnowMelt.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
Total Snow storage	StorageDepth	millimeter	Instantaneous	100
Dry Snow storage	StorageDepth	millimeter	Instantaneous	254
Wet Snow storage	StorageDepth	millimeter	Instantaneous	255
Wet Snow storage Fraction	Fraction	-	Instantaneous	256
Fraction of cell area covered by Snow	Fraction	-	Instantaneous	257
Precipitation + Irrigation added to Snow	Precipitation-Rate	millimeter/Day	MeanStepAcc	258
Total Snow converted to Overland Flow	Precipitation-Rate	millimeter/Day	MeanStepAcc	259
Freezing due to Air temperature	storagechang-erate	millimeter/Day	MeanStepAcc	260
Melting due to Air temperature	storage change rate	millimeter/Day	MeanStepAcc	261
Melting due to SW Solar Radiation	storage change rate	millimeter/Day	MeanStepAcc	262
Melting due to energy in Rain	storage change rate	millimeter/Day	MeanStepAcc	263
Snow evaporation	ETRate	millimeter/Day	MeanStepAcc	99

Table A.1.5 Gridded output items in projectname_overland.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
depth of overland water	WaterDepth	meter	Instantaneous	61
depth of overland water (TS average)	WaterDepth	meter	Instantaneous	
depth of overland water (TS min)	WaterDepth	meter	Instantaneous	
depth of overland water (TS max)	WaterDepth	meter	Instantaneous	
overland flow in x-direction	Discharge	m3/Sec	MeanStepAcc	58
overland flow in y-direction	Discharge	m3/Sec	MeanStepAcc	59
flow from overland to river	Discharge	m3/Sec	MeanStepAcc	342
overland flow velocity (TS average)	Flow velocity	meter/sec	Instantaneous	
overland flow velocity (TS min)	Flow velocity	meter/sec	Instantaneous	
overland flow velocity (TS max)	Flow velocity	meter/sec	Instantaneous	
OL drainage to river or MOUSE	Discharge	m3/Sec	MeanStepAcc	62
OL Drain Stored Volume	WaterVolume	m3	Instantaneous	356
OL Drain Storage Depth	StorageDepth	millimeter	Instantaneous	367
OL Drain Storage Inflow (flow)	Discharge	m3/Sec	MeanStepAcc	357
OL Drain Storage Inflow (flux)	storagechang-erate	millimeter/Day	MeanStepAcc	366
drain flow with OC storing frequency	Discharge	m3/Sec	MeanStepAcc	114
flooded (yes no)	Logical	-	Instantaneous	90
flow from flooded areas to river	Discharge	m3/Sec	MeanStepAcc	91
net precipitation rate for AD	Precipitation-Rate	millimeter/Day	MeanStepAcc	60
Overland flow to MOUSE	Discharge	m3/Sec	MeanStepAcc	152
External sources to Overland (for OpenMI)	Discharge	m3/Sec	MeanStepAcc	238
overland water elevation	Elevation	meter	Instantaneous	241
Mean OL Wave Courant number (explicit OL)	Fraction	-	MeanStepAcc	304
Max OL Wave Courant number (explicit OL)	Fraction	-	MeanStepAcc	305

Table A.1.5 Gridded output items in projectname_overland.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
Max Outflow OL-OL per Cell Volume (explicit OL)	Fraction	-	MeanStepAcc	306
External inflow to OL drain (for OpenMI)	Discharge	m3/Sec	MeanStepAcc	359

Table A.1.6 Gridded output items in projectname_OLMultiCell.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
depth of Multi-Cell overland water	WaterDepth	meter	Instantaneous	307
Multi-Cell overland water elevation	Elevation	meter	Instantaneous	317

Table A.1.7 Gridded output items in projectname_flood.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
H Water Depth	WaterDepth	meter	Instantaneous	369
P Flux	FlowFlux	m3/Sec/m	Instantaneous	370
Q Flux	FlowFlux	m3/Sec/m	Instantaneous	371

Table A.1.8 Gridded output items in projectname_wetland.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
Water content in root zone (2-layer UZ)	Volumetric WaterContent	-	Instantaneous	141
Water content below root zone (2-layer UZ)	Volumetric WaterContent	-	Instantaneous	142
Maximum water content (2-layer UZ)	Volumetric WaterContent	-	Instantaneous	143
Minimum water content (2-layer UZ)	Volumetric WaterContent	-	Instantaneous	144

Table A.1.9 Gridded output items in projectname_2DUZ_AllCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
infiltration to UZ (negative)	Infiltration	millimeter/Day	MeanStepAcc	121
exchange between UZ and SZ (pos.up)	Recharge	millimeter/Day	MeanStepAcc	122
bypass flow UZ (negative)	Infiltration	millimeter/Day	MeanStepAcc	124
UZ deficit	Deficit	millimeter	Instantaneous	57
infiltration to macropores (negative)	Infiltration	millimeter/Day	MeanStepAcc	125
macropore recharge to SZ (negative)	Recharge	millimeter/Day	MeanStepAcc	126
Total recharge to SZ (pos.down)	Precipitation-Rate	millimeter/Day	MeanStepAcc	310
average soil moisture content in top 5 compartments	Volumetric WaterContent	-	Instantaneous	37

Table A.1.10 Gridded output items in projectname_2DUZ_UzCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
rate of change in UZ storage	storagechange rate	millimeter/Day	MeanStepAcc	119
epsilon calculated in UZ	EpsilonUZ	millimeter	Instantaneous	123
groundwater levels used by UZ	Elevation	meter	Instantaneous	140
column mean macropore water content above GW	Volumetric WaterContent	-	Instantaneous	139
column total net exchange matrix to macropores	ExchangeRate	millimeter/Day	MeanStepAcc	136
column total exchange matrix to macropores	ExchangeRate	millimeter/Day	MeanStepAcc	137
column total exchange macropores to matrix	ExchangeRate	millimeter/Day	MeanStepAcc	138
groundwater feedback to the unsaturated zone	Recharge	millimeter/Day	MeanStepAcc	45
External sources to UZ columns (for OpenMI)	ExchangeRate	millimeter/Day	MeanStepAcc	334

Table A.1.11 Gridded output items in projectname_3DUZ.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
unsaturated zone flow	DarcyVelocity	millimeter/Day	MeanStepAcc	117
water content in unsaturated zone	Volumetric WaterContent	-	Instantaneous	118
pressure head in unsaturated zone	PressureHead	meter	Instantaneous	159
saturation in unsaturated zone	Fraction	-	Instantaneous	
root water uptake	ETRate	millimeter/Day	MeanStepAcc	129
macropore water content	Volumetric WaterContent	-	Instantaneous	130
macropore flow	DarcyVelocity	millimeter/Day	MeanStepAcc	131
exchange from matrix to macropores	ExchangeRate	millimeter/Day	MeanStepAcc	132
exchange from macropores to matrix	ExchangeRate	millimeter/Day	MeanStepAcc	133
net exchange from matrix to macropores	ExchangeRate	millimeter/Day	MeanStepAcc	134
microbiological clogging UZ conductivity	HydrConductiv-ity	meter/Sec	Instantaneous	127
UZ soil temperature	Temperature	degreeKelvin	MeanStepAcc	210
External sources to UZ nodes (for OpenMI)	ExchangeRate	millimeter/Day	MeanStepAcc	333

Table A.1.12 Gridded output items in projectname_2DIrrigation.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
irrigation: actual water content in root zone	VolumetricWa-terContent	-	Instantaneous	20
irrigation: soil moisture deficit in root zone	Deficit	millimeter	Instantaneous	135
total irrigation	IrrigationRate	millimeter/Day	MeanStepAcc	21
irrigation from river	IrrigationRate	millimeter/Day	MeanStepAcc	26
irrigation from wells	IrrigationRate	millimeter/Day	MeanStepAcc	28
irrigation from external source	IrrigationRate	millimeter/Day	MeanStepAcc	22
irrigation index	IrrigationIndex	-	Instantaneous	23
irrigation shortage	IrrigationRate	millimeter/Day	MeanStepAcc	24
irrigation total demand	IrrigationRate	millimeter/Day	MeanStepAcc	25
sprinkler irrigation	IrrigationRate	millimeter/Day	MeanStepAcc	153
drip and sheet irrigation	IrrigationRate	millimeter/Day	MeanStepAcc	154

Table A.1.13 Gridded output items in projectname_3DIrrigation.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
ground water extraction for irrigation	Discharge	m3/Sec	MeanStepAcc	27

Table A.1.14 Gridded output items in projectname_2DSZ.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
depth to phreatic surface (negative)	HeightAbove Ground	meter	Instantaneous	106
elevation of phreatic surface	Elevation	meter	Instantaneous	368

Table A.1.15 Gridded output items in projectname_3DSZ.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
head elevation in saturated zone	Elevation	meter	Instantaneous	101
saturated thickness of SZ layer	Length	meter	Instantaneous	327
SZ horizontal conductivity (for DA-OpenMI)	HydrConductivity	meter/Sec	Instantaneous	110
SZ vertical conductivity (for DA-OpenMI)	HydrConductivity	meter/Sec	Instantaneous	111

Table A.1.16 Gridded output items in projectname_2DSZFlow.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
seepage flow SZ -overland	DarcyVelocity	millimeter/Day	MeanStepAcc	107
seepage flow overland - SZ (negative)	DarcyVelocity	millimeter/Day	MeanStepAcc	108
qSZToFlood	Discharge	m3/Sec	MeanStepAcc	116
External inflow to SZ drain (for OpenMI)	Discharge	m3/Sec	MeanStepAcc	300
seepage flow Deficit (FEFLOW coupling)	StorageDepth	millimeter	Instantaneous	329

Table A.1.17 Gridded output items in projectname_3DSZflow.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
3D UZ recharge to SZ (negative)	Recharge	millimeter/Day	MeanStepAcc	113
groundwater flow in x-direction	Discharge	m3/Sec	MeanStepAcc	102
groundwater flow in y-direction	Discharge	m3/Sec	MeanStepAcc	103
groundwater flow in z-direction	DarcyVelocity	millimeter/Day	MeanStepAcc	104
SZ head elevation stored with SZ flows	Elevation	meter	Instantaneous	251
groundwater extraction	PumpingRate	m3/Sec	MeanStepAcc	109
SZ exchange flow with river	Discharge	m3/Sec	MeanStepAcc	115
SZ drainage flow from point	Discharge	m3/Sec	MeanStepAcc	112
SZ flow to general head boundary	Discharge	m3/Sec	MeanStepAcc	105
SZ flow to MOUSE	Discharge	m3/Sec	MeanStepAcc	151
External sources to SZ (for OpenMI)	Discharge	m3/Sec	MeanStepAcc	239

Table A.1.18 Gridded output items in projectname_RivEx.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
overland to river flow (positive) (RivEx-File)	Discharge	m3/Sec	MeanStepAcc	350
river to overland flow (negative) (RivEx-File)	Discharge	m3/Sec	MeanStepAcc	351
negative baseflow (river to SZ) (RivEx-File)	Discharge	m3/Sec	MeanStepAcc	352
exchange from mike 11 river to SZ (RivExFile)	Discharge	m3/Sec	MeanStepAcc	353
exchange from mike 11 river to OL (RivExFile)	Discharge	m3/Sec	MeanStepAcc	354

Table A.1.19 Gridded output items in projectname_LR.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
recharge to interflow reservoirs	Discharge	m3/Sec	MeanStepAcc	29
interflow from interflow reservoirs	Discharge	m3/Sec	MeanStepAcc	30
percolation from interflow reservoirs	Discharge	m3/Sec	MeanStepAcc	31
interflow reservoir storage	WaterVolume	m3	Instantaneous	32
change in interflow reservoir storage	Discharge	m3/Sec	MeanStepAcc	33
percolation to baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	34
percolation to dead zone of baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	211
river discharge to baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	252
river discharge to dead zone of baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	253
baseflow from baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	35
groundwater feedback from baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	36
pumping from baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	44
pumping for irrigation from baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	248
storage in baseflow reservoir	WaterVolume	m3	Instantaneous	46
dead zone storage in baseflow reservoir	WaterVolume	m3	Instantaneous	212
change in subcatchment storage in baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	38
change in dead zone storage in baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	213

Table A.1.20 Gridded output items in projectname_SimpleOL.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
simple overland water depth	WaterDepth	meter	Instantaneous	155
simple overland exchange to lower zone or river	Discharge	m3/Sec	Instantaneous	156
simple overland recharge	Discharge	m3/Sec	Instantaneous	157
water level in M11 h-point	Elevation	meter	Instantaneous	214
discharge in M11 q-point	Discharge	m3/Sec	MeanStepAcc	215

Table A.1.21 Gridded output items in projectname_ADOC.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
Overland concentration	Concentration	gram/M3	Instantaneous	216
Overland sorbed concentration	Concentration_4	gram/gram	Instantaneous	217
Overland fixed(undef) concentration	ItemUndefined	UnitUndefined	Instantaneous	335
Overland mass per area	MassPerUnitArea	gram/M2	Instantaneous	218
Air temperature	Temperature	degreeCelsius	Instantaneous	219
Overland temperature	Temperature	degreeCelsius	Instantaneous	331
External sources to Overland (for OpenMI)	SpecificSoluteFluxPerArea	gram/M2/Day	MeanStepAcc	266
Overland MIKE ECO Lab Output	ItemUndefined	UnitUndefined	MeanStepAcc	311
Overland Drainage concentration	Concentration	gram/M3	Instantaneous	362
Overland Drainage fixed(undef) concentration	ItemUndefined	UnitUndefined	Instantaneous	365
Overland Drainage mass per area	MassPerUnitArea	gram/M2	Instantaneous	363
Overland Drainage temperature	Temperature	degreeCelsius	Instantaneous	364

Table A.1.22 Gridded output items in projectname_ADUZ3.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
UZ concentration (matrix phase)	Concentration	gram/M3	Instantaneous	220
UZ sorbed concentration (matrix phase)	Concentration_4	gram/gram	Instantaneous	221
UZ fixed(undef) concentration (matrix phase)	ItemUndefined	UnitUndefined	Instantaneous	336
UZ mass flux (matrix phase)	SpecificSoluteFluxPerArea	gram/M2/Sec	MeanStepAcc	224
UZ concentration (macropore phase)	Concentration	gram/M3	Instantaneous	222
UZ sorbed concentration (macropore phase)	Concentration_4	gram/gram	Instantaneous	223
UZ fixed(undef) concentration (macropore phase)	ItemUndefined	UnitUndefined	Instantaneous	337
UZ mass flux (macropore phase)	SpecificSoluteFluxPerArea	gram/M2/Sec	MeanStepAcc	225
UZ soil temperature	Temperature	degreeCelsius	Instantaneous	226
UZ MIKE ECO Lab Output	ItemUndefined	UnitUndefined	MeanStepAcc	312

Table A.1.23 Gridded output items in projectname_ADSZx.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
External mass flux to SZ drain (for OpenMI)	SpecificSoluteFluxPerArea	gram/M2/Day	MeanStepAcc	302
Drain temperature	Temperature	degreeCelsius	Instantaneous	332

Table A.1.24 Gridded output items in projectname_ADSZ3D.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
SZ concentration (mobile phase)	Concentration	gram/M3	Instantaneous	227
SZ sorbed concentration (mobile phase)	Concentration_4	gram/gram	Instantaneous	228
SZ fixed(undef) concentration (mobile phase)	ItemUndefined	UnitUndefined	Instantaneous	338
SZ concentration (immobile phase)	Concentration	gram/M3	Instantaneous	229
SZ sorbed concentration (immobile phase)	Concentration_4	gram/gram	Instantaneous	230
SZ fixed(undef) concentration (immobile phase)	ItemUndefined	UnitUndefined	Instantaneous	339
SZ soil temperature	Temperature	degreeCelsius	Instantaneous	231
SZ porosity	PorosityCoefficient	-	Instantaneous	232
External sources to SZ (for OpenMI)	SpecificSoluteFluxPerArea	gram/M2/Day	MeanStepAcc	267
SZ MIKE ECO Lab Output	ItemUndefined	UnitUndefined	MeanStepAcc	313

Table A.1.25 Gridded output items in projectname_PTSZ3D.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
Number of particles	IntegerCode	intCode	Instantaneous	233
Accumulated particle count	IntegerCode	intCode	Instantaneous	250
Number of registered particles	IntegerCode	intCode	Instantaneous	234
Most recent registration zone code	IntegerCode	intCode	Instantaneous	235
Average age	Age	Year	Instantaneous	236
Average transport time to nearest registration cell	Age	Year	Instantaneous	237

INDEX