MIKE
Powered by DHI

MIKE SHE

Volume 2: Reference Guide

PLEASE NOTE

CONTENTS

REFERENCE MANUAL FOR WATER MOVEMENT

1 Water Movement Overview

This section includes detailed descriptions of the numeric engines used for moving water in MIKE SHE, as well as descriptions and suggestions on working with the different components. The following topics are included in this section of the manual:

- Evapotranspiration - Technical Reference *(V2 p. 17)*
  - Kristensen and Jensen method *(V2 p. 19)*
  - The 2-Layer Water Balance Method *(V2 p. 27)*
- Snow Melt - Technical Reference *(V2 p. 41)*
- Overland Flow - Technical Reference *(V2 p. 51)*
  - Finite Difference Method *(V2 p. 51)*
  - Ponded drainage *(V2 p. 61)*
  - Simplified Overland Flow Routing *(V2 p. 68)*
- Channel Flow - Technical Reference *(V2 p. 109)*
- Unsaturated Zone - Technical Reference *(V2 p. 141)*
  - Richards Equation *(V2 p. 142)*
  - Gravity Flow *(V2 p. 153)*
  - Two-Layer Water Balance *(V2 p. 155)*
- Saturated Flow - Technical Reference *(V2 p. 189)*
  - 3D Finite Difference Method *(V2 p. 189)*
  - Linear Reservoir Method *(V2 p. 205)*

2 Evapotranspiration - Technical Reference

In MIKE SHE, the ET processes are split up and modelled in the following order:

1. A proportion of the rainfall is intercepted by the vegetation canopy, from which part of the water evaporates.
2. The remaining water reaches the soil surface, producing either surface water runoff or percolating to the unsaturated zone.
3. Part of the infiltrating water is evaporated from the upper part of the root zone or transpired by the plant roots.
4. The remainder of the infiltrating water recharges the groundwater in the saturated zone where it will be extracted directly if the roots reach the water table, or indirectly if capillarity draws groundwater upwards to replace water removed from the unsaturated zone by the roots.

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.

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.

2.1 ET from Snow

ET from snow is a sum of the ET from wet snow and dry snow storages.

$$ET_{snow} = ET_{wetsnow} + ET_{drysnow} \tag{2.1}$$

First, ET will be removed from the wet snow storage, if it exists,

$$ET_{wetsnow} = ET_{ref} \cdot \Delta t$$

where ET_{ref} is the Reference Evapotranspiration *(V1 p. 225)* before being reduce by the Crop Coefficient, k_c, that is specified in the Vegetation Development Table *(V1 p. 375)*.

However, if there is insufficient wet snow storage, then ET will also be removed from dry snow as sublimation using

$$ET_{drysnow} = ET_{ref} \cdot S_f \cdot \Delta t \quad (2.2)$$

where S_f is the sublimation reduction factor found on the Snow Melt *(V1 p. 230)*, which reduces the amount of ET that can be removed from dry snow due to the extra energy required to sublimate snow.

If there is not enough snow storage then ET_{snow} will reduce the snow storage to zero.

2.2 ET from Canopy Interception

Interception is defined as the process whereby precipitation is retained on the leaves, branches, and stems of vegetation. This intercepted water evaporates directly without adding to the moisture storage in the soil.

The interception process is modelled as an interception storage, which must be filled before stem flow to the ground surface takes place. The size of the interception storage capacity, I_{max}, depends on the vegetation type and its stage of development, which is characterised by the leaf area index, *LAI*. Thus,

$$I_{max} = C_{int} \cdot LAI \quad (2.3)$$

where C_{int} is an interception coefficient [L] and *LAI* is leaf area index [-].

The coefficient C_{int} defines the interception storage capacity of the vegetation. A typical value is about 0.05 mm but a more exact value may be determined through calibration.

Note The interception coefficient is a unit of length [mm] - not a rate. This means that the full amount is intercepted in every time step, if precipitation is available and the storage is not full. Thus, the total amount of intercepted water is time step dependent. For example, if you have a precipitation rate of 2 mm/hour over 12 hours, the total precipitation will be 24 mm. However, the total interception could range between 2 mm if the time step length is 12 hours to the full 24 mm, if the time step length is 1 hour, assuming that there is 2 mm of evapotranspiration per time step.

2.2.1 Evaporation from the Canopy

The evaporation from the canopy storage is equal to the reference evapotranspiration, if sufficient water has been intercepted on the leaves, that is

$$E_{can} = min(I_{max}, ET_{ref}\Delta t)$$

where *Ecan* is the canopy evaporation [LT^{-1}], *Eref* is the reference evapotranspiration rate [LT^{-1}] and Δt is the time step length for the simulation.

Note: The amount of evaporation from the canopy is time dependent, since the interception on the canopy is calculated for every time step. So, if you half the time step, then the total amount of water stored in the canopy will double. The total amount of water stored in the canopy in temperate climates is generally small compared to the precipitation. However, semi-arid climates, this may impact your water balance.

2.3 Kristensen and Jensen method

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.

The primary assumption in the Kristensen and Jensen (KJ) method is Actual Evapotranspiration (AET) cannot exceed ET_0. The AET is reduced primarily by insufficient water in the root zone and by the density of the vegetation. In MIKE SHE, the current moisture content in the root zone is the limiting factor for AET in a time step. The root depth defines the depth to which ET can be extracted, and the Leaf Area Index (LAI) is the measure of the density of the vegetation.

The KJ method is based on empirical equations derived from field measurements. The model generally assumes the temperature to be above freezing.

2.3.1 Plant Transpiration

The transpiration from the vegetation, E_{at}, depends on the density of the vegetation, (i.e. the leaf area index, LAI) the soil moisture content in the root zone and the root density. Thus,

$$E_{at} = f_1(LAI) \cdot f_2(\theta) \cdot RDF \cdot ET_{ref} \tag{2.4}$$

where E_{at} is the actual transpiration [LT^{-1}], $f_1(LAI)$ is a function based on the leaf area index [-], $f_2(\theta)$ is a function based on the soil moisture content in the root zone [-], and *RDF* is a root distribution function [-].

$f_1(LAI)$
The function, $f_1(LAI)$, expresses the dependency of the transpiration on the leaf area of the plant by (see Figure 2.1)

$$f_1(LAI) = C_2 + C_1 LAI \tag{2.5}$$

where C_1 and C_2 are empirical parameters [-].

$f_2(\theta)$

The second function, *$f_2(\theta)$*, is given by

$$f_2(\theta) = 1 - \left(\frac{\theta_{FC} - \theta}{\theta_{FC} - \theta_W} \right)^{\frac{C_3}{E_p}} \tag{2.6}$$

where θ_{FC} is the volumetric moisture content at field capacity [-], θ_W is the volumetric moisture content at the wilting point [-], θ is the actual volumetric moisture content [-] and C_3 is an empirical parameter [LT^{-1}].

As illustrated in Figure 2.2, higher values of C_3 will lead to higher values of transpiration, which means that the soil will dry out faster, assuming all other factors constant. In a simulation, the actual transpiration will decrease more quickly for larger values of C_3.

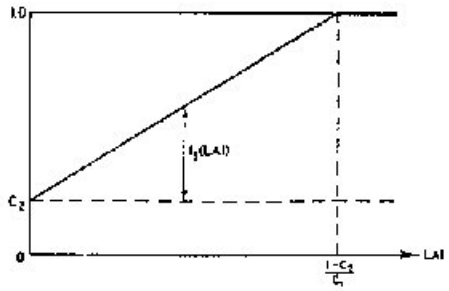


Figure 2.1 The function f*1* versus LAI.

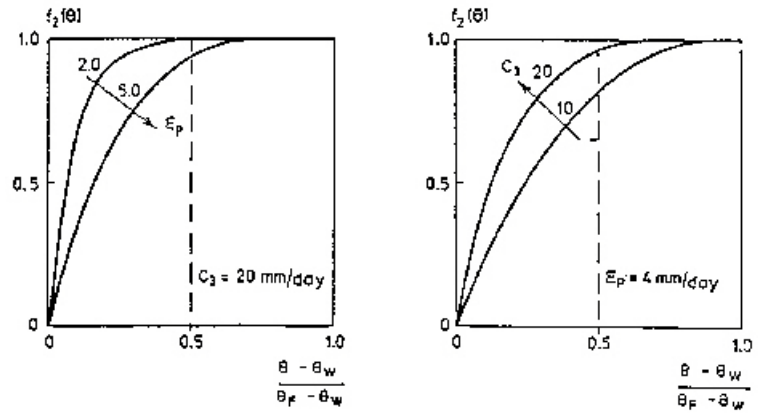


Figure 2.2 The soil moisture function f*2*(θ) for constant C*3* (20 mm/day) and varying E*p* (left), and for constant ET*ref* (4 mm/day) and varying C*3* (right).

Root Distribution Function, RDF

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 user-defined, time-varying root distribution determined by the root depth (time varying) and a general, vertical root-density distribution, see Figure 2.3.

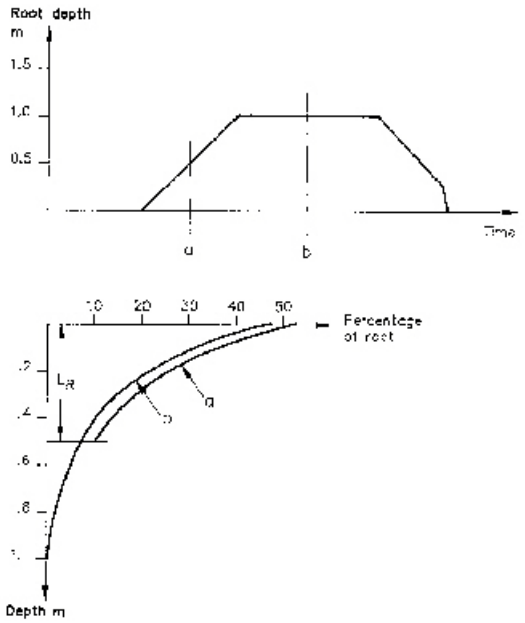


Figure 2.3 Root distribution in time and with depth.

The root extraction is assumed to vary logarithmically with depth by (see Figure 2.3)

$$\log R(z) = \log R_o - AROOT \cdot z \tag{2.7}$$

where *Ro* is the root extraction at the soil surface, *AROOT* is a parameter that describes the root mass distribution [-], and *z* is the depth below ground surface [L].

The value of the Root Distribution Function, RDF, in each layer is then calculated by dividing the amount of water extracted in the layer by the total amount of water extracted by the roots. Thus,

$$RDF_i = \int_{z1}^{z2} R(z)dz \; / \; \int_{o}^{L_R} R(z)dz \qquad (2.8)$$

where the numerator is the total amount of water extracted in layer *i* bounded above by Z_1 and below by Z_2 and the denominator is the total amount of water extracted by the roots between the ground surface and the maximum root depth, L_R.

AROOT

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 potential 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 with depth 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.

Thus, the factors *AROOT* and root depth are important parameters for estimating how much water can be drawn from the soil profile under dry conditions.

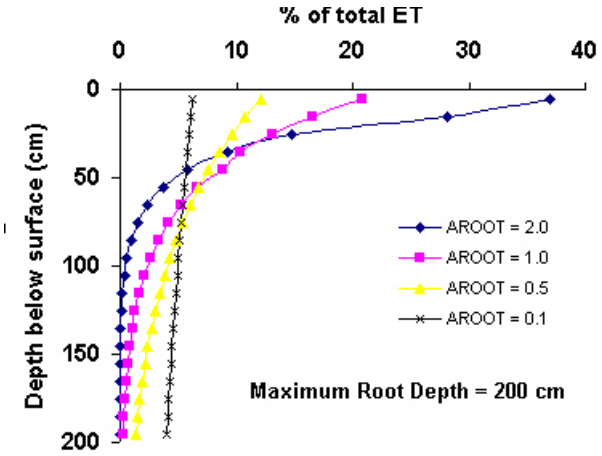


Figure 2.4 Fraction of ET extracted as a function of depth for different values of AROOT.

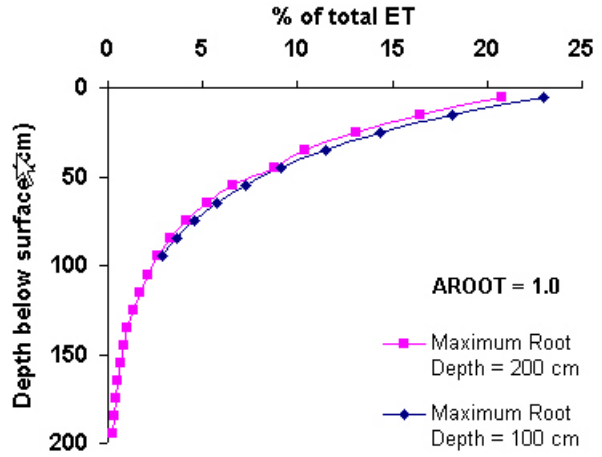


Figure 2.5 Fraction of ET extracted as a function of depth for different maximum root depths.

ET distribution between UZ and SZ

Evapotranspiration is extracted from saturated groundwater only when the roots are in contact with the water table - that is, when the root zone is connected to the water table. This can happen in two ways.

- The first is when the water table is physically in the root zone. In most natural systems this will only occur temporarily. If the roots are submerged for any length of time the roots will be starved of oxygen and drown.
- More commonly, the roots reach the capillary fringe. In this case, the evapotranspiration from the capillary fringe is replaced by water from the saturated zone.

In both cases the Kristensen and Jensen method calculates the ET from the subsurface. The MIKE SHE UZ module distributes this ET between the UZ and SZ cells.

In the first case, the ET is simply divided between the UZ and SZ depending on the fraction that would be removed from UZ cells that are below the water table.

In the second case, Richards Equation takes care of the natural replacement of water from the saturated zone by capillarity. However, the Gravity Flow method will ignore capillarity and the lower root zone may become too dry.

2.3.2 Soil Evaporation

Soil evaporation, E_s, occurs from the upper part of the unsaturated zone and consists of a basic amount of evaporation, $ET_{ref} \cdot f_3(\theta)$, plus additional evaporation from excess soil water as the soil saturation reaches field capacity. This can be described by the following function:

$$E_s = ET_{ref} \cdot f_3(\theta) + (ET_{ref} - E_{at} - ET_{ref} \cdot f_3(\theta)) \cdot f_4(\theta) \cdot (1 - f_1(LAI)) \quad (2.9)$$

where ET_{ref} is the reference evapotranspiration, E_{at} is the actual transpiration (Eq. (2.4)), $f_1(LAI)$ is from Eq. (2.5) and the functions $f_3(\theta)$ and $f_4(\theta)$ are given by

$$f_3(\theta) = \begin{cases} C_2 & \text{for } \theta \geq \theta_W \\ C_2 \dfrac{\theta}{\theta_W} & \text{for } \theta_r \leq \theta \leq \theta_W \\ 0 & \text{for } \theta \leq \theta_r \end{cases}$$

$$f_4(\theta) = \begin{cases} \dfrac{\theta - \dfrac{\theta_W + \theta_{FC}}{2}}{\theta_{FC} - \dfrac{\theta_W + \theta_{FC}}{2}} & \text{for } \theta \geq \dfrac{(\theta_W + \theta_F)}{2} \\ 0 & \text{for } \theta < \dfrac{(\theta_W + \theta_F)}{2} \end{cases} \quad (2.10)$$

In the absence of vegetation $f_1(LAI)$ can be set to zero and E_{at} in Eq. (2.9) goes to zero. This allows us to see how E_s varies in relation to ET_{ref} for different values of q. Thus, Eq. (2.9) can be simplified to

$$\frac{E_s}{ET_{ref}} = f_3(\theta) + f_4(\theta) - f_3(\theta) \cdot f_4(\theta) \quad (2.11)$$

which is plotted in Figure 2.6.

In the MIKE SHE, soil evaporation is restricted to the upper node in the unsaturated zone, which, generally, should be about 10 centimetres deep, or less.

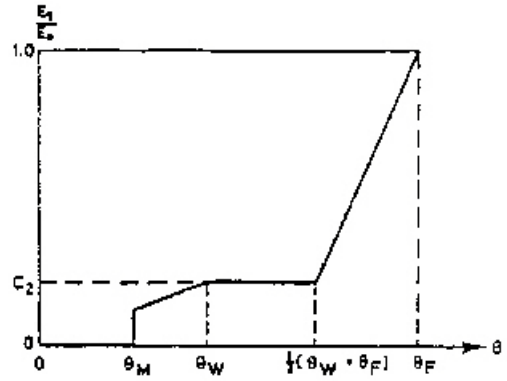


Figure 2.6 Soil evaporation E_s in relation to ET_{ref} as a function of q in the top layer when f(LAI) = 0.

2.3.3 Evapotranspiration Coefficients C_1, C_2 and C_3

The equations for actual transpiration, Eq. (2.4), and soil evaporation, Eq. (2.9), 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) (Eq.(2.5)). C_3 is also part of Eq. (2.4), but is only found in the soil moisture function, Eq. (2.6).

C_1

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.

C_2

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.

C_3

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).

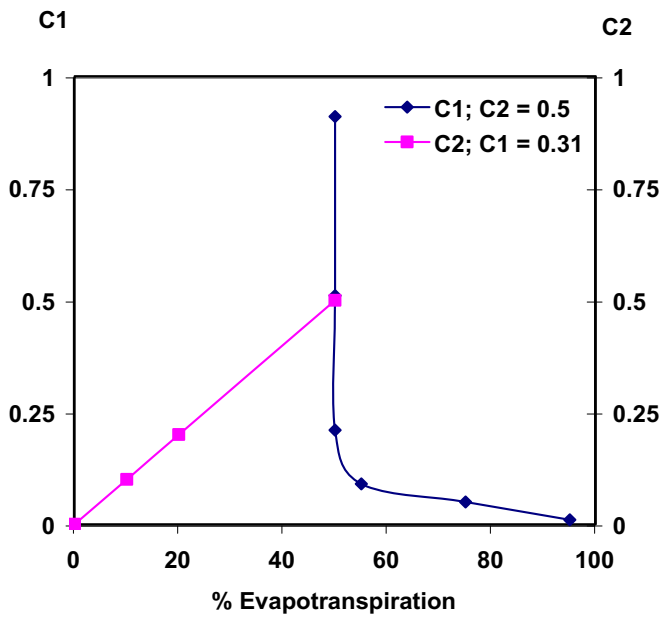


Figure 2.7 The influence of the C_1 and C_2 on the ratio between soil evaporation and transpiration. The values were obtained from model runs assuming C_{int} = 0, the moisture content above field capacity, and LAI = 5

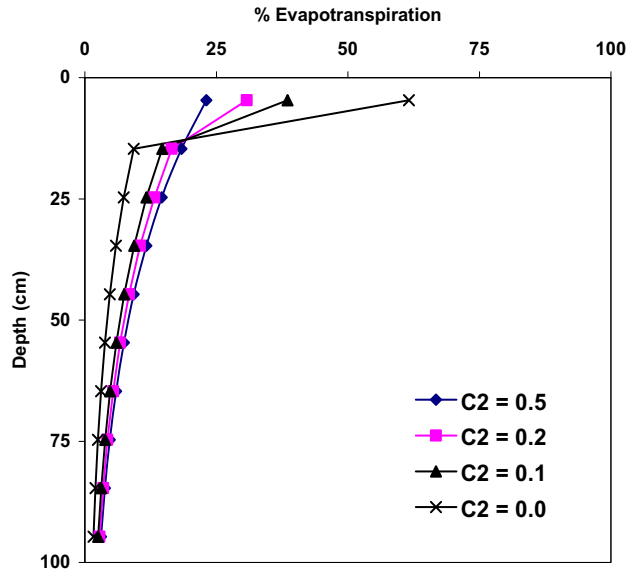


Figure 2.8 Distribution of actual evapotranspiration (in per cent) over depth for different values of C_2. C_2=0 corresponds to pure transpiration.

2.4 The 2-Layer Water Balance Method

The 2-Layer Water Balance Method is an alternative to the more complex unsaturated flow process coupled to the Kristensen and Jensen module for describing evapotranspiration. The 2-Layer Water Balance Method is based on a formulation presented in Yan and Smith (1994). The main purpose of the module is to calculate actual evapotranspiration and the amount of water that recharges the saturated zone.

The module is particularly useful for areas with a shallow ground water table, such as swamps or wetlands areas, where the actual evapotranspiration rate is close to the reference rate. In areas with deeper and drier unsaturated zones, the model does not realistically represent the flow dynamics in the unsaturated zone. The model only considers average conditions and does not account for the relation between unsaturated hydraulic conductivity and soil moisture content and, thereby, the ability of the soil to transport water to the roots. The model simply assumes that if sufficient water is available in the root zone, the water will be available for evapotranspiration. However, it is usually possible to “calibrate” the input parameters so that the model performs reasonably well under most conditions.

The 2-Layer Water Balance Method includes the processes of interception, ponding, and evapotranspiration, while considering the entire unsaturated zone to consist of two ‘layers’ representing average conditions in the unsaturated zone. The vegetation is described in terms of leaf area index (LAI) and

root depth. The soil properties include a constant infiltration capacity and the soil moisture contents at the wilting point, field capacity and saturation.

The output is an estimate of the actual evapotranspiration and the ground water recharge.

Leaf Area Index (LAI)

The area of leaves above a unit area of the ground surface is defined by the leaf area index, *LAI*. Generalised time varying functions of the LAI for most crops and types of vegetation are available in the literature. In MIKE SHE, you must specify the temporal variation of the LAI for each vegetation type during the growing seasons to be simulated. Different climatic conditions from year to year may require a shift of the LAI curves in time but will generally not change the shape of the curve. Typically, the LAI varies between 0 and 7.

Root Depth

The root depth is defined as the maximum depth of active roots in the root zone.

2.4.1 Explicit ET calculation

The calculation of ET is explicit and the removal of ET from the various storages in MIKE SHE happens only at specific times in the calculation process, as described in the following:

1. Evaporation and sublimation is removed from snow (if present),
2. Bypass infiltration is calculated (if included),
3. Flow from paved areas is calculated (if included),
4. Infiltration to UZ and recharge to SZ calculated,
5. Evaporation from the canopy is calculated,
6. Evaporation from Ponded water is calculated,
7. Transpiration from the unsaturated zone is calculated, and then
8. Transpiration from the saturated zone is calculated.
9. the UZ/SZ water balance is checked and the SZ groundwater table is corrected if necessary.

However, if the Extra Parameter option Transpiration during ponding *(V2 p. 343)* is specified, then the order above is changed, where

1. Transpiration from the unsaturated zone is calculated,
2. Transpiration from the saturated zone is calculated, and then
3. Evaporation from Ponded water is calculated.

2.4.2 Soil Moisture

ET Extinction depth - If the capillary fringe reaches the bottom of the root zone, then water removed from the root zone by ET will be replaced by water drawn up by capillary action. The ET extinction depth is then the maximum depth where ET can be removed from the saturated zone by the roots - that is the root depth + the thickness of the capillary fringe.

In the 2-Layer Water Balance Method the unsaturated zone can consist of one or two layers.

- The upper layer extends from the ground surface to the ET extinction depth. If the water table is at the ground surface then the thickness of the upper layer is zero.
- If the water table is below the ET extinction depth, then a second layer is added that extends from the bottom of first layer to the water table. If the water table is above the ET extinction depth, the thickness of the lower layer is zero.

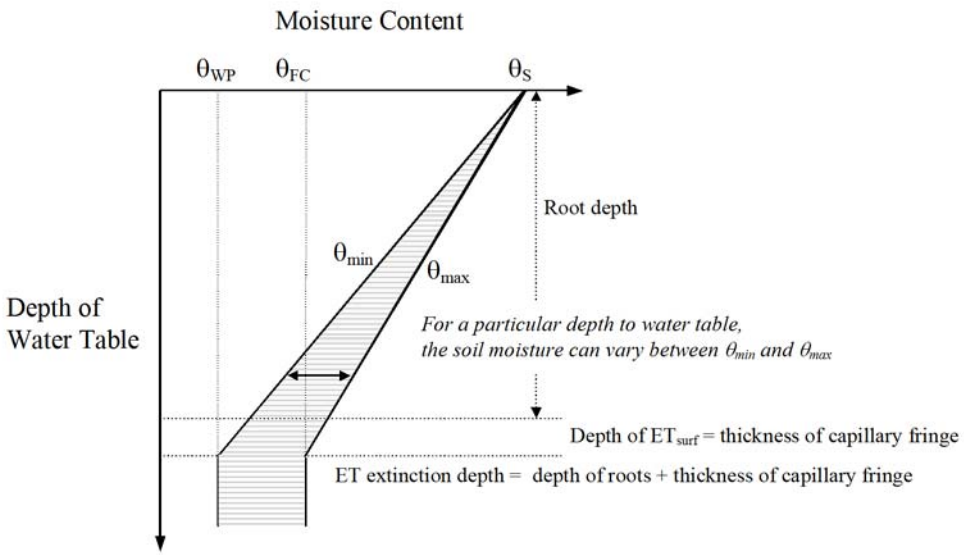


Figure 2.9 Allowable range for soil moisture in the upper ET layer as a function of the depth to the water table.

Water table at ground surface

If the water table is at the ground surface then the moisture content equals the saturated moisture content, θs.

Water table above ET extinction depth

If the water table is above the ET extinction depth, then the average water content in the upper UZ layer will vary between a minimum, θmin, and a max-

imum, θmax (see Figure 2.9). θmax is the water content that would be present if no ET occurred. θmin is the minimum water content that can exist in the upper ET layer when ET is active. Both θmin and θmax decrease linearly with the depth to the water table. In other words, the average water content of the entire column decreases as the water table drops because the upper part of the column is draining to the field capacity. When the water table is at the extinction depth, the average water content of will be field capacity if there is no ET.

When ET is removed, the water content will decrease from θmax. The water content will increase again up to θmax when infiltration occurs. Thus, the total storage deficit of the top layer of the unsaturated zone, D_{UZ} is

$$D_{uz} = (\theta_{max} - \theta_{act}) \cdot z_{wt} \tag{2.12}$$

and the total storage available for ET, S_{UZ} is

$$S_{uz} = (\theta_{act} - \theta_{min}) \cdot z_{wt} \tag{2.13}$$

where θact is the actual water content at the end of the previous time-step and z_{wt} is the thickness of the upper UZ layer.

Vertical infiltration to the lower UZ layer or to the saturated zone occurs when the water content is equal to θmax.

Water table below ET extinction depth

If the water table is below the ET extinction depth, then a lower UZ layer is added. The moisture content of the lower UZ layer is generally equal to the field capacity because ET is not removed from the lower UZ layer. However, the water content of the lower UZ layer can range between the field capacity and the wilting point. If the water content is equal to field capacity then an infiltration from the upper UZ layer will be added to the top layer of the SZ model as groundwater recharge.

Changing layer thicknesses

The thickness of the upper and lower UZ layers changes whenever the root depth or the water table changes. Thus, even if there is no ET or infiltration, a changing layer thickness will change the average water content of the layer because the current water content will be redisributed within a smaller or larger volume.

If the water content in the upper UZ layer is currently at or near θmin, a decrease in the root depth (e.g. crop harvest) can lead to a redistributed water content that is less than the new θmin. In this case, if the water table is above the extinction depth, water will be transfered from the SZ model to the upper UZ layer to bring the water content back up to θmin. If the water table is below the extinction depth, the water content will simply remain below θmin

until infiltration increases the water content, or until the water table rises to the new extinction depth.

If the water table is above the extinction depth and falls, then the restributed water content of the upper UZ layer will be calculated based on the added layer thickness times the saturated water content.

A redistributed water content above θmax in the upper UZ layer will result in discharge to either the lower UZ layer or the upper SZ layer depending on the location of the water table.

The water content of the lower UZ layer is generally at field capacity because ET is not removed from the lower UZ layer. However, changing layer thicknesses will cause a redistribution of water content which can result in a water content below field capacity. For example, if the upper UZ layer water content is at or near θmin and the root depth suddenly decrease (e.g. the root depth could suddenly become zero due to a crop harvest) then the average water content of the lower UZ layer will drop well below field capacity.

2.4.3 Canopy Interception

Interception is defined as the process whereby precipitation is retained on the leaves, branches, and stems of vegetation. This intercepted water evaporates directly without adding to the moisture storage in the soil.

The interception process is modelled as an interception storage, which must be filled before stem flow to the ground surface takes place. The maximum amount of interception storage, $S_{int\,max}$, depends on the vegetation type and its stage of development, which is characterised by the leaf area index, LAI. Thus,

$$S_{intmax} = C_{int} \cdot LAI \tag{2.14}$$

where C_{int} is the interception storage capacity that defines the maximum amount of interception storage for the vegetation. A typical value is about 0.05 mm but a more exact value may be determined through calibration.

The interception storage, S_{int}, is accumulated over time. So, if the amount of precipitation in the time step, plus the current amount of storage, exceeds $S_{int\,max}$, then the interception storage will equal $S_{int\,max}$ and any excess will be added ponded water on the ground surface. If $S_{int\,max}$ is not exceeded, then the canopy will capture all of the available precipitation in the time step.

If the LAI decreases between time steps (e.g. after a crop harvest), $S_{int\,max}$ will decrease and any excess interception storage will be added to the ponded water on the ground surface.

2.4.4 Infiltration

At the beginning of each time step, sprinkler irrigation and snowmelt is added to the available precipitation. This available precipitation first fills the interception storage. If $S_{\text{int max}}$ is exceeded, the excess water is added to the current amount of ponded water on the ground surface.

Then direct bypass infiltration to SZ is calculated and the depth of ponded water is updated.

After the calculation of the bypass infiltration, the depth of ponded water is updated for drainage to paved areas (see).

After the removal of bypass flow and drainage to paved areas, the infiltration to the upper UZ layer is calculated, limited by

- the amount of available water for infiltration - the depth of ponded water, *dOL*,
- the rate of infiltration, and
- the estimated amount of water than would raise the water table to the ground surface in the current time step.

The rate of infiltration is adjusted for two special conditions. In the first case, the infiltration can be higher when the soil is very dry (see Saturated Zone *(V2 p. 345)*). In the second case, the infiltration rate will be reduced if you have specified an Surface-Subsurface Leakage Coefficient *(V1 p. 260)* that is lower than the specified leakage coefficient in the 2-Layer UZ soil properties *(V1 p. 285)* dialogue.

The amount of infiltration can also be limited when the amount of ponded water is below a specified threshold, since the 2-Layer water balance method does not include evaporation from the soil surface. Thus, even a small amount of water on the ground surface will infiltrate. If you define a depth of overland water that must be exceeded before infiltration will occur, small amounts of precipitation will not infiltrate, but rather evaporate instead. The calculated infiltration is simply reduced if the remaining overland water depth will be smaller than the specified threshold value. See Threshold depth for infiltration (2-Layer UZ) *(V2 p. 344)*.

The actual infiltration to the upper UZ layer is then calculated - subject to the limits described above. Any surplus water above θmax is added to the lower UZ layer. In the lower UZ layer, any surplus water above θFC is added to the upper SZ layer. Finally, ponding depth and the water contents in the upper and lower UZ layers are recalculated and updated.

2.4.5 Evapotranspiration

MIKE SHE uses the Crop Reference ET rate for all calculations of ET. The Crop Reference ET rate is

$$ET_{rate} = ET_{crop} = ET_{ref} \cdot k_c \tag{2.15}$$

where ET_{ref} is the Reference Evapotranspiration *(V1 p. 225)* and k_c is the Crop Coefficient specified in the Vegetation Development Table *(V1 p. 375)* that adjusts the Reference ET rate for different vegetation types. The maximum amount of ET that can be removed in one time step is

$$ET_{max} = ET_{rate} \cdot \Delta t \tag{2.16}$$

ET_{max} is satisfied in the following order:

1. ET is first removed from snow storage.
2. If the snow storage cannot satisfy ET_{max}, water is evaporated from the interception storage.
3. If the interception storage cannot satisfy ET_{max}, water is evaporated from the ponded water until the ponded water is exhausted.
4. If ET_{max} has not yet been satisfied, water is removed from the unsaturated zone until ET_{max} is satisfied or the water content of the upper UZ layer is reduced to θmin.
5. If ET_{max} has still not yet been satisfied, water is extracted from the saturated zone if the water table is above the ET extinction depth.

However, if Transpiration during ponding *(V2 p. 343)* is allowed, then the order of the last three calculations above are changed and ET is removed from the ponded water after being removed from the UZ and SZ.

ET from Snow

ET from snow is a sum of the ET from wet snow and and dry snow storages.

$$ET_{snow} = ET_{wetsnow} + ET_{drysnow} \tag{2.17}$$

ET will be first removed from the wet snow storage, if it exists,

$$ET_{wetsnow} = ET_{ref} \cdot \Delta t$$

where ET_{ref} is the Reference Evapotranspiration *(V1 p. 225)* before being reduce by the Crop Coefficient, k_c, that is specified in the Vegetation Development Table *(V1 p. 375)*.

However, if there is insufficient wet snow storage, then ET will be removed from dry snow as sublimation using

$$ET_{drysnow} = ET_{ref} \cdot S_f \cdot \Delta t \quad (2.18)$$

where S_f is the sublimation reduction factor found on the Snow Melt *(V1 p. 230)* dialog, which reduces the amount of ET that can be removed from dry snow due to the extra energy required to sublimate snow.

If there is not enough snow storage then ET_{snow} will reduce the snow storage to zero.

ET from the Canopy

After the ET is removed from any available snow storage, ET is removed from the canopy storage until the canopy storage is exhausted or ET_{max} is satisfied using

$$ET_{canopy} = ET_{rate} \cdot \Delta t \quad (2.19)$$

If there is not enough canopy storage then ET_{canopy} will reduce the canopy storage to zero.

ET from Ponded Water

ET is removed from the ponded storage until ET_{max} is satisfied or the ponded storage is exhausted. Thus,

$$ET_{ponded} = ET_{rate} \cdot \Delta t \quad (2.20)$$

If there is not enough ponded storage then ET_{ponded} will reduce the ponded storage to zero.

ET from the Unsaturated Zone

ET is only removed from the upper UZ layer. However, as the water content of the root zone decreases, plants will find it harder to remove the water. Finally, when the wilting point is reached ET will stop. However, the reduction in ET does not occur immediately. Plants will remove ET from the root zone at the maximum rate until a plant specific water content is reached, at which point the rate of ET removal will be restricted. In the 2-Layer Water Balance method this phenomena is accounted for by a plant specific deficit fraction. ET will be removed at the maximum rate until this deficit fraction is reached, then linearily reduced to zero as the water content falls to the wilting point.

Thus, the ET_{rate} is reduced by

$$F_{ETUZ} = \frac{min(\theta_{act}, \theta_p) - \theta_{min}}{\theta_p - \theta_{min}} \quad (2.21)$$

where θ_p is water content when ET begins to be restricted given by

$$\theta_p = \theta_{min} + F_p \cdot (\theta_{max} - \theta_{min}) \quad (2.22)$$

θ_{act} is the current water content in the upper layer, θ_{min} and θ_{max} are the minimum and maximum water contents in Figure 2.9 and F_p is the user-specified, plant-specific deficit fraction when ET begins to be restricted by a lack of soil water. From Equations (2.21) and (2.22) θ_p is variable if the ET extinction depth is above the bottom of the root zone. If the ET extinction depth is below the root zone, then θ_p represents the minimum water content for full ET. Crop specific values for θ_p can be found in the literature or from the FAO.

ET is removed from the upper UZ layer until ET_{max} is satisfied or the average water content is reduced to the wilting point. Thus,

$$ET_{UZ} = ET_{rate} \cdot F_{ETUZ} \cdot \Delta t \quad (2.23)$$

However, if Transpiration during ponding *(V2 p. 343)* is allowed, then F_{ETUZ} is multiplied by the anaerobic tolerance factor to account for the decreased capacity for ET when there is ponded water on the ground surface. Also, the maximum remaining amount of ET that can be removed is also reduced by the anaerobic tolerance factor, F_{antol}. That is,

$$ET_{left} = (ET_{max} - ET_{snow} - ET_{canopy}) \cdot F_{antol} \quad (2.24)$$

ET from the Saturated Zone

If the water table is above the extinction depth, then ET is removed from SZ until ET_{max} is satisfied using

$$ET_{SZ} = ET_{rate} \cdot F_{ETSZ} \cdot \Delta t \quad (2.25)$$

where F_{ETSZ} is 1.0 when the water table is in the root zone and decreases linearly from 1.0 to zero when the water table is below the root zone, but above the extinction depth.

If the water table is below the extinction depth, then F_{ETSZ} is zero, and no ET is removed from the saturated zone.

In the case where Transpiration during ponding *(V2 p. 343)* is allowed, then, F_{ETSZ} is multiplied by the anaerobic tolerance factor to account for the decreased capacity for ET when there is ponded water on the ground sur-

face. Also, the maximum remaining amount of ET that can be removed is also reduced by the anaerobic tolerance factor, F_{antol}. That is,

$$ET_{left} = (ET_{max} - ET_{snow} - ET_{canopy} - ET_{UZ}) \cdot F_{antol} \quad (2.26)$$

Actual ET

Finally, the actual evapotranspiration is the sum of the above contributions

$$ET_{actual} = ET_{snow} + ET_{canopy} + ET_{ponded} + ET_{UZ} + ET_{SZ} \quad (2.27)$$

remembering that ET_{actual} cannot be greater than ET_{max} and that the ET is calculated in a specific order until ET_{max} is reached.

2.4.6 Coupling to the Saturated Zone

At the end of the ET calculation step, the change in storage in the upper SZ cell is compared to the change in storage in upper and lower UZ layers, plus the amount of water removed by ET and any infiltration and recharge to SZ. If this water balance exceeds the limit specified in the UZ Computational Control Parameters *(V1 p. 188)* dialog, then a depth correction is calculated for the water table and the water table elevation is adjusted, while ensuring that the exchange between UZ and SZ does not exceed the UZ infiltration rate.

3 Working with Evapotranspiration - User Guide

The calculation of evapotranspiration uses meteorological and vegetative data to predict the total actual evapotranspiration 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, and its replenishment from the saturated zone due to capillarity.

In MIKE SHE, the ET processes are split up and modelled in the following order:

1. A proportion of the rainfall is intercepted by the vegetation canopy, from which part of the water evaporates.
2. The remaining water reaches the soil surface, producing either surface water runoff or percolating to the unsaturated zone.
3. Part of the infiltrating water is evaporated from the upper part of the root zone or transpired by the plant roots.
4. The remainder of the infiltrating water recharges the groundwater in the saturated zone where it will be extracted directly if the roots reach the water table, or indirectly if capillarity draws groundwater upwards to replace water removed from the unsaturated zone by the roots.

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.

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.

3.1 Some ET Definitions

Evaporation is the when liquid water is converted to vapour due to the addition of energy. Evaporation of water occurs from any free water surface, such

as a lake. Evaporation requires energy, which is mostly supplied by solar radiation, and to a lesser extent by air temperature. Evaporation will slowly decrease to zero as the humidity of the air above the water surface increases. Thus, wind is critical to carry moist air away, and replace it with drier air. In the context of soil evaporation, plants will shade the soil, reducing solar radiation and reducing evaporation.

Transpiration is the evaporation of liquid water from the plant tissues. Plants primarily lose water through pores in their leaves called stomata. Different plant species are able to regulate their stomata openings to greater or lesser degrees. Thus, different plants will transpire at different rates. Similar to evaporation, the overall rate is determined by solar radiation, humidity and the wind. However, transpiration also depends on the status of the crop, cultivation practices, etc.

Evapotranspiration (ET) is the combination of both evaporation and transpiration on vegetated surfaces. Both processes occur simultaneously and cannot be separated. In the case of crops sown in bare soil, after sowing ET is 100% evaporation but as the plants grow and shade the soil, the transpiration begins to dominate.

Reference ET (ET0) is the maximum ET that can be extracted from a standard grass surface (reference crop) that is fully watered. The ET_0 is a climate parameter and is dependent only on the climate variables. It is NOT dependent on the vegetation or crop. ET_0 is defined by the Penman-Montieth method. The most widely used reference document for the Penman-Montieth method is the FAO-56 document by Allen et al (1998).

The units of ET are normally the same as precipitation, eg mm/day. Evapotranspiration is a significant loss of water. One hectare is 10,000 m^2. So an ET rate of 1mm/d is equivalent to 10 m^3 of water per hectare per day.

Table 3.1 Average ET_0 for different climatic regions in mm/day (from FAO-56)

Regions	Mean daily temperature		
	Cool (~10C)	Moderate (20C)	Warm (>30C)
Sub-/Tropics			
humid/sub-humid	2-3	3-5	5-7
arid/semi-arid	2-4	4-6	6-8
Temperate			
humid/sub-humid	1-2	2-4	4-7
arid/semi-arid	1-3	4-7	6-9

Leaf Area Index (LAI)

The Leaf Area Index (LAI) is defined as the one-sided, area of leaves above a unit area of the ground surface. Generalised time varying functions of the LAI

for most crops and types of vegetation are available in the literature. In MIKE SHE, you must specify the temporal variation of the LAI for each vegetation type during the growing seasons to be simulated. Different climatic conditions from year to year may require a shift of the LAI curves in time but will generally not change the shape of the curve. Typically, the LAI varies from 0 to a about 7 for forests and up to about 10 for dense plantation forests.

Root Depth

The root depth is defined as the maximum depth of active roots in the root zone.

3.2 ET from Canopy Interception

Interception is defined as the process whereby precipitation is retained on the leaves, branches, and stems of vegetation. This intercepted water evaporates directly without adding to the moisture storage in the soil.

The interception process is modelled as an interception storage, which must be filled before stem flow to the ground surface takes place. The size of the interception storage capacity depends on the vegetation type and its stage of development, which is characterised by the leaf area index, *LAI*.

Note The interception coefficient is a unit of length [mm] - not a rate. This means that the full amount is intercepted in every time step, if precipitation is available and the storage is not full. Thus, the total amount of intercepted water is time step dependent. For example, if you have a precipitation rate of 2 mm/hour over 12 hours, the total precipitation will be 24 mm. However, the total interception could range between 2 mm if the time step length is 12 hours to the full 24 mm, if the time step length is 1 hour, assuming that there is 2 mm of evapotranspiration per time step.

The amount of soil water, which can be intercepted by the vegetation canopy is determined by multiplying the interception capacity, C_{int}, by the LAI. C_{int} depends on the surface characteristics of the vegetation type. The units of C_{int} are [L], but they should be interpreted as [L]/(area of leaves)/(ground area). A typical value is 0.05 mm.

The calculation of soil evaporation contains two components, the basic soil evaporation which occurs regardless of soil dryness at moisture contents in the range θ_W - ½(θ_W + θ_F) and enhanced soil evaporation at moisture contents above ½(θ_W + θ_F). The fraction of the potential evapotranspiration, which is always allocated to the basic soil evaporation, is determined by C2. In the two-layer soil model described by Kristensen & Jensen (1975), this value was found to be 0.15. For dynamic simulation using the unsaturated zone description in MIKE SHE, a value of 0.2 was, however, found to give better results (Miljøstyrelsen,1981; Jensen, 1983).

The transpiration from the vegetation is regulated by two parameters. C1 is the slope of the linear relation between LAI and Ea/Ep, which determines at

which LAI the actual evapotranspiration equals the potential evapotranspiration at ample water supply. A typical value of C1 is 0.3. C3 regulates the influence of water stress on the transpiration process and may depend on the soil type with higher values for light soils than for heavier soils. The influence of soil dryness is reduced when C3 is increased. In Kristensen & Jensen (1975), a value of 10 mm was found for loamy soils. For simulations with the unsaturated zone description in MIKE SHE, a value of 20 mm was found more appropriate (Miljøstyrelsen,1981; Jensen, 1983).

The root distribution in the soil is regulated by the Aroot parameter. The value of Aroot may depend on soil bulk density with higher values for soils with high bulk density where root development may be more restricted than for soils with low bulk density. A typical value is 1 at which 60% of the root mass is located in the upper 20 cm of the soil at a root depth of 1 m. Lower Aroot values decrease this fraction and give a more even root distribution.

4 Snow Melt - Technical Reference

Snow melt is an important phenomena that can dramatically affect the spring runoff timing and volume. Therefore, a realistic description of the snow melt process is important.

The current snow melt module is a modified degree-day method. That is, the rate of melting increases as the air temperature increases. Degree-day based methods are commonly used in hydrology because the models require limited additional data and are relatively simple to calibrate.

Energy balance methods are also widely used, but require considerably more detailed input data. Further, the energy balance methods are often layer based (i.e. multi-layered snow models) and the energy balance is often calculated implicitly. Both of these factors could significantly impact the solution time.

In general, degree-day methods are most applicable in open and lightly forested areas. Whereas, energy balance methods are better for mountainous and densely forested areas. However, well-parameterized degree-day method, such as the one available in MIKE SHE, can usually be calibrated for all climatic conditions.

4.1 Air temperature correction

Air temperature varies linearily with elevation. Snow typically accumulates at higher elevations, but the temperature measuring network is rarely dense enough to represent the spatial variation of temperature that is known to exist in the catchment. Furthermore, the change in temperature with elevation depends on the relative humidity. Thus, the elevation corrected temperature in a cell, under dry conditions, is

$$T_{cell} = T_{ref} + (H_{cell} - H_{ref}) \cdot \beta_{dry} \tag{4.1}$$

and under wet conditions (when it is raining or snowing) is

$$T_{cell} = T_{ref} + (H_{cell} - H_{ref}) \cdot \beta_{wet} \tag{4.2}$$

where T_{cell} is the air temperature in the cell, T_{ref} is the measured air temperature at a weather station, H_{cell} is the elevation of the cell, H_{ref} is the elevation of the weather station, and b_{dry} and b_{wet} are the elevation correction factors (lapse rate) for temperature under dry and wet conditions. Wet conditions are defined as any time step were the precipitation rate is greater than zero.

Note The air temperature is also used for temperature dependent decay, so the air temperature elevation correction is available whenever air temperature needs to be specified.

Note Air temperatures are specified as instantaneous values. Thus, for air temperature time series, an average air temperature is used in each time step based on a linear interpolation between two instantaneous values.

4.2 Precipitation correction

Precipitation varies linearly with elevation. Similarly, precipitation varies spatially across the catchment, but the amount of local precipitaiton is also a function of the elevation. However, the different areas will have precipitation-elevation relationships. Thus, the elevation corrected precipitation in a cell, when the precipitation is greater than zero, is

$$P_{cell} = P_{ref} + (H_{cell} - H_{ref}) \cdot \beta_p \tag{4.3}$$

where P_{cell} is the precipitation in the cell, T_{ref} is the measured precipitation at a weather station, H_{cell} is the elevation of the cell, H_{ref} is the elevation of the weather station, and b_p is the elevation correction factor (lapse rate) for precipitation.

Note The precipitation is a basic input parameter in MIKE SHE, so the precipitation elevation correction is always available.

4.3 Snow melting and freezing

Snow melts in response to several climatic conditions, including air temperature, sun shine (solar radiation), the heat content of rain, and the heat transfer from condensing moisture in the air.

Snow moisture content

It is well known that melting snow does not immediately genenerate runoff. Rather, the snow gradually becomes wetter, like a sponge, until liquid water starts to drain out of the snow pack. If the temperature drops below freezing again, the liquid water will re-freeze. In MIKE SHE, this is conceptualized as two seperate snow storages - dry (or frozen) snow storage and wet (or liquid) snow storage.

Snow melt occurs by converting dry snow to wet snow. Surface runoff occurs when the ratio of dry to wet snow storage reaches a user specified maximum - the maximum wet snow storage fraction, where the wet snow storage fraction, W_{frac}, is calculated by

$$W_{frac} = \frac{S_{wet}}{S_{wet} + S_{dry}} \tag{4.4}$$

where S_{wet} and S_{dry} are the wet and dry snow storages respectively.

Air temperature melting

If the air temperature is above the threshold melting temperature, then the snow will begin to melt and the snow storage will be reduced by

$$M_T = C_T \cdot (T_{air} - T_0) \tag{4.5}$$

where M_T is the rate of melting due to the air temperature, C_T is the degree-day factor for snow melting (e.g in units of mm/day/C degree), T_{air} is the air temperature of the cell, and T_0 is the threshold melting temperature. The air temperature melting will be positive if the air temperature is above the threshold temperature and negative if it is below. Thus, if the air temperature falls below the threshold melting temperature, then wet snow will be reconverted back to dry snow.

Note In MIKE SHE, the degree-day factor is a time varying, spatially distributed value. This allows you to calibrate your snow melt over the winter season as the snow properties (e.g. albedo, density, etc) change.

Radiation melting

Solar radiation will cause the snow to melt at a rate proportional to the amount of incoming radiation. On cloudy days, the radiation intensity will be less, but still non-zero. Thus,

$$M_R = -C_{rad} \cdot R_{sw} \tag{4.6}$$

where M_R is the rate of melting due to incoming short wave radiation, C_{rad} is the radiation melting factor for snow melting (e.g in units of mm/kJ/m^2), R_{sw} is the amount of incoming solar radiation (e.g. in units of kJ/m^2/hour).

Energy melting

The condensation of moist air on snow and the heat released from liquid rain as it cools are important contributors to snow melt. Even though these energy sources are not physically simulated, the following linear relationship allows these processes to be included.

$$M_E = C_E \cdot P \cdot (T_{air} - T_0) \tag{4.7}$$

where M_E is the rate of melting due to the energy in liquid rain, C_E is the energy melting coefficient for the energy in liquid rain (e.g in units of mm/mm rain/C degree), T_{air} is the air temperature of the cell, and T_0 is the threshold melting temperature.

Energy melting only occurs if the air temperature is above the threshold melting temperature. In other words, the energy melting is aways positive. The temperature of the rain is assumed to be the same as the air temperature.

Note The energy melting coefficient is a constant value for the entire model.

Snow balance

If the air temperature is above the threshold melting temperature, then dry snow storage will be reduced (converted to wet snow) by combining the three melting rates.

$$M_{total} = M_T + M_R + M_E \tag{4.8}$$

If, on the other hand, the air temperature is below the threshold melting temperature, then the dry snow storage will be increased (wet snow converted to dry snow) by combining the freezing rate and the radiation melting rate, until the wet snow storage goes to zero

$$M_{total} = M_T + M_R \tag{4.9}$$

In this case, the temperature melting will be positive (i.e. freezing) and radiation melting will be negative.

4.3.1 Snow pack area coverage

When the snow storage is small, snow will not cover the enter cell area. Rather, the snow will be concentrated in sheltered areas. If a small snow thickness was spread out over the entire cell area, then melting would occur too quickly. Thus, MIKE SHE also includes a snow coverage function, where you provide a minimum snow depth for which you can assume that the entire cell is covered with snow. For snow depths less than this minimum, the area fraction is linearly reduced to zero when the snow storage goes to zero. Thus, the area factor, Af, is

$$A_f = \frac{S_T}{S_{min}} \tag{4.10}$$

where S_T is the total snow storage (wet + dry) and S_{min} is the minimum snow depth for full area coverage.

The melting/freezing rates are multiplied by the area factor before being applied. Furthermore, if the area factor is less than one, then the precipitation and ET is distributed between the snow and the normal ground surface based on the area factor.

4.3.2 ET from 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 has 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. This is found on the Snow Melt *(V1 p. 230)* dialog. 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)*

See, Reference Evapotranspiration *(V1 p. 225)* and the Crop Coefficient specified in the Vegetation Properties Editor *(V1 p. 373)*.

5 Working with Freezing and Melting - User Guide

Snow melt is an important phenomena that can dramatically affect the spring runoff timing and volume. Therefore, a realistic description of the snow melt process is important.

The snowmelt module in MIKE SHE is a modified degree-day method, whereby the rate of melting increases as the air temperature increases. The main input parameters required for the snowmelt process includes melting threshold temperature, degree day coefficient, minimum snow storage for full coverage, and maximum wet snow fraction as well as functions to account for incoming solar radiation, heat content of rainfall, and sublimation.

Degree-day based methods are commonly used in hydrology because the models require limited additional data and are relatively simple to calibrate. In general, degree-day methods are most applicable in open and lightly forested areas. A well-parameterized degree-day method, such as the one available in MIKE SHE, can usually be calibrated for all climatic conditions.

5.1 Elevation lapse rates

Snow typically accumulates at higher elevations, but the temperature measuring network is rarely dense enough to represent the spatial variation of temperature that is known to exist in the catchment. Thus, MIKE SHE allows you to correct the temperature and precipitation with elevation. The change in value with elevation is called the lapse rate.

5.1.1 Air temperature lapse rate

Air temperature decreases with elevation. MIKE SHE assumes that this decrease is linear with the elevation. However, the lapse rate depends on the relative humidity. So, MIKE SHE includes two different lapse rates - one for dry conditions and the other for wet conditions (i.e. when it is raining or snowing). The exact formulae are shown in Equations(4.1) and (4.2). Wet conditions are defined as any time step were the precipitation rate is greater than zero.

Note Air temperatures are specified as instantaneous values. Thus, for air temperature time series, an average air temperature is used in each time step based on a linear interpolation between two instantaneous values.

5.1.2 Precipitation lapse rate

Precipitation tends to increase with elevation. MIKE SHE assumes that this increase is linear. Similarly, precipitation varies spatially across the catchment, but the amount of local precipitaiton is also a function of the elevation. However, the different areas will have precipitation-elevation relationships.

So, for precipitation, it may be important to have a distribution of lapse rates that depends on the slope orientation, etc. Thus, it is likely necessary to have locally varying lapse rates that depend on the orientation of valleys.

5.2 Evapotranspiration

ET is also very sensitive to elevation. However, the function is rather complex. The lower temperatures will reduce ET with elevation. However, the wind speed may be higher which will increase ET. If there is more rainfall at higher elevations, or if the humidity is higher, then this will also act to reduce ET. And, solar radiation may be more intense at higher elevations.

There is currently no means to automatically adjust ET with elevation. So, if you need to consider ET versus elevation, then you will have to generate either a spatially varying ET grid, or local time series of ET and apply these in different elevation zones.

Overall, if your winter evapotranspiration is too high, you will underestimate your snow pack accumulation.

5.3 Time varying infiltration

A common characteristic in cold climates is that infiltration is reduced during the winter months. When the air temperature is cold enough to maintain precipitation as snow, then infiltration will be limited in any case. However, in the spring, when snow storage is melting, then infiltration may still be limited for some period of time.

There is an Extra Parameter time varying infiltration function to reduce infiltration in winter. The time varying infiltration function is a modification of the Surface-Subsurface Leakage Coefficient *(V1 p. 260)* to allow it to be time varying.

The time varying dfs2 file must cover the model domain and the model simulation period.

Parameter Name	Type	Value
time varying ol leakage coefficient	Boolean	On
leakage coefficient dfs2 file name	filename	.dfs2 file
leakage coefficient item number	integer	item number in dfs2 file, greater than zero
Optional		
mean step accumulated leakage coefficient	Boolean	On

By default, the Time Series Types *(V1 p. 143)* is Instantaneous, but their is an option that allows you to use Mean Step Accumulated values. The use of Mean Step Accumulated does not change the meaning of the item, but changes the way the values are interpolated.

Note: The code does not check for the time series type.

Note The areas in which these values will be applied has not changed. The areas are defined in the original Surface-Subsurface Leakage Coefficient *(V1 p. 260)* dialogue. That is, the values specified in the User Interface act as a mask, where the leakage coefficient is active if a non-delete value is specified in the user interface.

5.4 Snow water equivalent

MIKE SHE does not distinguish between rainfall and snowfall. If the rainfall is in a period where the air temperature is below the Melting threshold, then the rainfall accumulates as snow - as a Snow Water Equivalent (SWE). In other words, you cannot compare the MIKE SHE results against snow pillow depths. These must be converted to an equivalent depth of water - the Snow Water Equivalent.

6 Overland Flow - Technical Reference

When the net inflow of water onto the cell (eg rainfall or runofff to the cell) exceeds the infiltration capacity of the soil, water ponds on the ground surface. This water is available as surface runoff, to be routed downhill towards the river system. The exact route and quantity is determined by the topography and flow resistance, as well as the losses due to evaporation and infiltration along the flow path.

The water flow on the ground surface is calculated by MIKE SHE's Overland Flow Module, using the diffusive wave approximation of the Saint Venant equations, or using a semi-distributed approach based on the Mannings equation. Alternatively ponded water can be routed directly to a river, boundary or internal depression using the OL Drainage function.

This chapter is the technical reference for the Overland Flow Module in MIKE SHE.

6.1 Finite Difference Method

The Finite Difference method is the most common method used in MIKE SHE. With the Finite Difference Method, you have the choice of two different numerical solvers. The SOR solver is an implicit matrix solver that is somewhat faster than the Explicit solver, but the increased speed is traded off for lower accuracy. Both are using the Diffusive Wave Approximation to the St Venant Equations.

6.1.1 Diffusive Wave Approximation

Using rectangular Cartesian (x, y) coordinates in the horizontal plane, let the ground surface level be z_g (x, y), the flow depth (above the ground surface be $h(x, y)$, and the flow velocities in the x- and y-directions be $u(x, y)$ and $v(x, y)$ respectively. Let $i(x, y)$ be the net input into overland flow (net rainfall less infiltration). Then the conservation of mass gives

$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x}(uh) + \frac{\partial}{\partial y}(vh) = i \tag{6.1}$$

and the momentum equation gives

$$S_{fx} = S_{Ox} - \frac{\partial h}{\partial x} - \frac{u}{g}\frac{\partial u}{\partial x} - \frac{1}{g}\frac{\partial u}{\partial t} - \frac{qu}{gh} \tag{6.2a}$$

$$S_{fy} = S_{Oy} - \frac{\partial h}{\partial y} - \frac{v}{g}\frac{\partial v}{\partial y} - \frac{1}{g}\frac{\partial v}{\partial t} - \frac{qv}{gh} \tag{6.2b}$$

where S_f is the friction slopes in the x- and y-directions and S_O is the slope of the ground surface. Equations (6.1), (6.2a) and (6.2b) are known as the St. Venant equations and when solved yield a fully dynamic description of shallow, (two-dimensional) free surface flow.

The dynamic solution of the two-dimensional St. Venant equations is numerically challenging. Therefore, it is common to reduce the complexity of the problem by dropping the last three terms of the momentum equation. Thereby, we are ignoring momentum losses due to local and convective acceleration and lateral inflows perpendicular to the flow direction. This is known as the diffusive wave approximation, which is implemented in MIKE SHE.

Considering only flow in the x-direction the diffusive wave approximation is

$$S_{fx} = S_{Ox} - \frac{\partial h}{\partial x} = -\frac{\partial z_g}{\partial x} - \frac{\partial h}{\partial x} \tag{6.3}$$

If we further simplify Equation (6.3) using the relationship $z = z_g + h$ it reduces to

$$S_{fx} = -\frac{\partial}{\partial x}(z_g + h) = -\frac{\partial z}{\partial x} \tag{6.4}$$

in the x-direction. In the y-direction Equation (6.4) becomes

$$S_{fy} = -\frac{\partial}{\partial y}(z_g + h) = -\frac{\partial z}{\partial y} \tag{6.5}$$

Use of the diffusive wave approximation allows the depth of flow to vary significantly between neighbouring cells and backwater conditions to be simulated. However, as with any numerical solution of non-linear differential equations numerical problems can occur when the slope of the water surface profile is very shallow and the velocities are very low.

Now, if a Strickler/Manning-type law for each friction slope is used; with Strickler coefficients *Kx* and *Ky* in the two directions, then

$$S_{fx} = \frac{u^2}{K_x^2 h^{4/3}} \tag{6.6a}$$

$$S_{fy} = \frac{v^2}{K_y^2 h^{4/3}} \tag{6.6b}$$

Substituting Equations (6.4) and (6.5) into Equations (6.6a) and (6.6b) results in

$$\frac{u^2}{K_x^2 h^{4/3}} = -\frac{\partial z}{\partial x} \tag{6.7a}$$

$$\frac{v^2}{K_y^2 h^{4/3}} = -\frac{\partial z}{\partial y} \tag{6.7b}$$

After simplifying Equations (6.7a) and (6.7b) and multiply both sides of the equations by *h*, the relationship between the velocities and the depths may be written as

$$uh = K_x\left(-\frac{\partial z}{\partial x}\right)^{1/2} h^{5/3} \tag{6.8a}$$

$$vh = K_y\left(-\frac{\partial z}{\partial y}\right)^{1/2} h^{5/3} \tag{6.8b}$$

Note that the quantities *uh* and *vh* represent discharge per unit length along the cell boundary, in the x- and y-directions respectively.

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

6.1.2 Finite Difference Formulation

Consider the overland flow in a small region (see Figure 6.1) of a MIKE SHE model, having sides of length *Dx* and *Dy* and a water depth of *h*(*t*) at time *t*.

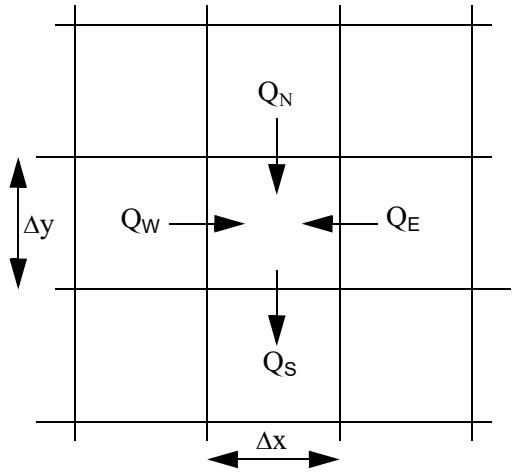


Figure 6.1 Square Grid System in a small Region of a MIKE SHE model

A finite-difference form of the velocity terms in Eq. (6.1) may be derived from the approximations

$$\frac{\partial}{\partial x}(uh) \cong \frac{1}{\Delta x}\{(uh)_{east} - (uh)_{west}\} \tag{6.9}$$

and

$$\frac{\partial}{\partial y}(vh) \cong \frac{1}{\Delta y}\{(vh)_{north} - (vh)_{south}\} \tag{6.10}$$

where the subscripts denote the evaluation of the quantity on the appropriate side of the square, and noting that, for example, *Dx (uh)west* is the volume flow across the western boundary

$$\Delta h = h(t+\Delta t) - h(t) = I + \frac{\Sigma Q \Delta t}{\Delta x^2} \tag{6.11}$$

where

$$I = i\Delta x^2, \tag{6.12}$$
$$\Sigma Q = Q_N + Q_S + Q_E + Q_W$$

and where, i is the net input to overland flow in Eq. (6.1) and the Q's are the flows into the square across its north, south, east and west boundaries evaluated at time *t*.

Now consider the flow across any boundary between squares (see Figure 6.2), where Z_U and Z_D are the higher and lower of the two water levels referred to datum. Let the depth of water in the square corresponding to Z_U be h_U and that in the square corresponding to Z_D to h_D.

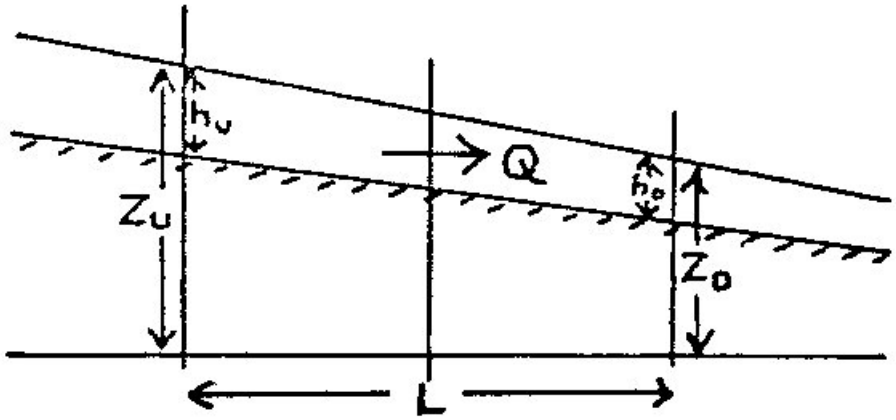


Figure 6.2 Overland flow across grid square boundary.

Equations (6.8a) and (6.8b) may be used to estimate the flow, Q, between grid squares by

$$Q = \frac{K\Delta x}{\Delta x^{1/2}}(Z_U - Z_D)^{1/2}\ h_u^{5/3} \tag{6.13}$$

where *K* is the appropriate Strickler coefficient and the water depth, h_u, is the depth of water that can freely flow into the next cell. This depth is equal to the actual water depth minus detention storage, since detention storage is ponded water that is trapped in shallow surface depressions. Equation (6.13) also implies that the overland flow into the cell will be zero if the upstream depth is zero. The flow across open boundaries at the edge of the model is also calculated with Eq. (6.13), using the specified boundary water levels.

6.1.3 Successive Over-Relaxation (SOR) Numerical Solution

The method for solving the overland flow equations is similar to the method applied to the saturated zone flow. A linear matrix system of N equations with N unknown water levels is derived. The matrix is then solved iteratively, using the modified Gauss Seidel method. Because of the non-linear relationship between water levels and flows, the 2nd order term is included in the Taylor series expressing the correction of water levels as a function of the residuals.

The flow is calculated for the remainder of any iteration using Eq. (6.13) whenever there is sufficient water in a cell, that is, whenever h_u exceeds the minimum threshold that is specified by the user.

The exchange between ponded water and the other hydrologic components (e.g. direct exchange with the saturated zone, unsaturated infiltration, and evaporation) is added or subtracted from the amount of ponded water in the cell at the beginning of every overland flow time step.

Water balance correction

As the flow equations, so to speak, are explicit during one iteration, it is necessary to reduce the calculated flows in some situations to avoid internal water balance errors and divergence of the solution scheme.

Thus, requiring that the water depth cannot be negative, which implies that *Dh ³ -h(t)*, rearranging Eq. (6.11) gives:

$$\Sigma Q \geq \frac{-\Delta x^2 h(t)}{\Delta t} - I \tag{6.14}$$

where *SQ* is the sum of outflows and inflows.

Splitting *SQ* into inflows and outflows and remembering that outflow is negative, gives:

$$\Sigma \left| Q_{out} \right| \leq \Sigma Q_{in} + I + \frac{\Delta x^2 h(t)}{\Delta t} \tag{6.15}$$

remembering that $I = i\Delta x^2$ and *i* is the net input into overland flow (net rainfall less infiltration).

If necessary during an iteration, these calculated outflows are reduced to satisfy the equal sign of (6.15).

To ensure that the inflows, *SQin*, have been summed before calculating *SQout*, the grid squares are treated in order of descending ground levels during each iteration.

6.1.4 Explict Numerical Solution

The explicit solution is different from the SOR solution in the sense that there is no iterative matrix solution. In other words, the exchange flows between every cell and to the river are simply calculated based on the individual cell heads. However, the explicit solution is much more restrictive in terms of time step. For the explicit solution to be stable, the flow must be slow relative to the time step. For example, a flood wave cannot cross a cell in one time step. So, this leads to the following 3-step calculation process for the explicit solution of the overland flow:

1. Calculate all flow rates and discharges between cells and between the overland cells and river links based on the current water levels
2. Loop over all the cells and calculate the maximum allowed time step length for the current time step, based on the following criteria
   - Courant criteria (see next section)
   - Cell volume criteria - the volume in the cell divided by the flow rate
   - River link volume criteria - the volume in the river link divided by the flow rate
   - River bank criteria - the exchangeable volume in the river link based on the river bank elevation divided by the flow rate

   In most cases, the Courant criteria is the critical criteria for the maximum time step, with the Cell volume criteria sometimes being critical. The River link and River bank criteria are less commonly critical, but may become critical when the river is very shallow.
3. Calculate the actual flows between the all the cells and to/from the river links using the maximum allowed time step and update all the cell water depths.

Courant criteria

Courant number, C, represents the ratio of physical wave speed to the 'grid speed' and is calculated as

$$C = \frac{\frac{dQ}{dA}}{\frac{dx}{dt}} = \frac{1}{dx} \times \frac{dQ}{dh} \times \frac{dt}{dx} \tag{6.16}$$

where dQ is the change in flow rate, dA is the change in cross-sectional area and dh is the change in water depth.

For a stable explicit solution the courant number must be less than one, which means that a wave cannot travel across a cell in less than one time step.

Solving for the time step, yields,

$$\Delta t = \frac{dh}{dQ} \cdot \Delta x^2 \cdot C \tag{6.17}$$

where Δt is the time step and Δx is the cell width.

The differential term in (6.17) is the inverse of the derivative of the Mannings equation, (6.13), with respect to h, which goes to zero as the change in water

depth approaches zero. Thus, flat areas with flat water levels will require very small time steps. Likewise, smaller grid spacing will also lead to smaller time steps.

6.1.5 Boundary conditions

The default outer boundary condition for the overland flow solver is a specified head, based on the initial water depth in the outer nodes of the model domain. Thus, if the water depth inside the model domain is greater than the initial depth on the boundary, water will flow out of the model. If the water depth is less than the initial depth on the boundary, the boundary will act as a source of water.

There is an Extra Parameter option that allows you to define a time varying outer boundary condition for Overland Flow. In this case, the outer boundary is still a fixed head boundary, but the fixed head is defined by a time-varying dfs2 file, instead of the initial water depth.

The boundary condition to the River can be calculated based on a Mannings resistance formulation, where the length is the distance from the node to the edge of the cell, the head difference is that between the cell and the River Link, and the depth of water is the depth above the detention storage.

The boundary condition to the river can also be calculated based on a standard weir formulation. **The weir formulation must be used if two-way flow is calculated between the river and the cell.**

In this case, the weir formula is a standard expression based on the Villemonte formula:

$$Q = W \cdot C \cdot (H_{US} - H_W)^k \cdot \left[1 - \left(\frac{H_{DS} - H_w}{H_{US} - H_w}\right)^k\right]^{0.385} \quad (6.18)$$

where Q is the discharge over the weir, W is the weir width, C is the weir coefficient, k is the weir exponent, H_{US} is the upstream depth, H_{DS} is the downstream depth and H_W is the crest level of the weir.

As the upstream water depth approaches zero, the flow over the weir becomes undefined. To prevent numerical problems, the flow is reduced linearly to zero when the water depth is below the user-defined minimum depth for full weir flow (assumes a triangular weir cross-section).

A minimum flow area threshold is defined to reduce numerical issues with small volumes of transfer from the river to the floodplain, The flow area is calculated by dividing the volume of water in the Coupling Reach by the length of the Coupling Reach. If the calculated area is less than the minimum flow area, then overbank spilling from the river is excluded.

Note: the minimum flow area threshold could unduly restrict overbank spilling if you have a long Coupling Reach and a small river section where overbank spilling occurs.

6.1.6 Low gradient damping function

In flat areas with stable ponded water, both the flow gradient and the water depth difference between grid cells will be zero or nearly zero. Equation (6.17) implies that as *dh* goes to zero Δt also goes to zero. To allow the simulation to run with longer time steps and dampen any numerical instabilities in areas with very low lateral flow, the calculated intercell flows are multiplied by a damping factor when the flow gradient between cells is close to zero.

Essentially, the damping factor reduces the flow between cells. You can think of the damping function as an increased resistance to flow as the flow between cells goes to zero. In other words, the flow goes to zero faster than the time steps goes to zero. This makes the solution more stable and allows for larger time steps. However, the resulting intercell flow will be lower than that calculated using the Mannings equation in the affected cells and neighbouring water levels will take longer to equilibrate. At very low flow gradients this is normally insignificant, but as the flow gradient increases the differences could become noticeable. Therefore, the damping function is only applied below a user-defined flow gradient.

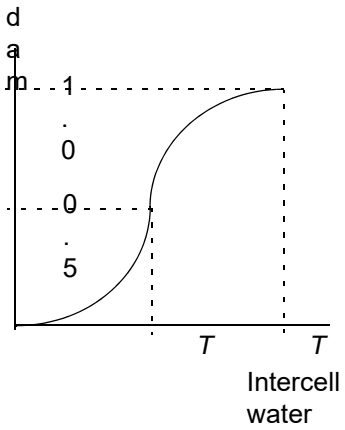


Figure 6.3 Default damping function for numerical stability at low gradients

The default damping function is a pair of parabolic equations (see Figure 6.3). When the flow gradient between cells reaches the threshold the following damping function is applied

$$F_D = 1 - 2\left(\frac{T \cdot dx - dZ}{T \cdot dx}\right)^2 \qquad (6.19)$$

where T is the user-specified threshold gradient, dx is the cell size and dZ is the water level difference between the two nodes. When the water level difference reaches $T\,dx/2$, the damping function changes to

$$F_D = 2\left(\frac{dZ}{T \cdot dx}\right)^2 \tag{6.20}$$

which goes to zero as the water level difference goes to zero.

In the log-log graph in Figure 6.4, you can see that the lateral flow betwen cells drops off very quickly when the Threshold is active. For example, if the difference between the surface water gradient and the Threshold gradient is 0.1, then the actual flow will only be 2% of the original flow. If the difference is 0.01, then overland flow will be essentially turned off between cells.

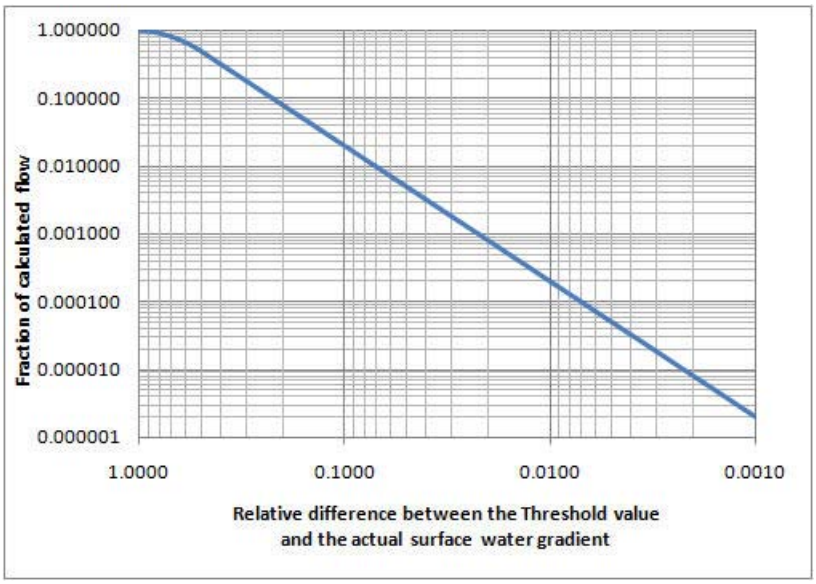


Figure 6.4 Actual overland flow as a function of the difference between the threshold gradient and the surface water gradient based on Equations (6.19) and (6.20).

Alternative damping function

An alternative damping function is available as an Extra Parameter that goes to zero more quickly and is consistent with the function used in MIKE FLOOD.

The alternative function is a single parabolic function (see Figure 6.5)

$$F_D = 1 - \left(\frac{T \cdot dx - dZ}{T \cdot dx}\right)^2 \tag{6.21}$$

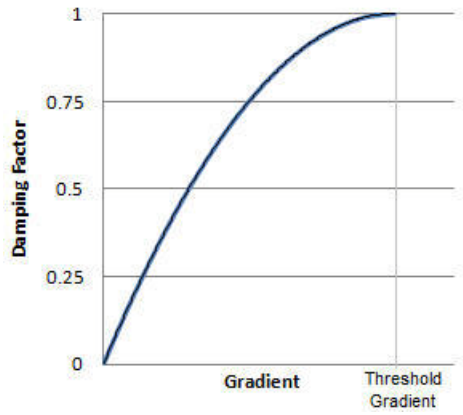


Figure 6.5 Alternative damping function activated by an Extra Parameter.

To activate the alternate function, you must specify the Alternative low gradient damping function *(V2 p. 337)* boolean parameter in the Extra Parameters *(V1 p. 334)* dialog.

For both functions and both the explicit and implicit solution methods, each calculated intercell flow in the current timestep is multiplied by the local damping factor, F_D, to obtain the actual intercell flow. In the explicit method, the flow used to calculate the courant criteria are also corrected by F_D.

The damping function is controlled by the user-specified threshold gradient (see Common stability parameters *(V1 p. 185)* for the Overland Flow), below which the damping function becomes active.

6.1.7 Ponded drainage

The amount of runoff generated by MIKE SHE depends on the density of the stream network defined in MIKE Hydro River. If the stream network is dense, then you will get more runoff in the model because the lateral travel time to the streams will be short, and the infiltration and evaporation losses will be smaller.

The Ponded Drainage function has been designed to facilitate a flexible definition of storm water drainage in both natural and urbanized catchments. In particular, the Ponded Drainage function supports the following features:

- In urban areas, paving and surface sealing (eg roof tops) limits infiltration and enhances runoff.
- Ponded water is routed to user-defined locations (boundaries, depressions, manholes and streams).
- The time varying storage in the drainage system is accounted for by different inflow and outflow rates.

- Drainage rates can be controlled by drain levels, drainage time constants and maximum outflow rates.
- Drainage is restricted if the destination water level is higher than the source water level.
- Urban development over time can be simulated by time varying drainage parameters.
- Overland flow is calculated normally for ponded water that does not drain in the current time step.

Conceptually, the Ponded Drainage function is shown in Figure 6.6.

Figure 6.6 Conceptualization of the Ponded Drainage function

Calculation order

The Ponded Drainage is calculate explicitly in a particular order with respect to the other hydrologic processes.

1. Rainfall is added to the ponded depth in the cell.
2. Then, OL Drainage will be immediately calculated.

In this way, the OL Drainage essentially acts on rainfall and removes rainfall before anything else happens.

If there is any ponded water remaining on the cell after the drainage has been removed, then it will be removed in the following order until no ponded water remains.

1. Evaporation is removed from the ponded water;
2. Then, Infiltration to the unsaturated zone will be calculated; and
3. Finally, Lateral Overland Flow will be calculated to the adjacent cells.

The Ponded Drainage is calculated for every cell independently. That is, the drainage that is calculated is not added to the destination cells until the end of the calculation for all the cells. This prevents circular drainage, and means that the order in which the calculation occurs is not relevant. After the ponded drainage is calculated, the depths in all the cells are updated - both sources and destinations. The calculation of ET, infiltration and lateral OL flow is based on the updated ponded depths.

Calculation method

Mathematically, the Ponded Drainage is calculated similarly to the Saturated Zone Drainage with the SOL solver.

The new ponded water level at the end of the time step is calculated explicitly from the flow balance equation

$$\Delta S = (Q_{dr} + \Sigma q)\Delta t \tag{6.22}$$

where ΔS is the storage change per unit area as a result of a drop in ponding, Q_{dr} is the specific outflow into the drain per unit area and Σq represents all other flow terms in a computational node.

Since, the ponded drainage is calculated independently in every cell, and before ET, infiltration and lateral overland flow, the Σq is zero in this formulation.

The change in storage per unit area can also be calculated from

$$\Delta S = d_0 - d_t \tag{6.23}$$

where d_0 is the depth of water above the drain at the beginning of the time step and d_t is the depth of water above the drain after drainage occurs.

Q_{dr} is calculated based on the mean depth of ponded water during the time step. Thus,

$$Q_{dr} = C_{dr} \cdot \frac{d_0 + d_t}{2} \tag{6.24}$$

where C_{dr} is the drain leakage coefficient, or time constant, in units of [1/time].

Substituting (6.23) and (6.24) into (6.22) and rearranging, the water depth after drainage is calculated, d_t, can be determined by

$$d_t = \frac{d_0\left(1 - \frac{C_{dr}\Delta t}{2}\right)}{1 + \frac{C_{dr}\Delta t}{2}} \tag{6.25}$$

From which the new ponded water level elevation, h_t, after drainage is calculated can be determined by

$$h_t = Z_{dr} + d_t \tag{6.26}$$

where Z_{dr} is the elevation of the drain.

Constraints

Several constraints have been added to the formulation to make the Ponded Drainage function more practical.

- **Paved Fraction** - a paved fraction for the cell is used to reduce infiltration in urban areas.
- **Runoff Fraction** - a runoff fraction for the cell is used to partition the ponded water into a fraction that can drain to the Ponded Drainage network, and a fraction that flows laterally with the Finite Difference Overland Flow.
- **Detention storage** - the Ponded Drainage will only affect the depth of ponding above the Detention storage. However, this can be changed by means of an Extra Parameter.
- **Drain Level** - the Ponded Drainage will occur only when the ponded depth exceeds the Drain Level
- **Maximum drainage rate** - a maximum drainage rate ensures that culvert capacities are not exceeded
- **Downstream depth criteria** - there is an optional check to ensure that the downstream drainage destination does not have a higher water level.
- **Inflow time constant** - the rate of inflow to the drains is controlled by a time constant.
- **Outflow time constant** - the rate of outflow to the destination is controlled by a time constant. If the outflow rate is less than the inflow rate, then water will accumulate on a cell-by-cell basis in the “drain”. There is currently no limit on the amount of water than can accumulate in the drain.

Reference Drainage System

The Reference System for linking drainage source and destination is calculated by the Pre-Processor. By default the drainage reference system is calculated downhill until either a local depression is found, or a stream or local boundary is encountered.

Optionally, the drainage can be routed to specified MIKE Hydro River nodes, or MIKE Urban Manholes.

6.1.8 Multi-cell Overland Flow Method

The resolution of available topography data is often greater than the practical discretization of the overland flow model. This means that important topographic detail is often neglected in the overland flow model. The multi-cell overland flow method attempts to mitigate this discrepancy.

In the Multi-cell overland flow method, high resolution topography data is used to modify the flow area used in the St Venant equation and the courant criteria. The method utilizes two grids - a fine-scale topography grid and a

coarser scale overland flow calculation grid. However, both grids are calculated from the same reference data - that is the detailed topography digital elevation model.

In the Multi-cell method, the principle assumption is that the volume of water in the fine grid and the coarse grid is the same. Thus, given a volume of water, a depth and flooded area can be calculated for both the fine grid and the coarse grid. See Figure 6.7.

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

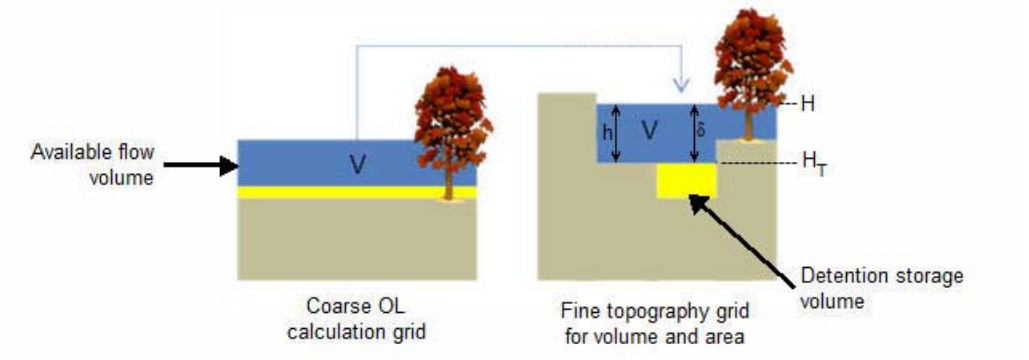


Figure 6.7 The constant volume from the coarse grid is transfered to the fine scale grid.

According to the Gauckler-Manning-Strickler formula, overland sheet flow can be expressed as

$$Q = A_f(h) \cdot M \cdot R(h)^{2/3} \cdot \sqrt{\frac{\Delta h}{\Delta x}} \tag{6.27}$$

Where A_f is the flow area, M is the Manning number, h is the water depth, Δx is the grid size and R is the hydraulic radius which, for practical reasons, is replaced by the resistance radius in the multi-cell method. Thus,

$$\frac{\partial Q}{\partial A_f} = \frac{\partial Q}{\partial d} \cdot \frac{\partial h}{\partial A_f} \tag{6.28}$$

which expands to

$$\frac{\partial Q}{\partial A_f} = \frac{\partial h}{\partial A_f} \cdot \left(\frac{\partial A_f}{\partial h} \cdot M \cdot R^{2/3} \cdot \sqrt{\frac{\Delta h}{\Delta x}} + A_f \cdot M \cdot \frac{2}{3} R^{(-1/3)} \cdot \right. \tag{6.29}$$

$$\left. \frac{\partial R}{\partial h} \cdot \sqrt{\frac{\Delta h}{\Delta x}} + A_f \cdot M \cdot R^{2/3} \cdot \frac{1}{2\sqrt{\frac{\Delta h}{\Delta x}}} \cdot \frac{1}{\Delta x} \cdot \frac{\partial \Delta h}{\partial h} \right)$$

For any cross-section of flow, the resistance radius is calculated by

$$\sqrt{R} = \frac{1}{A_f} \cdot \int_0^B h^{\frac{3}{2}} dx \tag{6.30}$$

where again h is the local water depth and B is the total width of the cross-section.

Thus, in a cross-section divided into n equal sub-grids,

$$\sqrt{R} = \frac{1}{A_f} \sum_1^n h_i^{\frac{3}{2}} \frac{B}{n} \tag{6.31}$$

where h_i is the local water depth in each sub-grid. Since, A_f is equal to

$$A_f = \sum_1^n h_i \frac{B}{n} \tag{6.32}$$

The resistance radius can be simplified to

$$R = \frac{\left(\sum h_i^{\frac{3}{2}} \right)^2}{\left(\sum h_i \right)^2} \tag{6.33}$$

However, since parts of the sub-section may be dry, or the entire depth may not be available for flow because of detention storage, the actual resistance radius in the cross section is

$$R = \frac{\left(\sum \delta_i^{\frac{3}{2}} \right)^2}{\left(\sum \delta_i \right)^2} \tag{6.34}$$

where

$$\delta_i = \begin{cases} \{H - H_T\}_i & H > H_T \\ 0 & H \le H_T \end{cases} \tag{6.35}$$

in which H is the water level in the refined cross-section and H_T is the maximum of the topography and level of detention storage in the current sub-grid.

Now, we need to apply the above formulation to a 2D coarse grid cell. In this case, the flow area and resistance radius is an average of the values along each row and column in the cell.

Thus, from Equation (6.32), the flow area for the x and y flow directions for each coarse grid cell is

$$A_{fx} = A_{fy} = \frac{1}{n}\sum_1^n \sum_1^n \delta_i \frac{\Delta X}{n} = \frac{\Delta X}{n^2}\sum_1^n \delta_i \tag{6.36}$$

and from Equation (6.34), the resistance radius for the x and y flow directions for each coarse grid cell is

$$R_x = R_y = \frac{1}{n}\sum_1^n \frac{\left(\sum_1^n \delta_i^{\frac{3}{2}}\right)^2}{\left(\sum_1^n \delta_i\right)^2} \tag{6.37}$$

Finally, the courant criteria also includes the flow area. Thus, from Equation

$$C = \frac{\frac{dQ}{dA_f}}{\frac{dx}{dt}} = \frac{dQ}{dA_f} \cdot \frac{dt}{dx} \tag{6.38}$$

which implies that the maximum allowed time step, Δt,when using the multi-grid method is

$$\Delta t = \frac{\Delta x}{\frac{dQ}{dA_f}} \cdot C \tag{6.39}$$

where Δx is the coarse overland flow grid size, the denominator is defined by Equation (6.29), and C is the user-specified courant criteria.

Implementation and Limitations

When using the multi-cell overland flow method, each overland flow cell is divided into an integer number of sub-grid cells (e.g. 4, 9, 16, 25, etc). The elevation of the coarse grid nodes and the fine grid nodes are calculated based on the input data and the selected interpolation method. However, the coarse grid elevation is adjusted such that it equals the average of the fine grid nodal elevations. This provides consistency between the coarse grid and fine grid elevations and storage volumes. Therefore, there may be slight differences between the cell topography elevations if the multi-cell method is turned on or off. This could affect your model inputs and results that depend on the topography. For example, if you initial water table is defined as a depth to the water table from the topography.

Overland flow exchange with MIKE Hydro River does not consider the multi-cell method. That is, flow into and out of the River Links is controlled by the water level calculated from the elevation defined in the coarse grid cell. Likewise the flow area for exchange with MIKE Hydro River is calculated as the coarse water depth times the overall grid size. Also, the elevation used when calculating flood inundation with flood codes only considers the average cell depth of the coarse grid.

However, if you choose to modify the topography based on a bathymetry file, or the MIKE Hydro River cross-sections, then this information will be used when calculating the multi-cell elevations.

Evaporation is adjusted for the area of ponded water in the coarse grid cell. That is, evaporation from ponded water is reduced by a ponded area fraction, calculated by dividing the area of ponding in the fine grid cell by the total cell area.

Infiltration from ponded water to the underlying UZ column or exchange between SZ and OL does not consider the multi-cell elevations. In both of these cases, only the average coarse grid elevation is used.

6.2 Simplified Overland Flow Routing

The conceptual reservoir representation of overland flow in MIKE SHE is based on an empirical relation between flow depth and surface detention, together with the Manning equation describing the discharge under turbulent flow conditions (Crawford and Linsley, 1966). This was implemented in the Standford watershed model and in its descendants, such as HSPF (Donigian et al., 1995), and has been applied in other codes such as the WATBAL model (Knudsen, J. 1985a,b; Refsgaard and Knudsen, 1996).

In the following, a description of the principles behind the model and the governing equations implemented and solved in MIKE SHE are presented. It is implicitly assumed that the equations derived for a hill slope can be applied to describe overland flow, in a lumped manner, across a catchment.

6.2.1 Theoretical basis

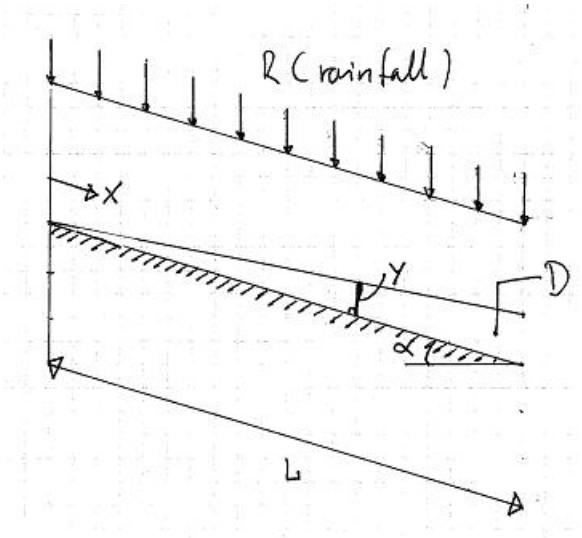


Figure 6.8 Schematic of overland flow on a plane

Figure 6.8 represents a schematic of overland flow on a planar surface of infinite width with uniform rainfall. Precipitation falls on the plane, builds on the surface in response to the surface roughness, and flows down the slope in the positive x-direction. In the figure, *L* is the length of the slope, *y* is the local depth of water on the surface at any point along the surface and α is the slope. Then, from the continuity equation

$$\frac{\partial q}{\partial x} = R - \frac{\partial y}{\partial t} \tag{6.40}$$

where q is the specific discharge.

For turbulent flow on a plane of infinite width, the Manning equation is

$$q = M \cdot y^{5/3} \sqrt{\alpha} \ [m^2/s] \tag{6.41}$$

where M is the Mannings M.

Now, at equilibrium, the depth no longer changes and the specific discharge approaches the rainfall rate

$$\frac{\partial y}{\partial t} = 0 \Rightarrow \frac{\partial q}{\partial x} = R \Rightarrow q_e = R \cdot x \tag{6.42}$$

where q_e is the equilibrium specific discharge.

Then, at equilibrium, the volume of water detained on the surface, D_e, can be calculated by

$$De = \int_o^L y dx = \int_o^L \left(\frac{q_e}{M \cdot \sqrt{\alpha}}\right)^{3/5} dx = \int_o^L \left(\frac{R \cdot x}{M \cdot \sqrt{\alpha}}\right)^{3/5} dx$$

$$De = \frac{5}{8} \cdot \frac{R^{3/5} \cdot L^{8/5}}{M^{3/5} \cdot \alpha^{3/10}} \; [m^3/m] \tag{6.43}$$

The depth, y, near the leading edge of the flow plane can be related to the depth at equilibrium by

$$y = \left(\frac{t}{t_e}\right) \cdot y_e \tag{6.44}$$

where t is the time and t_e is the time until the equilibrium is reached.

Then from (6.41) we can write

$$q = M \cdot \left(\frac{t}{t_e}\right)^{5/3} \cdot y_e^{5/3} \sqrt{\alpha} \; [m^2/s] \tag{6.45}$$

Now if we integrated the specific discharge from time 0 to when the equilibrium is reached, we can calculate the total volume discharged, Q, (per unit width of the plane) by

$$Q = \int_o^{t_e} q dt = \int_o^{t_e} M \cdot \left(\frac{t}{t_e}\right)^{5/3} \cdot y_e^{5/3} \cdot \sqrt{\alpha} (dt)$$

$$Q = \frac{3}{8} \cdot M \cdot y_e^{5/3} \cdot \sqrt{\alpha} \cdot t_e \; [m^2] \tag{6.46}$$

From (6.45), at equilibrium, ($t = t_e$), the depth of water at the leading edge of the plane (x = L) is

$$q = M \cdot y_e^{5/3} \sqrt{\alpha} = R \cdot L \tag{6.47}$$

which yields

$$y_e^{5/3} = \frac{R \cdot L}{M \cdot \sqrt{\alpha}} \; [m^{5/3}] \tag{6.48}$$

From continuity, the total volume of inflow up until equilibrium must equal the total outflow minus the amount retained on the surface. Thus,

Inflow - Outflow = Surface storage

which from equations (6.46) and (6.43) yields

$$(R \cdot L \cdot t_e) - \left(\frac{3}{8} \cdot M \cdot y_e^{5/3} \cdot \sqrt{\alpha} \cdot t_e\right) = \left(\frac{5}{8} \cdot \frac{R^{3/5} \cdot L^{8/5}}{M^{3/5} \cdot \alpha^{3/10}}\right) \quad (6.49)$$

which, when simplified, yields the time to reach equilibrium

$$t_e = \frac{L^{3/5}}{R^{2/5} \cdot M^{3/5} \cdot \alpha^{3/10}} = \frac{8}{5} \cdot \frac{De}{R \cdot L} \quad (6.50)$$

If we now assume that the flow on the sloping plane is uniform, that is the change in discharge as a function of x is zero, then the depth prior to equilibrium is simply

$$y = R \cdot t \quad (6.51)$$

and the relationship between the depth, *y*, and the surface storage at equilibrium, D_e, is given by

$$y = \frac{8}{5} \cdot \frac{De}{L} \ [m] \quad (6.52)$$

The relationship between the depth, *y*, and the detained surface storage prior to equilibrium, *D*, is given by an empirical model (Fleming, 1975; Crawford and Linsley, 1966)

$$y = \frac{D}{L}\left(1 + 3/5 \cdot \left(\frac{D}{De}\right)^3\right) \ [m] \quad (6.53)$$

where during the recession part of the hydrograph, when D/D_e is greater than 1, D/D_e is assumed to be equal to 1.

Substituting (6.53) into the Manning equation (6.41) yields

$$q = M \cdot \sqrt{\alpha} \cdot \left[\frac{D}{L}\left(1 + 3/5 \cdot \left(\frac{D}{De}\right)^3\right)\right]^{5/3} \ [m^2/s] \quad (6.54)$$

6.2.2 Implementation in MIKE SHE

In MIKE SHE the current level of surface detention storage is continually estimated by solving the continuity equation

$$D_2 = D_1 + (\bar{q}_{supply} - \bar{q}) \cdot \Delta t \tag{6.55}$$

where D1 is the detained storage volume at the start of the time step and D2 is the detained storage volume at the end of the time step, q is the overland flow during the time interval, and q_{supply} is the amount of water being added to overland flow during the time step. Since q is a function of the average detained storage volume, (D1+D2)/2, equation (6.54) is solved iteratively until a solution satisfies both equations.

6.2.3 Coupling to other processes

Overland flow interacts with the other process components, such as evapotranspiration from the water surface, infiltration into the underlying soils, interaction with soil drains, drainage into the channel network, etc. This is an integral part of the MIKE SHE framework and these interactions are treated in the same manner in both the Semi-distributed Overland Flow Routing model and the 2D Finite Difference Method, based on the diffusive wave approximation.

The Semi-distributed Overland Flow Routing model simulates flow down a hillslope. To apply this at the catchment scale, it is assumed that the overland flow response for a catchment is similar to that of an equivalent hillslope. Furthermore, the drainage of overland flow from one catchment to the next, and from the catchment to the river channels is represented conceptually as a cascade of overland flow reservoirs.

6.2.4 Avoiding the redistribution of ponded water

In the standard version of the Simplified Overland Flow solver, the solver calculates a mean water depth for the entire flow zone using the available overland water from all of the cells in the flow zone. During the Overland flow time step, ET and infiltration are calculated for each cell and lateral flows to and from the zone are calculated. At the end of the time step, a new. average water depth is calculated, which is assigned to all cells in the flow zone.

In practice, this results in a redistribution of water from cells with ponded water (e.g. due to high rainfall or low infiltration) to the rest of the flow zone where cells potentially have a higher infiltration capacity. To avoid this redistribution, an option has been added where the solver only calculates overland flow for the cells that can potentially produce runoff, that is, only in the cells for which the water depth exceeds the detention storage depth.

Example application

To illustrate the effect of this option, it was applied to a model with a 10 x 10 square domain, one subcatchment and 3 different soil types in the unsaturated zone with the following saturated hydraulic conductivities

- coarse1e-5 m/s
- medium 1e-7 m/s
- fine 1e-9 m/s

For rainfall, a synthetic time series with alternating daily values of 50 and 0 mm/day was used. The simulation period was 2 weeks. Thus, the cumulative rainfall input was 350 mm.

For the case, where the ponding water was not redistributed, the cumulative runoff was 96 mm. Whereas, when the ponded water was redistributed, the cumulative runoff was essentially zero.

Activating the option

This option is activated by means of the boolean Extra Parameter, **Only Simple OL from ponded**, set to **On**. For more information on the use of extra parameters, see Extra Parameters *(V2 p. 325)*.

6.2.5 Routing simple overland flow directly to the river

In the standard version of the Simplified Overland Flow solver, the water is routed from 'higher' zones to 'lower' zones within a subcatchment. Thus, overland flow generated in the upper zone is routed to the next lowest flow zone based on the integer code values of the two zones. In other words, at the beginning of the time step the overland flow leaving the upper zone (calculated in the previous time step) is distributed evenly across all of the cells in the receiving zone. In practice, this results in a distribution of water from cells in the upstream zone with ponded water (e.g. due to high rainfall or low infiltration) to all of the cells in the downstream zone with potentially a large number of those cells having a higher infiltration capacity. In this case, then, overland flow generated in the upper flow zone may never reach the stream network because it is distributed thinly across the entire downstream zone.

To avoid excess infiltration or evaporation in the downstream zone, an option was added that allows you to route overland flow directly to the stream network. In this case, overland flow generated in any of the overland flow zones is not distributed across the downstream zone, but rather it is added directly to the MIKE Hydro River stream network as lateral inflow.

Example application

To illustrate the effect of this option, it was applied to a model with a 10 x 10 square domain, one subcatchment and two overland flow zones. The upper zone included an unsaturated zone with a low infiltration capacity, whereas the lower zone had a high infiltration capacity. The saturated hydraulic conductivities of the two zones were

- upper zone 1e-9 m/s
- lower zone 1e-5 m/s

For rainfall, a synthetic time series with alternating daily values of 50 and 0 mm/day was used. The simulation period was 2 weeks. Thus, the cumulative rainfall input was 350 mm.

For the case, where the overland flow was routed directly to the river, the cumulative runoff to the river was 167 mm. Whereas, when the overland flow was routed first to the lower zone, the cumulative runoff reaching the river was only 1 mm.

Activating the option

This option is activated by means of the boolean Extra Parameter, **No Simple OL routing**, set to **On**. For more information on the use of extra parameters, see Extra Parameters *(V2 p. 325)*.

7 Working with Overland Flow and Ponding- User Guide

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

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

This Chapter provides additional information and tips to help you understand how overland flow and ponding work, as well as help you speed up your simulations. It is divided into five sections:

- Overland Flow Control *(V2 p. 75)* - this section describes additional information on parameter values;
- Overland Flow Performance *(V2 p. 78)* - this section provides additional information on optimizing your overland simulation;
- Multi-cell Overland Flow *(V2 p. 82)* - this section describe how to include a sub-grid scale topography in your overland flow calculations;
- Overland and Ponded Drainage *(V2 p. 93)* - this section describes how to use route ponded water directly to rivers, boundaries and internal depressions; and
- Reduced OL leakage to UZ and to/from SZ *(V2 p. 103)* - this section describes how to limit exchange across the land surface in areas that are compacted or paved.

7.1 Overland Flow Control

The overland flow calculations are controlled by parameters and some basic options.

7.1.1 Principal parameters

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

Surface Roughness

The Stickler roughness coefficient is equivalent to the **Manning M**. The Manning M is the inverse of the commonly used **Manning's n**. The value of **n** is

typically in the range of 0.01 (smooth channels) to 0.10 (thickly vegetated channels), which correspond to values of **M** between 100 and 10, respectively.

If you don't want to simulate overland flow in an area, a 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.

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

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

The OL Drainage is also linked to the detention storage. Only the available ponded water will be routed to the OL Drainage network - that is the ponded depth above the detention storage. If you want to route all of the ponded water to the OL Drainage network, then you can use the Extra Parameter: OL Drainage Options *(V2 p. 338)*.

Initial and Boundary Conditions

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

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

Time varying OL boundary conditions

If you need to specify time varying overland flow boundary conditions, you can use the Extra Parameter option Time-varying Overland Flow Boundary Conditions *(V2 p. 332)*.

7.1.2 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.

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

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

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

Also, you should note that Overland flow cannot cross a river link. So, the cell faces with river links always define a separated flow boundary.

7.1.3 Output: Overland Flow Velocities

MIKE SHE calculates overland flow based on the diffusive wave approximation, which neglects the momentum. Further, the depth and flow rates are averages for a cell, which does not take into account the actual distribution of velocities and water depths in a natural topography. Finally, if the area of interest is next to a river, then the physical exchange with the river depends on the calculation method used. Even in the best case, exchange between the river and the flood plain is conceptual. There is no velocity calculated for the river-OL exchange, nor is there any momentum transfer between the overland flow and the flow in the river. Water is simply taken from the river and put on the flood plain cell, or vice versa. The rate of exchange depends the water level difference and the weir coefficients used.

Thus, the calculated velocity is less useful for things like damage assessment. If velocities are important, then MIKE FLOOD is a much better tool. MIKE SHE, on the other hand, is good at calculating overland water depths,

general flow directions and the exchange of ponded water with the subsurface and rivers.

MIKE SHE does, however, generate several output items related to velocity.

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.

For example, if your storing time step is a month, your flows will be a monthly average. The flow will be saved at midnight on the last day of the month. Depending on the timing of your events, you could easily have a high average flow and a zero depth.

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.

7.2 Overland Flow Performance

The overland flow can be a significant source of numerical instabilities in MIKE SHE. Depending on the setup, the overland flow time step can become very short - leading to very long.

Overland flow has both an Implicit and Explicit solver. Your choice of solver affects both the accuracy of your results and the simulation run time.

The Implicit solver is faster than the Explicit solver because it can run with longer time steps. However, the it must iterate to converge on a solution. Thus, if each time step takes several iterations because of the dynamics of the overland flow, then the implicit solver can become slow. The most obvious sign of poor convergence is the presence of warning messages in the *project-name_WM_Print.log* file about the overland flow solver not converging. You may be able to live with a few warning messages, but the if the Implicit solver frequently fails to converge then this will significantly slow down your simulation. If this happens, then you have a few options.

The first option is to reduce your OL time step. This make increase the stability of the solver and actually reduce your run times. You can also increase the convergence criteria. This will decrease the accuracy, but if there is a troublesome area outside of your area of interest, then this may be acceptable.

If you switch to the Explicit solver, then the time step becomes dynamic depending on the Courant criteria. This will likely reduce your numerical instabilities because the Courant criteria is very restrictive, but the simulation is likely to be slower. However, the difference may not be that great if you are having a lot of convergence problems.

7.2.1 Stagnant or slow moving flow

The solution of overland flow is sensitive to the surface water gradient. If the surface water gradient is very small (or zero), then a numerically stable solution will generally require a very short time step.

Slow moving flow is a problem when you have long term ponded water, for example in wetlands. If you are only interested in the water levels in the wetland areas, but not the flow velocity and flow directions, then solving the overland flow equations is not necessary for decision making.

If you want to turn off the overland flow solver in slow moving or stagnant areas, then you can convert these areas to flood codes and allow MIKE Hydro River to control the water levels. Lateral overland runoff to these areas will still be calculated, as will evapotranspiration and infiltration. For more detail on using Flood Codes, see Overland Flow Exchange with MIKE Hydro River *(V2 p. 128)*.

An alternative is to turn off the overland flow calculation in these cells. You can turn off the overland flow in a cell by setting the Manning's M number to zero. However, this also turns off the lateral overland inflow into the cell, as well.

Another option is to use the detention storage parameter to restrict the amount of available water. In this case, overland flow is allowed into and out of the cell, but overland flow is not actually calculated until the depth of water in the cell exceeds the detention storage.

The Threshold gradient for overland flow (see next Section) is also a way to reduce the influence of stagnant water on the time step. However, you cannot specify a spatially varying threshold. So, the appropriate value may be difficult to select if you want to restrict flow in one area, yet keep surface flow in other less stagnant areas.

If you are using the Explicit OL solver, there are several dfs2 output options that make it easier to find the model areas that are contributing to reducing the time step. These include:

- Mean OL Wave Courant number,
- Max OL Wave Courant number, and
- Max Outflow OL-OL per Cell Volume.

7.2.2 Threshold gradient for overland flow

In flat areas with ponded water, the head gradient between grid cells will be zero or nearly zero. As the head gradient goes to zero, Δt must also become very small to maintain accuracy. To allow the simulation to run with longer time steps and dampen any numerical instabilities in areas with low lateral gradients, the calculated intercell flows are multiplied by a damping factor when the gradients are close to zero.

Essentially, the damping factor reduces the flow between cells. You can think of the damping function as an increased resistance to flow as the gradient goes to zero. In other words, the flow goes to zero faster than the time steps goes to zero. This makes the solution more stable and allows for larger time steps. However, the resulting gradients will be artificially high in the affected cells and the solution will begin to diverge from the Manning solution. At very low gradients this is normally insignificant, but as the gradient increases the differences may become noticeable. Therefore, the damping function is only applied when the gradient between cells is below a user-defined threshold.

The details of the available functions can be found in the Section Low gradient damping function *(V2 p. 59)*.

For both functions and both the explicit and implicit solution methods, each calculated intercell flow in the current time step is multiplied by the local damping factor, F_D, to obtain the actual intercell flow. In the explicit method, the flow used to calculate the Courant criteria are also corrected by F_D.

The damping function is controlled by the user-specified threshold gradient (see Common stability parameters *(V1 p. 185)* for the Overland Flow), below which the damping function becomes active.

The choice of appropriate threshold value depends on the slope of the flow surface. Based on both actual model tests in Florida and synthetic setups, the following conclusions can be reached:

- A Threshold gradient greater than the surface slope can lead to excessive OL storage on the surface that takes a long time to drain away.
- A Threshold gradient equal to the surface slope is often reasonable, but there may still be some excess storage on the surface.
- Threshold values less than the surface slope typically cause rapid drainage and give nearly the same answers.
- Threshold values below 1e-7 do not significantly improve the results even if the topography is perfectly flat.
- In general, you should used the highest value possible. Lower values may increase accuracy but at the expense of run time.

Therefore, we can safely recommend a Threshold gradient in the range of 1e-4 to 1e-5, with a default value of 1e-4. For many floodplains, 1e-4 or 1e-5 should be sufficient. In flood plains with very flat relief, 1e-6 may be used. Lower values are probably never necessary.

Since most discharge happens during and immediately after an event, the Threshold gradient is likely to be most important when there is significant ponding that lasts over several time steps and drains to a boundary or MIKE Hydro River. Ponded water that infiltrates or evaporates and experiences limited lateral flow will not be affected by the Threshold value.

If the topography slope requires a low Threshold, but the solution is unstable at low threshold values, solution stability may be improved with the Explicit solver by reducing the Maximum Courant number until the solution becomes stable. With the Implicit solver, you may need to change the solver parameters.

Performance Impact

A low Threshold gradient will increase your simulation time. So, the final value that you use, may be a compromise between simulation length and accuracy of the flow in low gradient conditions.

If you have stagnant ponded water in your model, then the intercell gradient in these areas will be nearly zero. If you lower your Threshold gradient, your simulation performance may be adversely impacted, simply because the OL solver will begin to calculate flow sloshing back and forth in these areas. Not only will the OL solver have to work harder, the OL time step will likely also decrease because of the very low gradients. Thus, the Threshold gradient effectively reduces intercell flow in stagnant areas to zero allowing the Courant criteria to be satisfied at much higher time step lengths.

7.3 Multi-cell Overland Flow

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

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

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

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

In the Multi-cell method, the principle assumption is that the volume of water in the fine grid and the coarse grid is the same. Thus, given a volume of water, a depth and flooded area can be calculated for both the fine grid and the coarse grid. See Figure 7.1.

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

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

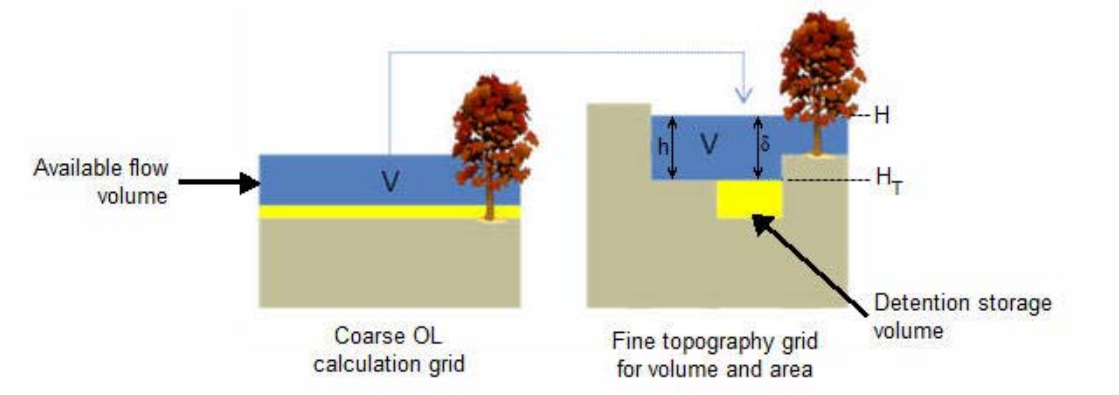


Figure 7.1 The constant volume from the coarse grid is transferred to the fine scale grid

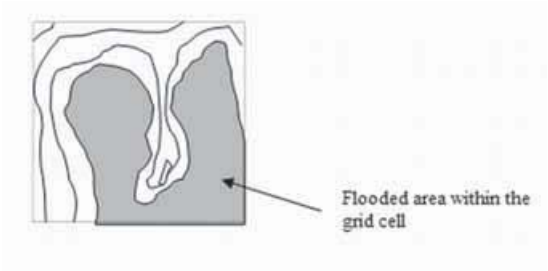


Figure 7.2 Flooded area is a function of the surface water level in the grid cell.

Elevations

The elevation of the coarse grid nodes and the fine grid nodes are calculated based on the input data and the selected interpolation method. However, the coarse grid elevation is adjusted such that it equals the average of the fine grid nodal elevations. This provides consistency between the coarse grid and fine grid elevations and storage volumes. Therefore, there may be slight differences between the cell topography elevations if the multi-cell method is turned on or off. This could affect your model inputs and results that depend on the topography. For example, if you initial water table is defined as a depth to the water table from the topography.

7.3.1 Evaporation

Evaporation is adjusted for the area of ponded water in the coarse grid cell. That is, evaporation from ponded water is reduced by a ponded area fraction, calculated by dividing the area of ponding in the fine grid cell by the total cell area.

The ET from soil evaporation is also reduced to the areas where there is not ponding.

Total transpiration does not need to be adjusted for the non-ponded area because evaporation from the ponded area is calculated prior to transpiration, and this is already adjusted for the ponded area in the cell. Thus, if the cell is fully ponded, then the Reference ET will be satisfied from ponded storage and there will be no transpiration. If the cell is only partially ponded, then the area fraction of the RefET will be first extracted from the ponded water and the remainder from the root zone. Since, there is only one UZ column to extract from, the entire root zone will be available for transpiration.

The Extra Parameter option,Transpiration during ponding *(V2 p. 343)*, allows transpiration from the root zone beneath ponded areas. In this case, transpiration is calculated before evaporation from ponding. This option includes a reduction factor to account for the reduced ET under saturated conditions. The application of this factor will be changed so that it only applies to the ponded fraction of the cell.

7.3.2 Infiltration to SZ and UZ with the Multi-Grid OL

If ponded water is flowing between cells, the multi-scale topography will ensure that only the lowest part of each cell will be flooded, and the rate of flow between the cells will be adjusted for the flooded depth. However, the infiltration also needs to be adjusted to account for the fact that there is a driving pressure head in some parts of the cell.

Infiltration to UZ

The UZ infiltration is calculated based on three UZ calculations: one for the ponded fraction of the cell, one for the non-ponded fraction of the cell and finally a calculation is made using the area weighted infiltration of the two first UZ calculations. The last step is needed as there is only one UZ column below the multi-cells.

In a MIKE SHE simulation without multi-cell infiltration, the engine calculates an average storage depth, which is available for infiltration. This storage depth is then used for the infiltration calculations. The storage depth is calculated as follows:

1. Assuming that the OL depth from the previous OL time step is known,
2. the OL depth is updated using the current net precipitation and any sink and source terms (irrigation, by pass flow, paved area drainage etc.). Noting that:
   - Bypass flow is extracted from the net precipitation before the infiltration calculation, and
   - OL Drainage is also extracted before the infiltration calculation. If you want to calculate the infiltration before paved area drainage this is available as an Extra parameter:

Parameter Name	Type	Value
infiltration before paved routing	Boolean	On

- The updated OL depth is used for the infiltration calculation;
- If the reduced contact option is used, the leakage coefficient is used to calculate the maximum infiltration rate.

When using the multi-cell infiltration, the infiltration is calculated based on three cases which depend on the ponded area fraction from the latest OL time step:

1. **Non ponded** (Ponded Area Fraction = 0). Only one infiltration calculation based on the available storage depth. This is done in the same way as a situation without the multi-cell option.

2. **Fully ponded** (Ponded Area Fraction = 1). Only one infiltration calculation based on the available storage depth. This is done in the same way as a situation without the multi-cell option.

3. **Partly ponded** (0 < Ponded Area Fraction < 1). Three infiltration calculations are made; ponded area, non-ponded area and a final calculation using the area weighted storage depth.

In the partly ponded case, it is assumed that the net-precipitation is equally distributed across the whole cell, while ponding from the previous OL time step only occurs in the ponded part of the cell. For the infiltration calculation in the non-ponded are, the available water depth is calculated as

$$DepPrec = precipitation \times dt \qquad (7.1)$$

The remaining part of the available water (ponding + precipitation on the ponded part) is scaled to an equivalent water depth in the ponded area:

$$DPonded = (OLDepth + DepPrec) \times (1-PAreaFrac) / PAreaFrac \qquad (7.2)$$

where OLDepth is the depth of ponded water from the previous time step, and PAreaFrac is the ponded area from the previous time step.

Disabling multi-cell infiltration

Multi-cell infiltration is automatically activated when the multi-cell option is invoked. However, an Extra Parameter option is available if you want to disable this function - perhaps for backwards compatibility with older models.

Parameter Name	Type	Value
disable multi-cell infiltration	Boolean	On

When this is specified, the infiltration will be calculated based on the values of the course grid, and any ponding occurring in any sub-grid cells will not be included.

7.3.3 Multi-cell Overland Flow + Saturated Zone drainage

The topography is often used to define the SZ drainage network. Thus, a refined topography more accurately reflects the SZ drainage network.

The SZ drainage function uses a drain level and drain time constant. The drain level defines the depth at which the water starts to drain. Typically, this is set to some value below the topography to represent the depth of surface drainage features below the average topography. This depth should probably be much smaller if the topography is more finely defined in the sub-grid model. The drain time constant reflects the density of the drainage network. If there are a lot of drainage features in a cell then the time constant is higher and vice versa.

Details related to the use of the Multi-cell OL with the SZ Drainage function are found along with the rest of the user guidance on SZ Drainage in the section: Saturated Zone drainage + Multi-cell Overland Flow *(V1 p. 228)*.

7.3.4 Test example for impact on simulation time

The increased accuracy of the multi-cell overland flow method does not come for free. There is a performance penalty when you turn on the multi-cell option. However, the penalty relative to the increased accuracy of water depths is small.

A test done on a large, complex model in Florida, USA illustrates the performance penalty of the multi-cell method.

In the model the grid cell size is 457.2 m (1500 ft). However, a high resolution (5-ft) DEM is available for the whole model domain based on LIDAR data. This makes it attractive to use of higher resolution map with the Multi-cell option to account more accurately for the OL flow between 1500-ft grid cells.

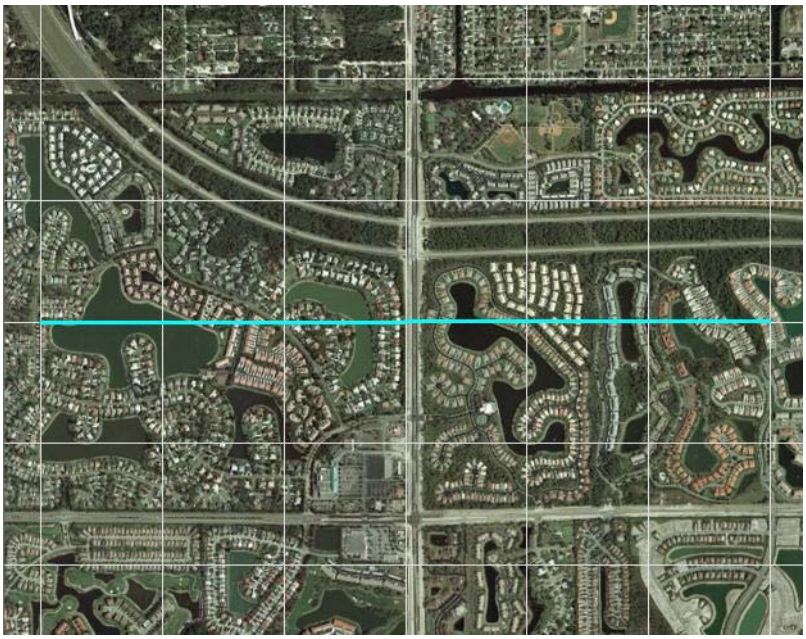


Figure 7.3 Aerial photo of part of the model area

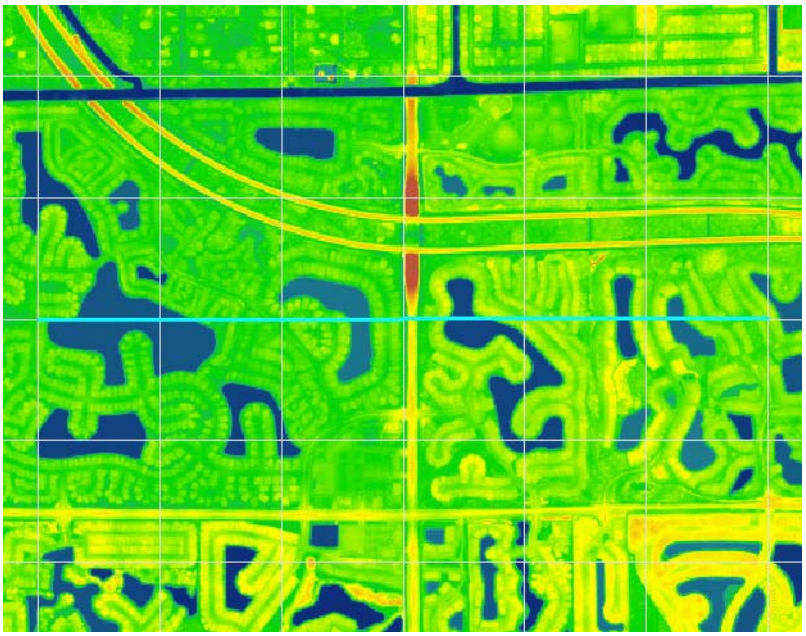


Figure 7.4 5-foot LIDAR data for part of the model area

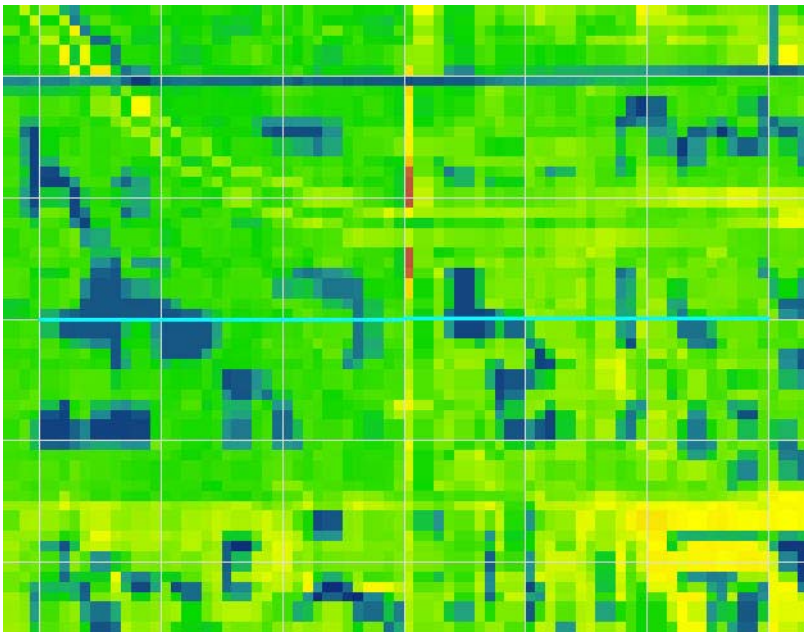


Figure 7.5 Interpolation of the 5-foot LIDAR data to the 125ft model grid

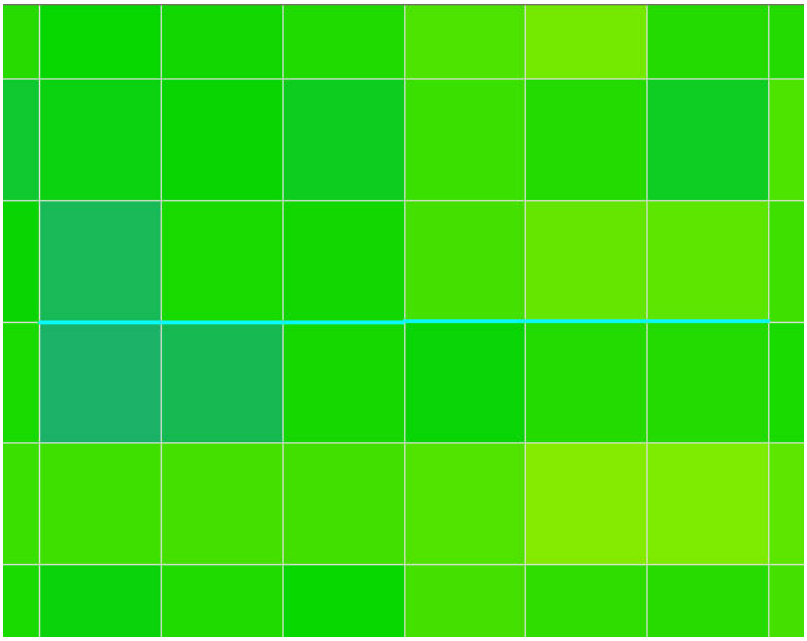


Figure 7.6 Interpolation of the 5-foot LIDAR data to the 1500ft model grid

Impact on simulation time

In this test, we tested multi-cell factors of 1, 2, 3, 4, 6, and 12. The smallest grid size was 125-ft, which is 12 times smaller than the coarse 1500-ft grid.

The following graphs illustrate the impact of the multi-grid option on the running times for the test model. Figure 7.7 shows that the OL run time increases linearly with higher multi-cell factors. In the test model, a multi-cell factor of 12 caused the OL portion of the simulation time took 30 times longer. Figure 7.8 shows that the multi-cell factor also impacts the run time for MIKE Hydro River. However, this impact is not linear, with the impact on MIKE Hydro River leveling off after a multi-cell factor of four.

The test model run time is dominated by MIKE Hydro River. In this case, the original run time for the OL is not very long and the multi-cell factor increases the OL run time considerably. However, as a fraction of the total run time, the OL is still small. When the OL cells are subdivided, there is probably some significant changes in the lateral inflow to MIKE Hydro River. However, as the multi-cell factor increases, the increased resolution of the inflows is not significant above a factor of about four.

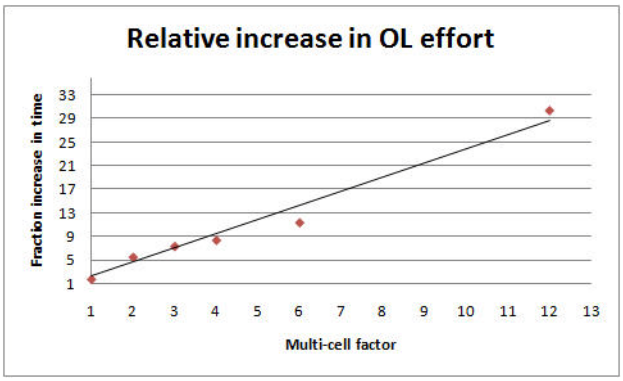


Figure 7.7 Increase in OL run time as a function of multi-cell factor

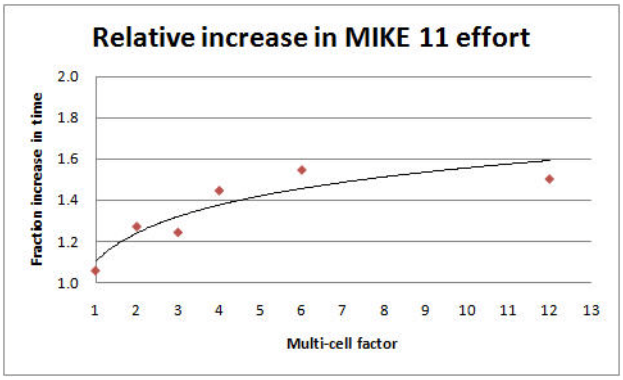


Figure 7.8 Increase in MIKE Hydro River run time as a function of multi-cell factor.

Impact on model results

The model contains a mix of natural, urban, and agricultural areas. The model also includes a complex river network with the relevant man-made canals and structures, and most of the natural flow ways. However, there is a natural flow way in the southeast part of the model that is not conceptualized in MIKE Hydro River. Since, the surface water flow in this area is relying only on overland flow, the multi-cell option should significant changes in the OL flow prediction in this natural flow way.

In urban and agricultural areas, the drainage and the OL flow components in MIKE SHE route the water into the canal network. The drainage component would keep the water table level below the ground most of the time in those areas. However, it is of interest to test the OL flows predicted with the multi-cell option in those areas during storms events.

7.3.5 Limitations of the Multi-cell Overland Flow Method

In principle, all of the exchange terms in MIKE SHE could be adjusted to reflect the fine scale water levels and flooded areas. However, some of these are easier to implement than others and of greater importance. Thus, in current release, the exchange with MIKE Hydro River, as well as UZ and SZ, depend only on the coarse scale grid elevations.

Overland flow exchange with MIKE Hydro River

Overland flow exchange with MIKE Hydro River does not consider the multi-cell method. That is, flow into and out of the River Links is controlled by the water level calculated from the elevation defined in the coarse grid cell. Likewise the flow area for exchange with MIKE Hydro River is calculated as the coarse water depth times the overall grid size. Also, the elevation used when calculating flood inundation with flood codes only considers the average cell depth of the coarse grid. See Overland Flow Exchange with MIKE Hydro River *(V2 p. 128)*.

However, if you choose to modify the topography based on a bathymetry file, or the MIKE Hydro River cross-sections, then this information will be used when calculating the multi-cell elevations.

7.3.6 Setting up and evaluating the multi-grid OL

The multi-cell overland flow method is activated in the OL Computational Control Parameters *(V1 p. 183)* dialogue. In this dialogue, you can check on the option and then specify a sub-division factor. The coarse grid will be divided into this number of cells in both directions. That is, for a factor of two, the coarse grid will be divided into four cells. Likewise a factor of five will lead to 25 fine cells per coarse grid cell.

In addition

Pre-processed data

When you enable the Multi-grid OL option the following new items will be available in the preprocessed data:

- Multi-Cell ground levels (subscale topography) In Figure 7.9, two interpolations of the same DEM are shown. The top figure is a plan view of the interpolation to a 100m grid resolution and a 25m grid resolution respectively. The bottom figure is a cross-section across the middle of the top figure, where you can clearly see the more accurate resolution of the drainage features.

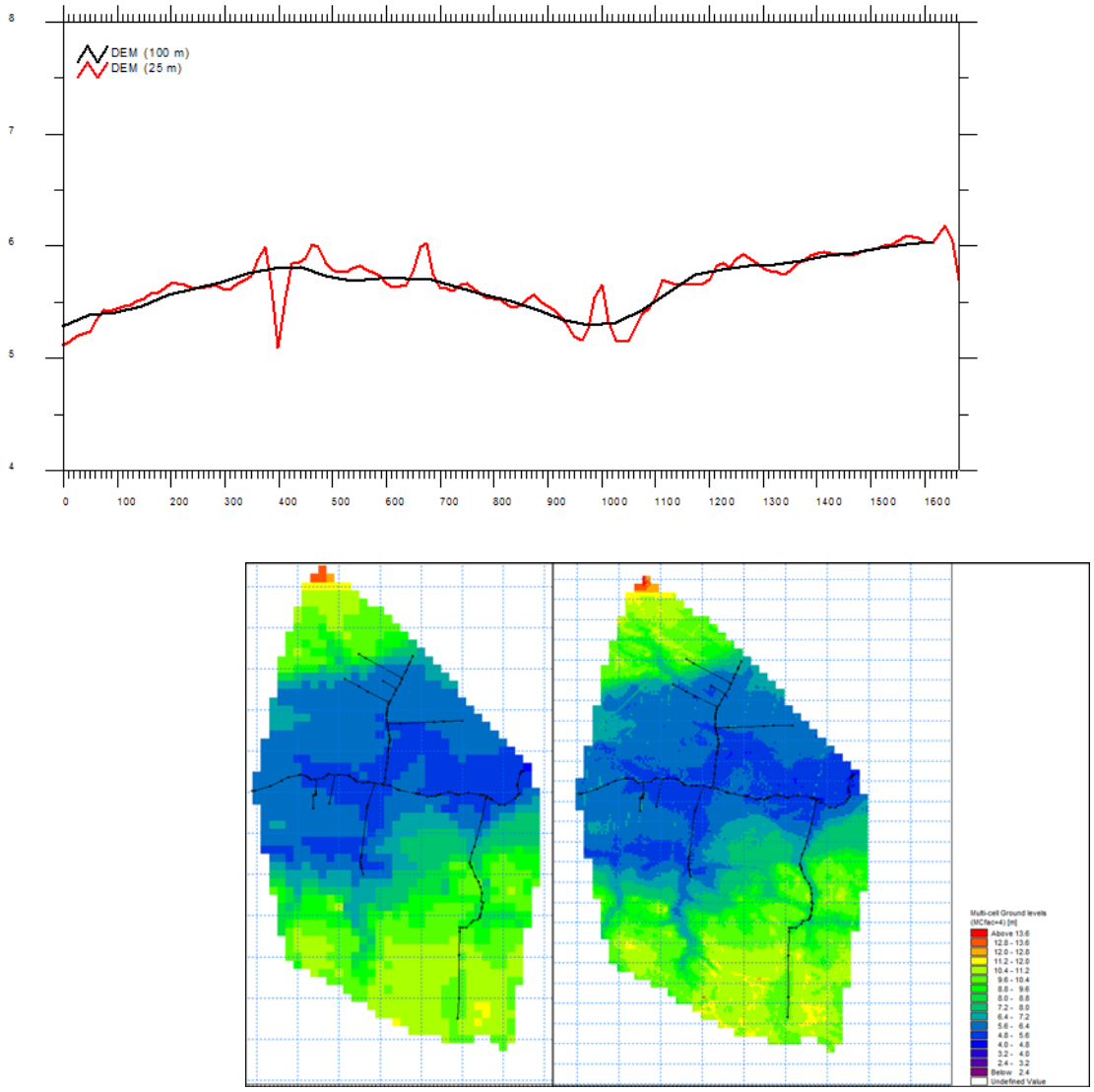


Figure 7.9 Example of preprocessed data - topography using a 100 meter resolution and a sub-scale factor of 4 (25 m sub-scale resolution)

When SZ Drainage is also active, then the following items are also available:

- Max. MC Drain Level - displays the maximum drain level of the sub-cells within each of the model cells
- Min MC Drain level - displays the minimum drain level of the sub-cells within each of the model cells
- Min MC Drain Depth - displays the minimum drain depth of the sub-cells within each of the model cells

- Max MC Drain Depth - displays the maximum drain depth of the sub-cells within each of the model cells

Additional results options

When the Multi-grid OL option is active, the following additional items will be available in the result items:

- Depth of Multi-Cell overland water - displays the depth of overland water using the sub-scale resolution
- Multi-Cell overland water elevation - displays the overland water elevation using the sub-scale resolution

7.4 Overland and 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 amount of runoff generated by MIKE SHE depends on the density of the stream network defined in MIKE Hydro River. If the stream network is dense, then you will get more runoff in the model because the lateral travel time to the streams will be short, and the infiltration and evaporation losses will be smaller.

Fine-scale, natural and man-made drainage systems are difficult to simulate in MIKE SHE using the traditional finite difference Overland Flow method. This is largely because the drainage features are often smaller than the grid scale, and shallow overland flow will not travel far laterally before it infiltrates or evaporates.

In large catchments, it may be impractical to defined the detailed natural drainage network. In urban catchments, the detail of the drainage network may be either unknown or impractical to define.

The Ponded Drainage (OL Drainage) option was introduced in the 2017 Release to alleviate some of these issues. 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.

However, as in the SZ Drainage, the pathway is only a reference system. It does not actually contain any water. Drainage from a particular cell is released directly to the destination cell. If the destination is a boundary cell, the water is added directly to the boundary. If the destination is a river link, then the water is added to the cell beside the river link.

A key difference from the SZ Drainage feature is that the OL Drainage includes an intermediate storage. The SZ Drainage is instantly added to the destination, whereas the OL Drainage is first added to an internal storage. Technically, this internal storage is “on the cell”, not “in the drain”. Thus, any cell that is generating OL Drainage will also have an amount of OL Drain Storage. The amount of OL Drain Storage retained on the cell depends on the difference between the Inflow and Outflow time constants. This allows you to simulate a rapid drainage of rainfall, but a slower discharge to the receiving destination. Effectively, this is the same as having storage in a drainage network, but the drainage from different cells are not summed together, and their is no physical drainage network to interact with while the water is in storage.

The Ponded Drainage function has been designed to facilitate a flexible definition of storm water drainage in both natural and urbanized catchments. In particular, the Ponded Drainage function supports the following features:

- In urban areas, paving and surface sealing (eg roof tops) limits infiltration and enhances runoff.
- Ponded water is routed to user-defined locations (boundaries, depressions, manholes and streams).
- The time varying storage in the drainage system is accounted for by different inflow and outflow rates.
- Drainage rates can be controlled by drain levels, drainage time constants and maximum outflow rates.
- Drainage is restricted if the destination water level is higher than the source water level.
- Urban development over time can be simulated by time varying drainage parameters.
- Overland flow is calculated normally for ponded water that does not drain in the current time step.

Conceptually, the Ponded Drainage function is shown in Figure 7.10.

Figure 7.10 Conceptualization of the Ponded Drainage function

Calculation order

The Ponded Drainage is calculate explicitly in a particular order with respect to the other hydrologic processes.

1. Rainfall is added to the ponded depth in the cell.
2. Then, OL Drainage will be immediately calculated.

In this way, the OL Drainage essentially acts on rainfall and removes rainfall before anything else happens.

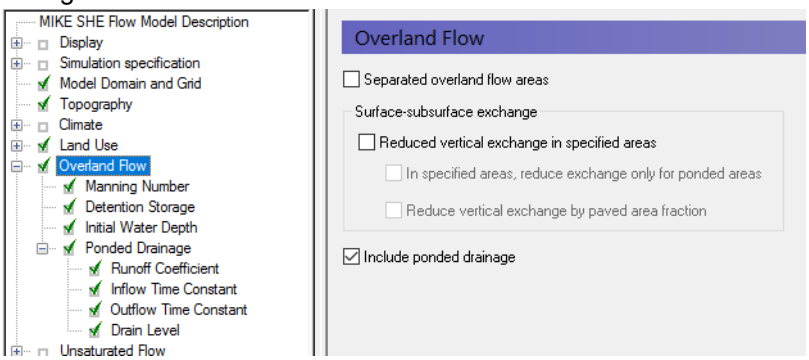
If there is any ponded water remaining on the cell after the drainage has been removed, then it will be removed in the following order until no ponded water remains.

1. Evaporation is removed from the ponded water;
2. Then, Infiltration to the unsaturated zone will be calculated; and
3. Finally, Lateral Overland Flow will be calculated to the adjacent cells.

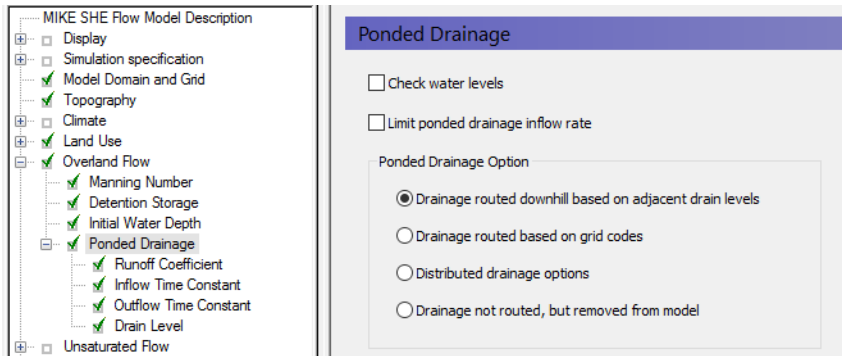
The Ponded Drainage is calculated for every cell independently. That is, the drainage that is calculated is not added to the destination cells until the end of the calculation for all the cells. This prevents circular drainage, and means that the order in which the calculation occurs is not relevant. After the ponded drainage is calculated, the depths in all the cells are updated - both sources and destinations. The calculation of ET, infiltration and lateral OL flow is based on the updated ponded depths.

7.4.1 Specifying Ponded Drainage

Ponded drainage is specified by selecting the Include ponded drainage checkbox in the Overland Flow dialog. This will activate the Ponded Drainage dialog and the default additional data tree items.



The options for defining OL Drainage are selected in the Ponded Drainage dialog:



There are two check box options for controlling the OL Drainage:

Check water levels

If the destination of the OL drainage is a river or an internal depression, then it will have a water level. If you chose the option to Check water levels, then the drainage will not be added to the destination if the water level in the destination is equal to or higher than the water level in the drain.

The source water level for the comparison is calculated based on the OL Drain level plus the depth of water in the OL Drain storage in the time step.

Limit inflow rate to OL Drain Storage

The amount of ponding that is removed in one time step is a function of the leakage coefficient. Usually the leakage coefficient is large enough to ensure that all ponding is removed in the current time step. However, often the conveyance through the drain is restricted, such as a culvert. This is addressed by an option for specifying a maximum discharge rate into the drain. The maximum inflow rate can be specified as a constant value or a distributed (dfs2) value. With an Extra Parameter option, the maximum discharge rate can be time varying (dfs2).

The maximum inflow rate can be used to control the inflow to the OL drainage system. At every time step, the available drainage volume will be checked against the maximum drainage rate. Thus, ponded water will be retained on a grid cell and drained at a controlled rate into the drainage system. While the water is ponded it will be subject to infiltration and ET. The rate of leakage below the cell can be controlled by the Surface-Subsurface Leakage Coefficient *(V1 p. 260)*.

Note: Prior to Release 2017, this was called the “Max Storage Change Rate” with Item Type [Storage Change Rate] and default EUM Units of [mm/hour]. There is no automatic backwards compatibility conversion for this. If an existing model uses this option, a uniform value has to be manually converted to the new units - Default units [m^3/s]. If a dfs2 file is used, the Item Type also needs to be converted to the new Item Type [Discharge].

Destination options for OL Drainage

The default option is to calculate the drainage destination by looking downhill until a river, boundary or local depression is reached. The elevation for calculating this is the Drain Level. Typically, this will be the same as the Topography (default). However, it could be different, for example, in urban areas where the drain level might be based on sewer inverts.

If Drain codes are selected, then the Drain Levels are used only within the cells that have the same Drain code.

7.4.2 Runoff Coefficient

The OL Drainage is calculated from **ponded water**. It is not calculated from rainfall directly. However, the order of operations in the three UZ solvers (Richards, Gravity, and 2Layer Water Balance) is such that rainfall is added to existing ponded storage after interception and before ET and infiltration are calculated, and, thus, before the OL Drainage is calculated. Therefore, the OL Drainage effectively acts on net rainfall (i.e. net rainfall = rainfall minus leaf interception).

The amount of OL Drainage is calculated based on the available ponded water. That is, a specified fraction of the amount of available ponded water is routed directly to the OL Drainage system. The OL Drainage system immediately adds this water into the OL Drainage Storage.

Water discharging from the OL Drainage Storage is instantly added to the destination. This outflow is analogous to a full-pipe of water. That is, for any inflow an equal amount of outflow is generated instantaneously.

Rainfall is added to the ponded depth and then the OL Drainage fraction is removed. However, if at the end of the time step, there is still ponded water in the cell, the Runoff Coefficient will be applied to the remainder again in the next time step. Thus, the ponded water will eventually drain away.

The OL Drainage is also linked to the detention storage. Only the available ponded water will be routed to the OL drainage network - that is the ponded depth above the detention storage. If you want to route all of the ponded water to the OL Drainage network, then you can use an Extra Parameter found in OL Drainage Options *(V2 p. 338)*.

The OL Drainage function does not check to make sure that you do not create any physically impossible feedback loops. So, flood code cells, and overbank spilling from MIKE Hydro River should not be directed to cells where OL Drainage is active. If this happens, you may encounter excessive feedback between MIKE SHE overland flow and MIKE Hydro River.

Note: The OL Drainage is calculated on the UZ time step, and stored in the UZ output file. Thus, the OL Drainage depends on the UZ and/or the OL time step length. For example, if the UZ time step is reduced by the precipitation

controls, then the amount of inflow to the drain may change because the amount of rainfall in the time step changes. Thus, you may want to adjust your precipitation controls such that the UZ time step is not reduced by these parameters.

7.4.3 Time constants

The time constant for OL Drainage Inflow controls the rate of water entering the drain storage. The time constant for Outflow controls the rate of water draining to the destination. Thus, if the Outflow time constant is greater than the Inflow time constant, the drain storage will not accumulate any water. In this case, inflow to the drain will be instantly discharged to the destination.

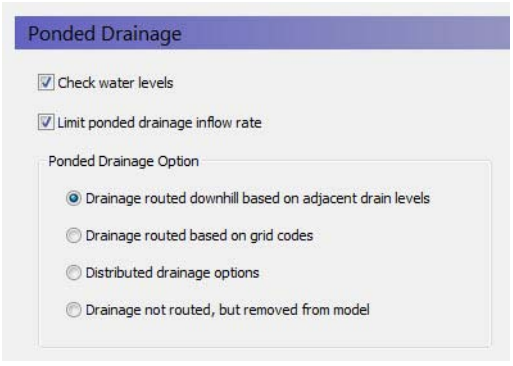
The default values for both time constants is 0.001. This is quite high and will normally drain nearly all ponded water from the cell. Since the two time constants by default are the same, the default storage will be very low or zero. This will normally result in a very sharp peak in your streams in response to the instantaneous drainage of rainfall to the streams.

During model calibration, you can start decreasing the outflow time constant to dampen these drainage peaks.

7.4.4 OL Drainage Reference System

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.

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



.

Drainage routed downhill based on adjacent drain levels

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-processor calculates the drainage source-recipient reference system by

1. looking at each cell in turn and then
2. looking for the neighbouring cell with the lowest drain level.
3. If this cell is an outer boundary cell or contains a river link, the search stops.
4. If this cell does not contain a boundary or river link, then the search is repeated with the next downstream neighbour until either a local minimum is found or a boundary cell or river link is found.

The result of the above search from each cell is used to build the source-recipient reference system.

If local depressions in the drainage levels exist, the OL 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.

The drain-slope based reference system has been used in MIKE SHE for many years in SZ and works well in most situations. However, when MIKE SHE is applied where there is very little surface relief, it is often difficult to establish a suitable reference system based on the surface topography/drain slopes. In flat areas, this may generate many undesired local depressions, which may receive drainage water from a large area, thus generating lakes in places where there should not be a lake.

If the drain level is perfectly flat, drainage is turned off. In other words, if the drain-slope method cannot find a downhill neighbour because all the neighbours have the same elevation as the cell, the drain slope method assumes that the cell is a local depression. However, the depression has no sources of drainage except itself. Thus, the drainage function is effectively turned off.

Tip: MIKE SHE considers a grid point to be a local depression even if the drainage level in the four surrounding model grids is only 1 mm higher. The only way to avoid such problems is to create a drain level map that does not contain artificial local depressions. For large models this may be difficult and time consuming. In this case, one of the other drainage options may be better.

Remember, the drainage is routed to a destination. It does not physically flow downhill. The drain levels are only used to build the drainage source-recipient reference system, and to calculate the amount of drainage.

Drainage routing based on grid codes

This method is often used when the topography is very flat, which can result in artificial depressions, or when the drainage system is very well defined, such as in urban applications.

In this method, the drainage levels and the time constants are defined as in the previous method and the amount of drainage is calculated based on the drain levels and the time constant.

If the drainage routing is specified by Drain Codes, a grid code map is required that is used to restrict the search area for the source-recipient reference system. In this case, the pre-processer calculates the reference system within each grid code zone, such that all drainage generated within one zone is routed to recipient nodes with the same drain code value.

When building the reference system, the pre-processor looks at each cell and then

1. looks for the nearest cell with a river link with the same grid code value,
2. if there is no cells with river links, then it looks for the nearest outer boundary cell with the same grid code,
3. if there are no cells with outer boundary conditions, then it looks for the cell with the same grid code value that has the lowest drain level. In this case, the reference system is calculated as if it was based on Drain Levels (see previous section).

The result of the above search for each cell is used to build the source-recipient reference system.

The above search algorithm is valid for all **positive** Drain Code values. However, all cells where

- **Drain Code = 0** - will not produce any drain flow and will not receive any drain flow, and
- **Drain Code < 0 (negative)** - will not drain to river links, but will start at Step 2 above and only drain to either a outer boundary or the lowest drain level.

Tip: One method that can be used is to specify one Drain Code value for the entire model area (e.g. Drain Code = 1). Thus, all nodes can drain and any drain flow is routed to the nearest river link. If there are no rivers, the drain flow will be routed to the nearest boundary. If you want to route all drain flow to the boundaries instead of the rivers, a negative drain code can be specified for the entire area (e.g. Drain Code = -1).

Distributed drainage options

Choosing this method, adds the Option Distribution item to the data tree. With the Option Distribution, you can specify an integer grid code distribution that can be used to specify different drainage options in different areas of your model.

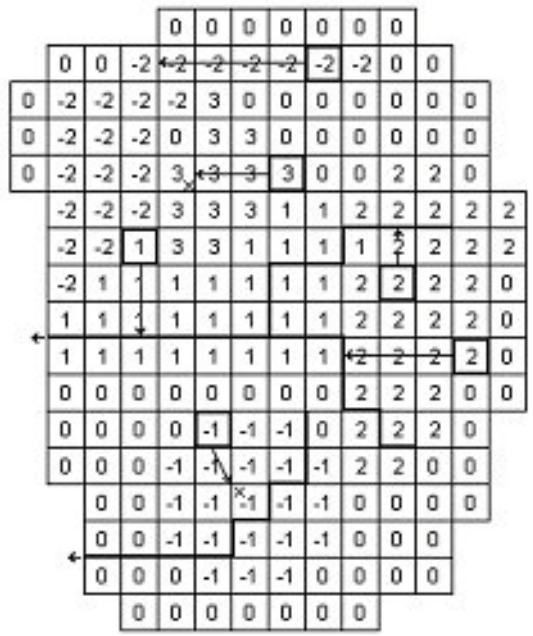
- **Code = 1** - In grid cells with a value of 1, the drainage reference system is calculated based on the Drain Levels.

- **Code =2** - In grid cells with a value of 2, the drainage reference system is calculated based the Drain Codes.
- **Code = 3** - Drainage in grid cells with a value of 3 is routed to a specified MIKE Hydro River branch and chainage. At the moment, this options requires the use of Extra Parameters *(V1 p. 334)* and is described in OL Drainage to Specified MIKE Hydro River H-points *(V2 p. 339)*.
- **Code = 4** - Drainage in grid cells with a value of 4 is routed to a specified MIKE URBAN 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 simply removes the water from the model. This is equivalent to routing all the drainage to a boundary.

Drain Code Example



- The grid cells with Drain Code 3 drain to a local depression since no boundary or river link is found adjacent to a grid with the same drain code.
- The grid cells with Drain Code 1 or 2 drain to nearest river link located adjacent to a grid with the same drain code.

- The grid cells with drain code 0 do not contain drains and thus no drainage is produced.
- The grid cells with Drain Code -1 drains to local depression since no boundary is found adjacent to a grid with the same drain code.
- The grid cells with Drain Code -2 drains to nearest boundary grid with the same drain code.

7.4.5 Preprocessed data

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

- **Drain Codes** - The value in the pre-processed Drain Codes map reflects the Option Distribution specified. For example, those cells with an Option Distribution equal to 1 (Drainage routed based on Drain Levels) will have a pre-processed Drain Code equal to 0, because the Drain Codes are not being used for those cells.
- **Drainage to local depressions and boundary** - This grid displays all the cells that drain to local depressions or to the outer boundaries. All drainage from cells with the same negative value are drained to the cell with the corresponding positive code. If there is no corresponding positive code, then that cell drains to the outer boundary, and the water is simply removed from the model. Cells with a delete value either do not generate drainage, or they drain to a river link.
- **Drainage to river** - This grid displays the river link number that the cell drains to. Adjacent to the river links, the cells are labeled with negative numbers to facilitate the interpretation of flow from cells to river links. Thus, in principle, all drainage from cells with the same positive code are drained to the cell with the corresponding negative code.

  However, this is slightly too simple because the cells actually drain directly to the river links. In complex river systems, when the river branches are close together, you can easily have cells connected to multiple branches on different sides. In this case, the river link numbers along the river may not reflect the drainage-river link reference used in the model.

  If you want to see the actual river links used in all cells, you can use the Extra Parameter, OL Drainage to Specified MIKE Hydro River H-points *(V2 p. 339)*, to generate a table of all the river link-cell references in the PP_Print.log file.

7.4.6 Time varying OL Drainage parameters

In projects where you want to simulate the build out of an OL Drainage network over time, or changes in the OL Drainage time constants over time, then you can use the Extra Parameters: Time varying OL drainage parameters *(V2 p. 341)*. Without this set of Extra Parameters you would have to hot start your simulation at regular time intervals with the new OL Drainage parameters.

Note: If you specify time varying OL Drainage parameters, you will not be able to use any of the drainage routing methods that depend on the OL drain level. The drain level methods are not allowed because the source-recipient reference system is only calculated once at the beginning of the simulation.

The preprocessor checks this and gives an error if you have specified

- option 1 (routing based on levels), or
- option 3 (distributed options) AND any of the distributed option codes are 1 (routing based on levels in these cells).

7.5 Reduced OL leakage to UZ and to/from SZ

In MIKE SHE the exchange across the land surface is normally controlled by the UZ vertical hydraulic conductivity. If there is no UZ because the groundwater is at or above the topography then the SZ vertical hydraulic conductivity is used.

However, there are many cases where the exchange across the land surface is restricted. An additional reduced leakage coefficient can be used to account for soil compaction and fine sediment deposits on flood plains in areas that otherwise have similar soil profiles, as well as for paving in urban areas

The Leakage Coefficient will reduce the amount of infiltration from OL to UZ and OL to SZ. It will also reduce groundwater discharge to the surface, SZ to OL. Thus, the leakage coefficient reduces the infiltration rate at the ground surface in both directions. It reduces both the infiltration rate and the seepage outflow rate across the ground surface.

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

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

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

The value of the leakage coefficient may be found by model calibration, but a rough estimate can be made based on the saturated hydraulic conductivities of the unsaturated zone or in the low permeable sediment layer, if such data is available. For example, if the surface layer has a K of 1e-6 m/s, and thickness of 10cm, your leakage coefficient would be (K/thickness) 1e-7 1/s.

7.5.1 Surface-Subsurface Leakage Coefficient

The specified Surface-Subsurface Leakage Coefficient is used wherever it is specified. In areas where a delete value is specified, the model calculated leakage is used.

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

7.5.2 Simulation of paved areas

Paving is common in urban areas and has a significant impact on infiltration and runoff.

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: Paving acts only as an extra reduction in leakage coefficient. It does not separate runoff from paved areas. Direct runoff from paved areas can be managed with the OL Drainage function.

7.5.3 Backwards compatibility of Paving prior to Release 2017

Prior to Release 2017, paving in urban areas was simulated in MIKE SHE as a combination of the Surface-Subsurface Leakage Coefficient under Overland Flow, a Paved Area Fraction under Land Use, and the SZ Drainage. This was awkward and ineffective.

In Release 2017, these functions have been both improved and consolidated under Overland Flow. This means that the functions cannot be 100% automatically carried over. But, the functions should be 100% compatible after some minor manual adjustments.

In Release 2016 and earlier, the SZ Drainage network was used to define the River Link destination of Paved Drainage. Boundaries and Internal Depressions were ignored. There was no inflow time constant and drainage was added instantly to the destination. These limitations have been removed in Release 2017.

When opening a model built in Release 2016 or earlier,

- The default values are used for all the new parameters.
- If Paving was specified, then the Paved Area Fraction is carried over to the Runoff Coefficient, which has the same effect.

The Runoff Coefficient is the fraction of ponding that is allowed to drain into the drainage network. The new Paved Area fraction is a scaling factor on the Surface-Subsurface Leakage Coefficient.

The OL Drainage Inflow and Outflow time constants are both set to 1e-3 [1/s]. Since both are the same, all inflow will be instantly discharged to the destination. There will be no accumulation in the OL Drainage Storage. The default value is sufficient for most applications, but may still restrict the inflow in some cases. If the inflow rate is too slow, then the time constant can be manually increased.

Recreating the previous Drainage reference system

A significant limitation of the Paved function in Release 2016 and earlier was that the drainage could only go to a River Link based on the SZ Drainage reference system. The Paved function ignored any nodes where the SZ Drainage reference system directed water to the boundary or an internal depression. In Release 2017, this restriction has been eliminated.

If you now open and existing model with paving, then the drainage will be “correctly” routed. To recreate the previous drainage reference system:

1. Pre-process your model and in the Processsed Data tab save the SZ drainage Drain to River item to a new dfs2 file.
2. Open this dfs2 file and select all the Delete Values. Set the Delete Values equal to 0. Then, select all the Negative Values and multiply them by -1. Save the file.
3. Change the OL Drainage reference method to be based on Grid Codes. Then use the new, modified dfs2 file as the Grid Code reference system.

These three steps should ensure that the OL Drainage only drains to the River Links, and drains to the same River Links as previously. OL Drainage in all the cells that would drain to boundaries or local depressions is turned off.

7.5.4 Reduced Leakage with Multi-cell OL

The Surface-Subsurface Leakage Coefficient is used if it is specified. However, it is used differently if the option to use it only in ponded areas is selected (this option is only available if the Multi-cell OL is turned on.

Thus, the two mechanisms are:

- Reduced contact only in ponded areas (activated). Leakage coefficient is only used in the ponded areas and not in the non-ponded areas.
- Reduced contact only in ponded areas (not activated). Leakage coefficient is used in both the ponded and the non-ponded case.

The two infiltration calculations (for the ponded and the non-ponded case) result in two infiltration rates from which an area weighted infiltration rate, *QinfAWghtd*, is calculated:

$$QinfAWghtd = Qinfntpd \times (1 - PAreaFrac) + Qinfpd \times (PAreaFrac) \qquad (7.3)$$

where *Qinfntpd* is the infiltration rate from the non-ponded area, and *Qinfpd* is the infiltration rate for the ponded area. The area weighted infiltration rate is then used in the final UZ calculation (when reduced contact is not used).

7.5.5 Reduced leakage in ponded areas only

In many cases, the ponded areas will have a lower infiltration rate than the surrounding dry areas. The land surface in the dry areas will tend to be broken up macropores, etc. Whereas, surface sealing will occur beneath ponded areas. To yield a more realistic flow surface drainage for flooded areas, an option for reduced contact (Ol leakage coefficient) in only the ponded part of the cell is available.

Note: This is only used in the UZ infiltration and NOT in the exchange between SZ and OL.

Activating this option will allow you to include a distributed dfs2 integer grid code file. The reduced leakage will be applied in all areas with a positive integer value. In all other areas (with a negative, zero or delete value), the reduced leakage condition will be applied to the whole cell with the following constraints:

- The option will be applied to the ponded area from the previous time step. This will ensure that rainfall infiltrates normally in the non-ponded areas and currently ponded water will be retained.
- After the rainfall in non-ponded areas is infiltrated, then intercell lateral flow will be calculated and a new ponded area determined.

This method ensures that ponded water is able to flow laterally between cells with limited losses. By adjusting the leakage rate, you can decrease the losses along the OL flow path. This will essentially lead to a sub-grid scale drainage network that will ensure that runoff will eventually reach the river.

However, this option only applies to cells that are ponded, and thereby ensuring that ponded water remains on the surface. During high intensity rainfall in the current time step, this option will not encourage the creation of flooded areas, as the reduced leakage coefficient will first be applied in the following time step if ponded water is present at the end of the time step.

On flood plains, where the ponding occurs from overbank spilling from rivers or streams, the option will likely result in a more realistic description of the flow paths on the flood plain, as it prevents the flooded water from infiltrating.

Note: When the Multi-cell option is used, a uniform value for the maximum discharge rate will be used within each coarse cell. Further, the depth of ponded water is calculated on the sub-scale, and used to calculate the OL Drainage flow for each sub-scale cell

If the maximum discharge rate is set low, then the paved area fraction can be used to control inflow to and outflow from, for example, small scale surface impoundments.

The combination of maximum discharge rate and the OL leakage coefficient, along with the multi-cell OL, allows you to simulate distributed on grid surface water storages. You can use the combination to define, for example, distributed farm dams that release water to streams at a fixed rate. The volume of the storage is defined using the multi-grid OL. The ponded water is subject to evaporation, and you can use the OL leakage coefficient to control leakage to groundwater.

7.6 Simulating surface water infrastructure

The combination of maximum discharge rate and the OL leakage coefficient, allows you to simulate distributed on grid surface water storages. You can use the combination to define, for example, distributed farm dams that release water to streams at a fixed rate. The ponded water is subject to evaporation, and you can use the OL leakage coefficient to control leakage to groundwater.

8 Channel Flow - Technical Reference

The hydrologic components of MIKE SHE are directly coupled to DHI's river hydraulic program MIKE Hydro River. The MIKE SHE-MIKE Hydro coupling enables

- the one-dimensional simulation of river flows and water levels using the fully dynamic Saint Venant equations.
- the simulation of a wide range of hydraulic control structures, such as weirs, gates and culverts.
- area-inundation modelling, using a simple flood-mapping procedure that is based on simulated river water levels and a digital terrain model.
- dynamic overland flooding flow to and from the MIKE Hydro River network.
- the full, dynamic coupling of surface and sub-surface flow processes in MIKE Hydro River and MIKE SHE.

For technical information on MIKE Hydro River, please refer to either the .pdf version of the MIKE Hydro River Technical Reference Manual that is installed with MIKE SHE, or the MIKE Hydro River documentation in the on-line help.

Note: In Release 2017, MIKE SHE can also be coupled to MIKE 11. MIKE 11 is being replaced by MIKE Hydro River, and the coupling to MIKE 11 will be slowly phased out over time. All references in the MIKE SHE manuals to MIKE 11 and MIKE Hydro River are largely interchangeable.

8.1 Surface Water Exchange Mechanisms

In a catchment scale model, it is usually sufficient to consider a river as a line located between model grid cells. In this case, the river-aquifer exchange can be calculated inflow to and from both sides of the river, depending on the head gradient to the adjacent groundwater cells. Further, the overland-river exchange is also along a line seperating inflow over the right and left river banks.

The line assumption is generally valid if the river width is small relative to the model cells - in other words, in catchment or basin scale models. However, very often a more precise description of the interactions between rivers, flood plains, aquifers and the atmosphere (evapotranspiration) must be adopted. In this context, a reliable description of area-inundation and flood dynamics is crucial.

Thus, the MIKE SHE-River coupling considers three principally different surface water exchange mechanisms, which are described in detail in the section **Coupling of MIKE SHE and MIKE Hydro River** *(V2 p. 115)*:

- **Groundwater exchange with MIKE Hydro River** *(V2 p. 123)* The river is located on the edge between two adjacent model grid cells. The river is considered a line source/sink to the groundwater and the river is a one-way sink for overland flow.
- **Flooding from MIKE Hydro River to MIKE SHE using Flood Codes** *(V2 p. 129)* The river has a wide cross-section containing the flood plain and designated cells are “flooded” if the river water level is above the topography.
- **Direct Overbank Spilling to and from MIKE Hydro River** *(V2 p. 131)* The river is a line source/sink, but water above the bank elevation is allowed to flood onto the topography as overland flow.

The above options can be mixed in the river network. This allows, for example, Flood Codes in the major flood plain and overbank spilling in the upstream secondary branches, but no flooding in the upland regions with steep slopes and narrow channels. MIKE SHE also automatically converts between the line source/sink option and the flooding options. Thus, during low flow conditions, when the river is narrow (less than one grid size) and water flow is confined to the main river channel, the river-aquifer exchange method is adopted. If the river starts to flood one or more model grid cells, MIKE SHE switches to the area-inundation method or floods the grid cells directly via overbank spilling.

9 Working with Rivers and Streams - User Guide

9.1 Channel Flow

9.1.1 MIKE Hydro River Overview

MIKE Hydro River is a comprehensive 1D channel flow model for simulating rivers and surface water bodies that can be approximated as 1-dimensional flow (as strict 1-Dimensional flow does not occur in nature). Basically, MIKE Hydro River can be applied anywhere average values of levels, velocities, concentrations etc. at a point are acceptable, including:

- River hydrodynamics
- Structure/reservoir operational control
- Water quality (e.g. wetlands, salinity)
- Sediment transport & morphology
- Flood studies (e.g. mapping, hazard assessment)
- Flood forecasting (on-line, real-time)
- Dam break
- Sediment transport (e.g. Long term morphology)
- River restoration
- Integrated with groundwater and flooding

MIKE Hydro River plays a critical role in MIKE SHE Both the overland flow and groundwater flow modules are linked directly to MIKE Hydro River. The MIKE SHE-MIKE Hydro River coupling enables:

- the one-dimensional simulation of river flows and water levels using the fully dynamic Saint Venant equations.
- the simulation of a wide range of hydraulic control structures, such as weirs, gates and culverts.
- area-inundation modelling, using a simple flood-mapping procedure that is based on simulated river water levels and a digital terrain model.
- dynamic overland flooding flow to and from the MIKE Hydro River network.
- the full, dynamic coupling of surface and sub-surface flow processes in MIKE Hydro River and MIKE SHE.

9.2 Building a MIKE Hydro River model

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

1. Build a stand-alone MIKE Hydro River HD hydraulic model and make a performance test and, if possible, a rough calibration using prescribed inflow and stage boundaries. If needed, you can specify a default groundwater table (e.g. MIKE SHE's initial groundwater level) and leakage coefficients for any leakage calculations.

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

3. Couple MIKE SHE and MIKE Hydro River by defining branches (reaches) where MIKE Hydro River HD 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.

In the above scheme, the first step in coupling MIKE Hydro River to MIKE SHE is to create a normal MIKE Hydro River HD model without coupling it with MIKE SHE. In this regard, a few things should be emphasised:

- In a normal MIKE Hydro River model only the river chainage (dx) is important for the results. Geographic positioning of river branches and cross-sections are only important for the graphical presentation. When interfacing MIKE Hydro River to MIKE SHE geographic positioning is critical, as MIKE SHE needs information on the river location.

- A reasonably high number of river cross-sections should be included to ensure that the river elevations are reasonably consistent with the surface topographic features.

9.2.1 MIKE Hydro River network limitations

There are a few features of MIKE Hydro River that do not relate well to MIKE SHE.

Short branches

In MIKE Hydro River there is no restriction on how short your branches are. If you are trying to simulate discontuous lakes or structures on the flood plain, for example. you may have very short branches. However, MIKE SHE does

not allow MIKE Hydro River branches to be shorter than the cell size. Generally, though, short branches are a sign that you should probably reconsider your model conceptualization - or switch to MIKE FLOOD, which allows flood plain structures.

Parallel branches

Like short branches, MIKE SHE does not like it when your branches are too close together. If you have parallel branches that are too close together, then the branches may be mapped to the same river link. However, each river link must be mapped to a unique branch. As a rule of thumb, parallel branches should be greater than a cell width apart. However, this is not uniformly true, since the two close parallel branches may map onto opposite sides of a cell, if they are located on either side of a cell mid-point. Thus, you may have unexpected problems, if you change the cell size in a model that was working and you have branches that are closer together than one cell size.

Long coupling links

MIKE SHE links to MIKE Hydro River branches. However, when two branches are connected, water is passed between the branches directly. The link has not physical length or storage itself. If your links are too long, there will be an error in the timing of the flows between the two branches. So, the links should be kept short. MIKE Hydro River does not have any restrictions on how long the links can be, but MIKE SHE will issue a warning if the links are longer than a cell size. The warning is simply to informing you that there is no possibility for groundwater-surface water exchange in the link.

Long distances between cross sections

MIKE Hydro River controls the distance between the calculation nodes. The properties at the calculation nodes are linearly interpolated from the available cross-sections. This includes geometric properties such as bank and bottom elevations, marker locations, etc. However, linear interpolation can easily result in inconsistencies between elevations in MIKE SHE.and marker elevations in MIKE Hydro River. If the bank elevation is higher than the topography, then overland flow into the river will be restricted. If the downstream river bottom elevation is higher than the side branch bottom elevation, then MIKE Hydro River will likely be unstable.

Long distances between calculation nodes

This is not the same as long distances between cross-sections. MIKE Hydro River manages the water at the q-points directly linked to the river links. MIKE SHE and the river link system automatically interpolates the nearest river link. However, if the calculation nodes are very far apart or very close together, then the linear interpolation of water volumes between the calculation points may lead to discrepancies in the available water volumes especially if the river links are being used for irrigation or the river is losing water. In this sense, the distance between the calculation nodes, should be similar to the MIKE SHE grid spacing.

9.2.2 MIKE Hydro River Cross-sections

Whenever there is a significant change in the bed slope there should, in principle, be a cross-section defined in MIKE Hydro River. If only a few cross-sections are available, it may be sufficient to estimate the cross-section shape based on neighbouring cross-sections and estimate the bank/bed elevation based on the surface topographic information in MIKE SHE or other topographic maps.

Cross-sections vs. time step

However, every cross-section in MIKE Hydro River is a calculation node. The time step in MIKE Hydro River is sensitive to the Courant number, which is proportional to the distance between calculation nodes. So, if the cross-sections are close together, then you may experience very short time steps in MIKE Hydro River.

Thus, if you are have very short MIKE Hydro River time steps, then you might want to check your river network to make sure you do not have cross-sections that are too close together. This frequently occurs when the cross-sections have been imported. If you do have cross-sections that are too close together, then you can easily eliminate one or more of them, as long as the conveyance of the different cross-sections is roughly the same. In other words, you can eliminate duplicate cross-sections if their Q/H relationships are roughly the same, even though the physical shape of the two cross-sections may appear quite different. This is often the case in braided stream networks, where the location of the main channels may move left or right, but the overall conveyance of the river bed is relatively constant.

Cross-sections versus MIKE SHE topography

In the absence of flooding, ponded water discharges to the MIKE Hydro River as overland flow. As a general rule, the topography must be higher than or equal to the bank elevation. If the bank elevation is higher than the topography, water will not be able to flow into the river in that cell, but will run laterally along the river until it reaches a place for it to flow into the river. An easy trick to see where this is happening is to run a simulation with no infiltration, ET, or detention storage and set the initial water depth at 1m. Then look at the results to find places were the water is piling up against the river links.

In the pre-processor log file, a table is create that lists all the river links where the bank elevation is different than the topography.of the adjacent cell. The critical river links with bank elevations above the topography are highlighted with the ==> symbol. This list can be surprisingly long because the river link bank elevations are interpolated from the neighbouring cross-sections. Whereas the topography is already defined. So, frequently the interpolated bank elevations do not line up precisely with the topography.

If overland flow on the flood plain is essentially absent, for example, due to infiltration or evapotranspiration, then these differences are not relevant and there is no need to modify the topography. However, if the overland to river

exchange is important then you may have to carefully modify your topography file or your bank elevations so that they are consistent.

Hint In many cases, your topography is from a DEM that is different from your model grid - either because it is a .shp or xyz file, or if it is a different resolution than your model grid. In this case, it may be easier to save the pre-processed topography to a dfs2 file (right click on the topography map in the pre-processed tab). Then modify and use the new dfs2 file as the topography in your model setup. The disadvantage of this, is that if you change your model domain or grid, then you will have to redo your topography modifications.

Hint You can also use one of the Flood code options to automatically modify your topography, if you have wide cross-sections or a detailed DEM of the floodplain. In this case, after you have set up your MIKE Hydro River model, you can specify a constant grid code for the whole model and let MIKE SHE calculate a modified topography based on the cross-sections or bathymetry. Then save the topography file as above and then use it as the model topography.

9.3 Coupling of MIKE SHE and MIKE Hydro River

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 MIKE SHE's set-up program, based on a user-specified sub-set of the MIKE Hydro River model, called the **coupling reaches**. The entire river system is always included in the hydraulic model, but MIKE SHE will only exchange water with the coupling reaches. Figure 9.1 shows part of a MIKE SHE model grid with the MIKE SHE river links, the corresponding MIKE Hydro River coupling reaches, and the MIKE Hydro River h-points (points where MIKE Hydro River calculates the water levels).

The location of each of MIKE SHE river link is determined from the co-ordinates of the MIKE Hydro River points, where the river points include both digitised points and h-points on the specified coupling reaches. Since the MIKE SHE river links are located on the edges between grid cells, the details of the MIKE Hydro River geometry can be only partly included in MIKE SHE, depending on the MIKE SHE grid size. The more refined the MIKE SHE grid, the more accurately the river network can be reproduced.

If flooding is not allowed, the MIKE Hydro River levels at the h-points are interpolated to the MIKE SHE river links, where the exchange flows from overland flow and the saturated zone are calculated.

If flooding is allowed, via Flood Codes, then the water levels at the MIKE Hydro River h-points are interpolated to specified MIKE SHE grid cells to determine if ponded water exists on the cell surface. If ponded water exists, then the unsaturated or saturated exchange flows are calculated based on the ponded water level above the cell.

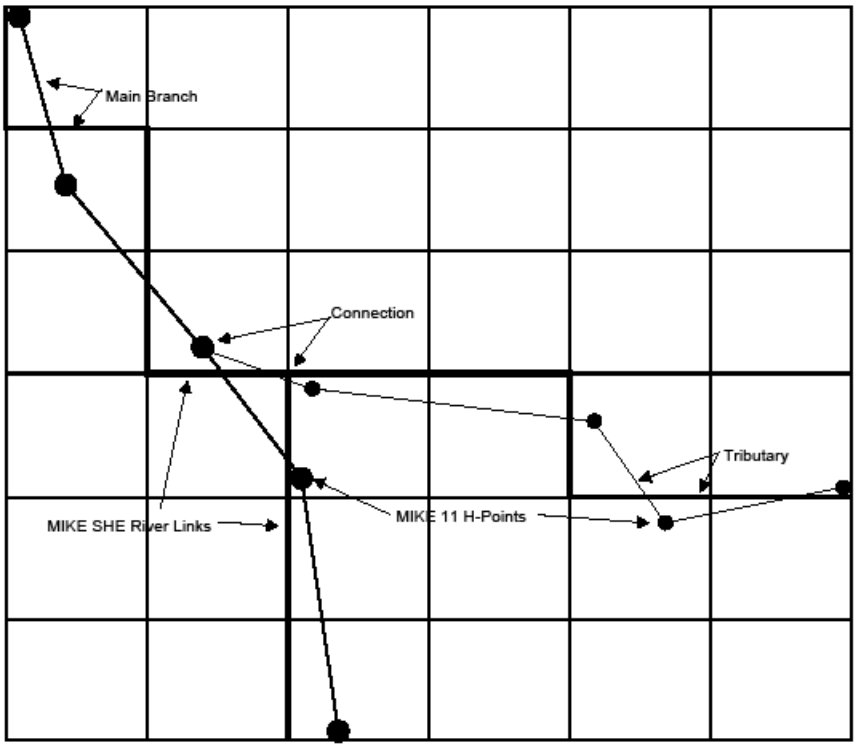


Figure 9.1 MIKE Hydro River Branches and h-points in a MIKE SHE Grid with River Links

If flooding is allowed via overbank spilling, then the river water is allowed to spill onto the MIKE SHE model as overland flow.

In each case, the calculated exchange flows are fed back to MIKE Hydro River as lateral inflow or outflow.

Each MIKE SHE river link can only be associated with one coupling reach, which restricts the coupling reaches from being too close together. This can lead to problems when you have a detailed drainage or river network with branches less than one half a cell width apart. It will also lead to problems if your MIKE Hydro River branches are shorter than your MIKE SHE cell size.

If you have coupling reaches that are too short or too close together, you will receive an error message. If this happens, you can

- decide not to include one of the branches as a coupling reach (it is still included in the MIKE Hydro River HD model), or
- remove some of the branches (this error often occurs when you have a detailed looped drainage network), or

- refine your MIKE SHE grid until all coupling reaches are assigned to unique river links.

If you have a regional model with large cells (say 1-2km wide), then you cannot expect the river-aquifer interaction to be accurate at the individual cell level (e.g. all your cell properties – topography, conductivity, Manning's M, etc. – are all average values over 1-4 km^2). Rather, most often you will be interested in having a correct overall water balance along the stream. Typically, this is achieved by calibrating a uniform average river bed leakage coefficient against a measured outflow hydrograph. In such a model, you may also be tolerant of higher groundwater residuals.

On the other hand, if you need more detailed site specific results (and you have data and measurements to calibrate against), then you will use a local scale model, with a smaller grid (say 50-200m) and discrepancies between topography and river bank elevation will largely disappear. In this case, you will be more likely to be able to make accurate local scale predictions of groundwater-surface water exchange.

9.3.1 MIKE SHE Branches vs. MIKE Hydro River Branches

A **MIKE Hydro River branch** is a continuous river segment defined in MIKE Hydro River. A MIKE Hydro River branch can be sub-divided into several coupling reaches.

A **MIKE SHE branch** is an unbroken series of coupling reaches of one MIKE Hydro River branch.

One reason for dividing a MIKE Hydro River branch into several coupling reaches could be to define different riverbed leakage coefficients for different sections of the river.

If there are gaps between the specified coupling reaches, the sub-division will result in more than one MIKE SHE branch. Gaps of this type are not important to the calculation of the exchange flows between the hydrologic components (e.g. overland to river, or SZ to river). The exchange flows depend on the water level in the MIKE Hydro River , which is unaffected by gaps in the coupling reaches.

However, MIKE SHE can calculate how much of the water in the river is from the various hydrologic sources (e.g. fraction from overland flow and SZ exfiltration). However, this sort of calculation is only possible if the MIKE SHE branch is continuous. If there is a gap in a MIKE SHE branch, then the calculated contributions from the different hydrologic sources downstream of the gap will be incorrect. If there are gaps in the MIKE SHE branch network, then the correct contributions from the different sources must be determined from the MIKE Hydro River output directly.

Furthermore, the MIKE Hydro River/MIKE SHE coupling for the water quality (AD) module will not work correctly if there are gaps in the MIKE SHE branch network.

There is one further limitation in MIKE SHE. That is, no coupling branch can be located entirely within one grid cell. This limitation is to prevent multiple coupling branches being located within a single grid cell.

Connections Between Tributaries and the Main Branch

Likewise, the connections between the tributaries and the main branch are only important for correctly calculating the downstream hydrologic contributions to the river flow and in the advection-dispersion (AD) simulations. The connections are not important to the calculation of the exchange flows between the hydrologic components (e.g. overland to river, or SZ to river).

In the example shown in Figure 9.1, the river links of the tributary are correctly connected to the main branch. This will happen automatically when

- the hydraulic connection is defined in the MIKE Hydro River network, AND
- the connection point (the chainage) on the main branch is included in a coupling reach, AND
- the connection point (the chainage) on the tributary is included in a coupling reach.

If the connection does not satisfy the above criteria, then there may be a gap in the MIKE SHE branch network and the limitations outlined above will apply.

9.3.2 The River-Link Cross-section

The MIKE Hydro River (HD) hydraulic model uses the precise cross-sections, as defined in the cross-section file, for calculating the river water levels and the river volumes. However, the exchange of water between MIKE Hydro River and MIKE SHE is calculated based the river-link cross-section.

The river-link uses a simplified, triangular cross-section interpolated (distance weighted) from the two nearest MIKE Hydro River cross-sections. The top width is equal to the distance between the cross-section's left and right bank markers. The elevation of the bottom of the triangle equals the lowest depth of the MIKE Hydro River cross-section (the elevation of Marker 2 in the cross-section). The left and right bank elevations in MIKE Hydro River (cross-section markers 1 and 3 in MIKE Hydro River) are used to define the left and right bank elevations of the river link (See Figure 9.2).

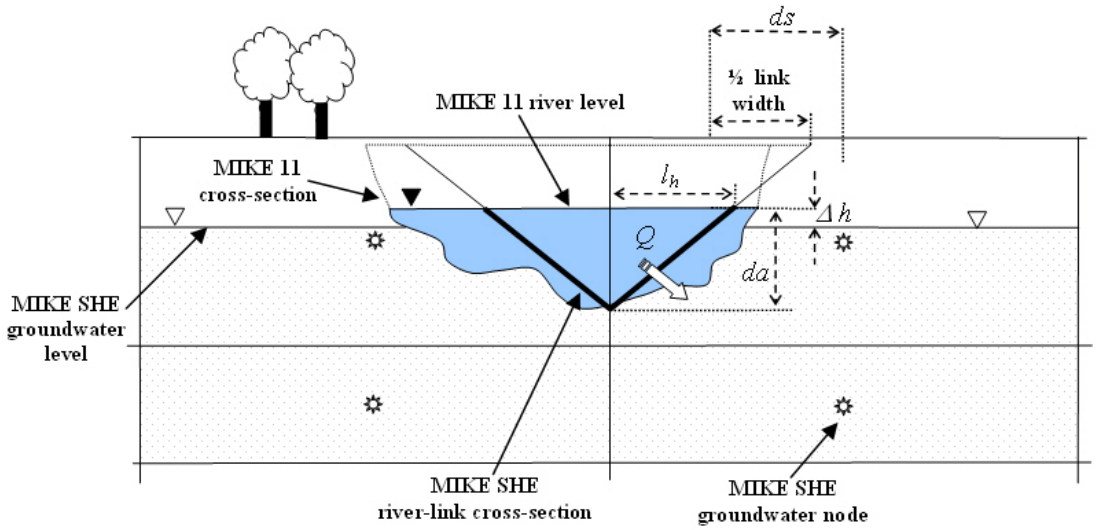


Figure 9.2 A typical simplified MIKE SHE river link cross-section compared to the equivalent MIKE Hydro River cross-section

If the MIKE Hydro River cross-section is wider than the MIKE SHE cell size, then the river-link cross-section is reduced to the cell width.This is a very important limitation, as it embodies the assumption that the river is narrower than the MIKE SHE cell width. If your river is wider than a cell width, and you want to simulate water on the flood plain, then you will need to use either the **Flooding from MIKE Hydro River to MIKE SHE using Flood Codes** *(V2 p. 129)* option or the **Direct Overbank Spilling to and from MIKE Hydro River** *(V2 p. 131)* option.

If you don't want to simulate flooding, then the reduction of the river link width to the cell width will not likely cause a problem, as MIKE SHE assumes that the primary exchange between the river and the aquifer takes place through the river banks. For more detail on the river aquifer exchange see **Groundwater exchange with MIKE Hydro River** *(V2 p. 123)*.

For more detail on flooding and overland exchange with MIKE Hydro River see Overland Flow Exchange with MIKE Hydro River *(V2 p. 128)*

9.3.3 Connecting MIKE Hydro River to MIKE SHE

In MIKE Hydro River, every node in the river network requires information on the river hydraulics, such as cross-section and roughness factors. These nodes are known as h-points, and MIKE Hydro River calculates the water level at every H-point (node) in the river network. Halfway between each H-point is a Storing Q-point, where MIKE Hydro River calculates the flow, which must be constant between the h-points.

The water levels at the MIKE Hydro River h-points are transferred to the MIKE SHE river links using a 2-point interpolation scheme. That is, the water level in each river link is interpolated from the two nearest h-points (upstream and downstream), calculated from the centre of the link. The interpolation is proportionally distance-weighted.

The volume of water stored in a river link is based on a sharing of the water in the nearest h-points. In Figure 9.3, River Link A includes all the water volume from H.points 1 and 2, plus part of the volume associated with H-point 3. The volume in River Link B is only related to the volume in H-point 3. While the volume in River Link C includes water from h-points 3 and 4. This is done to ensure consistency between the river volumes in MIKE Hydro River and MIKE SHE, as the amount of water that can infiltrate or be transferred to overland flow is limited by the amount of water stored in the river link.

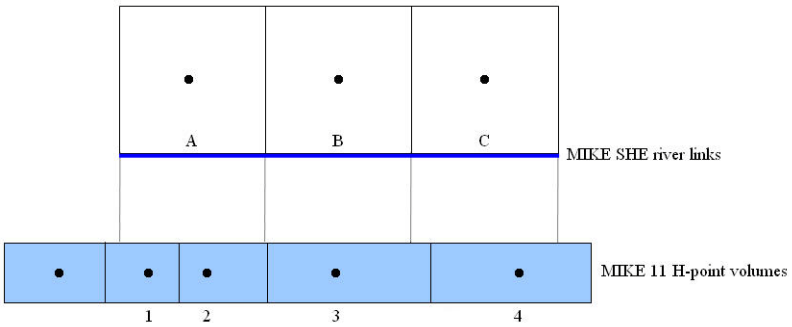


Figure 9.3 Sharing of MIKE Hydro River H-point volumes with MIKE SHE river links

The water levels and flows at all MIKE Hydro River h-points located within the coupling reaches can be retrieved from the MIKE SHE result file.

However, since the MIKE Hydro River flows are not used by MIKE SHE, the river flows stored in the MIKE SHE result file are not the flows calculated at the MIKE Hydro River Storing Q-points. Rather, the flows stored in the MIKE SHE result file are the estimated flows at the MIKE Hydro River h-points. That is, the flows in the MIKE SHE result file have been linearly interpolated from the calculated flows at the Storing Q-point locations to the H-point locations on either side of the Storing Q-point. If the exact Q-point discharges are needed, they must be retrieved or plotted directly from the MIKE Hydro River result file.

9.3.4 Evaluating your river links

The river links are evaluated during the pre-processing. In the pre-processor log file (*yourprojectnamePP_print.log*), there is a table that contains all of the river link details:

MIKE SHE River Link overview

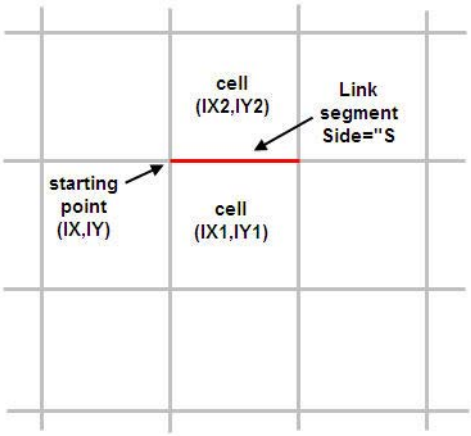
'==>' at start of line indicates that the bank is more than 0.010 m above ground level of a non-flood cell
'***' at start of line indicates that the bank is more than 0.010 m above ground level of a flood (inundation) cell

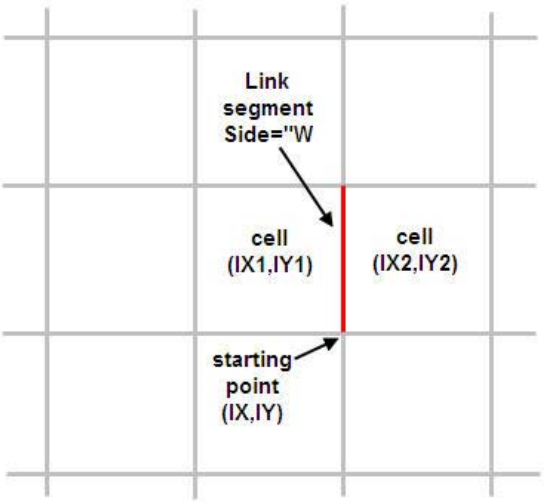
Link	IX	IY	Side	IX1	IY1	Topo1	Bank1	IX2	IY2	Topo2	Bank2	Bed	Width	Leak-opt	Leak-coeff	Spill
1	3	2	S	3	1	0.20	-0.08	3	2	0.22	-0.18	-0.97	71.66	Aq+Bed	0.1000E-04	On
2	3	2	W	2	2	0.12	-0.04	3	2	0.22	-0.14	-0.90	198.97	Aq+Bed	0.1000E-04	On
3	3	3	S	3	2	0.22	-0.10	3	3	0.24	0.00	-0.84	336.07	Aq+Bed	0.1000E-04	On
4	4	3	W	3	3	0.24	0.07	4	3	0.34	-0.03	-0.72	561.31	Aq+Bed	0.1000E-04	On
5	4	4	W	3	4	0.26	0.12	4	4	0.36	0.02	-0.63	741.77	Aq+Bed	0.1000E-04	On
6	3	5	S	3	4	0.26	0.17	3	5	0.28	0.07	-0.54	917.26	Aq+Bed	0.1000E-04	On

Spill	WeirCoeff	HExpo	FullWdepth	ThrVolSpill	Chainage	Branch
On	0.1838E+01	1.500	0.1000E+00	100.00	525.5563	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	453.8689	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	376.6670	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	249.8351	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	148.2199	BRANCH1
On	0.1838E+01	1.500	0.1000E+00	100.00	49.40662	BRANCH1

In this table, the locations where the river links are higher than the topography are marked in the outside left column.

The reference system used in the table is illustrated below:





The explanation of the columns is:

Link: River Link ID number. ID starts at 1 and increases by 1.

IX,IY: coordinate of one end of the link segment. They are referred to the pre-processed grid such that (IX,IY)=(1,1) at the left-bottom corner of the model grid. The link segment can be drawn starting from (IX,IY) coordinate, and then following east direction if Side="S" or following the north direction if Side="W".

Side: relative position of the (IX1,IY1) cell with respect to the link segment. "S" stands for south and "W" for west.

IX1,IY1: coordinate of the cell on the south side of the link if Side="S", or the cell on the west side of the link if Side= "W". The left-bottom corner cell of the model grid has coordinates (IX1,IY1)= (1,1).

Topo1: Pre-processed Topo elevation (in meters) of the cell (IX1,IY1).

Bank1: Interpolated cross section bank elevation (in meters) at marker 1 or 3 at the link chainage (last column). The marker (1 or 3) corresponding to Bank1 depends on the position of the cell (IX1,IY1) with respect to the direction of increasing chainage. Marker 1 is the left marker in the increasing chainage direction.

IX2,IY2: Coordinate of the cell on the opposite side to (IX1,IY1). In other words, it is the cell on the north side of the link if Side="S", or the cell on the east side of the link if Side= "W". The left-bottom corner cell of the model grid has coordinates (IX2,IY2)= (1,1).

Topo2: Pre-processed Topo elevation (in meters) of the cell (IX2,IY2).

Bank2: Interpolated cross section bank elevation (in meters) at marker 1 or 3 at the link chainage (last column). The marker (1 or 3) corresponding to Bank2 depends on the position of the cell (IX2,IY2) with respect to the direction of increasing chainage. Marker 1 is the left marker in the increasing chainage direction.

Bed: Interpolated cross section elevation (in meters) at marker 2 at the link chainage (last column). In other words, it is the river bed bottom elevation interpolated at that chainage.

Width: Interpolated cross section width (in meters) at the link chainage (last column). The cross section width is the distance between markers 1 and 3 in the cross section profile.

Leak-opt: The Conductance option used in the coupling reach in which this river link is contained. The value is from in the MIKE SHE links table of the MIKE Hydro River Coupling Reaches dialogue. The three possible options are "Aq+Bed", "Aq only", and "Bed only". See **Groundwater exchange with MIKE Hydro River** *(V2 p. 123)* and Figure 9.4.

Leak-coeff: The Leakage Coef. value used in the coupling reach in which this river link is contained found in the MIKE SHE links table of "Coupling Reaches". See **Groundwater exchange with MIKE Hydro River** *(V2 p. 123)* and Figure 9.4.

Spill: Indicates whether the Allow overbank spilling option is checked for the coupling reach in which the river link is contained. The two possible values are "On" and "Off". See Figure 9.4.

WeirCoeff: The Weir coefficient value used in the coupling reach in which the river link is contained. See Figure 9.4.

HExpo: The Head exponent value used in the coupling reach in which the river link is contained. See Figure 9.4.

FullWdepth: The Minimum upstream height above bank for full weir width value used in the coupling reach in which the river link is contained. See Figure 9.4.

ThrVolSpill: Threshold volume value in cubic meters, which is the product between the Minimum flow are for overbank spilling value (for the coupling reach in which this river link is contained. See Figure 9.4) and the MIKE SHE cell size.

Chainage: Chainage of the MIKE Hydro River network that corresponds to the center of the link segment. They are sorted from highest to lowest chainage values for the same branch.

Branch: Name of the MIKE Hydro River Branch. Branches are sorted alphabetically.

9.3.5 Groundwater exchange with MIKE Hydro River

The exchange flow, Q, between a saturated zone grid cell and the river link is calculated as a conductance, C, multiplied by the head difference between the river and the grid cell.

$$Q = C \cdot \Delta h \tag{9.1}$$

Note that Eq. (9.1) is calculated twice - once for each cell on either side of the river link. This allows for different flow to either side of the river if there is a groundwater head gradient across the river, or if the aquifer properties are different.

Referring to Figure 9.2, the head difference between a grid cell and the river is calculated as

$$\Delta h = h_{grid} - h_{riv} \tag{9.2}$$

where h_{grid} is the head in the grid cell and h_{riv} is the head in the river link, as interpolated from the MIKE Hydro River h-points.

If the ground water level drops below the river bed elevation, the head difference is calculated as

$$\Delta h = z_{bot} - h_{riv} \tag{9.3}$$

where z_{bot} is the bottom of the simplified river link cross section, which is equal to the lowest point in the MIKE Hydro River cross-section.

In Eq. (9.1), the conductance, *C*, between the cell and the river link can depend on

- the conductivity of the aquifer material only. See Aquifer Only Conductance *(V2 p. 124)*, or
- the conductivity of the river bed material only. See River bed only conductance *(V2 p. 125)*, or
- the conductivity of both the river bed and the aquifer material. See Both aquifer and river bed conductance *(V2 p. 126)*.

Aquifer Only Conductance

When the river is in full contact with the aquifer material, it is assumed that there is no low permeable lining of the river bed. The only head loss between the river and the grid node is that created by the flow from the grid node to the river itself. This is typical of gaining streams, or streams that are fast moving.

Thus, referring to Figure 9.2, the conductance, *C*, between the grid node and the river link is given by

$$C = \frac{K \cdot da \cdot dx}{ds} \tag{9.4}$$

where *K* is the horizontal hydraulic conductivity in the grid cell, *da* is the vertical surface available for exchange flow, *dx* is the grid size used in the SZ component, and *ds* is the average flow length. The average flow length, *ds,* is the distance from the grid node to the middle of the river bank in the triangular, river-link cross-section. *ds* is limited to between 1/2 and 1/4 of a cell width, since the maximum river-link width is one cell width (half cell width per side).

There are three variations for calculating *da*:

- If the water table is higher than the river water level, *da* is the saturated aquifer thickness above the bottom of the river bed. Note, however, that *da* is not limited by the bank elevation of the river cross-section, which means that if the water table in the cell is above the bank of the river, *da* accounts for overland seepage above the bank of the river.
- If the water table is below the river level, then *da* is the depth of water in the river.
- If the river cross-section crosses multiple model layers, then *da* (and therefore *C*) is limited by the available saturated thickness in each layer. The exchange with each layer is calculated independently, based on the *da* calculated for each layer. This makes the total exchange independent of the number of layers the river intersects.

This formulation for *da* assumes that the river-aquifer exchange is primarily via the river banks, which is consistent with the limitation that there is no unsaturated flow calculated beneath the river.

River bed only conductance

If there is a river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the conductivity of the river bed. However, when the head loss across the river bed is much greater than the head loss in the aquifer material, then the head loss in the aquifer can be ignored (e.g. if the bed material is thick and very fine and the aquifer material is coarse). This is the assumption used in many groundwater models, such as MODFLOW.

In this case, referring to Figure 9.2, the conductance, *C*, between the grid node and the river link is given by

$$C = L_c \cdot w \cdot dx \tag{9.5}$$

where *dx* is the grid size used in the SZ component, L_c is the leakage coefficient [1/T] of the bed material, and *w* is the wetted perimeter of the cross-section.

In Eq. (9.5), the wetted perimeter, *w*, is assumed to be equal to the sum of the vertical and horizontal areas available for exchange flow. From Figure 9.2, this is equal to $da + l_h$, respectively. The horizontal infiltration length, l_h, is calculated based on the depth of water in the river and the geometry of the triangular river-link cross-section.

The infiltration area of the river link closely approximates the infiltration area of natural channels when the river is well connected to the aquifer. In this case, the majority of the groundwater-surface water exchange occurs through the banks of the river and decreases to zero towards the centre of the river. However, for losing streams separated from the groundwater table by an unsaturated zone, the majority of the infiltration occurs vertically and not through the river banks. In this case, the triangular shape of the river link does not really approximate wide losing streams.and the calculated infiltration area may be too small - especially if the MIKE Hydro River bank elevations are much higher than the river level. This can be compensated for by either choosing a lower bank elevation or by increasing the leakage coefficient.

There are three variations for calculating *da*:

- If the water table is higher than the river water level, *da* is the saturated aquifer thickness above the bottom of the river bed. Note, however, that *da* is not limited by the bank elevation of the river cross-section, which means that if the water table in the cell is above the bank of the river, *da* accounts for overland seepage above the bank of the river.

- If the water table is below the river level, then *da* is the depth of water in the river.
- If the river cross-section crosses multiple model layers, then *da* (and therefore *C*) is limited by the available saturated thickness in each layer. The exchange with each layer is calculated independently, based on the *da* calculated for each layer. This makes the total exchange independent of the number of layers the river intersects.

This formulation for *da* assumes that the river-aquifer exchange is primarily via the river banks, which is consistent with the limitation that there is no unsaturated flow calculated beneath the river.

Both aquifer and river bed conductance

If there is a river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the conductivity of the river bed and can be calculated as a serial connection of the individual conductances. Thus, referring to Figure 9.2, the conductance, *C*, between the grid node and the river link is given by

$$C = \frac{1}{\frac{ds}{K \cdot da \cdot dx} + \frac{1}{L_c \cdot w \cdot dx}} \tag{9.6}$$

where *K* is the horizontal hydraulic conductivity in the grid cell, *da* is the vertical surface available for exchange flow, *dx* is the grid size used in the SZ component, *ds* is the average flow length, L_c is the leakage coefficient [1/T] of the bed material, and *w* is the wetted perimeter of the cross-section. The average flow length, *ds,* is the distance from the grid node to the middle of the river bank in the triangular, river-link cross-section. *ds* is limited to between 1/2 and 1/4 of a cell width, since the maximum river-link width is one cell width (half cell width per side).

In Eq. (9.5), the wetted perimeter, *w*, is assumed to be equal to the sum of the vertical and horizontal areas available for exchange flow. From Figure 9.2, this is equal to $da + l_h$, respectively. The horizontal infiltration length, l_h, is calculated based on the depth of water in the river and the geometry of the triangular river-link cross-section.

The infiltration area of the river link closely approximates the infiltration area of natural channels when the river is well connected to the aquifer. In this case, the majority of the groundwater-surface water exchange occurs through the banks of the river and decreases to zero towards the centre of the river. However, in the case of losing streams separated from the groundwater table by an unsaturated zone, the majority of the infiltration occurs vertically and not through the river banks. In this case, the horizontal infiltration area may be too small, if the MIKE Hydro River bank elevations are much higher than the river level. This can be compensated for by either choosing a lower bank elevation or by increasing the leakage coefficient.

- There are three variations for calculating *da*:
- If the water table is higher than the river water level, *da* is the saturated aquifer thickness above the bottom of the river bed. Note, however, that *da* is not limited by the bank elevation of the river cross-section, which means that if the water table in the cell is above the bank of the river, *da* accounts for overland seepage above the bank of the river.
- If the water table is below the river level, then *da* is the depth of water in the river.
- If the river cross-section crosses multiple model layers, then *da* (and therefore *C*) is limited by the available saturated thickness in each layer. The exchange with each layer is calculated independently, based on the *da* calculated for each layer. This makes the total exchange independent of the number of layers the river intersects.

This formulation for *da* assumes that the river-aquifer exchange is primarily via the river banks, which is consistent with the limitation that there is no unsaturated flow calculated beneath the river.

9.3.6 Steady-state groundwater simulations

For steady-state groundwater models, MIKE Hydro River is not actually run. Rather the initial water level in MIKE Hydro River is used for calculating *da* in the conductance formulas and h_{riv} for the head gradient.

To improve numerical stability during steady-state groundwater simulations, the actual conductance used in the current iteration is an average of the currently calculated conductance and the conductance used in the previous iteration.

Canyon option for steady-state groundwater simulations

In the case of a deep, narrow channel crossing multiple model layers, the head difference used in Equations (9.1) and (9.2) can optionally be limited by the bottom elevation of the layer. Thus,

$$\Delta h = h_{grid} - max(h_{riv}, z) \tag{9.7}$$

where *z* is the bottom of the current layer.

The above formulation reduces the infiltration from upper layers by reducing the available gradient. Without the 'Canyon' option, MIKE SHE effectively assumes that the river is hydraulically connected to the upper most model layer, since MIKE SHE calculates the exchange flow with all layers that intersect the river based on the difference between the river level and the water table.

Currently, this option is only available for steady-state models. It is activated by means of the boolean Extra Parameter, *Enable Canyon Exchange*. For more information on the use of extra parameters, see Extra Parameters *(V2 p. 325)*.

9.4 Overland Flow Exchange with MIKE Hydro River

The exchange between overland flow and MIKE Hydro River can be calculated in three different ways. If the flooding from MIKE Hydro River to MIKE SHE cells is ignored (the "no flooding" option) then the exchange from overland flow is one way - that is overland flow only discharges to MIKE Hydro River. If the you want to simulate flooding from MIKE Hydro River to MIKE SHE then the water can be transfered from MIKE Hydro River to MIKE SHE using "Flood Codes" or via direct overbank spilling using a wier formula. In principle, the flood code option does not impact the solution time signficantly, is relatively easy to set up for simple cases and is sufficient when detailed flood plain flow is not required. Direct overbank spilling combined with the explicit solution method requires more detailed topopgraphy data and is useful when detailed flood plain flow is required, but can be significantly slower from a numerical perspective.

Flooding with Overbank Spilling

If you are simulating flooding on the flood plain using the overbank spilling option, then the MIKE Hydro River cross-sections are normally restricted to the main channel. The flood plain is defined as part of the MIKE SHE topography. Since, the bank elevation is used to define when a cell floods, it is more critical that the cross-sections are consistent with your topography, in the areas where you want to simulate flooding. The table in the simulation log file mentioned above is useful to locate these inconsistencies. It is usually necessary to have a very fine grid and a detailed DEM for such simulations, which tends to reduce the inconsistencies because it reduces the amount of interpolation and averaging when creating the model topography.

Flooding with Flood Codes

If you are simulating flooding on the flood plain using the flood code option, then flood plain elevation should be consistent with the cross-sections. Otherwise, the flood plain storage will be inconsistent with the river storage based on the cross-sections.

When you are using Flood Codes, you typically specify wide cross-sections for your rivers. The wide cross-sections can then account for the increased flood plain storage during flood events. MIKE Hydro River then places water on the MIKE SHE cells that are defined by flood codes - if the water level in the river is above the cell topography. The flood water is then free to infiltrate or evaporate as determined by MIKE SHE.

In such flooded cells, overland flow is no longer calculated, so there is no longer any overland exchange to MIKE Hydro River in flooded cells. Thus,

the bank elevation is not so critical, as long as the cell is flooded. However, when the flood recedes, the cells revert back to normal overland flow cells and the same considerations apply as if the cells were not flooded - namely the bank elevation should be below the topography to ensure that overland flow can discharge to the river link.

Flood codes are also commonly used for lakes and reservoirs. In this case, you specify the lake bed bathymetry as the topography (or using the Bathymetry option). The lake area is defined using flood codes and the MIKE Hydro River cross-sections stretch across the lake. MIKE Hydro River calculates the lake level and floods the lake. Overland flow adjacent to the lake intersects the flooded cells and the overland water is added to the lake cell (and to MIKE Hydro River as lateral inflow). Groundwater exchange to the lake is through the lake bed as saturated zone discharge. In principle, the saturated zone could discharge to the river link, but the local groundwater gradients would probably make this exchange very small.

Combining Flood Codes and Overbank Spilling

Flooding using Overbank spilling and Flood Codes is possible in the same model and even in the same coupling reach. The only restriction is that there is no overland flow calculated in cells flooded by means of Flood Codes. So, in a long coupling reach, you could allow overbank spilling and calculate overland flow using the explicit solver, but define flood codes in the wide downstream flood plain were the surface water gradients are very low during flooding and in the wide shallow reservoir half way down the system.

9.4.1 Lateral inflow to MIKE Hydro River from MIKE SHE overland flow

MIKE SHE's overland flow solver calculates the overland flow across the boundary of the MIKE SHE cells. If a river link is located on the cell boundary, any overland flow is intercepted by the river link and added to the water balance of the river link. However, two checks are first made to ensure exchange to the river is physically possible. The level of ponded water in the cell must be above the

1. water level in the river link, and
2. the relevant bank elevation of the river link.

However, there is no exchange from MIKE Hydro River to overland flow unless Overbank spilling is turned on for the Coupling Reach. If the water level in the river rises above the bank elevation, then the bank elevation is simply extended vertically upwards.

9.4.2 Flooding from MIKE Hydro River to MIKE SHE using Flood Codes

The MIKE SHE/MIKE Hydro River coupling allows you to simulate large water bodies such as lakes and reservoirs, as well as flooded areas. If this option is used, MIKE SHE/MIKE Hydro River applies a simple flood-mapping

procedure where MIKE SHE grid points (e.g. grid points in a lake or on a flood plain) are linked to the nearest H-point in MIKE Hydro River (where the water levels are calculated). Surface water stages are then calculated in MIKE SHE by comparing the water levels in the h-points with the surface topographic elevations.

Conceptually, you can think of the flooded cells as “side storages”, where MIKE Hydro River continues to route water downstream as 1D flow. But, at the same time, the water is available to the rest of MIKE SHE for evaporation and infiltration.

Determination of the Flooded Area and Water Levels

The flooded area in MIKE SHE must be delineated by means of integer flood codes, where each coupling reach is assigned a flood code.

During the simulation, the flood-mapping procedure calculates the surface water level on top of each MIKE SHE cell with a flood code by comparing the MIKE Hydro River surface water level to the surface topography in the model grid. A grid cell is flooded when the MIKE Hydro River surface water level is above the topography. The MIKE Hydro River water level is then used as the level of ponded surface water.

The actual water level in the grid cell is calculated as a distance weighted average of the upstream and downstream MIKE Hydro River h-points.

Calculation of the Exchange Flows

After the MIKE SHE overland water levels have been updated, MIKE SHE calculates the infiltration to the unsaturated and saturated zones and evapotranspiration. Thus, MIKE SHE simply considers any water on the surface, including MIKE Hydro River flood water as ‘ponded water’, disregarding the water source. In other words, ponded rainfall and ponded flood water are indistinguishable.

MIKE SHE does not calculate overland flow between cells that are flooded by MIKE Hydro River. Nor, does MIKE SHE calculate overland exchange to MIKE Hydro River, if the cell is flooded by MIKE Hydro River. However, lateral overland flow to neighbouring non-flooded cells is allowed. Thus, if there is a neighbouring, non-flooded cell with a topography lower than a flooded cell’s water level , then MIKE SHE will calculate overland flow to the non-flooded cell as normal.

The calculated exchange flow between the flooded grid cells and the overland, saturated, unsaturated zone or other source/sink terms is fed back to MIKE Hydro River as lateral inflow or outflow to the corresponding H-point in the next MIKE Hydro River time step.

In terms of the water balance, the surface water in the inundated areas belongs to the MIKE Hydro River water balance. In other words, if there is ponded water on the surface when the grid cell floods, the existing ponded water is added to the MIKE Hydro River water flow in the river. As long as the

element is flooded, any exchange to or from the surface water is managed by MIKE Hydro River as lateral inflow and regular overland flow is not calculated.

If the element reverts back to a non-flooded state, then any subsequent ponded water is again treated as regular overland flow and the water balance is accounted for within the overland flow component.

9.4.3 Direct Overbank Spilling to and from MIKE Hydro River

If you want to calculate 2D overland flow on the flood plain during a storm event, then you cannot use the **Flooding from MIKE Hydro River to MIKE SHE using Flood Codes** *(V2 p. 129)* method. The Area Inundation method is primarily used as a way to spread river water onto the flood plain and make it available for interaction with the subsurface via infiltration and evapotranspiration.

The Overbank spilling option treats the river bank as a broad crested weir. When the overland flow water level or the river water level is above the left or right bank elevation, then water will spill across the bank based on the standard broad crested weir formula

$$Q = \Delta x \cdot C \cdot (H_{us} - H_w)^k \cdot \left[1 - \left(\frac{H_{ds} - H_w}{H_{us} - H_w}\right)^k\right]^{0.385} \quad (9.8)$$

where Q is the flow across the weir, Δx is the cell width, C is the weir coefficient, H_{us} and H_{ds} refer to the height of water on the upstream side and downstream side of the weir respectively, H_w is the height of the weir, and k is a head exponent.

The units of the weir coefficient depend on the head exponent. In MIKE SHE, the default head exponent is 1.5, which means that the weir coefficient has units of [$m^{2/3}/s$].

If the water levels are such that water is flowing to the river, then the overland flow to the river is added to MIKE Hydro River as lateral inflow. If the water level in the river is higher than the level of ponded water, then the river water will spill onto the MIKE SHE cell and become part of the overland flow.

If the upstream water depth over the weir approaches zero, the flow over the weir becomes undefined. Therefore, the calculated flow is reduced to zero linearly when the upstream height goes below a threshold.

If you use the overbank spilling option, then you should also use the Explict Numerical Solution *(V2 p. 56)* for overland flow.

9.4.4 Converting from Flood Codes to Overbank Spilling

The explicit solver and overbank spilling from MIKE Hydro River to overland flow are new in the 2007 Release. In principle, if you were careful setting up your flood codes, then the conversion to overbank spilling should result in the same flooded area, with similar depths. The only difference will be that the water on the flooded area is flowing.

However, in practice the conversion is not likely to be this smooth. Flood code setups are typically done manually and the topography is typically not very closely controlled - as long as it was inundated when it was supposed to be. Furthermore, the need for detailed surface roughness (Manning's M) will require additional data. Finally, the complication of fully dynamic (diffusive wave) 2D flow can lead to complicated water flows across the flood plain. So, there is likely to be substantial adjustment and re-calibration to get the flooding right.

Fortunately, you can mix Flood codes and Overbank spilling in the same model and even in the same coupling reach. This allows you to update only the parts of your model where the overbank spilling is important and leave the Flood code option intact elsewhere.

9.5 Unsaturated Flow Exchange with MIKE Hydro River

Direct exchange between MIKE Hydro River and the unsaturated zone is not currently supported. Groundwater exchange is assumed to be a line source and sink at the boundary between cells and the exchange mechanism assumes that the primary exchange takes place along the river banks. This is a suitable assumption when the river is well connected to the aquifer.

However, when MIKE Hydro River can exchange water with overland flow via overbank spilling or flood codes, then river water is added to the ponded water on a MIKE SHE cell, which can then infiltrate to the unsaturated zone.

9.6 Water balance with MIKE Hydro River

The water balance tool in MIKE SHE (Using the Water Balance Tool *(V1 p. 91)*) includes the exchange with MIKE Hydro River, but it does not include the water balance within MIKE Hydro River. In other words, once water enters MIKE Hydro River it is no longer part of the MIKE SHE water balance. Thus, there are numerous water balance items that detail the different exchanges to and from MIKE Hydro River.

Water exchanges within MIKE Hydro River can be evaluated using the MIKE View tool. In some cases, this may require you to include the additional output for MIKE Hydro River, which is selected in the Additional Output tab in MIKE Hydro River's HD editor.

Note: output in MIKE Hydro River is instantaneous, whereas the output in MIKE SHE is generally accumulated within a time step. Therefore, a flow at a rate at a point in MIKE Hydro River (e.g. a weir) will be the instantaneous flow at the end of the time step. In MIKE SHE, however, the flow into a cell will be the average flow over the time step.

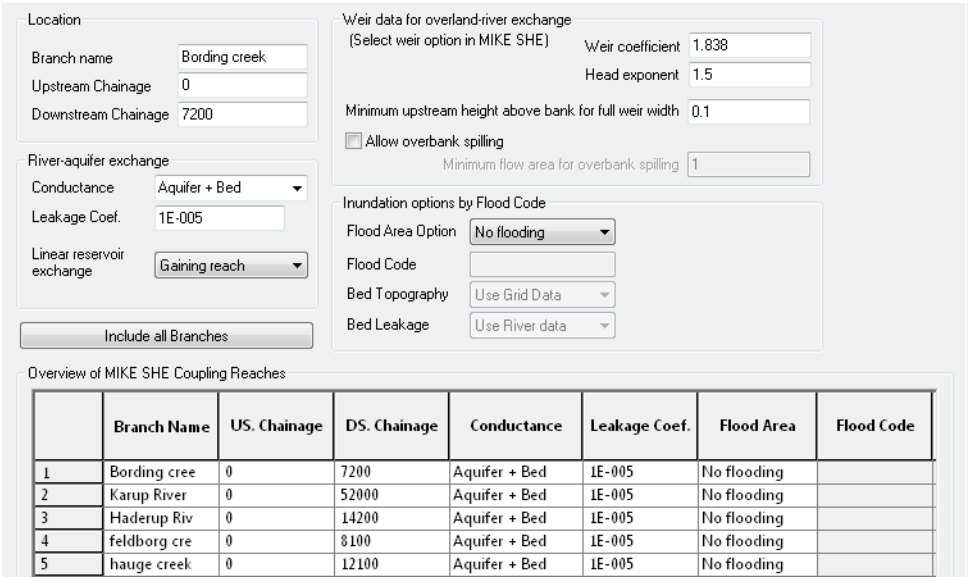
9.7 MIKE Hydro River User Interface

The following section provides additional information for the MIKE Hydro River dialogues that are commonly used with MIKE SHE.

9.7.1 MIKE SHE Coupling Reaches

Each MIKE Hydro River branch that exchanges water with MIKE SHE is called a coupling reach. A MIKE Hydro River branch can be sub-divided into several coupling reaches. A reason for doing so could be to allow different riverbed leakage coefficients for different parts of the river.

The upper half of the dialogue displays the properties of the current coupling reach. While, the bottom half of the dialogue is a table listing all of the coupling reaches defined.



	Branch Name	US. Chainage	DS. Chainage	Conductance	Leakage Coef.	Flood Area	Flood Code
1	Bording cree	0	7200	Aquifer + Bed	1E-005	No flooding	
2	Karup River	0	52000	Aquifer + Bed	1E-005	No flooding	
3	Haderup Riv	0	14200	Aquifer + Bed	1E-005	No flooding	
4	feldborg cre	0	8100	Aquifer + Bed	1E-005	No flooding	
5	hauge creek	0	12100	Aquifer + Bed	1E-005	No flooding	

Figure 9.4 MIKE SHE River Links dialogue in the tabular view of the MIKE Hydro River Network Editor

Include all branches button

If the Include all branches button is pressed all the branches in the MIKE Hydro River setup will be copied to the MIKE SHE Links table. Branches that

should not be in the coupling can subsequently be deleted manually and the specifications for the remaining branches completed. Thus, you may have a large and complex hydraulic model, but only couple certain reaches to MIKE SHE. All branches will still be in the hydraulic MIKE Hydro River model but MIKE SHE will only exchange water with branch reaches that are listed in the MIKE SHE links table.

Note The Include all branches button will erase all existing links that have been specified.

Location

The **branch name**, **upstream chainage** and **downstream chainage** define the stretch of river that can exchange water with MIKE SHE. A MIKE Hydro River branch can be sub-divided into several coupling reaches, to allow, for example, different riverbed leakage coefficients for different parts of the river.

River Aquifer Exchange

Conductance

The river bed conductance can be calculated in three ways.

Aquifer only - When the river is in full contact with the aquifer material, it is assumed that there is no low permeable lining of the river bed. The only head loss between the river and the grid node is that created by the flow from the grid node to the river itself. This is typical of gaining streams, or streams that are fast moving. More detailed information on this option can be found in Aquifer Only Conductance *(V2 p. 124)*.

River bed only - If there is a low conductivity river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the conductivity of the river bed. However, when the head loss across the river bed is much greater than the head loss in the aquifer material, then the head loss in the aquifer can be ignored (e.g. if the bed material is thick and very fine and the aquifer material is coarse). This is the assumption used in many groundwater models, such as MODFLOW. More detailed information on this option can be found in River bed only conductance *(V2 p. 125)*.

Aquifer + Bed - If there is a low conductivity river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the total conductivity of the between the river and the adjacent groundwater can be calculated as a serial connection of the individual conductances. This is commonly the case, when the aquifer material presents a significant head loss. For example, when the aquifer is relatively fine and the groundwater cells are quite large.More detailed information on this option can be found in Both aquifer and river bed conductance *(V2 p. 126)*.

Leakage Coefficient - [1/sec]

This is the leakage coefficient for the riverbed lining in units of [1/seconds]. The leakage coefficient is active only if the conductance calculation method includes the river bed leakage coefficient.

Linear Reservoir Exchange

If you are using the Linear Reservoir method for groundwater in MIKE SHE, then by default the Interflow and Baseflow reservoirs discharge uniformly to all the river links within the reservoir. This is generally true in the lower reaches. However, in the upper reaches many rivers discharge to the groundwater system.

In this dialogue, you can define whether or not a branch is a Gaining branch (default) or a Losing branch. If the branch is a:

- Gaining branch, then the leakage coefficient and wetted area are ignored and the rate is discharge from the Baseflow reservoir to the river is calculated based on the Linear Reservoir method.
- Losing branch, then the rate of discharge from the river to the Baseflow reservoir is calculated using:

  Q=water depth*bank width* branch length* leakage coefficient.

The gaining and losing calculations are done in MIKE SHE for every river link within the Baseflow reservoir. For the losing river links, the water level is interpolated from the nearest h-points, the bottom elevation and bank width is interpolated from the nearest cross-sections. The length is simply the cell size. MIKE SHE keeps track of the inflow volumes to ensure that sufficient water is available in the river link.

Weir Data for overland-river exchange

The choice of using the weir formula for overland-river exchange is a global choice made in the MIKE SHE OL Computational Control Parameters *(V1 p. 183)* dialogue. If the weir option is chosen in MIKE SHE, then all MIKE Hydro River coupling reaches will use the weir formula for moving water across the river bank.The weir option is typically used when you want to simulate overbank spilling and detailed 2D surface flow in the flood plains. The following parameters and options are available when you specify the weir option in MIKE SHE. If you chose the Manning equation option in MIKE SHE, then these parameters are ignored.

Weir coefficient and Head exponent

The Weir coefficient and head exponent refer to the C and k terms respectively in Equation (9.8). The default values are generally reasonable. Both the weir coefficient and the head exponent are dimensionless.

Minimum upstream height above bank for full weir width

In Equation (9.8), when the upstream water depth above the weir approaches zero, the flow over the weir becomes undefined. To prevent numerical problems, the flow is reduced linearly to zero when the water depth is below the minimum upstream height threshold. The EUM data type is Water Depth.

Allow overbank spilling

This checkbox lets you define which branches are allowed to flood over their banks. Thus, you can allow flooding from MIKE Hydro River only in branches with defined flood plains, or only in areas of particular interest.

If overbank spilling is not allowed for a particular branch, then the overland-river exchange is still calculated using the weir formula, but the exchange is only one way - that is from overland flow to the river.

Minimum flow area for overbank spilling

The minimum flow area threshold prevents overbank spilling when the river is nearly dry. The flow area is calculated by dividing the volume of water in the coupling reach by the length of the reach. So, you can think of this threshold as a minimum (river volume)/(length of river) before overbank spilling can occur. The EUM data type is Flow Area, which by default is m^2.

The default value is 1 m^3/m length of river. This is quite a small amount of water for most reasonable rivers and should be adjusted based on the river width. For example for the default value of 1 m^3/m,

- if the width of your river is 10m wide, then spilling will occur when the water level is 10cm above the bank elevation.
- if the width of your river is 200m wide, then spilling would start when the water level is only 5mm above the bank elevation.

The cell size also plays a role in determining a reasonable threshold value. When a cell is flooded, the entire cell is covered by water. If the cell size is 1000m x 1000m, then a discharge onto the flood plain of 1 m^3/m of river will be only 1mm deep across the cell.

Thus, a value of 1-5 m^3/m is probably reasonable for small rivers (10-20m wide) and small grid cells(50-100m). For larger rivers (+50m wide) and larger grid cells (200-500m), a value of 10-50 m^3/m is probably more reasonable.

Inundation options by Flood Code

The Inundation method allows specified model grid cells to be flooded if the MIKE Hydro River water level goes above the topography of the cell. In this case, water from MIKE Hydro River is “deposited” onto the flooded cell. The flood water can then infiltrate, or evaporate. However, overland flow between flooded cells and to the river is not calculated. Also, the flooded water remains as part of the MIKE Hydro River water balance and is only transferred to MIKE SHE when it infiltrates.

Inundation areas and their associated Flood codes are specified on a coupling reach basis.

Flood Area Option

The following three options are available for the Flood Area Option:

- **No Flooding** (default) With the No flooding option, the MIKE Hydro River is confined between the left and right banks. If the water level goes above the bank elevation, then the river is assumed to have vertical banks above the defined left and right bank locations. No flooding via flood codes will be calculated.

  Note If neither inundation nor overbank spilling is allowed, then the overland flow exchange to the river is one way only. The only mechanism for river water to flow back into MIKE SHE is through baseflow infiltration to the groundwater. If overland flow does spill into the river, there is first a check to make sure that the water level in the river is not higher than the ponded water.

- **Manual** If the Manual option is selected, then you must supply a Flood code map in MIKE SHE. This Flood code map is used to established the relationship between MIKE Hydro River h-points and individual model grids in MIKE SHE. MIKE SHE then calculates a simple flood-mapping during the pre-processing that is used during the simulation to assign river water stages to the MIKE SHE cells if the river level is above the topography.

- **Automatic** The automatic flood mapping option is useful if the river network geometry is not very complex or for setting up the initial flood mapping, for later refinement. The automatic method, maps out a polygon for each coupling reach based on the left and right bank locations of all the cross-sections along the coupling reach. All cells within this polygon are assigned an integer flood code, unique to the coupling reach. The automatic method works reasonably well along individual branches with cross-sections that represent the flood plain. At branch intersections the assigned flood code may not be correct. However, this is often not serious because at river confluences the water levels in the different branches are roughly the same anyway. In any case, the flood code map is available in MIKE SHE's preprocessed tab, where you can check its reasonableness. Right clicking on the map will give you the option of saving the map to a dfs2 file, which you can then correct and use with the Manual option.

Flood Code

If the Manual option is selected, then you must specify a Flood code for the coupling reach. The flood code is used for mapping MIKE SHE grids to MIKE Hydro River h-points. You must click on the Flood Code checkbox in Figure 9.4, and then specify an integer flood code file in MIKE SHE. The

specified flood code for the coupling reach must exist in the dfs2 Flood Code file. It is important to use unique flood codes to ensure correct flood-mapping.

Bed Topography

Since the flood mapping procedure will only flood a cell when the river water level is above the cell's topography, accurate flood inundation mapping requires accurate elevation data. If one of the flood options are selected, then you have the option to refine the topography of the flood plain cells based on the actual cross-section elevations or on a more detailed local-scale DEM, if it exists.

- **Use Grid Data** (default) If Grid Data option is selected, the MIKE SHE topography value is used to determine whether or not the cell is flooded. However, the program first checks to see if a Bathymetry file has been specified.

  If a Bathymetry file is available, the topography values of the cells with flood codes are re-interpolated based on the bathymetry data. The bathymetry option is useful when a more detailed DEM exists for the flood plain area compared to the regional terrain model.

- **Use Cross-section** If the Cross-section option is specified the topography values of the cells with flood codes are re-interpolated based on the cross-section data.

  When the cross-section option is selected, the pre-processor maps out a flood-plain polygon for the coupling reach, based on the left and right bank locations of all the cross-sections along the coupling reach. Interpolated cross-sections are created between the available actual cross-sections, if the cross-section spacing is greater than ½ Δx (grid size). All the cross-sections (real and interpolated) are sampled to obtain a set of point values for elevation in the flood plain. The topography values of all cells with the current flood code that are within the flood-plain polygon are re-interpolated using the bilinear interpolation method to obtain a new topography value.

  In principle, the Cross-section option ensures a good consistency between MIKE SHE grid elevations and MIKE Hydro River cross-sections. There will, however, often be interpolation problems related to river meandering, tributary connections, etc., where wide cross-sections of separate coupling reaches overlap. Thus, you can make the initial MIKE SHE set-up using the Cross-section option and then subsequently retrieve and check the resulting ground surface topography, from the pre-processed data. If needed, the pre-processed topography can be saved to a .dfs2 file (right click on the map), modified and then used as input for a new set-up, now using the Use Grid Data option.

Bed Leakage

If one of the flood options are selected, then you must also specify if and how the leakage coefficient will be applied on the flooded cells. The infiltration/seepage of MIKE SHE flood grids is calculated as ordinary overland exchange with the saturated or unsaturated zone. That is, the leakage coefficient, if it exists, is applied to both saturated exchange to and from the flooded cell and unsaturated leakage from the flooded cell. In the case of the unsaturated leakage, the actual leakage is controlled by either the leakage coefficient or the unsaturated zone hydraulic conductivity relationship - which yields the lowest infiltration rate.

- **Use grid data** In this case, the leakage coefficient specified in Surface-Subsurface Leakage Coefficient is used. If this item has not been specified, then the leakage coefficient will be calculated based on the aquifer material only.
- **Use river data** (default) In this case, the Leakage Coefficient - [1/sec] for the coupling reach is actually copied to the flooded cell and used for all flood grid points of the coupling reach. This makes sense if the flood plain is frequently flooded and covered with the same sediments as the river bed. However, in many cases the flood plain material is not the same as the river bed and the infiltration rate can be substantially different.

9.8 Common MIKE Hydro River Error Messages

There are a number of common MIKE Hydro River error messages that you are likely to encounter when using MIKE Hydro River with MIKE SHE.

Error No 25: At the h-point ____ the water depth greater than 4 times max. depth

This error message essentially says that your MIKE Hydro River model is unstable. It frequently occurs when there is an inconsistency in your bed elevations at the branch junctions. For example, if the bed elevation of the main branch is much greater than the side branch, then the water piles up and causes this error.

Warning No 47: At the h-point ____ the water level as fallen below the bottom of the slot x times

This warning message essentially says that your MIKE Hydro River model is unstable. The slot is a numerical trick that keeps a very small amount of water in the MIKE Hydro River cross-section when the river is dry. So, when the water level falls below the slot, it implies that your river has dried out. This warning frequently occurs when there is either an inconsistency in your bed elevations or there is an error in your boundary conditions that is keeping water from entering the system.

Warning No __: Bed levels not the same

This warning message is issued when the bed elevation of a side branch is not the same as the main branch. If the difference is small (say a few cm) it can usually be ignored. However, if the side branch is much lower than the main branch then this warning will often be accompanied by Error No 25: At the h-point ____ the water depth greater than 4 times max. depth, as the water will pile up and not be able to flow into the main branch. If the side branch is only slightly lower than the main branch or even if they are the same, then backward flows can occur in the side branch when the water level in the main branch rises. If this is realistic fine, but often it is not. More typically, the side branch is slightly higher than the main branch.

10 Unsaturated Zone - Technical Reference

Unsaturated flow is one of the central processes in MIKE SHE and in most model applications. The unsaturated zone is usually heterogeneous and characterized by cyclic fluctuations in the soil moisture as water is replenished by rainfall and removed by evapotranspiration and recharge to the groundwater table. Unsaturated flow is primarily vertical since gravity plays the major role during infiltration. Therefore, unsaturated flow in MIKE SHE is calculated only vertically in one-dimension, which is sufficient for most applications. However, this may limit the validity of the flow description in some situations, such as on steep hill slopes with contrasting soil properties in the soil profile. MIKE SHE includes an iterative coupling procedure between the unsaturated zone and the saturated zone to compute the correct soil moisture and the water table dynamics in the lower part of the soil profile.

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

Richards Equation

The full Richards equation requires a tabular or functional relationship for both the moisture-retention curve and the effective conductivity.

The full Richards equation is the most computationally intensive, but also the most accurate when the unsaturated flow is dynamic.

Gravity Flow

The simplified gravity flow procedure assumes a uniform vertical gradient and ignores capillary forces.

The simplified gravity flow procedure provides a suitable solution when you are primarily interested in the time varying recharge to the groundwater table based on actual precipitation and evapotranspiration and not the dynamics in the unsaturated zone.

Two Layer Water Balance

The simple two-layer water balance method divides the unsaturated zone into two zones: the root zone and the zone between the roots and the water table.

The simple two-layer water balance method is suitable when the water table is shallow and groundwater recharge is primarily influenced by evapotranspiration in the root zone.

10.1 Richards Equation

The driving force for transport of water in the unsaturated zone is the gradient of the hydraulic head, h, which includes a gravitational component, z, and a pressure component, ψ. Thus,

$$h = z + \psi \tag{10.1}$$

The gravitational head at a point is the elevation of the point above the datum (z is positive upwards). The reference level for the pressure head component is the atmospheric pressure. Under unsaturated conditions the pressure head, ψ, is negative due to capillary forces and short range adsorptive forces between the water molecules and the soil matrix. These forces are responsible for the retention of water in the soil. As these two forces are difficult to separate, they are incorporated into the same term. Although the physical phenomena creating the pressure head under unsaturated and saturated conditions are very different, the pressure head is considered to be a continuous function across the water table, with the pressure being negative above and positive below the water table.

For vertical flow, the driving force for the transport of water is the vertical gradient of the hydraulic head. Thus,

$$\Delta h = \frac{\partial h}{\partial z} \tag{10.2}$$

The volumetric flux is then obtained from Darcy's law:

$$q = -K(\theta)\frac{\partial h}{\partial z} \tag{10.3}$$

where $K(\theta)$ is the unsaturated hydraulic conductivity. Assuming that the soil matrix is incompressible and the soil water has a constant density, the continuity equation will be:

$$\frac{\partial \theta}{\partial t} = -\frac{\partial q}{\partial z} - S(z) \tag{10.4}$$

where θ is the volumetric soil moisture and S is the root extraction sink term. Combining Eqs. (10.1), (10.3) and (10.4) yields

$$\frac{\partial \theta}{\partial t} = \frac{\partial}{\partial z}\left(K(\theta)\frac{\partial \psi}{\partial z}\right) + \frac{\partial K(\theta)}{\partial z} - S(z) \tag{10.5}$$

The dependent variables, θ and ψ, in Eq. (10.5) are related through the hydraulic conductivity function, $K(\theta)$, and the soil moisture retention curve, $\psi(\theta)$

Eq. (10.5) is general, in the sense that it is equally valid in both homogeneous and heterogeneous soil profiles, and there are no constraints on the hydraulic functions.

Introducing the concept of soil water capacity

$$C = \frac{\partial \theta}{\partial \psi} \tag{10.6}$$

which is the slope on the soil moisture retention curve, then the tension-based version of Eq. (10.5) is

$$C\frac{\partial \psi}{\partial t} = \frac{\partial}{\partial z}\left(K(\theta)\frac{\partial \psi}{\partial z}\right) + \frac{\partial K(\theta)}{\partial z} - S \tag{10.7}$$

This equation is usually referred to as Richards equation, which is named after L.A. Richards who first used it in 1931. It still applies when ψ becomes positive, in which case the equation degenerates to the LaPlace equation.

The sink terms in Eq. (10.7) are calculated from the root extraction for the transpiration in the upper part of the unsaturated zone. The integral of the root extraction over the entire root zone depth equals the total actual evapotranspiration. Direct evaporation from the soil is calculated only for the first node below the ground surface.

10.1.1 Numerical Solution of Richards Equation

MIKE SHE uses a fully implicit formulation in which the space derivatives of Eq. (10.7) are described by their finite difference analogues at time level n+1. The values of $C(\theta)$ and $K(\theta)$ are referred to at time level $n+½$. These are evaluated in an iterative procedure averaging C^n, K^n with C^m, K^m respectively. C^m and K^m are calculated as a running average of the coefficients found in each iteration.

This solution technique has been found to eliminate stability and convergence problems arising from the non-linearity of the soil properties.

For an interior node, the implicit scheme yields the following discrete formulation of the vertical flow:

$$q_{J+1/2}^{n+1} = -K_{J+1/2}^{n+1/2}\left(\frac{\psi_{J+1}^{n+1} - \psi_J^{n+1}}{\Delta Z_{J+1}} + 1\right) \tag{10.8}$$

where the subscript *J* refers to the spatial increment and the superscript *n* refers to the time increment. The vertical grid system for a soil column is shown in Figure 10.1. Similar to Eq. (10.8) the discrete form of Eq. (10.1) gives

$$C_J^{n+1}\frac{\psi_J^{n+1}-\psi_J^n}{\Delta t}=\left|K_{J+\frac{1}{2}}^{n+\frac{1}{2}}\left(\frac{\psi_{J+1}^{n+1}-\psi_J^{n+1}}{\Delta Z_{J+1}}\right)\right.$$
$$\left.-K_{J-\frac{1}{2}}^{n+\frac{1}{2}}\left(\frac{\psi_J^{n+1}-\psi_{J-1}^{n+1}}{\Delta Z_J}\right)\right|\frac{1}{\frac{1}{2}(\Delta Z_{J+1}+\Delta Z_J)} \qquad (10.9)$$
$$-S_J^{n+1}$$

The soil property *K* is centred in space using the arithmetic mean:

$$K_{J+\frac{1}{2}}^{n+\frac{1}{2}}=\frac{K_{J+1}^{n+\frac{1}{2}}+K_J^{n+\frac{1}{2}}}{2} \qquad K_{J-\frac{1}{2}}^{n+\frac{1}{2}}=\frac{K_J^{n+\frac{1}{2}}+K_{J-1}^{n+\frac{1}{2}}}{2} \qquad (10.10)$$

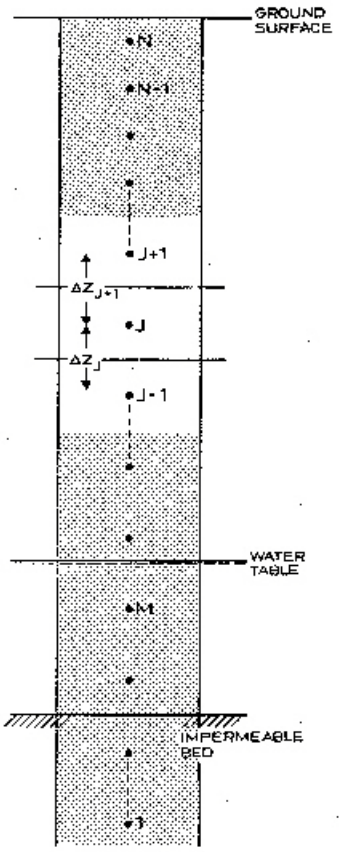


Figure 10.1 Vertical Discretisation in the Unsaturated Zone.

Eq. (10.9) involves three unknown values at time *n+1* and one known value at time *n* for each node. Written for all nodes with reference to Figure 10.1, a system of *N-M+1* equations with *N-M+1* unknowns is obtained. The system of equations forms a tri-diagonal matrix:

$$\begin{bmatrix} B_N & A_N & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ G_{N-1} & B_{N-1} & A_{N-1} & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & G_J & B_J & A_J & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & G_{M+1} & B_{M+1} & A_{M+1} \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & G_M & B_M \end{bmatrix} \begin{bmatrix} \psi_N \\ \psi_{N-1} \\ \dots \\ \dots \\ \psi_J \\ \dots \\ \dots \\ \psi_{M+1} \\ \psi_M \end{bmatrix} = \begin{bmatrix} D_N \\ D_{N-1} \\ \dots \\ \dots \\ D_J \\ \dots \\ \dots \\ D_{M+1} \\ D_M \end{bmatrix} \quad (10.11)$$

The *J*'th row in the matrix is

$$A_J^{n+1}\psi_{J-1} + B_J^{n+1}\psi_J + G_J^{n+1}\psi_{J+1} = D_J \tag{10.12}$$

where

$$\begin{aligned}
A_J &= -K_{J-\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_- \\
B_J &= C_J^{n+\frac{1}{2}} / \Delta t + K_{J+\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_+ + K_{J+\frac{1}{2}}^{n-\frac{1}{2}} / \Delta Z_- \\
G_J &= -K_{J+\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_+ \\
D_J &= C_J^{n+\frac{1}{2}}\psi_J^n / \Delta t + \frac{K_{J+\frac{1}{2}}^{n+\frac{1}{2}} - K_{J-\frac{1}{2}}^{n+\frac{1}{2}}}{\frac{1}{2}(\Delta Z_{J+1} + \Delta Z_J)} - S_J^{n+1} \\
\Delta Z_+ &= \frac{1}{2}\Delta Z_{J+1}(\Delta Z_{J+1} + \Delta Z_J) \\
\Delta Z_- &= \frac{1}{2}\Delta Z_J(\Delta Z_{J+1} + \Delta Z_J)
\end{aligned} \tag{10.13}$$

The solution to the matrix system Eq. (10.11) is solved by Gaussian elimination. Assuming that ψ_J^{n+1} and ψ_{J+1}^{n+1} can be related in the following recurrence relation

$$\psi_J^{n+1} = E_{J+1}\psi_{J+1}^{n+1} + F_{J+1} \tag{10.14}$$

The E_{J+1} and F_{J+1} can be calculated by combining Eqs. (10.12) and (10.14) as follows:

$$E_{J+1} = \frac{-G_J}{A_J E_J + B_J} \qquad F_{J+1} = \frac{D_J - A_J F_J}{A_J E_J + B_J} \tag{10.15}$$

Given the boundary conditions at the bottom and top nodes, ψ is computed for all nodes in a double sweep procedure:

1. *E* and *F* values are calculated from Eqs. 10.13 and 10.15 for all nodes from bottom-to-top in a *E*, *F*-sweep.
2. ψ is then calculated from Eq. 10.14 for all nodes in a top-to-bottom sweep.

Briefly, the iterative procedure within each time step is

1. the final result at time *n* (i.e. C_J^n and K_J^n) is used for the initial estimate of C_J^0 and K_J^0 for the first iteration,
2. then the following convergence criteria is checked for every node after each iteration, *i*

$$|\psi^i - \psi^{i-1}| < tolerance\ criteria \qquad if\ |\psi| < 0.5 \tag{10.16}$$

$$\left|\frac{\psi^i - \psi^{i-1}}{\psi^i}\right| < tolerance\ criteria \qquad if\ (|\psi| \geq 0.5) \tag{10.17}$$

3. if this convergence criteria is satisfied then a solution at the current time level (i.e. *n+1)* has been found.
4. if the criteria is not fulfilled then C_J^{i+1} and K_J^{i+1} are updated for the next iteration by

$$C_J^{i+1} = \frac{\sum^{i} C_J}{\frac{1}{i} + C_J^0} \tag{10.18}$$

$$K_J^{i+1} = \frac{\sum^{i} K_J}{\frac{1}{i} + K_J^0} \tag{10.19}$$

10.1.2 Boundary Conditions

The unsaturated zone extends from the ground surface to the groundwater table. The vertical flow is determined by the boundary conditions at each end of the column. However, the UZ column only exchanges water with the upper node of the SZ model, even if the UZ model extends below the top layer of the SZ model (see Limitations of the UZ - SZ coupling *(V2 p. 184)*).

Upper boundary

The upper boundary condition is either

- a constant flux condition within each time step (Neumann boundary condition), which is determined by the infiltration rate, or
- a constant head condition within each time step (Dirichlet boundary condition), which is determined by the level of ponded water on the surface.

If the infiltration is equal to the net rainfall rate at the soil surface, R, Eq. (10.9) can be written for the top node N as

$$C_N^{n+\frac{1}{2}}\frac{\psi_N^{n+1}\psi_N^n}{\Delta t} = \left(-R - K_{n-\frac{1}{2}}^{n+\frac{1}{2}}\left(\frac{\psi_N^{n+1}-\psi_{N-1}^{n+1}}{\Delta Z_N}+1\right)\right)\frac{1}{0.5(\Delta Z_{N+1}+\Delta Z_n)} - S_N^{n+1} \tag{10.20}$$

where R is defined negative downwards.

Writing Eq. (10.20) in a similar form to Eq. (10.12) yields

$$A_N\,\psi_{N-1}^{n+1} + B_N\,\psi_N^{n+1} = D_N \tag{10.21}$$

where

$$\begin{aligned} A_N &= -K_{N-\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_- \\ B_N &= C_N^{n+\frac{1}{2}} / \Delta t + K_{N-\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_- \\ D_N &= C_N^{n+\frac{1}{2}}\frac{\psi_N^n}{\Delta t} + \frac{-R - K_{N-\frac{1}{2}}^{n+\frac{1}{2}}}{\frac{1}{2}(\Delta Z_{N+1}+\Delta Z_N)} - S_N^{n+1} \end{aligned} \tag{10.22}$$

If water is ponded on the ground surface, the first node is assumed saturated and the boundary condition simply becomes

$$\psi_N^{n+1} = \psi_N^n = \Delta Z_{N+1} \tag{10.23}$$

At the beginning of each UZ time step, the amount of available water for infiltration is calculated as the amount of ponded water, plus the net rainfall at the ground surface, minus evaporation from ponded water.

The upper boundary condition is applied depending on the deficit in the uppermost UZ node:

- If the available water exceeds the deficit in the top UZ node, then the head boundary is used.
- If the available water is less then the deficit in the top UZ node, then a flux boundary is used.

If the head boundary is used, then when the solution is found, the amount of infiltration is compared against the available amount of infiltration. If the avail-

able infiltration is exceeded then the solution is repeated with the flux boundary.

If the flux boundary is used, then the available water for infiltration is divided by the time step length to get the infiltration rate. When the solution is found, the water content in the uppermost UZ node is compared to the saturated water content. If the saturated water content was reached or exceeded, then the solution is repeated using the head boundary.

The solution is restricted to a maximum of one repeat in each time step, to prevent an infinite loop.

Lower Boundary

In most cases, the lower boundary is a pressure boundary that is determined by the water table elevation. Then Eq. (10.22) consists of N-M equations. If node M is the first node below the water table, then

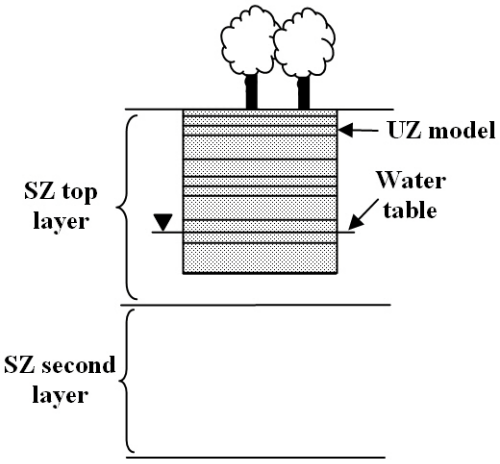
$$E_{M+1} = 0 \quad F_{M+1} = \psi_M^{n+1} = h \tag{10.24}$$

where h is the distance between the water table and node M. Noted that ψ_M is independent of ψ_{M+1} since $E_{M+1} = 0$.

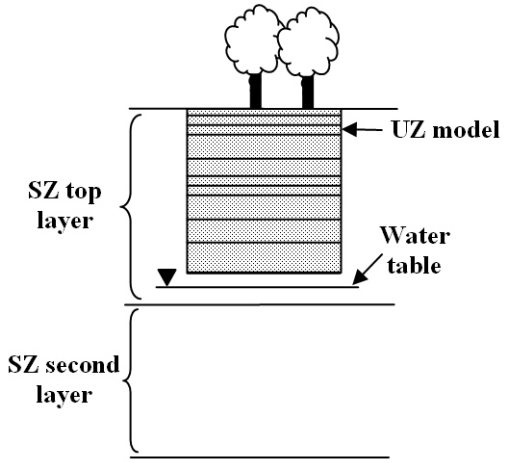
If the UZ model is not coupled to a SZ model, then the lower boundary is automatically converted from a pressure head boundary to a zero flux boundary (Q=0) if the water table falls below the impermeable bed (see Figure 10.1) and, at same time there is an upward flux in the lower part of the profile. The head boundary is re-started as soon as a positive hydraulic pressure gradient is calculated or the water table starts to rise in the profile.

However, when the UZ model is coupled to an SZ model, the UZ model exchanges water only with the top node of the SZ model. This can lead to three principle conditions:

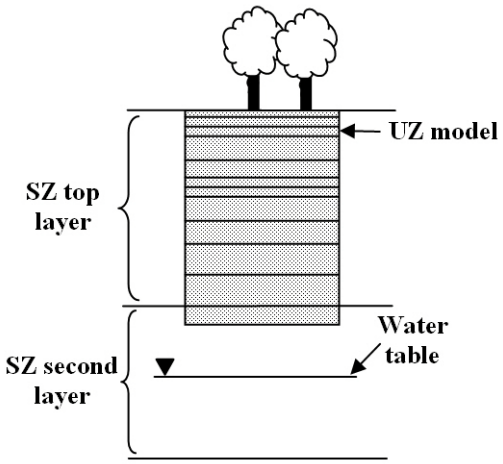
- If the UZ model intersects the water table in the top layer of the SZ model, then the lower boundary is a normal pressure boundary.



- If the UZ model does not extend to the bottom of the uppermost SZ layer, and the water table in the SZ model falls below the bottom UZ layer, then an error will be generated and the simulation will be stopped.



- If the UZ model extends below the top SZ layer and the top SZ layer dries out, then the UZ model treats the bottom boundary as either a pressure boundary with the pressure equal to the elevation of the bottom of the uppermost SZ layer, or a zero-flux boundary if there is an upward gradient at the lower boundary. The zero-flux boundary is only used for the Full Richards Equation option because the tension term can yield an upwards flow from the groundwater table, which is not physically possible when the upper SZ layer is dry.



In the first and last cases above, the flux out the bottom of the UZ column is added as a flux boundary condition to the uppermost SZ node.

10.1.3 Initial Conditions

By default, the initial conditions for ψ are generated by MIKE SHE assuming an equilibrium soil moisture/pressure profile with no-flow. The equilibrium profile is calculated assuming hydrostatic conditions, as illustrated in Figure 10.2. The pressure decreases linearly from zero at the groundwater table to ψ_{FC} when the moisture content reaches the field capacity and is then constant for all nodes above this point. The assumption is that the flow is (almost) zero at moisture contents below the field capacity.

The assumption that the initial water content is not below field capacity means that ET can occur from the first time step. If the water content was based on the saturation-pressure profile all the way to the residual water content, then the model would likely start with very dry soils. There would be no recharge or ET until the soils wetted up after significant rainfall.

However, in arid and semi-arid conditions the equilibrium water content may be quite low, as the soil profile may have drained for a long time with little infiltration. In this case, it is usually better to base the initial condition on the full saturation-pressure relationship - all the way to residual water content.

In both cases, an initial warm up of the UZ is usually required. In the first case, there will be some initial drainage from the UZ as the moisture content equilibrates with the rainfall rate. In the second case, the soil profile will gain water and the groundwater recharge will be initially very low as the soil profile absorbs all of the infiltration.

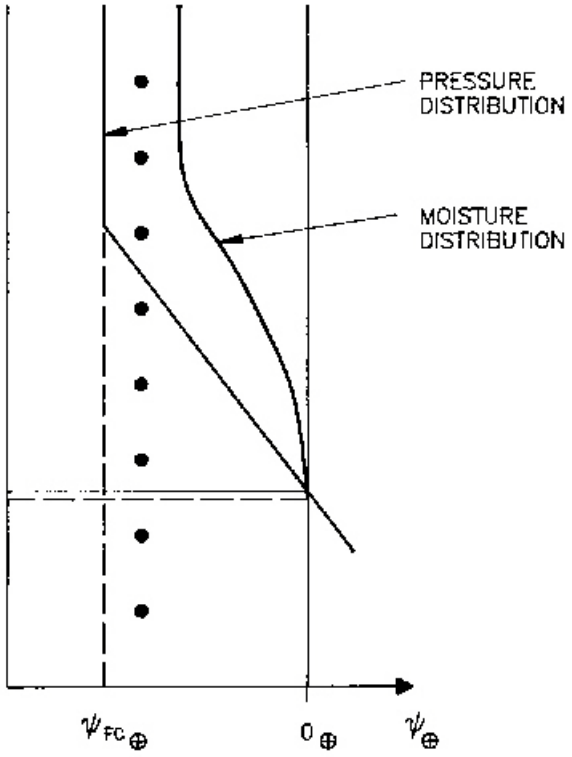


Figure 10.2 Illustration of initial soil moisture profile and pressure head profile.

10.1.4 Sources and sinks

There is a source/sink term for each computational node. These sink terms are calculated from the root extraction due to transpiration in the upper part of the unsaturated zone. The integral of the root extraction over the entire root zone depth equals the total actual evapotranspiration. Direct evaporation from the soil is calculated only for the first node in the soil column.

10.2 Gravity Flow

The driving force for transport of water in the unsaturated zone is the gradient of the hydraulic head, h, which includes a gravitational component, z, and a pressure component, ψ. Thus,

$$h = z + \psi \tag{10.25}$$

The gravitational head at a point is the elevation of the point above the datum (z is positive upwards). The reference level for the pressure head component is the atmospheric pressure. Under unsaturated conditions the pressure head, ψ, is negative due to capillary forces and short range adsorptive forces between the water molecules and the soil matrix. However, in the gravity flow module, the pressure head term is ignored and the driving force is due entirely to gravity.

Thus for vertical flow, the vertical gradient of the hydraulic head becomes,

$$\Delta h = \frac{\partial z}{\partial z} = 1 \tag{10.26}$$

The volumetric flux is then obtained from Darcy's law:

$$q = -K(\theta)\frac{\partial h}{\partial z} = -K(\theta) \tag{10.27}$$

where $K(\theta)$ is the unsaturated hydraulic conductivity. Assuming that the soil matrix is incompressible and the soil water has a constant density, the continuity equation will be:

$$\frac{\partial \theta}{\partial t} = -\frac{\partial q}{\partial z} - S(z) \tag{10.28}$$

where S is the root extraction sink term.

10.2.1 Solution method

In the Gravity Flow Module, Equation (10.28) is solved explicitly from the top of the soil column downward.

At the top of the soil column the infiltration rate is first set equal to the amount of water available for infiltration, which is the depth of overland water on the ground surface. This is reduced to the saturated conductivity of the first unsaturated soil cell, which is the maximum infiltration rate for the soil column (Equation (10.27)).

The infiltration rate is further reduced if a leakage coefficient has been specified for the overland-unsaturated zone interface, which may be done in paved areas or under lakes. A leakage coefficient must be explicitly specified for paved areas that are specified as part of the overland flow routing system. That is, paved areas may be defined as part of the overland flow module to route water to streams from parking lots, etc. However, any reduction in the leakage coefficient under such paved areas must be explicitly defined. For example, in an model cell where 25% of the land area is paved, a leakage coefficient may be specified equal to 0.25 times the hydraulic conductivity of the surficial soil.

If the water table is above the ground surface the infiltration is set to zero.

In the special case that the water table is above the top node of the soil column but below the ground surface, the infiltration rate is reduced to an estimate of the moisture deficit in the top cell. This is done to reduce or prevent artificial cycling of water between the unsaturated zone and ponded water on the surface.

If there is sufficient water in the top cell at the start of the time step (water content sufficiently above field capacity to satisfy root extraction), or if there is sufficient net infiltration to raise the moisture content above the field capacity, then the flux through the top cell is calculated based on the hydraulic conductivity, which is a function of the moisture content. The flux is first calculated based on the moisture content at the start of the time step and an updated moisture content is calculated. Then the flux is calculated again based on the updated moisture content and another moisture content is calculated. The actual flux through the cell is then set to the average of these two fluxes. Similarly, the actual updated moisture content is set to the average of the two moisture contents.

This flux is then added the cell below and the calculation repeated downwards for the remaining cells in the column.

Once the water table is reached, the water contents in the cells are rebalanced from the bottom up to ensure that no cell is over saturated.

The flux out the bottom of the soil column is accumulated over the UZ time steps and added as a source to the saturated zone calculation at the start of the next SZ time step.

10.2.2 Initial Conditions

By default, the initial conditions for ψ are generated by MIKE SHE assuming an equilibrium soil moisture/pressure profile with no-flow. The equilibrium profile is calculated assuming hydrostatic conditions, as illustrated in Figure 10.2. The pressure decreases linearly from zero at the groundwater table to ψ_{FC} when the moisture content reaches the field capacity and is then con-

stant for all nodes above this point. The assumption is that the flow is (almost) zero at moisture contents below the field capacity.

The assumption that the initial water content is not below field capacity means that ET can occur from the first time step. If the water content was based on the saturation-pressure profile all the way to the residual water content, then the model would likely start with very dry soils. There would be no recharge or ET until the soils wetted up after significant rainfall.

However, in arid and semi-arid conditions the equilibrium water content may be quite low, as the soil profile may have drained for a long time with little infiltration. In this case, it is usually better to base the initial condition on the full saturation-pressure relationship - all the way to residual water content.

In both cases, an initial warm up of the UZ is usually required. In the first case, there will be some initial drainage from the UZ as the moisture content equilibrates with the rainfall rate. In the second case, the soil profile will gain water and the groundwater recharge will be initially very low as the soil profile absorbs all of the infiltration.

10.2.3 Sources and sinks

There is a source/sink term for each computational node. These sink terms are calculated from the root extraction due to transpiration in the upper part of the unsaturated zone. The integral of the root extraction over the entire root zone depth equals the total actual evapotranspiration. Direct evaporation from the soil is not allowed in the Gravity Flow method.

10.3 Two-Layer Water Balance

The 2-Layer Water Balance Method is an alternative to the more complex unsaturated flow process coupled to the Kristensen and Jensen module for describing evapotranspiration. The 2-Layer Water Balance Method is based on a formulation presented in Yan and Smith (1994). The main purpose of the module is to calculate actual evapotranspiration and the amount of water that recharges the saturated zone.

The module is particularly useful for areas with a shallow ground water table, such as swamps or wetlands areas, where the actual evapotranspiration rate is close to the potential rate. In areas with deeper and drier unsaturated zones, the model does not realistically represent the flow dynamics in the unsaturated zone. The model only considers average conditions and does not account for the relation between unsaturated hydraulic conductivity and soil moisture content and, thereby, the ability of the soil to transport water to the roots. The model simply assumes that if sufficient water is available in the root zone, the water will be available for evapotranspiration. However, it is usually possible to “calibrate” the input parameters so that the model performs reasonably well under most conditions.

The detailed technical description is part of the ET chapter and found in the The 2-Layer Water Balance Method *(V2 p. 27)* section.

10.4 Simplified Macropore Flow (bypass flow)

Flow through macropores in unsaturated soil is important for many soil types. In the Unsaturated Zone module, a simple empirical function is used to describe this process. 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 (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.

Thus, the bypass flow, Q_{bypass}, is calculated as

$$Q_{bypass} = P_{net} \; P_{frac} \sqrt{\alpha_{10} \beta_{50}} \; / \; \Delta t \tag{10.29}$$

where P_{net} is the net rainfall rate, and P_{frac} is the maximum fraction of the net rainfall which can bypass the matrix (under wet conditions). α_{10} and β_{50} are used to reduce the total bypass fraction under dry conditions.

α_{10} and β_{50} are calculated internally by MIKE SHE and depend on the actual water contents of the unsaturated zone 10cm and 50cm below the ground surface, respectively. The relationship used to calculate α_{10} and β_{50} is illustrated in Figure 10.3. α_{10} and β_{50} vary linearly between 0.0 and 1.0 when the water content is between θ_2 and θ_1. If the water content is below θ_2, α_{10} and β_{50} equal 0.0. If the water content is above θ_1, α_{10} and β_{50} equal 1.0.

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})

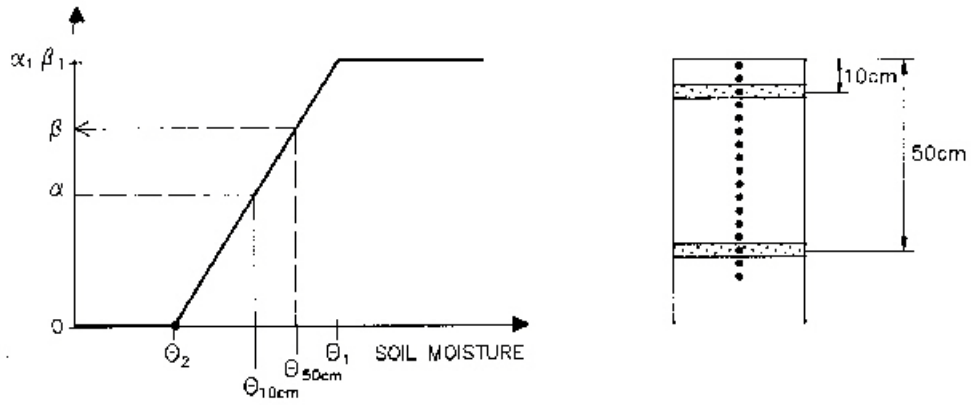


Figure 10.3 α and β as a function of the soil moisture content 10 cm and 50 cm below the ground surface, respectively.

10.4.1 Adjustment for the 2-Layer Water Balance method

In the 2-Layer Water balance method, there are only two UZ layers. Thus, the calculation of α_{10} and β_{50} is simplified somewhat, whereby θ_{10} and θ_{50} are respectively equal to

- the water content of the upper UZ layer, if they are located above the extinction depth,
- the water content of the lower UZ layer, if they are located below the extinction depth, but above the water table, and
- θ_s if they are located below the water table.

For example, if the extinction depth was at 40cm and the water table at 60cm, then θ_{10} would equal the water content of the Upper Layer and θ_{50} would equal the water content of the Lower Layer.

10.5 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, y_t ; 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, corresponding to gravitational flow. The vertical volumetric flux (positive upwards) q_{mp} is then given by

$$q_{mp} = -K_{mp}(\theta_{mp}) \tag{10.30}$$

where $K_{mp}(q_{mp})$ is the hydraulic conductivity of the macropores depending on the volumetric soil moisture content of the macropores, q_{mp}. The continuity equation is expressed as

$$\frac{\partial \theta_{mp}}{\partial t} = -\frac{\partial q_{mp}}{\partial t} - S_{mp} \tag{10.31}$$

where S_{mp} is a sink term for water exchange with the surrounding matrix. Combining (10.30) and (10.31) yields the governing equation for the macropores

$$\frac{\partial \theta_{mp}}{\partial t} = \frac{\partial}{\partial t} K_{mp}(\theta_{mp}) - S_{mp} \tag{10.32}$$

The term S_{mp} becomes a source/sink term in Richards equation used in the matrix domain. This term is given by

$$S_{mp} = \beta_{mp} \cdot K(\theta_{matrix}) \cdot (\psi_{mp} - \psi_{matrix}) \tag{10.33}$$

where y_{mp} and y_{matrix} are the capillary heads in the macropores and in the matrix, respectively, and $K(q_{matrix})$ is the hydraulic conductivity in the matrix depending on the volumetric soil moisture content of the matrix, q_{matrix}. The exchange flow from matrix to macropore is only considered when the capillary head in the matrix, y_{matrix}, exceeds the threshold pressure, y_t. β_{mp} is a first-order linear water transfer coefficient, which is expected to increase with decreasing distance between macropores and with increasing hydraulic matrix-macropore contact. It can be expressed as

$$\beta_{mp} = \frac{C_f}{d^2} \tag{10.34}$$

where d is an effective diffusion path length in metres. C_f is a dimensionless contact factor to take care of coatings on the interior walls of the macropores. Such a coating could be present due to, for example, root remnants, worm slime or mineral precipitation and can decrease the contact between matrix and macropore significantly. The contact factor ranges from 0.0 (no contact) to 1.0 (full contact).

In the macropores, a simple power law function is assumed to represent the conductivity relation

$$K_{mp}(\theta_{mp}) = K_{s,mp} \cdot \left(\frac{\theta_{mp}}{\theta_{s,mp}}\right)^n \tag{10.35}$$

where $K_{s,mp}$ is the saturated hydraulic conductivity of the macropores, $q_{s,mp}$ is the macroporosity, and n is an empirical exponent accounting for size distribution, tortuosity, and continuity of the macropores. n may vary from two to six, according to Jarvis (1994). The lower values represent soils of coarse structure with macropore networks of narrow pore size distribution and little tortuosity, whereas the higher values apply to soils with a wider macropore size distribution and larger tortuosity. If macropores are included in the simulation the hydraulic conductivity used to represent the soil matrix should exclude the effect of macropores.

The actual size, form and number of macropores are not explicitly considered in the model. Instead the macropore characteristics appear indirectly from y_t, n and β_{mp} that in the present formulation are dependent on soil type. The capillary pressure in the macropores, y_{mp}, is supposed to vary linearly with the macropore moisture content q_{mp} between zero (at $q_{mp} = q_{s,mp}$) and y_t (at $q_{mp} = 0$). Neither root water uptake nor soil evaporation are considered to take place from the macropore domain.

The infiltration process description includes water entering the macropores, as well as the soil matrix at the soil surface. In this case, water is only ponded on the ground surface when the infiltration capacities of both pore regions are exceeded. Water flow into the macropores commences as the matrix infiltration capacity is surpassed.

The bottom boundary condition for flow in the macropores is a vertical flux at a unit hydraulic gradient. This flux is input to the saturated zone. A coupling of the saturated zone and the unsaturated zone is necessary when the groundwater level fluctuates. During groundwater rise, the water present in the macropores in the bottom unsaturated zone layer is released instantaneously to the groundwater and during groundwater decline, the macropores are exposed as empty.

10.5.1 Macropore flow solution method

The numerical formulation has to take into account the fact that flows in the macropore and in the matrix domains occur with significantly different velocities. Priority in development of the numerical scheme has been put on preserving the water balance and ensuring numerical stability at time steps that are not much lower than the time steps used in solving the matrix flow equation (Richards) alone. The solution method is mass conserving. The time step length is controlled by specifying certain limits for flow and exchange flow (depths) per time step, partly for ensuring correct dynamics of the macropore flow description and partly to avoid instability of the Richards solution due to high source/sink terms. The time step is controlled by performing an extra (*a priori*) macropore computation at the start of each matrix flow time step—with the matrix conditions from the previous time step. In case the resulting maximum (*a priori*) macropore flows, infiltrations, and exchange flows exceed the specified limits, a reduced time step is estimated, assuming linear relation-

ship between time step length and flow volumes (unchanged flow rates). The procedure is repeated until the estimated maximum flows are within limits.

After the time step check, a normal time step simulation is performed, solving the Richards equation for the matrix flow—with reduced source/ sink terms from macropore-matrix exchange flows of the previous time step. After this the corresponding macropore time step is performed.

The calculation procedure consists of a double sweep algorithm with the following characteristics:

Downwards sweep (sweep 1)

First, a downwards sweep is performed, i.e. downwards flow from each cell to the cell below (or ground water table) in the macropore and in the matrix domains, and exchange flows, are calculated. Mass conservation is ensured by reducing the outflow from a cell if it exceeds the storage volume of that particular cell. The flow is not influenced by the water content of the receiving cell below, but the flow is set to 0 if the cell below has no macropores.

In the downwards sweep the downwards flow from a cell is calculated as the average of two estimates, and exchange flow as an average of four estimates:

- First, flow and exchange flow estimates are calculated as function of the macropore water content at the start of the time step. The estimates are reduced, if they exceed the start volume plus the inflow from the cell above.
- The second estimate is the flow and exchange flows as a function of the resulting macropore water content from the first estimate, including the inflow from the cell above (or the macropore infiltration if it is the upper cell). Again a reduction is made if the resulting volume would become negative.
- The final flow and exchange flow estimates are calculated as the average of the first two estimates. The flow is used as inflow to the cell below (or macropore recharge to groundwater if it is the lowest cell).

As mentioned above, each of the two macropore-matrix exchange flow estimates are calculated as two sub-estimates: The first estimate is calculated as function of the matrix water content at the start of the time step, and the second as function of the matrix water content as result of the first estimate. The same macropore water content (pressure) is used for both exchange estimates. Each sub-estimate is limited by two conditions:

- Flow from macropore to matrix is reduced if the resulting matrix water content would exceed saturation; and
- Flow from matrix to macropore is reduced if the resulting matrix pressure would be below the macropore pressure.

The macropore pressure used for exchange calculation is calculated as the macropore saturation (i.e. actual water content divided by porosity) multiplied by the cell height, and then reduced by the entry pressure. If the macropores of the cell are fully saturated, the macropore pressure of the cell above is added (hydrostatic conditions).

Upwards sweep (sweep 2)

In the second, upwards sweep, the macropore flows and the matrix-macropore exchange flows are reduced for situations where the macropores would otherwise become over-saturated. The resulting macropore water contents from downward sweep are checked, and when they exceed the macropore porosity the flow from the cell above and the exchange inflow from the matrix are reduced accordingly. If the cell receives exchange inflow from the matrix, the exchange flow and the flow from the cell above are reduced by the same proportion, otherwise only the flow from the cell above is reduced. Mass conservation is ensured by adding the flow reduction volume to the volume of the cell above (converted to water content).

The calculational time step is automatically adjusted in situations with high macropore and exchange flows so that these flows do not exceed certain limits, which from experience is known to generate numerical instability. In case of an increase of the ground water table, the water content in macropores now located below the groundwater table will be added to the groundwater recharge.

Notes

- Macropore porosity is added to the saturated water content to get the total porosity of the cell. This is important when adjusting the SZ-UZ exchange and the calculation of the relevant specific yield correction.
- The retention curve is only use for the matrix flow. It is not used for the macropore flow.

10.5.2 Verification of the Macropore Algorithm

Due to the complex flow description it was not possible to verify the code through tests against analytical solutions. Instead, several tests with different boundary conditions have indicated that the code is able to simulate the different aspects of the macropore flow events dynamically in accordance with the above theoretical basis. In addition, rigorous water balance tests have demonstrated that the continuity equation is simulated very well.

The numerical algorithm for the macropore component was previously tested by Thorsen et al. (1998). The only modification that has been made to the process descriptions since then is that the macroporosity that was described as a function of depth in the version used by Thorsen et al. (1998) now is described as a function of soil type.

An application of the macropore module is described in detail in Christiansen et al (2004).

10.6 Green and Ampt Infiltration

Infiltration is a complex process; however, numerical solution of the Richards equation is expensive. MIKE SHE includes two additional UZ models that require surface infiltration to be calculated: the 2-layer Water Balance (2LWB) method and the Gravity Flow (GF) method. In both of these methods, infiltration is calculated based on the infiltration capacity of the soil and ponding occurs if the rainfall rate exceeds the infiltration capacity. This method neglects the very important water adsorption processes that occur in dry soils, which means that the amount of infiltration is underestimated in dry soils and when the water table is relatively deep.

The Green and Ampt (GA) method is a widely used, physics based approximation to the Richards solution for calculating infiltration. As an infiltration model, the GA method can be used to calculate the infiltration in both the 2LWB and the GF UZ methods.

The GA method results in a high infiltration rate to dry soils. This is followed by ponding on the ground surface as the upper zone becomes saturated. This is followed by a slowly decreasing infiltration rate that is ultimately limited by the saturated hydraulic conductivity.

The Green and Ampt method assumes:

- an infinite thickness of soil
- no water table, capillary fringe, or lower soil boundary
- uniform soil characteristics with depth
- uniform water content with depth at t=0
- insignificant depth of ponded water

The necessary input parameters are:

- saturated hydraulic conductivity
- initial water content
- saturated water content
- rainfall rate, w
- effective soil suction ahead of wetting front

The GA method generates three key outputs:

- f(t), the infiltration rate at time t
- F(t), the total infiltration at time t

- the time until ponding after the start of the time step

The GA method distinguishes between two main cases:

Case 1: Rainfall rate less than the saturated hydraulic conductivity

In this case, no ponding occurs and the infiltration rate equals the rainfall rate and the total infiltration equals the rainfall rate times the time step length. Thus,

$f(t) = w$

$F(t) = w\ t$

Case 2: Rainfall rate greater than saturated hydraulic conductivity.

Initially, rainfall will be absorbed into the soil and ponding will occur when the top of the unsaturated zone becomes saturated. However, the initial infiltration rate will be greater than the saturated hydraulic conductivity because the soil will adsorb water ahead of the wetting front. Thus, ponding will occur after the start of the timestep. Prior to the start of ponding Case 1 applies. After ponding starts, the infiltration rate starts to decrease, and the infiltration time is a function of the total amount of infiltration. This must be solved implicitly to find the amount of infiltration corresponding to the time step.

Since the GA method assumes that the infiltration occurs as a sharp front plug flow, the depth of infiltration can be easily calculated by dividing the original deficit by the amount of infiltration.

Actually, there are a couple of special cases in addition to the two main cases.

Case 3: Initial ponding

The GA method assumes that there is no ponding on the ground surface. In fact, the infiltration rate depends on the degree of ponding but the ponding level is generally neglected. This is a reasonable assumption since the ponding due to rainfall is generally negligible. However, in MIKE SHE there is often ponding at the beginning of the time step due to flooding. In this case, the we can assume that the top of the soil is saturated and the infiltration rate will equal Ks.

Case 4: Rainfall + Ponded storage depleted during the time step

In some cases, there may be a small amount of initial ponded storage in the cell. This should be infiltrated as per Case 3, but once this has been depleted, ponded storage infiltration will cease if the rainfall rate is less than Ks. If the rainfall rate is greater than this, then water will continue to pond and infiltrate at the rate equal to Ks. However, if the rainfall rate is less than Ks, then the infiltration rate should continue at the rainfall rate.

Case 5: Insufficient UZ storage

The total amount of infiltration cannot exceed the amount of available UZ storage, so there must be a check to see that this amount is not exceeded. Excess must be added to OL.

10.6.1 Implementation of Green and Ampt in the 2LWB method

The implementation in MIKE SHE's 2LWB method is relatively straightforward. Since the GA method is only an infiltration model, the current infiltration method for the 2LWB method can nearly be substituted. In the current method, the amount of infiltration in the time step is the same as Case 1 with a maximum equal to Ks.

In the 2LWB method, there is an upper root zone layer (UL) and sometimes a lower layer down to the water table (LL). The calculation algorithm proceeds as follows:

1. Calculate UZdeficit based on deficit in UL and LL and the normally calculated recharge to SZ
2. If Case 1,

   a) Calculate time to ponding

   b) if time to ponding is greater than the time step, infiltrate all the rainfall in the time step

   c) If time to ponding is less than the time step, infiltrate only the allowed amount.

   d) Recalculate the water contents in the UL and LL, assuming

      UL is 100% saturated before the LL receives water, and

      UL and LL are 100% saturated if there is rainfall in excess of UZ storage.

3. If Case 2,

   a) Calculate time to ponding based on the Green and Ampt equation

   b) if time to ponding is greater than the time step, infiltrate the maximum rainfall using the Green and Ampt equation

   c) if time to ponding is less than the time step, infiltrate only the allowed amount.

   d) recalculate the water contents in the UL and LL, assuming

      UL is 100% saturated before the LL receives water, and

UL and LL are 100% saturated if there is rainfall in excess of UZ storage.

4. Add the excess rainfall to OL

Internal infiltration rates

The rate of exchange between the UL and LL, and between the LL and SZ is currently governed by Ks. This should be a function of the water content, defined by the water content of the draining layer. This will significantly reduce the rate of drainage in the layers - especially in the LL if it is thick.

Bypass Flow

The handling of bypass flow directly to SZ is not be affected by the GA implementation.

Overland-groundwater leakage coefficient

If the a leakage coefficient is specified that is lower than the specified Ks then the lower value should be used.

Extra Parameter: Threshold depth for infiltration

This extra parameter is used to prevent infiltration of very small amounts of rain. Instead the water remains on the ground surface and usually evaporates. This parameter is needed because the 2LWB does not remove ET from the soil surface. This parameter is unaffected by the GA method.

10.6.2 Implementation in the Gravity Flow method

Adding the GA method to the Gravity Flow model is slightly more complicated because the GA method assumes a uniform, infinite soil column.

Many GF models include a fine discretization near the ground surface to capture the infiltration dynamics. However, the GA method is an approximation to these infiltration dynamics and makes the refined grid unnecessary. Using multiple layers within the GA's depth of infiltration would often result in variations of water content with depth, which makes the infiltration calculations difficult.

Multiple UZ layers are handled by reducing the amount of infiltration depending on the actual storage available in the depth of infiltration. In other words, the amount of infiltration and depth based on the average water deficit in the GF layers in the root zone is calculated and then MIKE SHE calculates if the infiltration exceeds the actual available storage in the depth of infiltration. If it does then MIKE SHE only infiltrates the available storage. This is a reasonable physical approximation for the total amount of infiltration, but might underestimate the rate of infiltration if the upper layers are very dry.

The GF module allows you to have multiple soil types in the same column. Multilayer GA models are technically possible, but this is not implemented in MIKE SHE. If there are multiple soil types within the infiltration depth, a Warn-

ing is issued. The infiltration is only calculated based on the soil type in the uppermost UZ layer.

The GA infiltration with the GF method proceeds as follows

1. Calculate UZdeficit based on entire UZ deficit and an estimate of the current recharge to SZ
2. If Case 1,

   a) Calculate time to ponding

   b) if time to ponding is greater than the time step, infiltrate all the rainfall in the time step

   c) If time to ponding is less than the time step, infiltrate only the allowed amount.

   d) Recalculate the water contents in the layers, assuming, each layer is successively 100% saturated before the layer beneath receives water.

3. If Case 2,

   a) Calculate available storage in the

   b) If time to ponding is greater than the time step,

      i) Calculate the maximum rainfall

      ii) Calculate the maximum depth, and then calculate the maximum actual storage in the layers down to the max depth

      iii) Infiltrate either (i) or (ii)

      iv) Update all water contents assuming that each successive layer is 100% saturated.

   c) If time to ponding is less than the time step,

      i) infiltrate only the allowed amount

      ii) Update all water contents assuming that each successive layer is 100% saturated

      iii) Then add the excess rainfall to OL

Internal infiltration rates

The rate of exchange between the layers is currently governed by Ks ( ). This is unchanged, except that during the GA infiltration, it is ignored.

Bypass Flow

The handling of bypass flow directly to SZ is not affected by the GA implementation.

Overland-groundwater leakage coefficient

If a leakage coefficient is specified that is lower than the specified Ks then the lower value is used.

10.7 Coupling the Unsaturated Zone to the Saturated Zone

Briefly, the interaction between the unsaturated and saturated zones is solved by an iterative mass balance procedure, where the lower part of the unsaturated node system may be solved separately in a pseudo time step, between two real time steps. This coupling procedure ensures a realistic description of the water table fluctuations in situations with shallow soils. Particularly in these cases it is important to account for a variable specific yield above the water table, as the specific yield depends on the actual soil moisture profile and availability of that water.

The recharge to the groundwater is determined by the actual moisture distribution in the unsaturated 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, and 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., run in parallel) and not solved in a single matrix with an implicit flux coupling of the UZ and SZ differential equations. Explicit coupling of the UZ and SZ modules is used in MIKE SHE to optimize the time steps used and allows use of time steps that are representative of the UZ (minutes to hours) and the SZ (hours to days) regimes. MIKE SHE overcomes problems associated with the explicit coupling of the UZ and SZ modules by employing an iterative procedure that conserves mass for the entire column by considering outflows and source/sink terms in the saturated zone.

Error in the mass balance originates from two sources; 1) keeping the water table constant during a UZ time step and 2) using an incorrect estimate of the specific yield, S_y (the difference between the moisture content at saturation, θ_s, and moisture content at field capacity, θ_{fc}) in the SZ-calculations. This is illustrated in Figure 10.4a. If outflow from the SZ is neglected, it appears from the figure that during the time n to $n+m$, the column has lost V_1 mm of water (the light grey shaded area) and gained V_2 mm (the dark shaded area).

The changes calculated by the UZ module for the areas V_1 and V_2 represent a redistribution of water in the unsaturated zone to obtain an equilibrium moisture profile within the soil column. Comparing the equilibrium moisture content and the moisture content at UZ time n in Figure 10.4a shows that the

moisture content is too high in the upper portions of the soil column. This should result in downward flow in the unsaturated zone, loss of soil moisture in area V_1, increased soil moisture in area V_2, and a rise in the water table. However, the SZ module uses a constant specific yield (S_y) defined for each grid cell in each calculation layer. On the other hand, the UZ can have a unique S_y value for each UZ node, which may differ from the S_y value used by the SZ. Thus, mass balance errors can occur in exchange calculations between the two modules. A mass-conservative solution is achieved by using a step-wise adjustment of the water table and recalculation of the UZ solution until the area of V1 and V2 are equal (see Figure 10.4b).

The procedure to deal with this mass balance error consists of a bookkeeping of the accumulated mass balance error, E_{cum}, for each UZ column and the upper SZ calculation cell associated with the column on a cell-by-cell basis. If $|E_{cum}|$ exceeds a user specified value, E_{max}, the UZ coupling correction procedure corrects the water table of the upper SZ cell (i.e., the lower boundary condition for the UZ).

The correction procedure of step-wise water table adjustments and additional UZ calculations is repeated until $|E_{cum}|$ is less than E_{max} for each column that has failed the E_{max} criteria. During this correction procedure, the UZ module of MIKE SHE operates on a copy of the water table solution for the upper SZ calculation layer. After each SZ time step the UZ copy is updated with the new water table from the SZ solution and then adjusted during the succeeding UZ time step(s) until the next SZ time step. The calculated adjustments are converted to an additional flux term (multiplied with the specific yield of SZ and divided by the SZ time step length) and added to the uncorrected UZ-to-SZ flux term. The corrected UZ-to-SZ flux term is used by the SZ module as an explicit source/sink term during the next SZ time step.

The size of E_{max} determines the largest allowable mass balance before adjustments are made. Typically, an E_{max} value between 1-2 mm is an appropriate choice for regional MIKE SHE simulations. The E_{max} value is specified in the UZ Computational Control Parameters *(V1 p. 188)* dialogue.

10.7.1 Steps in the Coupling Procedure

The following outlines the actual steps in the coupling procedure used for each UZ time step:

1. If the total water content above the datum *Zo* (*Zo* should always be lower than the lowest elevation of the water table) is designated W^n, the UZ contribution to the E_{cum} error term in UZ-time step *n* to *n+1* is

$$q_u^{n+1} = \frac{(W^{n+1} - W^n)}{\Delta t^{n+1}} + q_I + q_E \qquad (10.36)$$

where W^{n+1} = the new water content; q_I = the infiltration rate (negative downwards); q_E = the evapotranspiration loss. (Note: negative values of *qu* indicate downward flow).

2. Assuming that the groundwater outflow in a cell is steady, the accumulated error at UZ time *n+1* is:

$$E_{cum}^{n+1} = \left(q_u^{n+1} + \frac{q_G^{n_G} \Delta t_{n_G}}{\Delta t_{n_G+1}} + q_s^{n+1} \right) \Delta t + E_{cum}^{n} \tag{10.37}$$

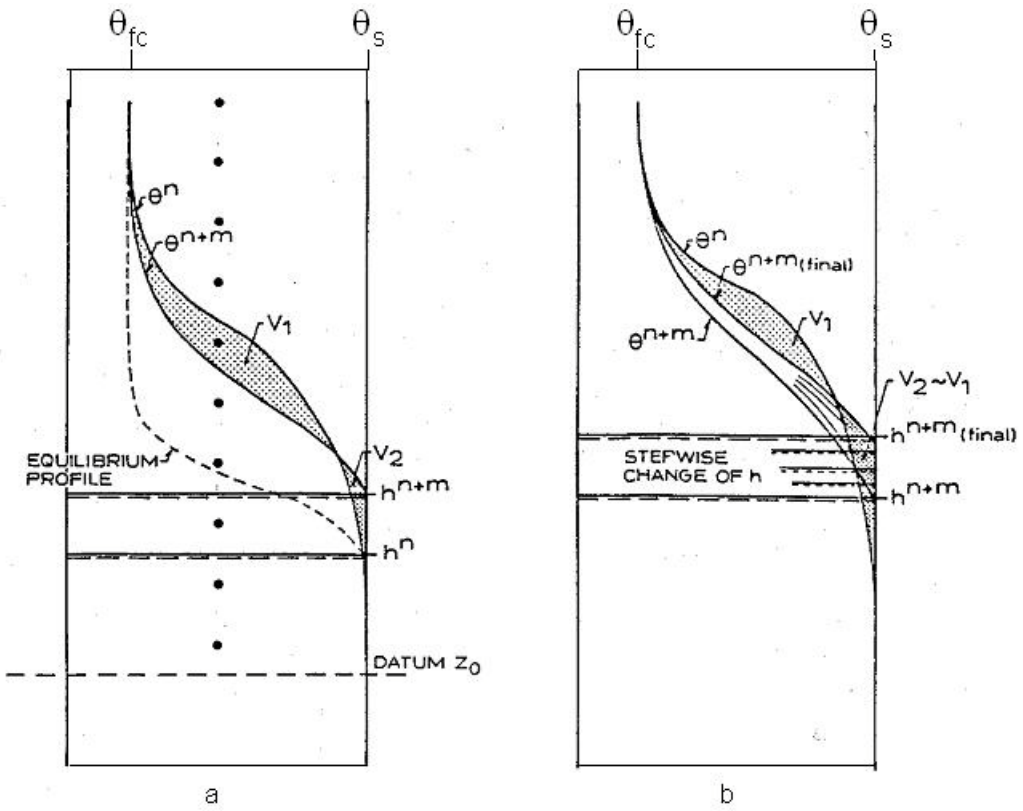


Figure 10.4 a) Soil moisture content at two times n and n+m without corrections, and b) Soil moisture content at time n+m before and after correction.

where $q_G^{n_G}$ (positive outwards) is the sum of the groundwater outflow rate for the cell in the last groundwater time step (n_G) scaled to the new SZ time step length (n_G+*1*) and q_s^{n+1} (positive outwards) is the sum of source sink terms calculated by the UZ module for the current time *n+1* (e.g., stream/aquifer exchanges, irrigation).

It should be noted that if E_{cum} is less than zero there is a deficit of water stored in the column and if E_{cum} is greater than zero there is an excess of water stored in the column.

3. If $\left|E_{cum}^{n+1}\right|$ less than E_{max} corrections are not made for the current UZ time step.

4. If $\left|E_{cum}^{n+1}\right|$ exceeds E_{max} the following corrections are made:

   a) If $\left|E_{cum}^{n+1}\right|$ is negative or positive, the water table is raised or lowered, respectively, in prescribed increments that depend on the distance between UZ nodes and the UZ-calculation in time step *n* to *n+1* is repeated as described above.

   b) In the Full Richards solution the UZ flow solution is repeated for the last three nodes above the water table to reduce numerical overhead. In the Gravity Flow option, the UZ flow solution is repeated for the entire column. The UZ flow solution is not repeated for the two-layer UZ option.

   c) The change in water volume W^{n+1*} over the entire column is computed and a new $\left|E_{cum}^{n+1}\right|$ is calculated.

   d) If $\left|E_{cum}^{n+1}\right|$ is less than $a{\cdot}E_{max}$, where a is a hard-coded correction factor equal to 0.9, the error associated with the solution is considered acceptable and the procedure stops. If $\left|E_{cum}^{n+1}\right|$ is greater than or equal to $a{\cdot}E_{max}$, the solution is unacceptable and steps a) through d) are repeated until criteria d) is satisfied. The value a defines a threshold for stopping the procedure lower than that used to initiate the procedure, which prevents correction overshoots.

   e) If E_{cum}^{n+1} changes sign the solution is considered acceptable and the procedure stops. The adjustment required to obtain a $\left|E_{cum}^{n+1}\right|$ value of zero is calculated using a secant line approach.

5. A new recharge rate, q_u^{n+1*} is calculated taking the adjustments into account.

$$q_u^{n+1*} = q_u^{n+1} - \frac{(h^{n*} - h^n)}{\Delta t} S_y \tag{10.38}$$

where h^{n*} is the new water table elevation after step d) calculated by the UZ module and Δt is the length of the current UZ time step (*n+1*). If SZ outflows for the next SZ time step (n_G+*1*) are unchanged, the water table from the SZ calculation will be $h^{n+1} = h^{n*}$ calculated in the last UZ time step before an SZ time step (see Eq. (10.38)).

If $(h^{n*} - h^n)S_y/\Delta t > q_{max}$, where q_{max}, is a maximum infiltration rate, the corrected rate is reduced to q_{max}. In the Richards Equation and Gravity Flow options q_{max} is $0.7K_s$ and $0.4K_s$ for rising and falling water

table conditions, respectively, where K_S is the saturated hydraulic conductivity of the UZ node at the water table. In the Two-layer UZ option, the infiltration rate is used to constrain the corrected rate.

Steps 1-5 are repeated for all UZ time steps within each SZ time step. The flows are accumulated and passed as an average rate, q_u, for the next SZ time step. The average q_u is used as a flux boundary condition in the SZ differential equations.

11 Working with Unsaturated Flow - User Guide

Unsaturated flow is one of the central processes in MIKE SHE. The unsaturated zone is responsible for the partitioning of rainfall into runoff, evapotranspiration and groundwater recharge.

Unsaturated flow is primarily vertical since gravity plays the major role during infiltration. Therefore, unsaturated flow in MIKE SHE is calculated only vertically, which is sufficient for most applications. However, this may limit the model's validity in some situations, such as on steep hill slopes with contrasting soil properties in the soil profile, or in small scale models with significant slopes (e.g. earth dams).

11.1 Physics of Unsaturated Zone Flow

The Unsaturated Zone (UZ) is generally considered the area between the land surface and the groundwater water table. The UZ is characterized by cyclic fluctuations in soil moisture as water is replenished by rainfall and removed by evapotranspiration and percolates to the groundwater table as recharge.

The flow of water in the UZ depends on the interplay between three processes:

- Infiltration from the land surface into the soil,
- Evapotranspiration (ET) from the root zone, and
- Flow in the soil pores and deep percolation to the groundwater table.

The entire amount of rainfall will infiltrate into the soil surface if the rate of rainfall is less than the maximum infiltration rate. Generally, the maximum rate of infiltration is when the soil is saturated.

The UZ is coincident with the root zone, which is normally always above the water table. Most plants can tolerate short periods of fully saturated conditions in their roots, but if the roots are saturated for longer periods the roots will typically die off. Thus, nearly all plant ET occurs in the UZ.

The only extraction of water in the unsaturated zone is ET, the water content in the soil below the root zone is in equilibrium with the rate of drainage. Thus, any infiltration that exceeds the capacity of the roots to extract it, will percolate past the root zone and eventually become groundwater recharge.

The UZ is generally differentiated from the Saturated Zone by the presence of air in the soil pores. Thus, the key characteristic of the UZ is the amount of water in the soil pores, or the water content. From this we can define several characteristic water contents:

- **Saturated water content,** θ_s, is equal to the porosity. This is the maximum amount of water that can be present in the soil.
- **Field capacity**, θ_{fc}, is amount of water in the soil after it has freely drained and when there is no evapotranspiration. Essentially, the water content is in equilibrium. It is an equilibrium between gravity drainage and the ability of the soil to hold on to moisture.
- **Wilting point**, θ_{wp}, is the minimum water content below which plants can no longer extract water from the soil using osmotic pressure. This is defined as a soil suction of -1500 KPa (-15 bar, pF 4.18).
- **Residual water content**, θ_r, is the effective minimum water content for a 'dry' soil.

Interestingly, Field Capacity is the language of the agronomist or soil scientist, where it is the related to the plant available water. However, the related Specific Yield is a groundwater term. As defined by a hydrogeologist, Specify Yield is the amount of water that can be abstracted from an unconfined groundwater aquifer as the water table declines due to pumping. In other words, the specific yield is the saturated water content minus the field capacity. This has been recognized for nearly 100 years evidenced by Figure 11.1, which is from a 1963 USGS report, which itself references the figure from a publication from 1934.

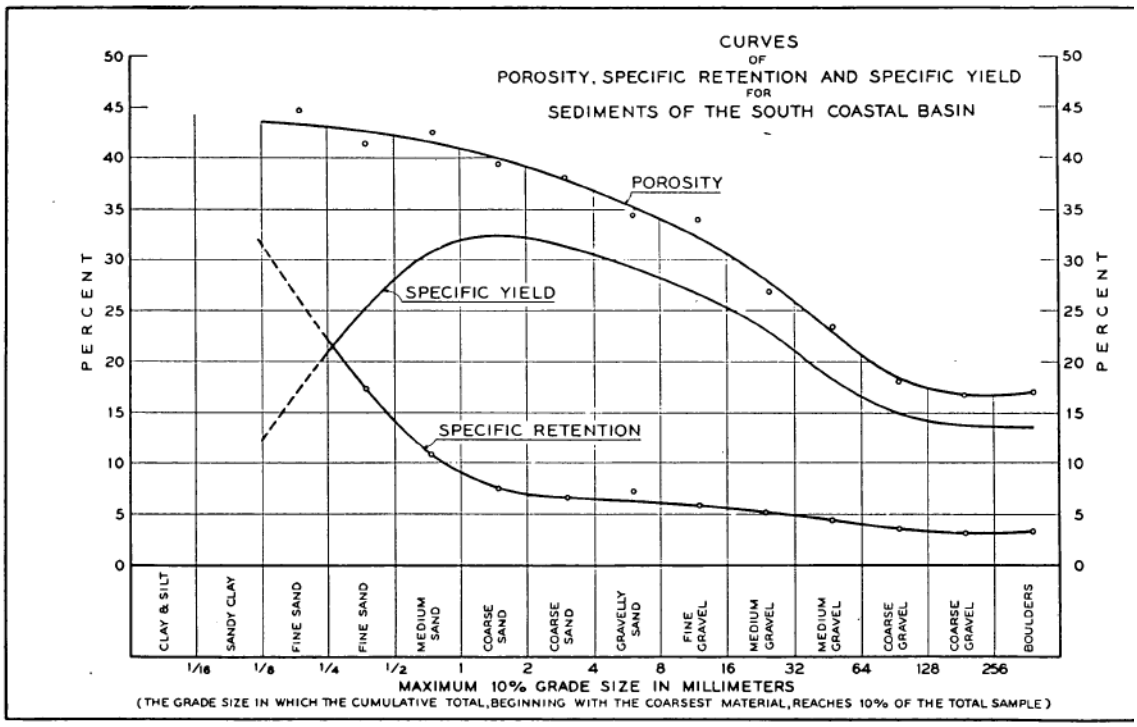


Figure 11.1 Relation between Specific Yield, Porosity and Field Capacity (from USGS OFR 63-59, Figure 3)

Note: in MIKE SHE the Specific Yield of the uppermost SZ layer is set equal to θ_s - θ_{fc}.at the beginning of the simulation. However, this is only done at the begin

11.1.1 The saturation-pressure function

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.

See Soil Moisture Retention Curve *(V1 p. 381)*.

11.1.2 Hydraulic conductivity as a function of saturation

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.

See Hydraulic Conductivity Function *(V1 p. 384)*.

11.1.3 Typical soil parameters

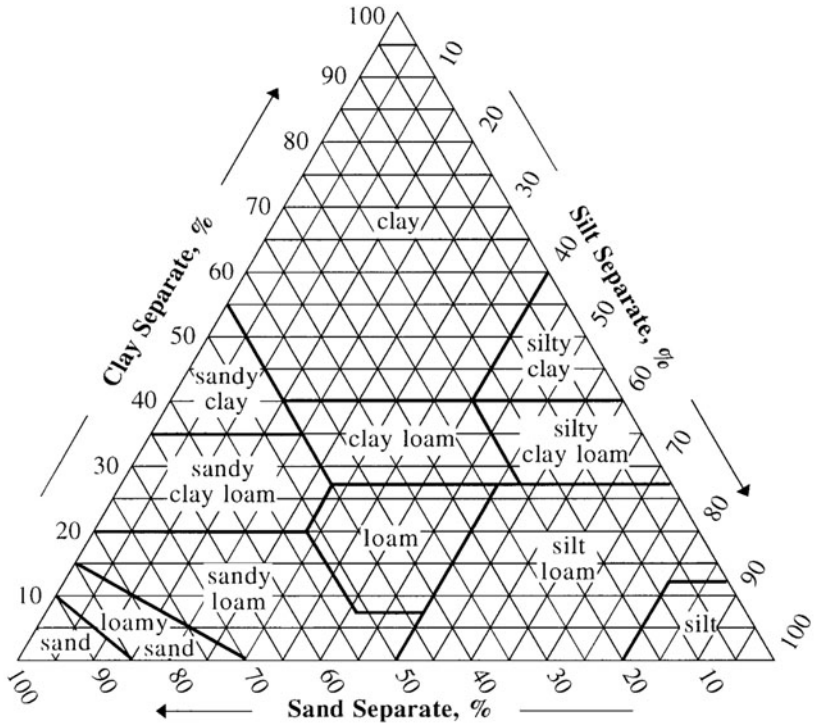


Figure 11.2 USDA Soil texture classification by grain size distribution

11.1.4 UZ Soils Editor

The UZ Soils Editor is used to define the saturation-pressure and the saturation-conductivity soil functions.

See UZ Soil Properties Editor *(V1 p. 379)*.

Note: the units of pressure in the UZ Soils Editor are pF. This is defined as the base 10 logarithm of the suction expressed in cm of water. To convert between MPa and pF, first convert MPa to cm of water. The conversion factor is 10200 cm/MPa. Drop the negative sign because you cannot take the logarithm of a negative number. Finally, take the base 10 logarithm to get pF. For example, -1 MPa equals pF 4.01.

11.2 Overview of the UZ User Interface

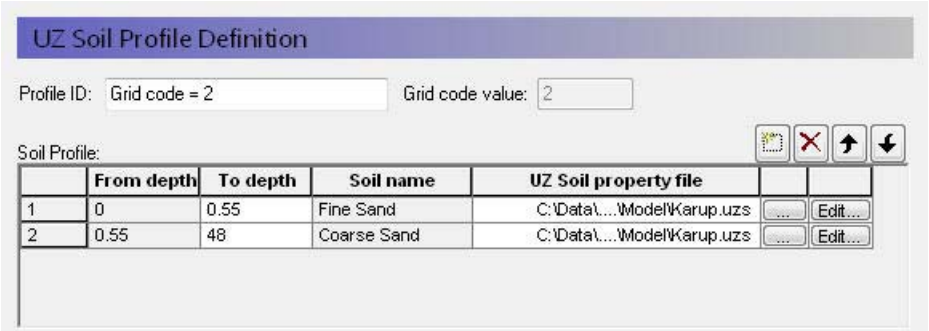
The setup of the UZ model can be divided into two steps: The definition of the soil profile and the definition of the vertical numerical grid. These two steps

are separate in the Gravity Flow and Richards Equation methods. In the Two-Layer UZ method the vertical discretisation is pre-defined (a root zone and area below the roots), so only the soil properties need to be defined.

11.2.1 Definition of the soil profile

Richards Equation and Gravity Flow

In the Gravity flow and Richards Equation methods, the soil profile section in the 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 refer 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. See UZ Soil Properties Editor *(V1 p. 379)*.

Note: The depth of the soil profile does not have any influence on the calculation. The only constraint is that the soil profile must be deeper than the numerical grid.

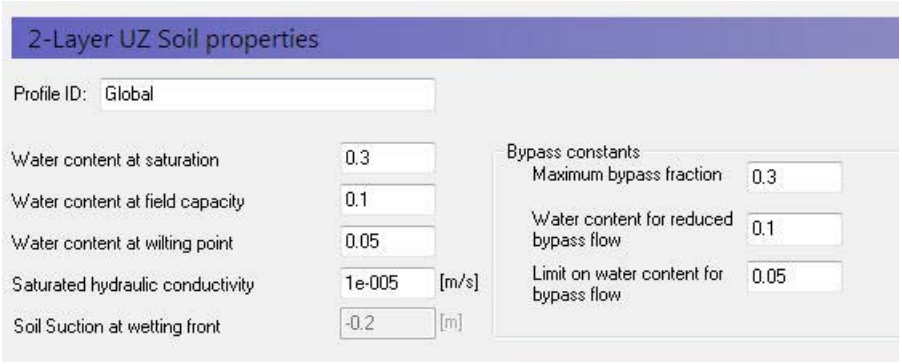
If you specify multiple soils in a soil profile, then the infiltration through the column will depend on the effective hydraulic conductivity function associated with water content, $K(\theta)$. This means that zones of high saturation may build up within the soil column. Depending on the situation, this may or may not be realistic. The UZ columns do not communicate laterally with one-another, so there is no way for such perched conditions to redistributed laterally to neighbouring cells.

In the case of Richards Equation, heterogeneous profiles can also lead to capillary barriers. This occurs when a fine grained soil is underlain by a

coarser soil. In this case, capillarity will retain the water in the fine grained soil because the gravity gradient is less than the capillary pressure holding the moisture in the fine grained soil. This does not occur in the Gravity Flow method, because capillarity is ignored.

Two-Layer UZ method

In the Two-Layer UZ method, only the soil properties need to be defined.



The soil properties are not defined from the Soils Editor, rather they must be input directly in the dialogue.

11.2.2 Definition of vertical UZ numerical grid

The Gravity and Richards Equation methods assume that the soil profile is divided into discrete computational nodes. The non-linearity of the unsaturated flow process creates large gradients in soil pressure and soil moisture content during infiltration. Therefore, it is important to select appropriate nodal increments, so as to describe the flow process with sufficient accuracy but at the same time keeping the computational time reasonable. This trade off can become a key constraint in catchment-scale simulations.

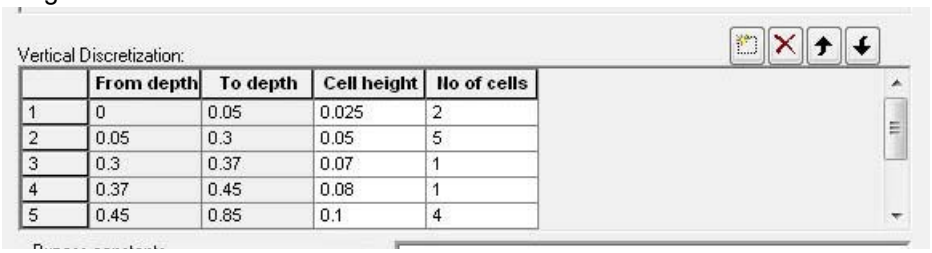
Note: The UZ model only connects to the top layer of the SZ model. All nodes below the water table are ignored. All nodes below the bottom of SZ layer 1 are also ignored. However, results will be output at all of these nodes, which can lead to unnecessarily large output files. There is also a small memory overhead associated with the extra nodes, but the extra nodes cause very little computational overhead

The simulation of Hortonian ponding at the ground surface (high rainfall intensity on dry, low permeable soil) requires a fine spatial resolution in the upper part of the profile (see Figure 11.3). Deeper in the profile the gradients are smaller and larger node increments can usually be selected. However, in the case of coarser grained soils, there may again be very high saturation gradients near the water table. Thus, as a general guideline, one should choose a finer spatial resolution in the top nodes.

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.

The vertical discretization is defined in the lower half of the soil profile dialogue.



	From depth	To depth	Cell height	No of cells
1	0	0.05	0.025	2
2	0.05	0.3	0.05	5
3	0.3	0.37	0.07	1
4	0.37	0.45	0.08	1
5	0.45	0.85	0.1	4

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.

Note: the soil properties are averaged if the cell boundaries and the soil boundaries do not align.

Note: The boundary between two blocks with different cell heights, the two adjacent boundary cells are automatically adjusted to give a smoother change in cell heights.

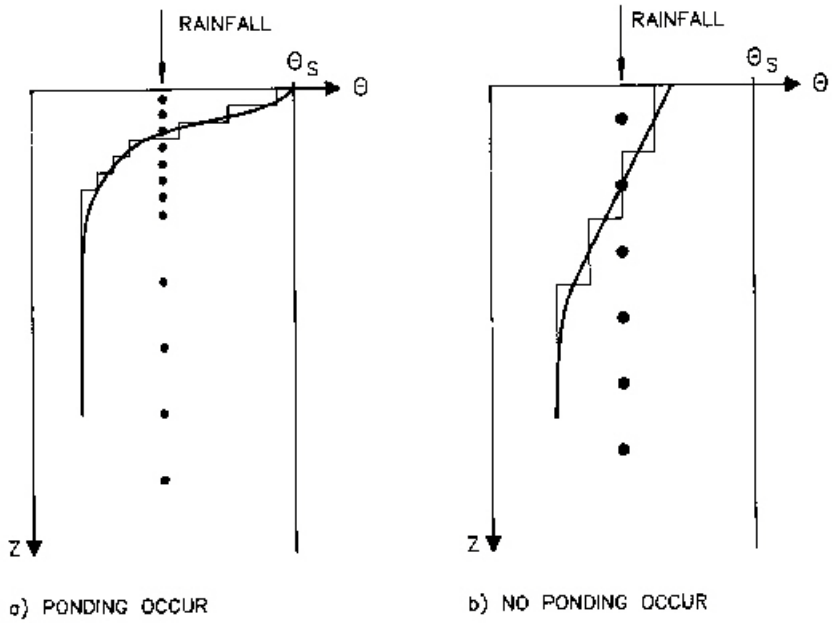


Figure 11.3 Examples of different increments in the soil profile and the resulting water content distribution of 1-3 cm for detailed studies and 3-5 cm for catchment studies. Further down in the profile larger increments can be chosen ranging from 10 cm to 30 cm.

11.2.3 Results

The UZ output contains all the information on pressure and saturation in the UZ columns, as well as information on the spatial processes, such as spatially distributed infiltration and groundwater recharge.

Gridded output items

The gridded output for the UZ solver are found in four files:

- The output related to the 2 Layer Water balance method is found in the file projectname_wetland.dfs2. (The name reflects the earlier name of this module - the former wetland module).
- The 2D results from the UZ model are found in two files: projectname_2DUZ_AllCells.dfs2 and projectname_2DUZ_UzCells.dfs2. The 2D output includes output items related to spatial distribution of UZ processes, such as UZ-SZ exchange and groundwater recharge.
- The 3D UZ outputs are found in projectname_3DUZ.dfs3. This file contains all the values in all the cells, such as the saturation in each UZ cell.

The detailed contents of these files are found in the Appendix: MIKE SHE Output Items *(V1 p. 397)*.

Transient UZ column plot

One of the most interesting UZ outputs is the transient UZ column plot. The UZ in MIKE SHE is based on 1D columns. This plot displays the property (water content, pressure, etc) of one column with depth versus time. This allows you to visualize, for example, the water content with depth at every saved time step in the simulation.

Evaluating the spatial distribution of UZ errors

The water balance program can be used to get an overview of errors due to a poor setup of the unsaturated zone. The follow procedure can be used to make a map of UZ-errors:

1. Create a sub catchment map by retrieving UZ-classification codes from the input file.
2. Replace negative values of the classification code map by positive values in the 2D graphical editor.
3. Use the sub catchment map in the water balance setup file to make a UZ map of the water balance, which will create your map of UZ-errors.

11.3 Overview of Numerical Methods

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

Richards Equation

The full Richards equation requires a tabular or functional relationship for both the moisture-retention curve and the effective conductivity.

The full Richards equation is the most computationally intensive, but also the most accurate when the unsaturated flow is dynamic.

Gravity Flow

The simplified gravity flow procedure assumes a uniform vertical gradient and ignores capillary forces.

The simplified gravity flow procedure provides a suitable solution when you are primarily interested in the time varying recharge to the groundwater table based on actual precipitation and evapotranspiration and not the dynamics in the unsaturated zone.

Two Layer Water Balance

The simple two-layer water balance method divides the unsaturated zone into two zones: the root zone and the zone between the roots and the water table.

The simple two-layer water balance method is suitable when the water table is shallow and groundwater recharge is primarily influenced by evapotranspiration in the root zone.

11.3.1 UZ-SZ coupling

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.

Ensuring the correct UZ thickness

The following procedure could be used to ensure that the unsaturated zone does not drop below the bottom of the first calculation layer of the saturated zone:

1. After a simulation, create a map of grid statistics of the potential head in the first calculation layer of the saturated zone
2. Subtract the map of the minimum potential head from the map of the bottom level of the first calculation layer of the saturated zone.
3. View the difference map. If the difference is very small in some areas of the map (e.g. <0.5 m), it is strongly advised to move the bottom level of the first calculation layer of the saturated zone downwards.
4. Repeat this procedure until there are no small differences.

Evaluation of the UZ-SZ Coupling

The WM_Print log file generated by MIKE SHE should be reviewed after each simulation to evaluate the performance of the UZ module. If the user specified maximum UZ iterations is exceeded an excessive number of times and there are no problems with the soil data used in the UZ module, the UZ and SZ time step should be evaluated. Sometimes it is possible to reduce the number of times the maximum UZ iterations is exceeded by making the UZ and SZ time steps more similar. Typically the SZ to UZ time step ratio should be no larger than four.

It is also useful to save the value of E_{cum} as a grid series output, or as a detailed time series output at critical locations. These plots can be used to determine if there are locations or periods of time during the simulation where the E_{cum} term exceeds E_{max}. This can occur if

- the water table drops below the first SZ calculation layer (positive value),
- the water table rises above the top of the first SZ calculation layer (negative value),
- the vertical hydraulic conductivity in the upper SZ calculation layer is much greater than the saturated hydraulic conductivity used in the UZ, or if
- the drainage time constant is too high.

In the first two cases above, the *epsilon* term can exceed E_{max} because the UZ module cannot get rid of *epsilon* because there is no available storage for the error term. In the third case, the UZ and SZ hydraulic properties should be consistent or it will be difficult for MIKE SHE to simulate consistent vertical flow rates. In the last case, the drainage time constant should be reduced to prevent excessive and unrealistic drainage outflows from the SZ module.

Limitations of the UZ - SZ coupling

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

- As a rule of thumb, the UZ soil profiles should extend to just below the bottom of the top SZ layer.
- However, if you have a very thick top SZ layer, then the UZ profiles must extend at least to below the deepest depth of the water table.
- If the top layer of the SZ model dries out, then the UZ model usually assumes a lower pressure head boundary equal to the bottom of the uppermost SZ layer.
- However, if the top layer of the SZ model dries out, and you are using the Richards Equation method, then you should ensure that there is one UZ **node** below the bottom of the top SZ layer. Otherwise, an error may be generated if there is an upwards capillary pressure gradient.
- 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.

More detail on interaction between the lower UZ boundary and the SZ Layer 1 is given in the Section: Lower Boundary *(V2 p. 149)*.

11.4 Options for the UZ model

11.4.1 Bypass flow

In the Bypass flow option, 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)*.

Bypass Constants

The available bypass parameters include:

Maximum bypass fraction - This is the maximum fraction of net rainfall that will infiltrate via bypass 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)*.

11.4.2 Full macropore flow

In the Full Macropore option, 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.

11.4.3 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.

11.4.4 Lumped UZ Calculations (Column Classification)

Note: 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 allows you to optionally 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. The unsaturated flow conditions in two cells are identical when they have

- identical soil- and vegetation characteristics, AND
- identical boundary conditions.

If these two conditions are met, then the calculations need only be made in one of the cells and the results transferred to the other cell.

In practical terms, the first condition is usually not a serious restriction since most models are divided into several homogeneous soil zones. The second condition, however, is much more restrictive. Fluctuations in the groundwater table usually vary from cell to cell, and spatial variations in rainfall and the topography cause overland flow and infiltration to vary continuously across the domain.

However, if homogeneous zones can be defined based on

- Topography,
- Meteorology,
- Vegetation,
- Soil, and
- Bypass characteristics,

then a representative cell for the zone can be defined and used for the UZ-calculations. If this is done, then the boundary conditions from the representative cell (i.e. infiltration rate, evapotranspiration loss and groundwater recharge) can be transferred to the other cells within the zone.

Such an approximation does not introduce any water balance errors, but it can influence the dynamics of the simulation. However, an intelligent grouping of the cells can reduce computational burden considerably.

The initial definition of homogeneous zones can be made using the depth to the groundwater table and the soil, vegetation, and rainfall distributions. It is often necessary to re-group the columns several times during the calibration

phase, until the groundwater regime is reasonable calibrated. Also, when the groundwater table is shallow, smaller intervals are usually required.

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.

  One tip is to extract a map of the calculated potential head in the very upper saturated zone layer from a previous simulation. The map should represent the time of the year when the largest variations of the groundwater table are expected (deep groundwater in the hills and shallow groundwater close to the rivers). Repeat the procedure as calibration improves.

  If the Linear Reservoir method is used for the groundwater, then the Interflow reservoirs are also used in the classification. However, since feedback to the UZ only occurs in the lowest Interflow reservoir of each subcatchment, the Interflow reservoirs are added to the Automatic Classification in two zones - those that receive feedback and those that don't.

- **Specified classification**

  Alternatively a data file specifying Integer Grid Codes, where UZ computations are carried out can be specified, with grid codes range from 2 up to the number of UZ columns (see Specified classification). The location

of the computational column is specified by a negative code and the simulation results are then transferred to all grids with the an equivalent positive code. For example, if a grid code holds the value -2 a UZ computation will be carried out for the profile located in that grid. Simulation results will subsequently be transferred to all grid codes with code value 2. An easy way to generate a .dfs2 file to be used for specification of UZ computational columns is to let the MIKE SHE setup program generate an automatic classification first, and subsequently extract the UZ classification grid codes. The extracted .dfs2 file can be edited in the 2D editor as desired and used to specify UZ computational grids.

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

  For most applications you should specify that computations are to be carried out in all soil columns.

- **Partial Automatic**

  Finally a combination of the Automatic classification and the Specified classification is available. If this option is chosen an Integer Grid Code file must be provide (see Partial automatic classification) with the following grid codes: In grid points where automatic classification should be used the grid code 1 must be given. In grid points where computation should be performed for all cells the grid code 2 must be given.

12 Saturated Flow - Technical Reference

The Saturated Zone (SZ) component of MIKE SHE WM calculates the saturated subsurface flow in the catchment. 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 gives the opportunity to chose between two groundwater solvers - the SOR groundwater solver based on a successive over-relaxation solution technique and the PCG groundwater solver based on a preconditioned conjugate gradient solution technique. The formulation of potential flow and sink/source terms differs between the two modules to some extent.

The Saturated Zone Component interacts with the other components of MIKE SHE WM mainly by using the boundary flows from other components implicitly or explicitly as sources and sinks.

12.1 3D Finite Difference Method

The governing flow equation for three-dimensional saturated flow in saturated porous media is:

$$\frac{\partial}{\partial x}\left(K_{xx}\frac{\partial h}{\partial x}\right)+\frac{\partial}{\partial y}\left(K_{yy}\frac{\partial h}{\partial y}\right)+\frac{\partial}{\partial z}\left(K_{zz}\frac{\partial h}{\partial z}\right)-Q=S\frac{\partial h}{\partial t} \tag{12.1}$$

where K_{xx},K_{yy},K_{zz} the hydraulic conductivity along the x, y and z axes of the model, which are assumed to be parallel to the principle axes of hydraulic conductivity tensor, h is the hydraulic head, Q represents the source/sink terms, and S_s is the specific storage coefficient.

Two special features of this apparently straightforward elliptic equation should be noted. First, the equations are non-linear when flow is unconfined and, second, the storage coefficient is not constant but switches between the specific storage coefficient for confined conditions and the specific yield for unconfined conditions.

12.1.1 The Pre-conditioned Conjugate Gradient (PCG) Solver

As an alternative to the SOR-solver, MIKE SHE's groundwater component also includes the pre-conditioned conjugate gradient solver (Hill, 1990). The PCG solver includes both an inner iteration loop, where the head dependent boundaries are kept constant, and an outer iteration loop where the (non-lin-

ear) head dependent terms are updated. The PCG solver includes a number of additional solver options that are used to improve convergence of the solver. The default values will generally ensure good performance. For the majority of applications, there is no need to adjust the default solver settings. If, on the other hand, non-convergence or extremely slow convergence is encountered in the SZ component, then some adjustment of the solver settings may help.

The PCG solver in MIKE SHE, which is identical to the one used in MODFLOW (McDonald and Harbaugh, 1988), requires a slightly different formulation of the hydraulic terms when compared to the SOR solver.

Potential flow terms

The potential flow is calculated using Darcy's law

$$Q = \Delta h C \tag{12.2}$$

where *Dh* is the piezometric head difference and *C* is the conductance.

The horizontal conductance in Eq. (12.2) is derived from the harmonic mean of the horizontal conductivity and the geometric mean of the layer thickness. Thus, the horizontal conductance between node i and node i-1 will be

$$C_{i-\frac{1}{2}} = \frac{KH_{i-1,j,k}\ KH_{i,j,k}\ (\Delta z_{i-1,j,k} + \Delta z_{i,j,k})}{(KH_{i-1,j,k} + KH_{i,j,k})} \tag{12.3}$$

where, KH is the horizontal hydraulic conductivity of the cell and Δz is the saturated layer thickness of the cell.

The vertical conductance between two cells is computed as a weighted serial connection of the hydraulic conductivity, calculated from the middle of layer k to the middle of the layer k+1. Thus,

$$C_v = \frac{\Delta x^2}{\frac{\Delta z_k}{2K_{z,k}} + \frac{\Delta z_{k+1}}{2K_{z,k+1}}} \tag{12.4}$$

where *Dz* is the layer thickness

Dewatering conditions

Consider the situation in Figure 12.1, where the cell below becomes dewatered.

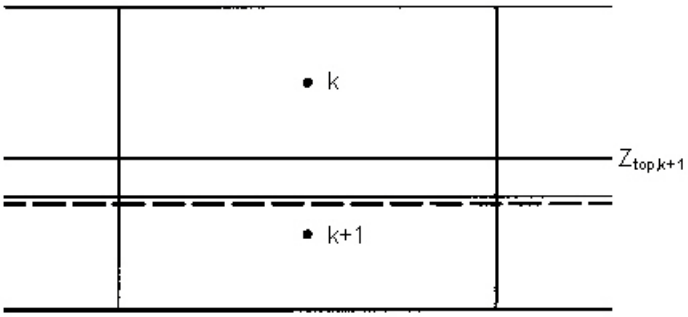


Figure 12.1 Dewatering conditions in a lower cell.

The actual flow between cell *k* and *k+1* is

$$q_{k+½} = Cv_{k+½}(z_{top,k+1} - h_k) \tag{12.5}$$

In the present solution scheme the flow will be computed as

$$q_{k+½} = Cv_{k+½}(h_{k+1} - h_k) \tag{12.6}$$

Subtracting Eqs.(12.5) from (12.6) gives the correction term

$$q_c = Cv_{k+½}(h_{k+1} - z_{top,k+1}) \tag{12.7}$$

which is added to the right-hand side of the finite difference equation using the last computed head.

A correction must also be applied to the finite difference equation if the cell above becomes dewatered.

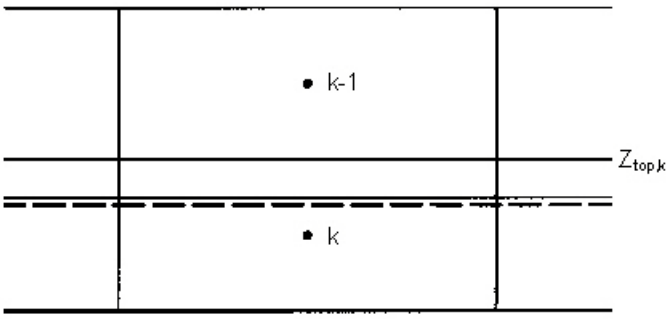


Figure 12.2 Dewatering conditions in the cell above

Thus, from Figure 12.2, the flow from cell *k-1* to *k* is

$$q_{k-½} = Cv_{k-½}(h_{k-1} - z_{top,k}) \tag{12.8}$$

where, again the computed flow is

$$q_{k-½} = Cv_{k-½}(h_{k-1} - h_k) \tag{12.9}$$

Subtracting Eqs. (12.8) from (12.9) gives the correction term

$$q_c = Cv_{k-½}(z_{top,k} - h_k) \tag{12.10}$$

which is added to the right-hand side of the finite difference equation using the last computed head

Storage terms

The storage capacity is computed by

$$\frac{\Delta w}{\Delta t} = \frac{S2(h^n - z_{top}) + S1(z_{top} - h^{n-1})}{\Delta t} \tag{12.11}$$

where *n* is time step, *S1* is the storage capacity at the start of the iteration at time step *n,* and *S2* is the storage capacity at the last iteration.

For confined cells the storage capacity is given as

$$S = \Delta x^2 \Delta z S_{art} \tag{12.12}$$

and for unconfined cells the storage capacity is given as

$$S = \Delta x^2 S_{free} \tag{12.13}$$

Maximum Residual Error

The maximum residual error is the largest allowable value of residual error during an iteration. The solution is obtained when the residual error during an iteration in any computational node is less than the specified tolerance.

The value of the maximum residual error should be selected according to aquifer properties and the dimensions of the model. In practice the maximum residual error value will always be a compromise between accuracy and computing time. It is recommended to check the water balance carefully at the end of the simulation, but it should be emphasized that large internal water balance errors between adjacent computational nodes may not be detected.

If large errors in the water balance are produced the maximum residual error should be reduced.

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).

Mean values of horizontal SZ-conductance:

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.

Under relaxation

The PCG solver can optionally use an under-relaxation factor between 0.0 and 1.0 to improve convergence. In general a low value will lead to convergence, but at a slower convergence rate (i.e. with many SZ iterations). Higher values will increase the convergence rate, but at the risk of non-convergence.

Automatic (dynamic) estimation of under-relaxation factors

If the automatic estimation of the under-relaxation factor is allowed, the under-relaxation factor is calculated automatically as part of the outer iteration loop in the PCG solver. 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 set to 90% of the factor used in the previous iteration and 10% of the current optimal factor.

The time used for automatic estimation of relaxation factors may be significant when compared to subsequently solving the equations and the option is only recommended in steady-state cases.

Under-relaxation by user-defined constant factor

This option allows the user to define a constant relaxation factor between 0.0 and 1.0. In general a value of 0.2 has been found suitable for most simulations.

2-norm reduction-criteria in the inner iteration loop

When the 2-norm option is active, the inner iteration loop of the PCG solver ends when the specified reduction of the 2-norm value is reached. Thus, if the 2-norm reduction criteria is set to 0.01, the inner iteration residual must be reduced by 99% before the inner iteration loop will exit. This option is sometimes efficient in achieving convergence in the linear matrix solution before updating the non-linear terms in the outer iteration loop. It may thus improve the convergence rate of the solver. Continued iterations to meet user-defined

criteria in the inner loop may not be feasible before the changes in the outer iteration loop have been minimised. On the other hand, very few iterations in the inner loop may not be sufficient. The 2-norm may be used to achieve a more optimal balance between the computational efforts spent in the respective solver loops.

Convergence is, however, not assumed until the user defined head and water balance criteria are fulfilled. A reasonable value for the 2-norm reduction criteria has been found to be 0.01.

12.1.2 The Successive Overrelaxation (SOR) Solver

Numerical formulation

Equation (12.1) is solved by approximating it to a set of finite difference equations by applying Darcy's law in combination with the mass balance equation for each computational node.

Considering a node i inside the model area, the total inflow ΣQ_{ij}^{n+1} from neighbouring nodes and source/sinks between time n and time n+1 is given by:

$$\Sigma Q_{ij}^{n+1} = \Sigma q_z^{n+1} + \Sigma q_x^{n+1} + RH_i \Delta x^2 \tag{12.14}$$

where q_z^{n+1} is the volumetric flow in vertical direction, q_x^{n+1} is the volumetric flow in horizontal directions, R is the volumetric flow rate per unit volume from any sources and sinks, Δx is the spatial resolution in the horizontal direction and H_i is either the saturated depth for unconfined layers or the layer thickness for confined layers. See Figure 12.3 and Figure 12.4 for a description of the geometric relationships between the cells.

The horizontal flow components in Eq. (12.14) are given by

$$q_x^{n+1} = C\Delta h^{n+1} \tag{12.15}$$

where C is the horizontal conductance between any of the adjacent nodes in the horizontal directions.

The horizontal conductance in Eq. (12.15) is derived from the harmonic mean of the horizontal conductivity and the geometric mean of the layer thickness. Thus, the horizontal conductance between node *i* and node *i-1* will be

$$C_{i-\frac{1}{2}} = \frac{KH_{i-1,j,k}\ KH_{i,j,k}\ (\Delta z_{i-1,j,k} + \Delta z_{i,j,k})}{(KH_{i-1,j,k} + KH_{i,j,k})} \tag{12.16}$$

where, KH is the horizontal hydraulic conductivity of the cell and Dz is the saturated layer thickness of the cell.

The vertical flow components in Eq. (12.14) are given by

$$q_z^{n+1} = \frac{K_v \Delta x^2 \Delta h^{n+1}}{\Delta z_{i,j,k} + \Delta z_{i,j,k+1}} \tag{12.17}$$

where K_v is the average vertical hydraulic conductivity between nodes in the vertical direction, and $\Delta z_{i,j,k}$ and $\Delta z_{i,j,k+1}$ are the saturated thicknesses of layers k and $k+1$ respectively. For the average vertical hydraulic conductivity, K_v, the SOR solver distinguishes between conditions where the hydraulic conductivity of layer k is greater than or less than the hydraulic conductivity of layer $k+1$. These cases are shown in Table 12.1 and Table 12.2.

Table 12.1 For the case when $K_k < K_{k+1}$.

		Layer k	
		confined	**unconfined**
Layer k+1	**confined**	$K_v = \dfrac{\Delta z_k + \Delta z_{k+1}}{\dfrac{\Delta z_{k+1}}{K_{k+1}}}$	$K_v = \dfrac{\Delta z_k + \Delta z_{k+1}}{\dfrac{\Delta z_{k+1}}{K_{k+1}} + \dfrac{z_{k+1} - h_k}{K_k}}$
	unconfined	$K_v = \dfrac{\Delta z_k + \Delta z_{k+1}}{\dfrac{h_{k+1} - z_{k+1}}{K_{k+1}}}$	$K_v = \dfrac{\Delta z_k + \Delta z_{k+1}}{\dfrac{h_{k+1} - z_{k+1}}{K_{k+1}} + \dfrac{z_{k+1} - h_k}{K_k}}$
		$\Delta h = h_k - h_{k+1}$	

Table 12.2 For the case when $K_k > K_{k+1}$.

		Layer k	
		confined	**unconfined**
Layer k+1	**confined**	$K_v = \dfrac{\Delta z_k + \Delta z_{k+1}}{\dfrac{\Delta z_{k+1}}{K_{k+1}}}$ $\Delta h = h_k - h_{k+1}$	$K_v = \dfrac{\Delta z_k + \Delta z_{k+1}}{\dfrac{\Delta z_{k+1}}{K_{k+1}}}$ $\Delta h = z_{k+1} - h_{k+1}$
	unconfined	$K_v = \dfrac{\Delta z_k + \Delta z_{k+1}}{\dfrac{h_{k+1} - z_{k+1}}{K_{k+1}}}$ $\Delta h = h_k - h_{k+1}$	$K_v = \dfrac{\Delta z_k + \Delta z_{k+1}}{\dfrac{h_{k+1} - z_{k+1}}{K_{k+1}}}$ $\Delta h = z_{k+1} - h_{k+1}$

The transient flow equation yields the finite difference expression

$$S\frac{h_i^{n+1} - h_i^n}{\Delta t} = \Sigma Q_{ij}^{n+1} \tag{12.18}$$

where S is the storage coefficient and Δt is the time step. Eq. (12.18) is written for all internal nodes N yielding a linear system of N equations with N unknowns. The matrix is solved iteratively using a modified Gauss Seidel method (Thomas,1973).

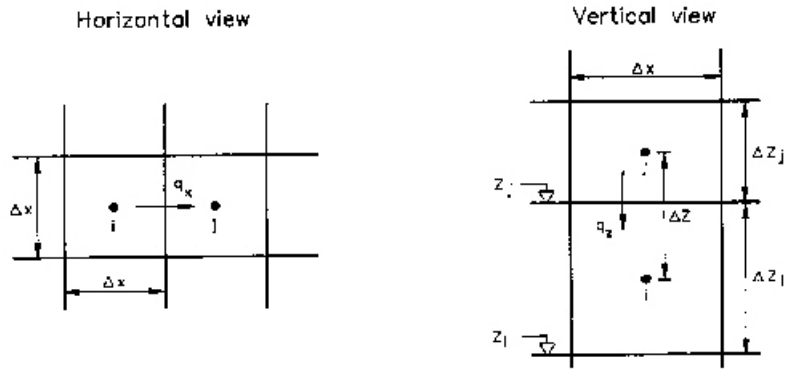


Figure 12.3 Spatial discretisation.

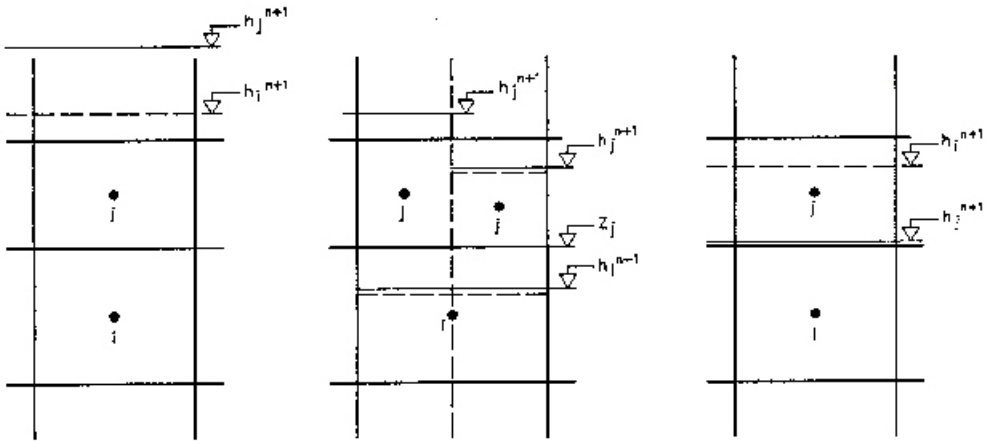


Figure 12.4 Types of vertical flow condition, a) confined conditions in nodes i and j, b) unconfined condition in node i, c) unconfined in nodes i and j, d) dry conditions in node j and confined conditions in node i.

Relaxation Coefficient

The relaxation coefficient, *w*, is used in the solution scheme to amplify the change in the dependent variable (hydraulic head h) during the iteration. The value of *w* should be less than 2 to ensure convergence but larger than 1 to accelerate the convergence.

The optimal value is the value for which the minimum number of iterations are required to obtain the desired tolerance. It is a complex function of the geometry of the grid and aquifer properties. Figure 12.5 illustrates the relation between w and the number of iterations for a given grid. In practice the optimal value of *w* can be found after setting up the grid. The model is run for a few time steps (e.g. ten) with a range of w values between 1 and 2, and the total number of iterations is plotted for each run against the w value as shown in Figure 12.5. The minimum number of iterations corresponds to the optimal value of *w*.

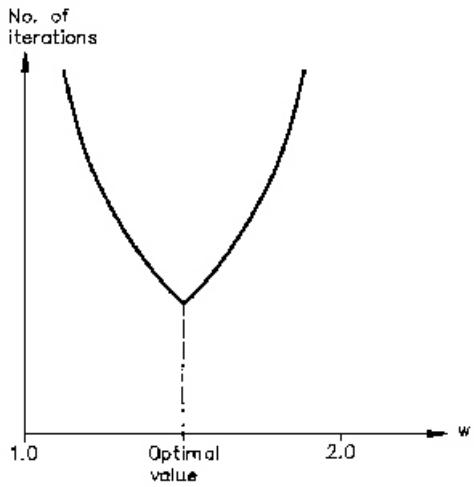


Figure 12.5 Empirically relationship between the relaxation coefficient w and number of iterations for a given model.

Maximum Residual Error

The maximum residual error is the largest allowable value of residual error during an iteration. The solution is obtained when the residual error during an iteration in any computational node is less than the specified tolerance.

The value of the maximum residual error should be selected according to aquifer properties and dimensions of the model. In practice, the maximum residual error value will always be a compromise between accuracy and computing time. It is recommended to check the water balance carefully at the

end of the simulation, but it should be emphasized that large internal water balance errors between adjacent computational nodes may not be detected. If large errors in the water balance are produced the maximum residual error should be reduced.

12.1.3 PCG Steady State Solver

The PCG Steady-State Solver is virtually identical to the transient PCG solver but has been implemented separately to enhance efficiency. In particular, experience has shown that different solver settings may be required when solving the system in steady state versus a transient solution. Furthermore, since the solvers have been implemented separately, there are a couple of options in the steady-state solver that are not available in the transient solver.

Average steady-state river conductance option

In the steady-state PCG solver, the conductance used for the SZ-river exchange in each iteration is averaged with the conductance of the previous iteration. This is done to reduce the risk of numerical instabilities when the conditions are changing between flow/no flow conditions in a computational cell. It is recommended to use this option, as it tends to enhance convergence.

Steady-state (constant) river water depth

When running a steady-state simulation that includes SZ-river exchange, a constant water depth may be specified for the river network, which can be used to calculate the head gradient driving the exchange flow.

If a constant river water depth is not specified, then the river water levels are determined in the following order

1. MIKE SHE hot-start water level, if MIKE SHE hot-start is specified
2. Initial water levels from MIKE Hydro River, if the MIKE Hydro River coupling is used
3. Water level equal to the river bed if not 1) or 2) - (i.e. dry river - no flow from the river to the aquifer).

Canyon exchange option

The Canyon exchange option is only available in the steady state PCG solver. It can be used to describe the exchange between a groundwater aquifer and a river when the river cuts deeply into the aquifer (e.g. through a narrow valley). If the river water level is below the bottom of the adjacent computational groundwater layer, the potential head gradient is reduced. In

this case, the head difference in layers above the river level is limited by the bottom elevation of the layer. Thus,

$$\Delta h_i = h_i - max(h_{riv}, z_i) \tag{12.19}$$

where h_i is the head in the adjacent groundwater node, h_{riv} is the head in the river, and z_i is the bottom of the current layer.

Without the 'Canyon' option, MIKE SHE effectively assumes that the river is hydraulically connected to the upper most model layer, since MIKE SHE calculates the exchange flow with all layers that intersect the river based on the difference between the river level and the water table.

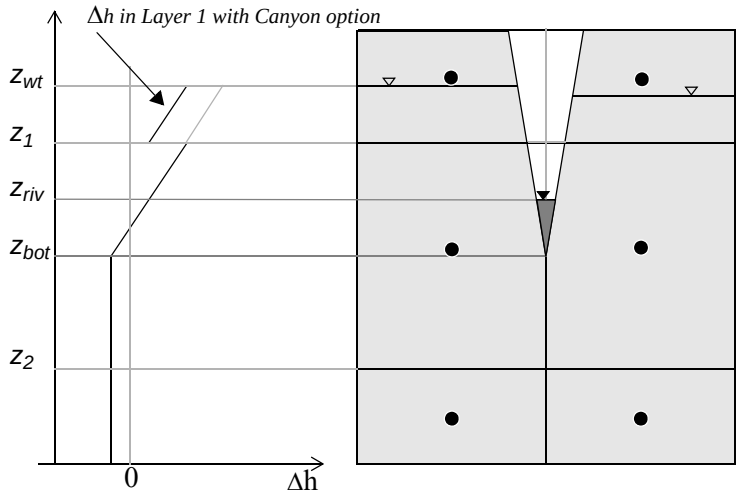


Figure 12.6 Water table elevation versus the head difference.

12.1.4 Boundary Conditions

The SZ module supports the following three types of boundary conditions:

1. Head boundary - Dirichlet conditions, (Type 1) where the hydraulic head is prescribed on the boundary
2. Gradient boundary - Neumann conditions, (Type 2) where the gradient of the hydraulic head (flux) across the boundary is prescribed
3. Head-dependent flux - Fourier conditions, (Type 3) where the head dependent flux is prescribed on the boundary.

The head can be prescribed for all grid nodes (i.e. at the catchment boundary, as well as inside the model area) and for all computational layers. The head may be time-invariant equal to the initial head or can vary in time as specified by the user. An important option is the transfer of space- and time interpo-

lated head boundaries from a larger model to a sub-area model with a finer discretisation.

Prescribed gradients and fluxes can be specified in all layers at the model boundary. Sinks and sources in terms of pumping or injection rates can be specified in all internal nodes. If the unsaturated zone component is not included in the model, the ground water recharge can be specified.

The exchange flow to the river system is included in the source/sink terms and can be regarded as a Type 3 boundary condition for cells with 'contact' to the river system. The exchange flow is a function of the water level in the river, the river width, the elevation of the riverbed, as well as the hydraulic properties at the riverbed and aquifer material.

Distribution of fluxes to the internal cells

Flux boundary

The total discharge of the actual time step (constant or time-varying) is distributed over the internal cells as a function of horizontal conductivities, cell sizes (constant in MIKE SHE) and full or saturated thicknesses (inflow / outflow, respectively).

Inflow (positive discharge):

Cell Factor = Conductivity X Size X Full layer thickness / SUMi=1,n(SizeiX-ConiXThicki)

Cell Dischargei Qi = Qtot X Cell Factori

Outflow (negative discharge):

Same as above, but using actual saturated thickness instead of full layer thickness. In the case that all cells are dry the solution will switch to full layer thickness. However, in this case, the solution will be unstable anyway.

Gradient boundary

Each cell receives a discharge calculated from the actual gradient (constant or time-varying), conductivity, cell size and saturated thickness. Positive gradient yields inflow:

Q = Gradient X Conductivity X Size X Saturated thickness

Notes on Grid size

Distribution of actual cell discharges to the X- and Y- flow velocities are stored on the MIKE SHE result file, where they are needed for presentation and water balance extraction.

The X/Y-flow stored in a cell represents the flow from this cell to the neighbour cell in positive X/Y direction. The discharge of a cell in contact with more than one boundary cells of the actual boundary section is distributed as X/Y

flow components to/from these boundary cells proportional to the grid size of the flow surfaces (constant in MIKE SHE).

Notes on the PCG solver

The discharges to/from the internal cells are updated at the start of each outer iteration and added to the “Right-Hand-Side” of the PCG solver. The discharges are distributed to the X/Y flow velocities (for results storing) at the end of each SZ time step.

Notes on the SOR solver

The discharges to/from the internal cells are updated at the start of each iteration and used as point source/sink terms in the solution. The discharges are distributed to the X/Y flow velocities (for results storing) at the end of each SZ time step.

12.1.5 SZ exchange with ponded water

The saturated zone component calculates the recharge/discharge between ponded water (OL) and the saturated zone (SZ) without the unsaturated zone, if the unsaturated zone is not included or if the phreatic surface is above the topography.

The exchange between SZ and OL is calculated implicitly using the Darcy equation

$$Q = \Delta h C_{1/2}$$

where $C_{1/2}$ is the conductance between the topography and the middle of the upper SZ calculation layer.

In the case of full contact between OL and SZ the conductance is calculated by

$$C_{1/2} = \frac{\Delta x^2}{\frac{\Delta z}{2K_z}} \tag{12.20}$$

where Dz is the thickness of layer 1 and K_z, is the vertical conductivity of layer 1.

In areas with reduced contact between overland and the saturated zone, the conductance between OL and layer 1 is calculated by

$$C_{1/2} = \frac{\Delta x^2}{\frac{\Delta z}{2K_z} + \frac{1}{K_{leak}}} \tag{12.21}$$

where *Dz* is the thickness of layer 1 and K_z, is the vertical conductivity of layer 1 and K_{leak} is the specified leakage coefficient.

12.1.6 Saturated Zone Drainage

The MIKE SHE allows for flow through drains in the soil. Drainage flow occurs in the layer of the ground water model where the drain level is located. In MIKE SHE the drainage system is conceptually modelled as one 'big' drain within a grid square. The outflow depends on the height of the water table above the drain level and a specified time constant, and is computed as a linear reservoir. The time constant characterises the density of the drainage system and the permeability conditions around the drains.

The drainage option may not only be used to simulate flow through drainpipes, but also in a conceptual mode to simulate saturated zone drainage to ditches and other surface drainage features. The drainage flow simulates the relatively fast surface runoff when the spatial resolution of the individual grid squares is too large to represent small scale variations in the topography.

Drainage water can be routed to local depressions, rivers or model boundaries. See Groundwater drainage *(V1 p. 57)* for further details about routing of drainage water.

There are also some additional Extra Parameter options for drainage routing:

- SZ Drainage to Specified MIKE Hydro River H-points *(V2 p. 350)*
- SZ Drainage Downstream Water Level Check *(V2 p. 353)*
- Time varying SZ drainage parameters *(V2 p. 353)*
- Time varying SZ drainage parameters *(V2 p. 353)*
- Canyon exchange option for deep narrow channels *(V2 p. 356)*

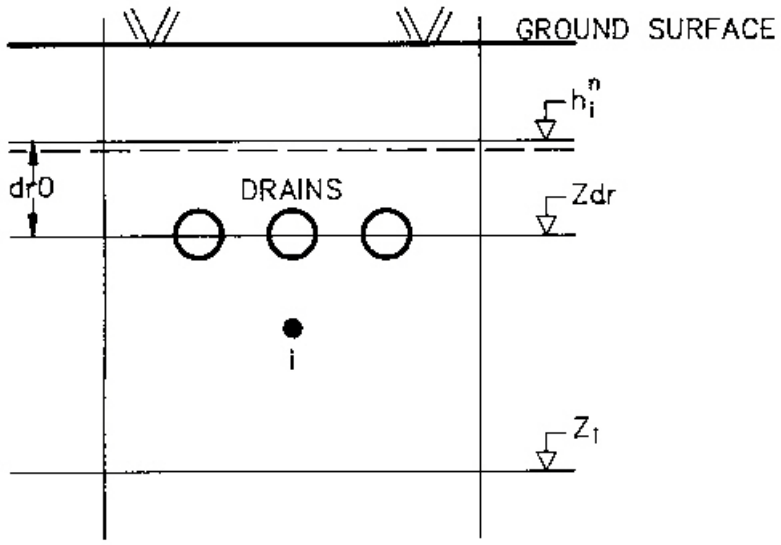


Figure 12.7 Schematic presentation of drains in the drainage flow computations.

SZ Drainage with the PCG solver

When the PCG solver is used, the drain flow is added directly in the matrix calculations as a head dependent boundary and solved implicitly, by

$$q = (h - Z_{dr})C_{dr} \tag{12.22}$$

where h head in drain cell, Z_{dr} is the drainage level and C_{dr} is the drain conductance or time constant.

Drainage with the SOR Solver

When the SOR solver is used, drainage is only allowed from the top layer of the saturated zone model. In this case, the new water table position at the end of the time step is calculated from the flow balance equation

$$\Delta S = (Q_{dr} + \Sigma q)\Delta t \tag{12.23}$$

where ΔS is the storage change as a result of a drop in the water table, Q_{dr} is the outflow through the drain and Σq represents all other flow terms in a computational node in the top layer (i.e. net outflow to neighbouring nodes, recharge, evapotranspiration, pumping and exchange to the river etc.).

The change in storage per unit area can also be calculated from

$$\Delta S = (d_0 - d_t)S_y \tag{12.24}$$

where d_0 is the depth of water above the drain at the beginning of the time step, d_t is the depth of water above the drain at the end of the time step and S_y is the specific yield.

Q_{dr} is calculated based on the mean depth of water in the drain during the time step. Thus,

$$Q_{dr} = C_{dr} \cdot \frac{d_0 + d_t}{2} \tag{12.25}$$

where C_{dr} is the drain conductance or time constant.

Substituting (12.24) and (12.25) into (12.23) and rearranging, the water depth at the end of a time step, d_t, can be calculated by

$$d_t = \frac{\left[d_0 \left(S_y - \frac{C_{dr}\Delta t}{2} \right) - \Sigma q \Delta t \right]}{S_y + \frac{C_{dr}\Delta t}{2}} \tag{12.26}$$

From which the new water table elevation, h_t, at the end of the time step can be calculated by

$$h_t = Z_{dr} + d_t \tag{12.27}$$

where Z_{dr} is the elevation of the drain.

The drainage outflow is added as a sink term using the hydraulic head explicitly. The computation for drainage flow uses the UZ time step, which is usually smaller than the SZ time step. The initial drainage depth d_0 at the beginning of an SZ time step is set equal to h - Zdr, where h is the water table elevation at the end of the previous SZ time step. d_0 is adjusted during the sequence of smaller time steps so that a successive lowering of the water table and the outflow occurs during an SZ time step. This approach often overcomes numerical problems when large time steps are selected by the user. If the drainage depth becomes zero during the calculations drainage flow stops until the water table rises again above the drain elevation.

12.1.7 Initial Conditions

The initial conditions are specified in the Setup Editor and can be either constant for the domain or distributed, using .dfs2 or .shp files. The initial conditions in boundary cells are held constant during the simulation, which means that the initial head in cells with Dirichlet's boundary conditions is the boundary head for the simulation.

If there is ponded water on the ground surface at the start of the simulation and the simulation is steady-state, then the depth of ponded water is treated as a fixed head for SZ.

12.1.8 The Sheet Pile Module

The Sheet Piling module is an add-on module for the PCG groundwater (SZ) solver in MIKE SHE that allows you to reduce the conductance between cells in both the horizontal and vertical directions. Vertical sheet piling reduces the horizontal flow between adjacent cells in the x- or y-direction. It is defined by a surface with a specified leakage coefficient located between two MIKE SHE grid cells in a specified computational layer. If the Sheet Piling module is active, then the horizontal conductance, C_h, for flow between two cells is calculated as

$$C_h = \frac{1}{\frac{1}{2 \cdot k_1 \cdot dz_1} + \frac{1}{k_{leak} \cdot dx \cdot dz^*} + \frac{1}{2 \cdot k_2 \cdot dz_2}} \tag{12.28}$$

where k_i is the hydraulic conductivity of the cells on either side of the sheet pile [L/T], $k_{leak}$is the leakage coefficient of the sheet pile between the cells[1/L], dz_isaturated layer thickness [L], and dz*maximium of dz1 and dz2 [L]

Similarly, reducing the vertical conductance between two layers can simulate a restriction in vertical flow (e.g. thin clay layer or liner). Thus, if the Sheet Piling module is active for vertical flow, the vertical conductance between two layers is calculated as

$$C_v = \frac{dx \cdot dx}{\frac{dz_1}{2 \cdot k_1} + \frac{1}{k_{leak}} + \frac{dz_2}{2 \cdot k_2}} \tag{12.29}$$

where k_i is the hydraulic conductivity of the cells above and below the sheet pile [L/T], k_{leak} is the leakage coefficient [1/T], and $dz_i$is the layer thickness [L].

12.2 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 infor-

mation 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. In many cases, subsurface flow can be described satisfactorily by a lumped conceptual approach such as the linear reservoir method.

The MIKE SHE modelling system used with the linear reservoir module 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. The combined lumped/physically distributed model was primarily developed to provide a reliable, efficient instrument in the following fields of application:

- Assessment of water balance and simulation of runoff for ungauged catchments
- Prediction of hydrological effects of land use changes
- Flood prediction

The following sections first provide an overview of the linear reservoir theory, followed by detailed descriptions of the implementation in MIKE SHE.

12.2.1 Linear Reservoir Theory

A linear reservoir is one, whose storage is linearly related to the output by a storage constant with the dimension time, also called a time constant, as follows:

$$S = kq \tag{12.30}$$

where S is storage in the reservoir with dimensions length, k is the time constant, and q is the outflow from the reservoir with dimensions length/time.

The concept of a linear reservoir was first introduced by Zoch (1934,1936,1937) in an analysis of the rainfall and runoff relationship. Also, a single linear reservoir is a special case of the Muskingum model, Chow (1988).

A Single Linear Reservoir with One Outlet

The continuity equation for a single, linear reservoir with one outlet can be written as

$$\frac{dS}{dt} = I - q \tag{12.31}$$

where t is time, and I is the inflow to the reservoir.

Combining equation (12.8) and (12.9) yields a first-order, linear differential equation which can be solved explicitly

$$\frac{dq}{dt} + \frac{1}{k}q(t) = \frac{1}{k}I(t) \tag{12.32}$$

If the inflow (I) to the reservoir is assumed constant, the outflow (q) at the end of a time step *dt* can be calculated by the following expression

$$q(t+dt) = q_t e^{-dt/k} + I(1 - e^{-dt/k}) \tag{12.33}$$

A Single Linear Reservoir with Two Outlets

The outflows from a linear reservoir with two outlets can also be calculated explicitly. In this case storage is merely, instead of Eq. (1) given as

$$S = k_p q_p = k_o q_o + h_{thresh} \tag{12.34}$$

where k_P is the time constant for the percolation outlet, q_p is percolation, k_o is the time constant for the overflow outlet, q_o is outflow from the overflow outlet, and h_{thresh} is the threshold value for the overflow outlet.

Combining equation (12.34) and (12.32) and solving for *S*, still assuming *I* is constant in time, yields the following expressions for q_p and q_o at time (t+dt).

$$q_p = q_{p_t} e^{\frac{k_o + k_p}{k_p k_o} dt} + \frac{k_o}{k_p + k_o}\left(I + \frac{h_{thresh}}{k_o}\right)\left(1 - e^{\frac{k_o + k_p}{k_p k_o} dt}\right) \tag{12.35}$$

$$q_o = \frac{k_p q_p - h_{thresh}}{k_o} \tag{12.36}$$

12.2.2 General Description

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. An example of a subdivision of a catchment area is outlined in Figure 12.8, where the topographical zones represent the interflow reservoirs in the model. If a river is present, water will be routed through the linear reservoirs as interflow and baseflow and subsequently added as lateral flow to the river. If no river is specified, the interflow and baseflow will be simply summed up and given as total outflow from the catchment area. 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.

The infiltrating water from the unsaturated zone may either contribute to the baseflow or move laterally as interflow towards the stream. Hence, the interflow reservoirs have two outlets, one outlet contributes to the next interflow reservoir or the river and the other contributes to the baseflow reservoirs. The baseflow reservoirs, which only have one outlet, are not interconnected.

Normally, one reservoir should be sufficient for modelling baseflow satisfactorily. However, in some cases, for example in a large catchment, hydraulic

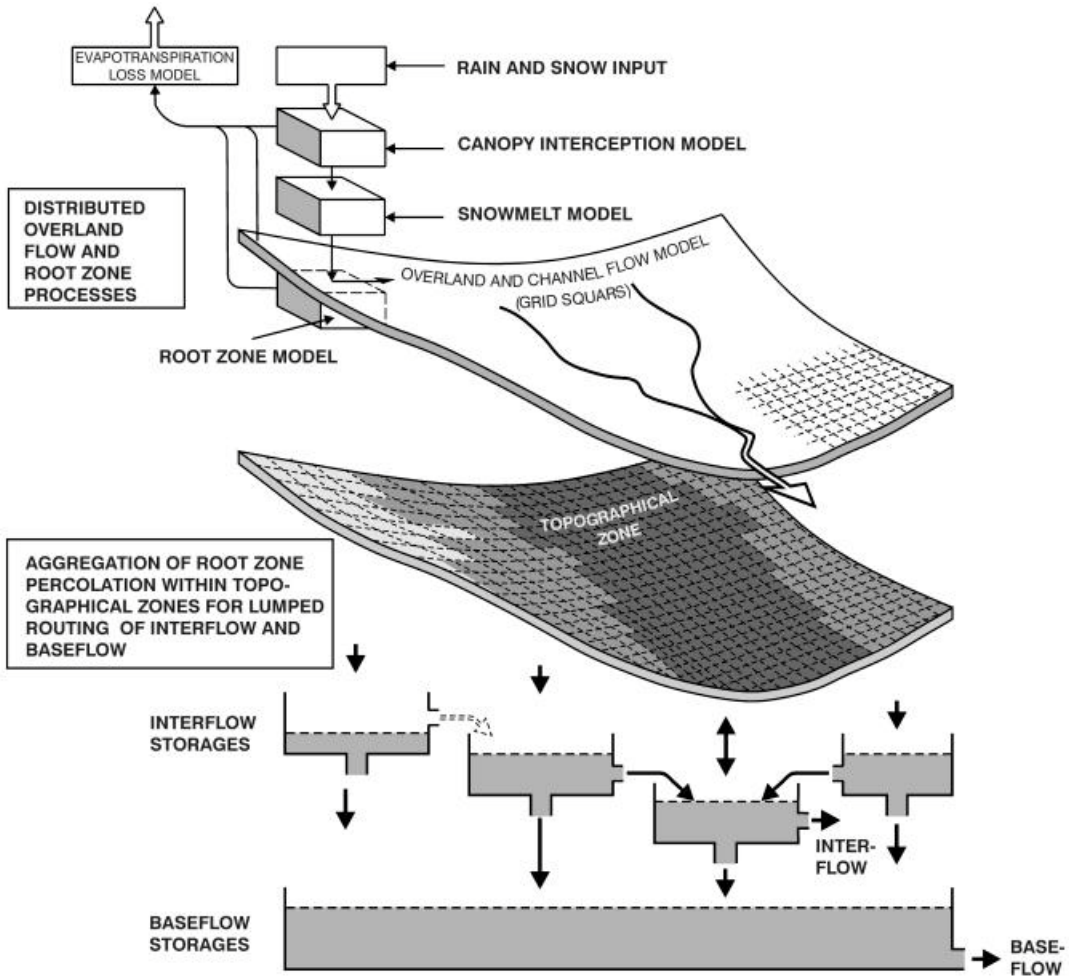


Figure 12.8 Model Structure for MIKE SHE with the linear reservoir module for the saturated zone

contact with a river is unlikely to be present everywhere. In this case, more than one reservoir can be specified.

In low areas adjacent to the river branches the water table may, in periods, be located above or immediately below the surface. In this case, it will contribute more to total catchment evaporation than the rest of the area. To strengthen this mechanism water held in the part of the baseflow reservoirs beneath the lowest interflow zone may be allowed to contribute to the root zone when the soil moisture is below field capacity.

Previous experience with lumped conceptual models shows that the linear reservoir approach is sufficient for an accurate simulation of the interflow and baseflow components, if the input to the reservoirs can be assessed correctly and the time constants of the outlets are known. Due to the distributed approach and physically based representation used in MIKE SHE in the over-

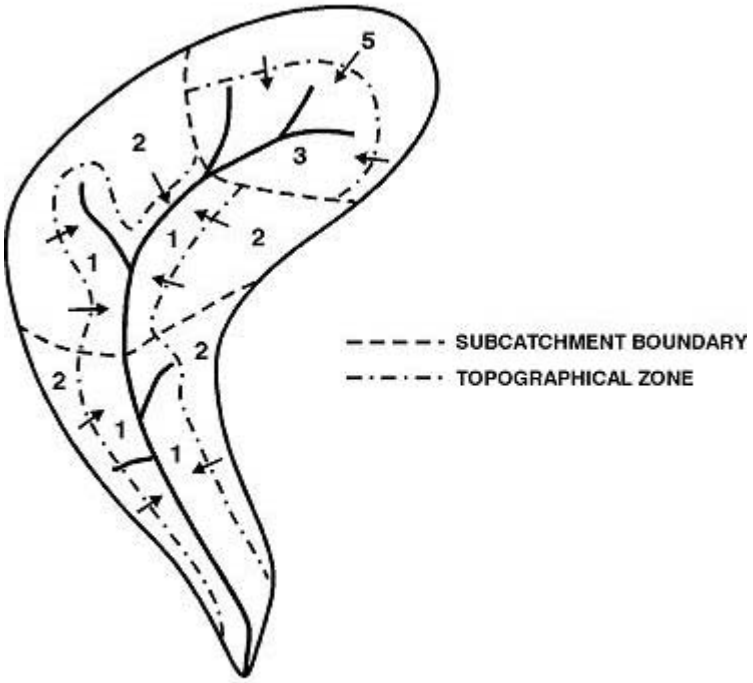


Figure 12.9 Example of Desegregation of a Catchment into Sub-catchments and Topographical Zones

land and unsaturated zone flow components, an accurate simulation of soil moisture drainage in space and time is provided in MIKE SHE for the linear reservoir module. The time constants on the other hand are basically unknown for an ungauged catchment but a fair estimate may be obtained from an evaluation of the hydrogeologic conditions and/or from gauged catchment with similar subsurface conditions.

If the UZ feedback is not included, uncertainty of the time constants will only affect routing of the baseflow and interflow components while the total volumes of runoff will remain unchanged. If UZ feedback from the baseflow reservoir is included, some of the baseflow to the stream will be transferred to the UZ storage because of ET in the unsaturated zone.

12.2.3 Subcatchments and Linear Reservoirs

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.

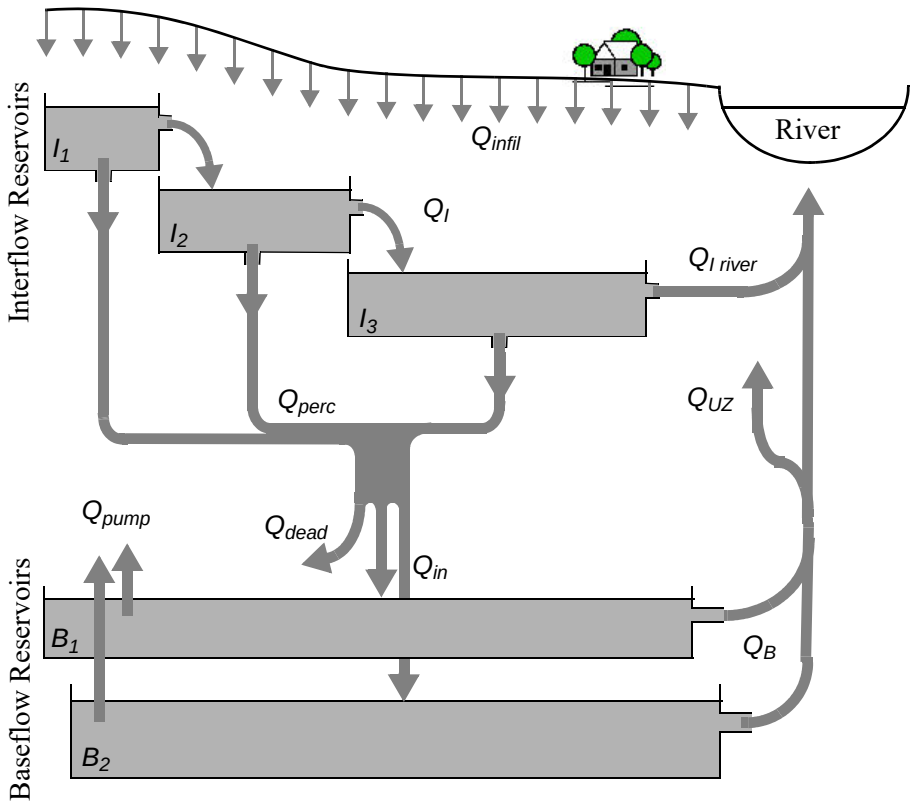


Figure 12.10 Schematic flow diagram for the Subcatchment-based, linear reservoir flow module

The Interflow Reservoirs are equivalent to what was called the Topographic Zones in earlier versions of the Linear Reservoir module in MIKE SHE. There is no limit on the number of subcatchments, or linear reservoirs that can be specified in the model.

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. If no river links are found in the reservoir, then the water will not drain to the river and a warning will be written to the run log file. For example, in Figure 12.9 Interflow Reservoir 2 always flows into Reservoir 1 and Reservoir 1 discharges to the stream network. Likewise, Reservoir 5 flows into 3, which discharges to the stream network.

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.

Figure 12.10 is a schematization of the flow to, from and within the system of linear reservoirs. Vertical infiltration from the unsaturated zone is distributed to the Interflow Reservoirs (Q_{infil}). Water flows between the Interflow Reservoirs sequentially (Q_I) and eventually discharges directly to the river network ($Q_{I\ river}$), or percolates vertically to the deeper Baseflow Reservoirs (Q_{perc}). The parallel Baseflow Reservoirs each receive a fraction of the percolation water (Q_{in}) and each discharges directly to the river network (Q_B). Groundwater can be removed from the Baseflow Reservoirs via groundwater pumping (Q_{pump}). If UZ feedback is included, then some of the baseflow to the stream will be added to the UZ storage (Q_{UZ}) and subsequently removed from the unsaturated zone via ET.

In some situations, the interflow reservoirs will not correspond to the areas of active baseflow in the current catchment. That is, some percolation from the interflow reservoirs may contribute to baseflow in a neighbouring watershed. This has been resolved by introducing a dead zone storage (Q_{dead}) between the Interflow and Baseflow Reservoirs

12.2.4 Calculation of Interflow

Each Interflow Reservoir is treated as A Single Linear Reservoir with Two Outlets *(p. 207)*. Thus, from Eq. 12.30, if the water level in the linear reservoir is above the threshold water level

$$q_I = (h - h_{thresh})/k_i \qquad (12.37)$$

where q_I is the specific interflow out of the reservoir in units of [L/T], h is the depth of water in the interflow reservoir, h_{thresh} is the depth of water required before interflow occurs, and k_i is the time constant for interflow. If the water level is below the threshold there is no interflow.

Similarly, if there is water in the linear reservoir, specific percolation outflow can be calculated from

$$q_{perc} = h/k_p \qquad (12.38)$$

where h is again the depth of water in the interflow reservoir, and k_p is the time constant for percolation flow. If the water level is at the bottom of the reservoir there is no percolation.

Combining Eqs 12.37 and 12.38 with the continuity equation

$$\frac{dh}{dt} = \frac{(q_{in} - q_I - q_{perc})}{S_y} \tag{12.39}$$

where, q_{in} is the sum of inflow from an upstream Interflow reservoir and the infiltration from UZ, S_y is the specific yield, gives the following expression for h at time t when there is both q_I and q_{perc} (linear reservoir with two outlets)

$$h_t = h_{t_0} e^{\frac{-(k_i + k_p)dt}{k_i k_p S_y}} + \frac{k_i k_p}{(k_i + k_p)}\left(q_{in} + \frac{h_{thresh}}{k_i}\right)\left(1 - e^{\frac{-(k_i + k_p)dt}{k_i k_p S_y}}\right) \tag{12.40}$$

In the case where the water level is below the threshold, the formulation for a linear reservoir with one outlet applies, which yields

$$h_t = h_{t_0} e^{\frac{-dt}{k_p S_y}} + k_p q_{in}\left(1 - e^{\frac{-dt}{k_p S_y}}\right) \tag{12.41}$$

Inflow to the Interflow reservoir, q_{in}, will normally be positive (i.e. water will be being added), but if evapotranspiration from the saturated zone is included or the net precipitation is used as input there might be a net discharge of water from the interflow reservoir. As the infiltration is a constant rate calculated explicitly in other parts of MIKE SHE, this will result in a water balance error if the interflow reservoir is empty. This will be reported in the log file at the end of the simulation.

From the level changes in the reservoir, the total average outflow can be calculated for the time step, *dt.* Thus, for the two outlet case,

$$q_{out} = -(h_t - h_{t0})S_y/dt + q_{in} \tag{12.42}$$

$$q_I = \frac{k_p}{k_i + k_p}(q_{out} - h_{thresh}/k_p) \tag{12.43}$$

$$q_{perc} = q_{out} - q_I \tag{12.44}$$

and for the single outlet case (no interflow),

$$q_{perc} = -(h_t - h_{t0})S_y/dt + q_{in} \tag{12.45}$$

If during a time step the reservoir level crosses one or more thresholds, an iterative procedure is used to subdivide the time step and the appropriate formulation is used for each sub-time step.

The discharge to the river, $q_{I\ river}$, in the lowest Interflow Reservoir is simply the q_I from that reservoir.

12.2.5 Calculation of Interflow Percolation and Dead Zone Storage

The inflow to the Baseflow Reservoirs from the Interflow Reservoirs, q_v, is weighted based on the overlapping areas. Thus, if the Baseflow and Interflow reservoirs overlap, then

$$q_v = \sum_{i=1}^{N} q_{perc_i} \frac{A_{interflow_i}}{A_{baseflow}} (1.0 - DZ_{frac}) \tag{12.46}$$

where $A_{interflow}$ is the area of the Interflow Reservoir that overlaps with the Baseflow Reservoir, $A_{baseflow}$ is the area of the Baseflow Reservoir and DZ_{frac} is the fraction of the total Q_{in} that goes to the dead zone storage.

Likewise, the amount of water going to deadzone storage is given by

$$q_{dead} = \sum_{i=1}^{N} q_{perc_i} \frac{A_{interflow}}{A_{baseflow}} DZ_{frac} \tag{12.47}$$

12.2.6 Calculation of Baseflow

From Eq. 12.30, if the water level in the linear reservoir is above the threshold water level

$$q_B = \frac{(h - h_{thresh})}{k_b} \tag{12.48}$$

where h is the depth of water in the baseflow reservoir, h_{thresh} is the depth of water required before baseflow occurs, and k_b is the time constant for baseflow. If the water level is below the threshold there is no baseflow.

Similar to Eq. (12.39), for each Baseflow Reservoir

$$\frac{dh}{dt} = \frac{(q_v - q_B - q_{pump})}{S_y} \tag{12.49}$$

where q_{IN} is the amount of inflow to each base flow reservoir, q_B is the specific baseflow out of the reservoir, and q_{pump} is the amount of water removed

via extraction wells from each reservoir. Both q_v and q_{pump} are controlled by split fractions that distribute q_v and q_{pump} between the two parallel baseflow reservoirs.

Each Baseflow Reservoir can be treated as A Single Linear Reservoir with One Outlet *(p. 206)*. Thus, as long as the water level is above the threshold water level for the reservoir (i.e. there is still baseflow out of the reservoir),

$$h_t = h_{t_0} e^{\frac{-dt}{k_b S_y}} + k_b (q_v - q_{pump}) \left(1 - e^{\frac{-dt}{k_b S_y}}\right) \tag{12.50}$$

where k_b is the time constant for the Baseflow Reservoir.

The formula for a single outlet is applicable because there is no time constant associated with the pumping. However, Q_{pump}, is also controlled by a threshold level, in this case, a minimum level below which the pump is turned off. Since this minimum level is independent of the threshold level for the reservoir itself, a case could arise, whereby there was pumping, but no baseflow from the reservoir. In this case,

$$h_t = h_{t_0} + \frac{(q_v - q_{pump}) \cdot dt}{S_y} \tag{12.51}$$

If there is no pumping and no baseflow out, then the expression for the water level in the reservoir simply becomes

$$h_t = h_{t_0} + \frac{q_v \cdot dt}{S_y} \tag{12.52}$$

In general, during a time step, the water level may cross one or more of the pumping or the baseflow thresholds. If this occurs, the program uses an iterative procedure to split the time step into sub-time steps and applies the appropriate formulation to each sub-time step.

12.2.7 UZ Feedback

A feedback mechanism to the unsaturated zone has been included in the module to model a redistribution of water in favour of evapotranspiration from the low wetland areas located adjacent to most rivers.

The UZ feedback is not relevant if the Richards Equation method is used and the feedback fraction should be set to zero.

Water is redistributed from the base flow reservoirs to the unsaturated zone model in the lowest interflow reservoir zone of the subcatchment, if there is a

water deficit in the root zone. The base flow discharge is distributed to all UZ columns in the lowest interflow reservoir.

The UZ deficit in the lowest interflow reservoir is called the Field Moisture Deficit (*FMD*) in units of metres, and is calculated as the total amount of water required to bring the entire root zone in all the UZ columns back up to field capacity. Thus,

$$FMD = \int_0^{\text{Rootdepth}} (\theta_{FC} - \theta) dz = \sum_{i=1}^{n} (\theta_{FC} - \theta_i) \Delta z_i \tag{12.53}$$

where θ_{FC} is the moisture content at field capacity, θ is the moisture content in the root zone, n is the number of UZ cells in the root zone, and Δz is the height of the UZ cell. This is then summed over all of the UZ columns in the lowest interflow reservoir.

Now, the amount of water that is available from the linear reservoirs to be redistributed to the unsaturated zone is calculated as a fraction of the base-flow:

$$S_{available} = Q_{B1} S_{y1} \cdot UZ_{frac1} \cdot dt + Q_{B2} S_{y2} \cdot UZ_{frac2} \cdot dt \tag{12.54}$$

where UZ_{frac} is a specified fraction of the baseflow that is allowed to recharge the unsaturated zone.

If $S_{available}$ is larger than or equal to the Field Moisture Deficit, then the water content of each of the root zone cells is increased to field capacity.

If $S_{available}$ is smaller than the Field Moisture Deficit, then the water content of each of the root zone cells is proportionally increased by

$$S_{feedback_i} = \theta_i + \frac{S_{available}}{FMD} (\theta_{FC} - \theta_i) \tag{12.55}$$

After the feedback calculation, the amount of baseflow to the river is reduced by the amount of water used to satisfy the Field Moisture Deficit in the unsaturated root zone, which is Q_{UZ}, in Figure 12.10.

12.2.8 Coupling to MIKE Hydro River

The discharge from the lowest Interflow Reservoir in each subcatchment is distributed evenly over the MIKE Hydro River nodes located in the reservoir. Likewise, the baseflow from the Baseflow Reservoirs is distributed over the same nodes. The default distribution can be overridden by specifying specific MIKE Hydro River Branches and chainages.

If the MIKE Hydro River branch is defined as a “losing branch” then water will be added to the baseflow reservoir based on the head in the river, the river width and the river bed conductivity specified.

Note: The lowest linear reservoir discharges to the MIKE Hydro River network. However, if the lowest linear reservoir is not connected to a MIKE Hydro River branch, then the linear reservoir will still discharge. The only difference is that no water will be added to MIKE Hydro River. This situation can arise when there are no MIKE Hydro River branches in small subcatchments. A warning will be written to the log file if a subcatchment is not connect to a MIKE Hydro River branch.

12.2.9 Limitations of the Linear Reservoir Method

The Linear Reservoir method is a simple method for calculating overall water balances in the saturated zone. As such, it is unsuitable when a detailed spatial distribution of the water table is required. However, even given the simplicity of the method, the following simplifications and limitations should be noted:

- In the Linear Reservoir Method, the same river links are used for both of the base flow reservoirs. This is a limitation in the sense that occasionally you may find that fast response and the slow response baseflow may contribute to different parts of the stream.
- When calculating the unsaturated flow, the bottom boundary condition is input from a separate file - not calculated during the simulation. This means that changes in the water levels in the reservoirs will have no effect on the UZ boundary condition.
- The numerical solution used for the Linear Reservoir module assumes that the inflow to each of the linear reservoirs is constant within a time step. Strictly speaking, this is not correct as outflow from each reservoir changes exponentially during a time step. The calculation procedure uses the mean outflow from the upper reservoir as inflow to the downstream reservoir. In this way, there is no water balance error but the dynamics are somewhat dampened. If this is a problem, smaller time steps can be chosen, which will lead to a more accurate solution, as the changes in flow become smaller during each time step.

13 Working with Groundwater - User Guide

The saturated groundwater component of MIKE SHE includes all of the water below the water table. If the water table is at or above the ground surface then the unsaturated zone is turned off this cell.

The unsaturated zone geology is not related to the saturated zone geology. Instead the unsaturated zone geology is essentially independent of the saturated zone geology.

13.1 Conceptualization of the Saturated Zone Geology

The development of the geological model is probably the most time consuming part of the initial model development. Before starting this task, you should have developed a conceptual model of your system and have at your disposal digital maps of all of the important hydrologic parameters, such as layer elevations and hydraulic conductivities.

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

The geologic model can include both geologic layers and geologic lenses. The former cover the entire model domain and the later may exist in only parts of your model area. Both geologic layers and lenses are assigned geologic parameters as either distributed values or as constant values.

The alternative is to define the hydrogeology based on geologic units. In this case, you define the distribution of the geologic units and the geologic properties are assigned to the unit.

Each geologic layer can be specified using a dfs2 file, a .shp file or a distribution of point values. However, you should be aware of the way these different types of files are interpolated to the numerical grid.

The simplest case is that of distributed point values. In this case, the point values are simply interpolated to the numerical grid cells based on the available interpolation methods.

In the case of shp files, at present, only point and line theme .shp files are supported. Since lines are simply a set of connected points, the .shp file is essentially identical to the case of distributed point values. Thus, it is interpolated in exactly the same manner.

The case of .dfs2 files is in fact two separate cases. If the .dfs2 file is aligned with the model grid then the cell value that is assigned is calculated using the bilinear method with the 4 nearest points to the centre of the cell. If the .dfs2 file is not aligned with the model grid then the file is treated exactly the same as if it were a .shp file or a set of distributed point values.

The geologic model is interpolated to the model grid during preprocessing, by a 2 step process.

1. **The horizontal geologic distribution is interpolated to the horizontal model grid.** If Geologic Units are specified then the integer grid codes are used to interpret the geologic distribution of the model grid. If distributed parameters are specified then the individual parameters are interpolated to the horizontal model grid as outlined above.

2. **The vertical geologic distribution is interpolated to the vertical model grid.** In each horizontal model grid cell, the vertical geologic model is scanned downwards and the soil properties are assigned to the cell based on the average of the values found in the cell weighted by the thickness of each of the zones present. Thus, for example, if there were 3 different geologic layers in a model cell each with a different Specific Yield, then the Specific Yield of the model cell would be

$$S_y = \frac{S_{y1} \cdot z_1 + S_{y2} \cdot z_2 + S_{y3} \cdot z_3}{z_1 + z_2 + z_3} \quad (13.1)$$

where z is the thickness of the geologic layer within the numerical cell.

Conductivity values

Hydraulic conductivity is a special parameter because it can vary by many orders of magnitude over a space of a only few metres or even centimetres. This necessitates some special interpolation strategies.

- **Horizontal Interpolation**
  The horizontal interpolation of hydraulic conductivity interpolates the raw data values. Thus, in Step 1 above, when interpolating point values that range over several orders of magnitude, such as hydraulic conductivity, the interpolation methods will strongly weight the larger values. That is, small values will be completely overshadowed by the large values.

  In fact, the interpolation in this case should be done on the logarithm of the value and then the cell values recalculated. Until this option is available in the user interface, you should interpolate conductivities outside of MIKE SHE using, for example, Surfer. Alternatively, the point values could be input as logarithmic values and the Grid Calculator Tool in the MIKE SHE Toolbox can be used to convert the logarithmic values in the .dfs2 file to conductivity values.

- **Vertical Interpolation**
  In Step 2 above, the geologic model is scanned down and interpreted to the model cell. Although, horizontal conductivity can vary by several orders of magnitude in the different geologic layers that are found in a model cell, the water will flow horizontally based on the highest transmissivity. Thus, the averaging of horizontal conductivity can be down the same as in the example for Specific Yield above. Vertical flow, however, depends mostly on the lowest hydraulic conductivity in the geologic layers present in the model cell. In this case a harmonic weighted mean is used instead. For a 3 layer geologic model in one model cell, the vertical conductivity would be calculated by

$$K_z = \frac{z_1 + z_2 + z_3}{\frac{z_1}{K_{z1}} + \frac{z_2}{K_{z2}} + \frac{z_3}{K_{z3}}} \tag{13.2}$$

  where z is the thickness of the geologic layer within the numerical cell.

13.1.1 Lenses

In building a geologic model, it is typical to find discontinuous layers and lenses within the geologic units. The MIKE SHE setup editor allows you to specify such units - again independent of the numerical model grid.

Lenses are also a very useful way to define a complex geology. In this case, the lenses are used to define the subsurface geologic units within a larger regional geologic unit.

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

In the case of lenses, an extra step is added to the beginning of the 2-step process outlined in the previous section. The location of the lenses is first interpolated to the horizontal numerical grid. Then the lenses become essentially extra geologic layers in the columns that contain lenses. However, there are a number of special considerations when working with lenses in the geologic model.

- **Lenses override layers**
  That is, if a lense has been specified then the lense properties take precedence over the layer properties and a new geologic layer is added in the vertical column.

- **Vertically overlapping lenses share the overlap**
  If the bottom of lense is below the top of the lense beneath, then the lenses are assumed to meet in the middle of the overlapping area.

- **Small lenses override larger lenses**
  If a small lense is completely contained within a larger lense the smaller lense dominates in the location where the small lense is present.

- **Negative or zero thicknesses are ignored**
  If the bottom of the lense intersects the top of the lense, the thickness is zero or negative and the lense is assumed not to exist in this area.

13.2 Numerical Layers

13.2.1 Specific Yield of the upper SZ numerical layer

The specified value for specific yield is not used for the specific yield of the upper most SZ numerical layer if UZ is included in the simulation.

By definition, the specific yield is the amount of water release from storage when the water table falls. The field capacity of a soil is the remaining water content after a period of free drainage. Thus, specific yield is equal to the saturated water content minus the field capacity.

To avoid water balance errors at the interface between the SZ and UZ models, the specific yield of the top SZ layer is set equal to the he saturated water content minus the field capacity. The value is determined once at the beginning of the simulation. The water content parameters are taken from the UZ layer in which the initial SZ water table is located.

In principle, having different values between the SZ and UZ models does not directly cause a water balance error, but it may cause numerical problems that could lead to water balance errors. By definition, the steady-state water table location will be identical in both the SZ and UZ models. Pumping from the SZ will lower the SZ water table by an amount equal to the specific yield divided by the cell area times the pumping rate. However, if the field capacity is not correlated to the specific yield, then the amount of water released from storage in the UZ will be more or less than the amount extracted from the SZ cell. This will result in different water tables in the SZ and UZ models. If pumping stops, the system will again reach an equilibrium with the same water table in both the SZ and UZ simply because of the pressure head redistribution.

As mentioned, the upper Sy value is calculated only at the beginning of the simulation based on the UZ layer in which the initial SZ water table is located. If the soil profile has multiple soil types with different field capacities and saturated water contents, then the specific yield in the SZ and UZ model may diverge during the simulation. With slowly moving water tables, the differ-

ences may not be that large and the errors generated will likely be tolerable. If the water table drops into a lower SZ layer, then the specified Specific Yield will be used.

The actual value used in the model is displayed in the pre-processed tab under Specific Yield.

13.2.2 SZ Boundary Conditions

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

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

In MIKE SHE, the rest of the boundary conditions can be divided into two types: Internal and Outer. A boundary condition in one of the cells on the edge of the model domiain is “outer boundary”. All other boundary conditions inside the domain are call “internal boundaries”.

Groundwater Drainage

Saturated zone drainage is a special boundary condition in MIKE SHE used to defined natural and artificial drainage systems that cannot be defined in MIKE Hydro River. It can also be used to simulate simple overland flow, if the overland flow system can be conceptualized as a shallow drainage network connected to the groundwater table - for example, on a flood plain.

Saturated zone drainage is removed from the layer of the Saturated Zone model containing the drain level. Water that is removed from the saturated zone by drains is routed to local surface water bodies, local topographic depressions, or out of the model.

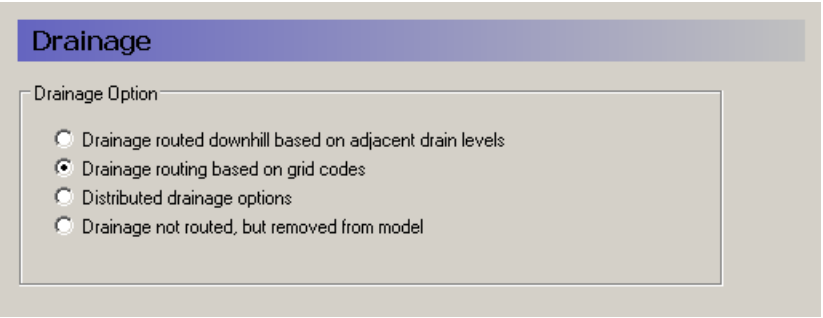
When water is removed from a drain, it is immediately moved to the recipient. In other words, the drain module assumes that the time step is longer than the time required for the drainage water to move to the recipient. This is the same as a “full pipe”. That is, water added to the end of a full pipe of water causes an equal amount of water to immediately flow out the opposite end - regardless of the length of the pipe.

Drain flow is simulated using an simple linear reservoir formula. Each cell requires a drain level and a time constant (leakage factor). Both drain levels and time constants can be spatially defined. A typical drainage level is 1m below the ground surface and a typical time constant is between 1e-6 and 1 e-7 1/s.

Drainage reference system

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

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



Drainage routed downhill based on adjacent drain levels
This option was originally the only option in MIKE SHE. The reference system is created automatically by the pre-processor using the slope of the drains calculated from the drainage levels in each cell.

Thus, the pre-processor calculates the drainage source-recipient reference system by

1. looking at each cell in turn and then
2. looking for the neighbouring cell with the lowest drain level.
3. If this cell is an outer boundary cell or contains a river link, the search stops.
4. If this cell does not contain a boundary or river link, then the search is repeated with the next downstream neighbour until either a local minimum is found or a boundary cell or river link is found.

The result of the above search from each cell is used to build the source-recipient reference system.

If local depressions in the drainage levels exist, the SZ nodes in these depressions may become the recipients for a number of drain flow producing nodes. This often results in the creation of a small lake at such local depressions. If overland flow is simulated, then the ponded drainage water will become part of the local overland flow system.

Drain levels above the topography are not allowed. In this case a warning will be written to the PP_Print.log and the drain level will be automatically adjusted to a value just below the topography.

The drain level method is not allowed when using Time varying SZ drainage parameters *(V2 p. 353)* because the source-recipient reference system is only calculated once at the beginning of the simulation.

The drain-slope based reference system has been used in MIKE SHE for many years and works well in most situations. However, when MIKE SHE is applied where there is very little surface topographic relief, it is often difficult to establish a suitable reference system based on the surface topography/drain slopes. For example, often it is assumed that the drains are located 50 to 100 cm below the terrain. In flat areas, this may generate many undesired local depressions, which may receive drainage water from a large area, thus generating lakes in places where there should not be a lake.

If the drain level is perfectly flat, drainage is turned off. In other words, if the drain-slope method cannot find a downhill neighbour because all the neighbours have the same elevation as the cell, the drain slope method assumes that the cell is a local depression. However, the depression has no sources of drainage except itself. Thus, the drainage function is effectively turned off.

Tip: MIKE SHE considers a grid point to be a local depression even if the drainage level in the four surrounding model grids is only 1 mm higher. The only way to avoid such problems is to create a drain level map that does not contain “wrong” local depressions. For large models this may be difficult and time consuming. In this case, one of the other drainage options may be better.

Remember, the drainage is routed to a destination. It does not phyisically flow downhill. The drain levels are only used to build the drainage source-recipient reference system, and to calculate the amount of drainage.

Drainage routing based on grid codes

This method is often used when the topography is very flat, which can result in artificial depressions, or when the drainage system is very well defined, such as in agricultural applications.

In this method, the drainage levels and the time constants are defined as in the previous method and the amount of drainage is calculated based on the drain levels and the time constant.

If the drainage routing is specified by Drain Codes, a grid code map is required that is used to restrict the search area for the source-recipient reference system. In this case, the pre-processer calculates the reference system within each grid code zone, such that all drainage generated within one zone is routed to recipient nodes with the same drain code value.

When building the reference system, the pre-processor looks at each cell and then

1. looks for the nearest cell with a river link with the same grid code value,

2. if there is no cells with river links, then it looks for the nearest outer boundary cell with the same grid code,

3. if there are no cells with outer boundary conditions, then it looks for the cell with the same grid code value that has the lowest drain level. In this case, the reference system is calculated as if it was based on Drain Levels (see previous section).

The result of the above search for each cell is used to build the source-recipient reference system.

The above search algorithm is valid for all **positive** Drain Code values. However, all cells where

Drain Code = 0 - will not produce any drain flow and will not receive any drain flow, and

Drain Code < 0 (negative) - will not drain to river links, but will start at Step 2 above and only drain to either a outer boundary or the lowest drain level.

Tip: One method that is often used is to specify only one Drain Code value for the entire model area (e.g. Drain Code = 1). Thus, all nodes can drain and any drain flow is routed to the nearest river link. If there are no rivers, the drain flow will be routed to the nearest boundary. If you want to route all drain flow to the boundaries instead of the rivers, a negative drain code can be specified for the entire area (e.g. Drain Code = -1).

Distributed drainage options

Choosing this method, adds the Option Distribution item to the data tree. With the Option Distribution, you can specify an integer grid code distribution that can be used to specify different drainage options in different areas of your model.

Code = 1 - In grid cells with a value of 1, the drainage reference system is calculated based on the Drain Levels.

Code =2 - In grid cells with a value of 2, the drainage reference system is calculated based the Drain Codes.

Code = 3 - Drainage in grid cells with a value of 3 is routed to a specified MIKE Hydro River branch and chainage. At the moment, this options 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)*.

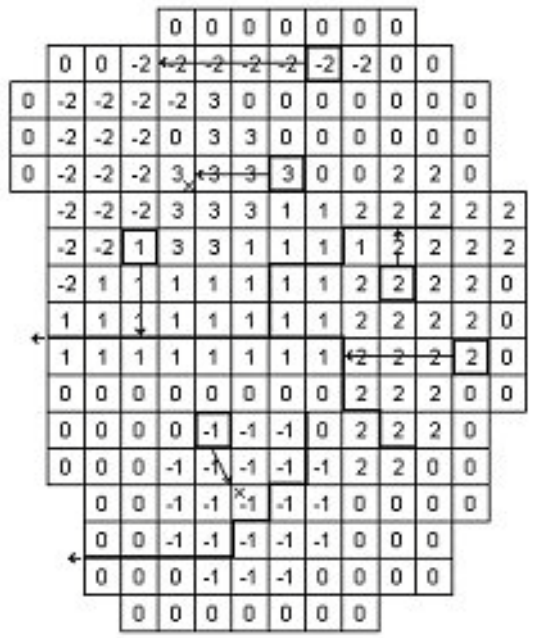
Code = 4 - Drainage in grid cells with a value of 4 is routed to a specified MOUSE man hole. At the moment, this options requires the use of

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.

Drain Code Example



- The grid cells with Drain Code 3 drain to a local depression since no boundary or river link is found adjacent to a grid with the same drain code.
- The grid cells with Drain Code 1 or 2 drain to nearest river link located adjacent to a grid with the same drain code.
- The grid cells with drain code 0 do not contain drains and thus no drainage is produced.
- The grid cells with Drain Code -1 drains to local depression since no boundary is found adjacent to a grid with the same drain code.
- The grid cells with Drain Code -2 drains to nearest boundary grid with the same drain code.

The Pre-processed Drainage Reference System

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

- **Drain Codes** - The value in the pre-processed Drain Codes map reflects the Option Distribution specified. For example, those cells with an Option Distribution equal to 1 (Drainage routed based on Drain Levels) will have a pre-processed Drain Code equal to 0, because the Drain Codes are not being used for those cells.

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

- **Drainage to river** - This grid displays the river link number that the cell drains to. Adjacent to the river links, the cells are labeled with negative numbers to facilitate the interpretation of flow from cells to river links. Thus, in principle, all drainage from cells with the same positive code are drained to the cell with the corresponding negative code.

  However, this is slightly too simple because the cells actually drain directly to the river links. In complex river systems, when the river branches are close together, you can easily have cells connected to multiple branches on different sides. In this case, the river link numbers along the river may not reflect the drainage-river link reference used in the model.

  If you want to see the actual river links used in all cells, you can use the Extra Parameter, Canyon exchange option for deep narrow channels *(V2 p. 356)*, to generate a table of all the river link-cell references in the PP_Print.log file.

Cells with a value of zero either do not generate drainage, or they drain to a the outer boundary or a local depression.

13.2.3 Saturated Zone drainage + Multi-cell Overland Flow

The topography is often used to define the SZ drainage network. Thus, a refined topography more accurately reflects the SZ drainage network.

The SZ drainage function uses a drain level and drain time constant. The drain level defines the depth at which the water starts to drain. Typically, this

is set to some value below the topography to represent the depth of surface drainage features below the average topography. This depth should probably be much smaller if the topography is more finely defined in the sub-grid model. The drain time constant reflects the density of the drainage network. If there are a lot of drainage features in a cell then the time constant is higher and vice versa.

When using the multi-cell OL, the drainage system is updated in the sense that the drain level will be defined using the sub-scale topography information. The SZ drainage will include the following when using sub-scale:

- Multi-scale SZ drainage supported only in the PCG transient SZ solver
- Each sub-grid cell will have the same drain time constant defined by the value in the coarse grid.
- If the drain level is defined as an elevation, then all sub-grids will have the same drain level.
- If the drain level is defined by depth below the surface, then each sub-grid may have a unique drain level, since each sub-grid can have a different elevation. Each coarse grid cell has a water table that is common for all fine scale grids within the coarse grid.
- If the coarse cell water table is above the fine scale drain level, then drainage is calculated based on the drain time constant and the depth of water above the fine scale drain level.
- Total drainage in a coarse cell is the sum of all the fine scale drainage volumes.
- Drainage routing by levels will be determined by the coarse grid. However to make it more realistic with respect to the fine scale hydrology, the drainage routing by levels will be based on the lowest drain level in a coarse cell.
- Drainage to local depressions will be added to the SZ cell, and resultant ponding will then follow the multi-scale OL flow.

Disabling Multi-Cell Drainage

By default, if the multi-cell OL option is invoked, multi-cell drainage will be active. If you want to disable multi-cell drainage, perhaps for backwards compatability with older models, an Extra parameter option is availble to switch off multi-cell drainage: .

Parameter Name	Type	Value
disable multi-cell drainage	Boolean	On

If this option is used, then the multi-cell drainage is switched off and the drainage will function using the groundwater level and drain level based on the course cells.

13.2.4 Internal validation of the drainage scheme

MIKE SHE performs an internal validation of the SZ drainage scheme. The following are used in connection with the sub-scale feature:

Drainage depths of zero

Zero depths are allowed and drainage depths above the topography are set to the topography. This allows drain levels at the ground surface. This check will be done on the coarse grid. That is, if the coarse grid drain level is above the coarse grid topography, a warning will be issued and all the sub-grid drain depths will be set to zero.

Note for Release 2011 In Release 2011 and prior releases, a drain level of zero turned off SZ drainage, and drain levels above topography were set to the topography (and turned off drainage). For backwards compatibility an Extra Parameter is available.

Parameter Name	Type	Value
disable drains at or above ground	Boolean	On

Drain levels vs River link elevations

There is an optional Extra Parameter check in the drainage routing by levels that checks on the river link bottom elevation.

Parameter Name	Type	Value
check drain level against bed level	Boolean	On

If the river link bottom elevation is higher than the drain level, the cell becomes a local depression. However, this will likely create a lot of local depressions beside the rivers.

When using the multi-grid OL option, the drainage in a coarse cell is controlled by the minimum drainage level in the cell. If one sub-grid cell has a drainage level below the bed level then the drainage in the entire cell is transferred to an internal depression.

Note for Release 2011 The check was originally added to prevent the "lifting" of drainage water up to a river link. However, in most cases, such lifting is probably unintentional. That is, the river bed has been poorly interpolated.

Prior to Release 2012, this was the default behaviour and the check above has been added for backwards compatibility.

There is a check on the drain levels below the bottom of the model. If the coarse grid drain level is below the coarse grid bottom of the model, then a warning will be printed and the drain level will be adjusted to the bottom of the model. In the sub-grids, you may have the situation where the sub-grid drain level is below the bottom of the model, but the average drain level is above. In this case, the sub-grid drain level will be the maximum elevation of the bottom of the model and the drain level. Meaning if the drain level of a sub-grid is below the bottom of the model, the drain level is adjusted to the maximum value of i) the bottom of the model and ii) the drainage elevation.

13.3 MIKE SHE versus MODFLOW

The MIKE SHE can be used to simulate all of the processes in the land phase of the hydrologic cycle, including overland flow, channel flow, groundwater flow in the unsaturated zone and saturated groundwater flow. MODFLOW, on the other hand, is restricted to simulating flow only in the saturated groundwater zone. Although many of the processes simulated in MIKE SHE are used in a similar way when simulating groundwater flow with MODFLOW, they are not actually “simulated” by MODFLOW.

Let’s take groundwater recharge as an example. MODFLOW allows you to include recharge as an upper boundary condition to the groundwater model, where recharge is defined as the amount of water reaching the groundwater table after accounting for evapotranspiration, surface runoff and changing storage in the unsaturated zone. In MODFLOW, the modeller has to account for these processes herself - usually by applying a constant rule-of-thumb fraction to the measured precipitation data. In most cases, the model results are very sensitive to this fraction and since the modeller has little data on this fraction, she will assume an initial value and use this parameter as a calibration parameter. Thus, she will adjust the amount of recharge during the calibration process until the measured groundwater levels match the calculated values.

However, the fraction of precipitation reaching the groundwater table is constant in neither space nor time. The actual amount of precipitation reaching the groundwater table depends strongly on the maximum rate of infiltration, which is a characteristic of the soil and will vary spatially over the model domain. Further, since the maximum rate occurs when the soil is saturated, different amounts of water will infiltrate during wet periods compared to dry periods. To complicate matters further, the length of the preceding dry period will determine the amount of available storage in the unsaturated zone. For example, if there has been a long dry summer period, then evapotranspiration may have created a large deficit of water in the unsaturated zone that must be satisfied before any water reaches the water table.

This example shows that infiltration of precipitation is a very dynamic process. It depends on a complex interaction between precipitation, unsaturated zone soil properties and the current soil moisture content, as well as vegetation properties.

In MIKE SHE, the saturated zone is only one component of an integrated groundwater/surface water model. The saturated zone interacts with all of the other components - overland flow, unsaturated flow, channel flow, and evapotranspiration.

In comparison, MODFLOW only simulates the saturated flow. All of the other components are either ignored (e.g. overland flow) or are simple boundary conditions for the saturated zone (e.g. evapotranspiration).

On the other hand, there are very few difference between the MIKE SHE Saturated Zone numerical engine and MODFLOW. In fact, they share the same PCG solver. The differences that are present are limited to differences in the discretisation and to some differences in the way boundary conditions are defined.

Setting up the saturated zone hydraulic model involves defining the:

- the geological model,
- the vertical numerical discretisation,
- the initial conditions, and
- the boundary conditions.

In the MIKE SHE GUI, the geological model and the vertical discretisation are essentially independent, while the initial conditions are defined as a property of the numerical layer. Similarly, subsurface boundary conditions are defined based on the numerical layers, while surface boundary conditions such as wells, drains and rivers (using MIKE 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 reused for different model discretisation and different model areas.

13.3.1 Importing a MODFLOW 96 or MODFLOW 2000 Model

A FORTRAN executable is automatically installed with MIKE SHE and located in the MIKE SHE bin directory. The program can be used to read a MODFLOW file set and extract the stationary distributed data to a set of point theme shape files. The shp files can then be used directly in MIKE SHE.

To extract data from a MODFLOW model, open a command prompt in the directory containing the input files. On the command prompt line, type

```
MShe_ModflowExtraction.exe file_name.pfs
```

The extraction will proceed silently - that is without any messages. To run the extraction with the messages, you need to use

```
MZLaunch file_name.pfs -e  MShe_ModflowExtrac-
tion.exe
```

which will start the MZLaunch utility. The `file_name.pfs` variable is the input file for the MODFLOW extractor. The input file has the standard MIKEZero Pfs format. The input fields of the file are explained below. Lines starting with '//' are not read, but rather can be used as comment lines.

Table 13.1 is an example .pfs file for the MODFLOW data extractor program:

Table 13.1 MODFLOW Extraction.pfs file format and description

Line item	Comment
[MIKESHE_ModflowExtraction] FileVersion = 3	File version 3 is for Release 2009 and up
ModflowModel = 'MODFLOW-96' \\ModflowModel = 'MODFLOW-2000'	The ModflowModel variable should be changed to MODFLOW-2000, if the MODFLOW model is a MODFLOW 2000 model.
NameFileName = |.\Airport5.nam|	The NameFileName is the name of the MODFLOW name file that contains all of the references to the other input files. The '|' around the name-file name and the path of the specified file name must be relative to the location of the pfs file.
XMin = 300. YMin = 400. XMax = 3032. YMax = 1132	The minimum and maximum (X,Y) coordinates are used to determine the exact spatial coordinates of the nodal points. XMin and YMin are the UTM coordinates of lower left MODFLOW corner. Xmax and Ymax are the UTM coordinates of the upper right MODFLOW corner. See figure next page.

Table 13.1 MODFLOW Extraction.pfs file format and description

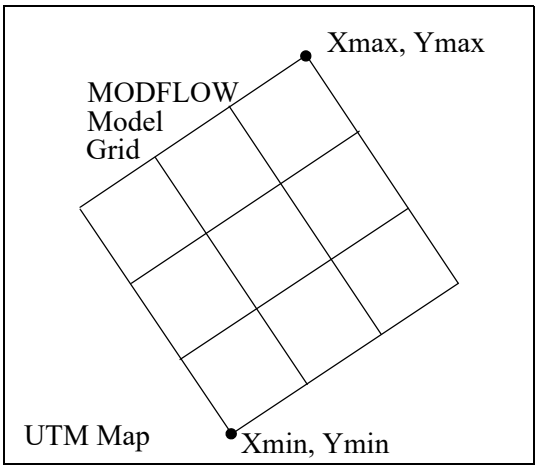
Line item	Comment
TimeUnit = 'DAYS'	The TimeUnit is not currently used, but must be input. Valid values for TimeUnit are DAYS, HOURS, MINUTES and SECONDS.
LengthUnit = 'METER'	The LengthUnit is not currently used, but must be input. Valid values for LengthUnit are METER and FEET.
StartDate = 2005,1,1,0,0	The start date and time of the MODFLOW simulation. Format: YYYY, MM, DD, HH, MM
WellExtraction = 1	Extract well data to a dfs0 file. On: Flag = 1 Off: Flag = 0
RechargeExtraction = 1	Extract recharge input to a dfs2 file. On: Flag = 1 Off: Flag = 0 Note: only works with uniform MODFLOW grids.
HeadExtraction = 1	Extract head results to a dfs2 file. On: Flag = 1 Off: Flag = 0 Note: only works with uniform MODFLOW grids
OutputFilePath = |.\MfExtractionDir|	The output path is the directory location where the output files will be written to.
Shape_Or_Dfs2_Output = 1	The parameter can have the values 1 or 2: 1 if shape file output is wanted, 2 if dfs2 output is wanted.
EndSect // MIKESHE_ModflowExtraction	

Note: MODFLOW does not have any internal unit checking. The units written in the MODFLOW file are only for display purposes. Also, the units that you define in your MODFLOW user interface may not be the same as those written to the MODFLOW files. So, you need to be careful of units and know what units the MODFLOW files are written in.

The MODFLOW name file has the usual MODFLOW format. However, you should

- Specify a new name for the LIST file not to overwrite the LIST file of an existing simulation, and
- Make copies of, or rename, all output files (lines starting with DATA). Existing result files might otherwise be overwritten during the execution of the extraction routine.

The coordinate information is the UTM coordinates of the lower left and upper right MODFLOW model corners - not the MODFLOW block-centered nodal coordinates.



These coordinates plus the DELR, DELC vectors from the MODFLOW files are used to defined the spatial location of the shape file and dfs2 output.

For a MODFLOW model, the extraction routine reads and outputs the following MODFLOW static parameters to a point theme shape file:

Top, Bot, Shead, Tran, Hy, Vcont, Sf1, and Sf2

Plus, it outputs the Specific storage, which is calculated as Sf1 divided by the layer thickness.

If the well output option is selected, a dfs0 file will be created. In this file, every cell in the MODFLOW file containing a well will have a seperate item in the dfs0 file.

If the recharge data and head results is selected, a dfs2 file will be created for each of these. However, the dfs2 format does not allow for variable grid spacing, which means that variable grid spacing will be ignored. The DELR and

DELC for the first column and row will be used as the grid spacing in the dfs2 file. Thus, the recharge and head results output option is really only useful for MODFLOW models with a uniform grid spacing.

The extraction routine outputs point theme shape files -one file per data type - with one item for each extracted layer. The shape file names reflect the MODFLOW manual naming convention (Top.shp, Vcont.shp, etc.). The points represent the centre of each grid square. The model orientation is calculated from the user-specified coordinates of lower left (origin) and upper right corner of the model.

To use the MODFLOW data in MIKE SHE, select the Point/Line .shp option for the static variable. Then browse to the appropriate .shp file. The .shp file will contain one item for each model layer in the MODFLOW model. The appropriate item is selected in the file browse dialogue. Once the file has been assigned, MIKE SHE will automatically interpolate the data to the model grid.

Internal inactive zones

Currently, it is not possible to extract the inactive zones from the MODFLOW model and convert these to inactive cells in MIKE SHE. MODFLOW and MIKE SHE treat internal inactive zones quite differently. In MIKE SHE, the internal inactive zones are simply treated as cells with a very low hydraulic conductivity, whereas, MODFLOW ignores them in the solution. Furthermore, the extraction program only writes points to the .shp file for the active nodes. Thus, when it comes to the interpolation in MIKE SHE, the interpolation does not know about the inactive zone and interpolates through the inactive zone - there are simply no data points in the inactive zones.

Errors

The extraction utility is based on MODFLOW 96 and MODFLOW 2000 source code downloaded from the USGS MODFLOW website. The entire MODFLOW input routines were copied from these codes and used directly in the extraction utility.

If you encounter errors during the extraction, then you need to evaluate the log files generated to see where the error is.

- The extraction utility generates a .log file that includes error messages related to the errors in the .pfs input file.
- The .out file is a log file generated by the core MODFLOW input rou-tine that includes errors related to reading the MODFLOW files.

The most frequent source of errors is that the MODFLOW input files are not compatible with the standard USGS MODFLOW code. These errors show up in the .out file. The extraction process will only run if the entire set of MODFLOW files is completely compatible with MODFLOW. The easiest way to test your MODFLOW input files is to try to run the standard USGS MOD-

FLOW executable from the command (DOS) line. The executable can be downloaded from the USGS website.

14 Using MIKE SHE with MIKE URBAN

Coupling MIKE URBAN and MIKE SHE allows you to simulate the effect of urban drainage and sewer systems on the surface/subsurface hydrology.

The use of the integrated MIKE SHE/MIKE URBAN system is not very different from establishing a stand-alone MIKE URBAN model and a stand-alone MIKE SHE model. In principle there are three basic set-up steps to have a coupled MIKE SHE-MIKE URBAN model:

1. Establish a MIKE URBANMIKE URBAN hydraulic model as a stand-alone model, make a performance test and, if possible, a rough calibration using prescribed inflow and boundaries.

2. Establish a MIKE SHE model that includes the overland flow component and (optionally) the saturated zone and unsaturated zone components.

3. Couple MIKE SHE and MIKE URBAN by defining the locations where MIKE URBAN should interact with MIKE SHE.

When MIKE SHE runs, it will call MIKE URBAN and ask it to perform a MIKE URBAN time step. If the end of the MIKE SHE time step has not yet been reached, MIKE SHE will ask MIKE URBAN to calculate the next MIKE URBAN time step. The MIKE URBAN model will run normally if it is launched directly from MIKE URBAN.

Note: The MIKE URBAN coupling was originally developed for the stand-alone sewer modelling product called MOUSE, which was later incorporated into MIKE URBAN. Thus, references in this chapter to MIKE URBAN can largely be substituted by “MOUSE”. Further, older MOUSE models can be coupled to MIKE SHE using the same method described here.

Important: In the command lines in the input files, the word “mouse” must still be used. For example, the Extra Parameters option to activate the MIKE URBAN coupling must be “mouse coupling”.

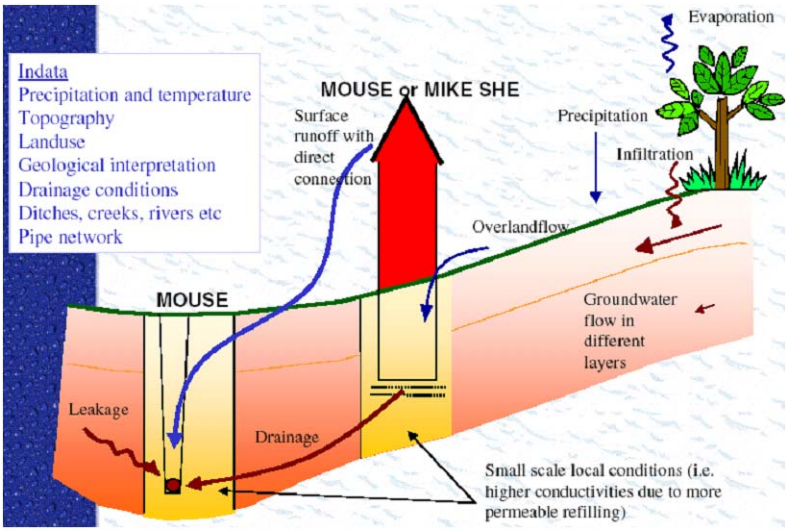


Figure 14.1 MIKE SHE to MIKE URBAN coupling linkages

The exchange between MIKE URBAN and MIKE SHE is calculated based on the following equation

$$Q = C \cdot (H_{SHE} - H_{MOUSE})^{k} \tag{14.1}$$

where Q is the exchange between MIKE URBAN and MIKE SHE, C is the exchange coefficient, k is a head difference exponent and

$$H_{SHE} = Max(H_{cell}, Z_T, Z_M) \tag{14.2}$$

$$H_{MOUSE} = Max(H_{pipe}, Z_T, Z_M) \tag{14.3}$$

where H_{cell} is the head in the MIKE SHE cell, H_{pipe} is the head in the MIKE URBAN pipe, Z_T is the topographic elevation in the cell and Z_M is the elevation of the manhole.

There are five variations on how to calculate the exchange based on above equations:

MIKE SHE SZ to MIKE URBAN LINKS

This is a leakage-based solution in which the head difference exponent is 1 and the exchange coefficient in Equation (14.1) for the flow to or from the pipe is calculated by

$$C = C_L \cdot P_w \cdot L \tag{14.4}$$

where C_L is the leakage coefficient (see below), P_w is the wetted perimeter for the flow (see below), and L is the length of the MIKE URBAN pipe (link) in the MIKE SHE cell.

Leakage Coefficient - The leakage coefficient can be defined in two ways.

Option 1 is the simple method, which is to use the pipe leakage coefficient specified in the MIKE URBAN .ADP file. See Telling MIKE URBAN that it is coupled to a MIKE SHE model *(V2 p. 243)*.

Option 2 uses a combination of the pipe leakage coefficient and the aquifer hydraulic conductivity. In this case, the leakage coefficient is calculated as a series connection of the pipe leakage coefficient (C_p) and the "average" leakage coefficient of the aquifer grid cell (C_{aq}). The average leakage coefficient of the grid cell is calculated assuming that the exchange of water between the pipe and the grid cell is both vertical and horizontal. The leakage coefficient calculation does not calculate a detailed flow path based on a geometric calculation, since a MIKE URBAN pipe can be located anywhere in a grid cell. Instead, an average vertical and horizontal flow distance is used based on 1/4 of the vertical and horizontal cell dimensions. Thus,

$$C_{aq} = C_{aqH} + C_{aqV} = \frac{K_x}{(\Delta x)/4} + \frac{K_z}{(\Delta z)/4} \tag{14.5}$$

where K_x and K_z are the horizontal and vertical hydraulic conductivities respectively and Δx and Δz are the horizontal and vertical cell dimensions.

The final leakage coefficient is then calculated as the harmonic mean of both the aquifer leakage coefficient and the pipe leakage coefficient:

$$\frac{1}{C_L} = \frac{1}{C_{aq}} + \frac{1}{C_p} \tag{14.6}$$

Wetted Perimeter - MIKE SHE uses the inner wetted perimeter if the flow is from MIKE URBAN to MIKE SHE. Whereas, it uses the outer wetted perimeter if the flow is from MIKE SHE to MIKE URBAN. The wetted perimeters are calculated by MIKE URBAN.

MIKE SHE Overland flow to MIKE URBAN LINKS

If a MIKE URBAN link is defined as link type *CRS* or *Natural Channel* and has a cross section which is "open", then MIKE SHE can exchange overland flow with it in both directions. In this case, the exchange coefficient in Equation (14.1) is defined as

$$C = C_L \cdot L \tag{14.7}$$

where C_L is the conductance and L is the length of the MIKE URBAN pipe (link) in the MIKE SHE cell.

If the exponent Equation (14.1) is 1.0, then this is a simple drain formulation and the conductance is per length with units of [m/s]. If the exponent is 1.5, then this is a weir formulation and the units of the conductance term are [$m^{1/2}$/s].

MIKE SHE Overland flow to MIKE URBAN Manholes

If the MIKE URBAN manholes are not sealed, then MIKE SHE can discharge overland flow into the MIKE URBAN manholes. In this case, the exchange coefficient in Equation (14.1) is defined as

$$C = C_L \tag{14.8}$$

where C_L is the conductance.

If the exponent Equation (14.1) is 1.0, then this is a simple drain formulation and the conductance, C_L, is per length with units of [m/s]. If the exponent is 1.5, then this is a weir formulation and the units of the conductance term are [$m^{1/2}$/s].

MIKE SHE SZ drain flow to MIKE URBAN Manholes

If drain flow is specified in MIKE SHE, then the drainage can be discharged to a MIKE URBAN manhole. The flow in the drain is calculated by MIKE SHE based on the groundwater height above the drain level. In MIKE SHE the distributed drainage option must be chosen (see Drainage *(V1 p. 315)*) and the cells that drain to a manhole must have an option value of 4 (see Option Distribution *(V1 p. 321)*). The references between the MIKE SHE drain codes and the MIKE URBAN manholes are defined in the *MsheMouse.pfs* file (see Creating a MsheMouse.pfs file *(V2 p. 244)*).

MIKE SHE Ponded Drainage to MIKE URBAN Manholes

If the Ponded Drainage option is used in MIKE SHE, then the ponded water can be discharged to a MIKE URBAN manhole. MIKE SHE's ponded drainage module uses a user-defined source-destination reference system.

MIKE URBAN Outlets to MIKE SHE

MIKE URBAN outlets cannot directly discharge to MIKE SHE's overland flow. To work around this, you can add a dummy manhole to your MIKE URBAN pipe and then couple the pipe to the outlet via a small diameter dummy pipe (See Figure 14.2). This will force most of the water out of the manhole and into MIKE SHE's overland flow. Downside of this method, is that the head loss at the outlet is over estimated, because the discharge velocity is zero at a manhole.

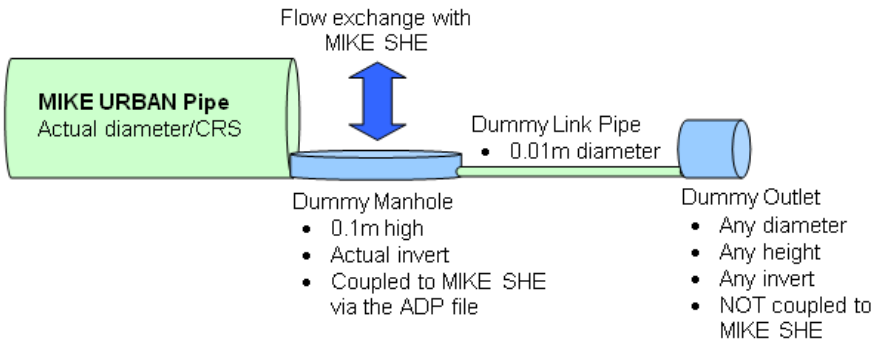


Figure 14.2 Work around for discharging MIKE URBAN outlets to MIKE SHE

14.1 Coupling MIKE SHE and MIKE URBAN

The MIKE URBAN coupling in MIKE SHE has not yet been added to the MIKE SHE user interface. Thus, to couple the models together, you must:

1. tell MIKE SHE to look for a MIKE URBAN model,
2. tell MIKE URBAN that it is coupled to a MIKE SHE model
3. create an *MsheMouse.pfs* file to define where and how the two models are coupled.

14.1.1 Telling MIKE SHE to couple to MIKE URBAN

To tell MIKE SHE that it needs to couple to a MIKE URBAN model, you must add the following two items in the Extra Parameters *(V1 p. 334)* section of the MIKE SHE Setup Editor.

Parameter Name	Type	Value
mouse coupling	Boolean	On
mouse coupling file	file name	the file name of the MIKE URBAN coupling .pfs input file

Note: The parameter names must be spelled exactly as shown, that is “mouse”. For more information see Extra Parameters *(V2 p. 325)*.

14.1.2 Telling MIKE URBAN that it is coupled to a MIKE SHE model

To couple a MIKE URBAN model to MIKE SHE, MIKE URBAN must be supplied with some extra information. This information is found in MIKE URBAN’s .ADP file.

Note: For more information on .ADP files, see the MIKE URBAN manual.

.

Line item	Comment
[MOUSE_COUPLING] SYNTAX_VERSION = 1 UNIT_TYPE = 1 CALLER = 'MSHE'	
// LineHeader = 'ID', 'LinkType', 'C ','OLExp', 'SzLeakageCoef'	Comment line for headers
COUPLINGMMSHE= 'NODE1', 1, 0.001, 2, ,	One line for each coupling item: ID = Link name LinkType = 1 for node; 2 for link C = conductance for Overland flow to MIKE URBAN, units depend on OLExp and whether it is a pipe or a manhole SzLeakageCoeff = leakage coefficient; needed only when the saturated zone is coupled to a link
COUPLINGMMSHE= 'LINK1', 2, 0.001, 2, 0.2	
[Endsect]	

14.1.3 Creating a MsheMouse.pfs file

The MsheMouse.pfs file is an ASCII file that includes all of the specifications for the coupling. The following table defines the structure of the file, along with some information on the parameters. When the MIKE URBAN coupling has been added to the user interface, the creation of this file will be automatic.

Note: The pfs format must be adhered to exactly. There is a small utility (pfsEditor.exe) in the installation \bin directly that you can use for editing and testing pfs files that you create.

Table 14.1 MsheMouse.pfs file format and description

Line item	Comment
[MIKESHE_MOUSE_Specifications] FileVersion = 2	
Link_SZ_Exchange_Option = 2	1 = Leakage coefficient based only on MIKE URBAN pipe leakage coefficient 2 = Leakage coefficient based on a series connection of the MIKE URBAN pipe leakage coefficient and the MIKE SHE aquifer properties
Mouse_MPR_file name = |.\MOUSE_NAS-SJO\handskeryd.mpr|	(**See Note below this table**) Name of the MOUSE .mpr file or the MIKE URBAN .mex file. The MIKEZero file name format (| |) indicates that the file name is relative to the location of this document.
SZ_Coupling = 1	1 or 0 to include/exclude SZ<->MIKE URBAN coupling
OL_Coupling = 1	1 or 0 to include/exclude Overland<->MIKE URBAN coupling
Dynamic_Coupling = 1	1 for dynamic coupling. Otherwise the initial MIKE URBAN conditions will be used.
Drainage_To_Manholes = 1	1 to include SZ (and paved area) drain to manholes. In this case the SZ drain option must be Levels and Codes (should rather be named Distributed Option). In the areas with drain to MIKE URBAN the Distributed option code must be 4. For each drain code value found in areas with Distributed code 4 a reference from the code to a MIKE URBAN manhole must be defined in the Drainage_Manholes section (see below).

Table 14.1 MsheMouse.pfs file format and description

Line item	Comment
Smooth_SZ_Inflow = 1 Smooth_OL_Inflow = 1	Ensures a more smooth calculation of flows to MIKE URBAN when the MIKE SHE time steps are large compared to the MIKE URBAN time step. The MIKE URBAN coupling is only made at every integer multiple of the MIKE SHE time step. If the Smooth option is not activated, the flows to MIKE URBAN can stop after a number of MIKE URBAN time steps because the calculated flow volume exceeds the volume of the MIKE SHE SZ/Overland grid cells. The Smooth option tries to use a reduced flow rate which equals the available volume / coupling time.
[Dynamic_Coupling_Specifications]	
Limit_Inflow = 0:	Specify 1 if the inflow to MIKE URBAN should be limited so the MIKE URBAN volume + inflow does not exceed a specified fraction of the maximum MIKE URBAN volume. This is used to avoid instabilities due to high pressure.
Limit_Outflow = 0:	Specify 1 if the outflow from MIKE URBAN should be limited so the MIKE URBAN volume - outflow doesn't come below a specified fraction of the maximum MIKE URBAN volume. This is used to avoid instabilities due to drying / negative volume.
[Inflow_Limitations] MaxVolFac_Links = 0.99 MaxVolFac_Manholes = 0.99 EndSect // Inflow_Limitations [Outflow_Limitations] MinVolFac_Links = 0.05 MinVolFac_Manholes = 0.05 EndSect // Outflow_Limitations	The inflow and outflow fractions are specified here:
EndSect // Dynamic_Coupling_Specifications	

Table 14.1 MsheMouse.pfs file format and description

Line item	Comment
No_Of_Storing_reaches = 2 [Storing_Reaches] [Storing_Reach_1] No_Of_Links = 2 LinkName_1 = 'Dike_01l1' LinkName_2 = 'Dike_03l1' EndSect // Storing_Reach_1 [Storing_Reach_2] No_Of_Links = 1 LinkName_1 = 'Dike_04l1' EndSect // Storing_Reach_2 EndSect // Storing_Reaches	When No_Of_Storing_reaches is greater than 0, the [Storing_Reaches] section must be specified, and inside this the [Storing_Reach_1], [Storing_Reach_2], ... defining the no. of links and link names for each reach.
[Drainage_Manholes] No_Of_DrainCodes = 8 [Draincode_1] Draincode= 12 ManholeName='DNB3182' Endsect // Draincode_1 . . Endsect // Draincode_8 EndSect // Drainage_Manholes	When No_Of_Storing_reaches is greater than 0, the [Storing_Reaches] section must be specified, and inside this the [Storing_Reach_1], [Storing_Reach_2], ... defining the no. of links and link names for each reach.
EndSect // MIKESHE_MOUSE_Specifications	

Note on file names:

The pfs file line item is always “Mouse_MPR_file name =”

When coupling MIKE SHE to an old MOUSE model, the MOUSE file name has the extension “.mpr”.

When coupling MIKE SHE to MIKE URBAN, the equivalent file is the “.mex” file. This file contains all the necessary information for the coupling and is generated automatically by MIKE URBAN.

To create the .mex file, you must start a sewer simulation from MIKE URBAN. However, since the .mex file is only created when the simulation is launched, if you make changes to the sewer network, then you must re-create the .mex file by first restarting the sewer simulation in MIKE URBAN. Otherwise, your changes to the sewer network will not be reflected in the coupled models.

14.1.4 Output Files

Output from the coupled run is written to a number of .dfs0 results files- all located in the standard results directory. In the case of storing reaches, there is one item in the .dfs0 file for each storing reach.

Table 14.2 File names and conditions for output for the MIKE SHE-MIKE URBAN coupling. 'setupname' refers to the name of the model setup file

file name	The file is created when...
.*setupname**setupname*_SZ2MouseReaches.dfs0	...the MIKE SHE SZ coupling is included.
.*setupname**setupname*_OL2MouseReaches.dfs0	...the MIKE SHE Overland coupling is included.
.*setupname**setupname*_OL2MouseManholes.dfs0	...the MIKE SHE Overland flow coupling to manholes is included.
.*setupname**setupname*_SZDrain2MouseManholes.dfs0	...the MIKE SHE SZ drain coupling to manholes is included.
.*setupname**setupname*_PavedDrain2MouseManholes.dfs0	...the MIKE SHE SZ paved areas to manholes is included.

14.2 Warning Messages

Exchange inflows reduced

```
Warning: Exchange inflows from Overland to MOUSE reduced
by Overland house-keeping in order to avoid instabilities

No. of time steps: 27000 of 27000

Total a priori inflows: 1332286 m3

Total reduced inflows:  920643.0 m3 (69.10%)
```

MIKE SHE calculates tine in/out flows after an overland time step and feeds them to MIKE URBAN for one or more MIKE URBAN time steps. The calculations of these flows are not included in the implicit overland flow solver. Thus, the "Total a priori flows" are the rough inflows calculated using Equation (14.1). However, to prevent water balance errors, MIKE SHE checks the volume of water available in the grid cell. If the volume is insufficient, then the inflow is reduced to the available amount. The final value of inflows is the "Total reduced inflows". Note though that the total NET inflow to MIKE URBAN will be less than this value if the flow goes from MIKE URBAN to MIKE SHE in other grid cells or other time steps.

Ideally, the Total reduced inflow should be 100%, but in practice this is rarely achieved.

14.3 Water Balance Limitations

The interaction with MIKE SHE is not included in the MIKE URBAN Summary HTM file. Thus, the water added from MIKE SHE appears as an error (i.e. 6: Continuity balance in MIKE Urban).

REFERENCE MANUAL FOR WATER QUALITY

15 Water Quality Overview

This section includes detailed descriptions of the numeric engines used for moving solutes and particles in MIKE SHE, including

- Advection Dispersion - Reference *(V2 p. 255)*
- Reactive Transport - Reference *(V2 p. 281)*
- Particle Tracking-Reference *(V2 p. 319)*

Also included in this section is detailed information on how to effectively use the Water Quality options in MIKE SHE, including

- Working with Solute Transport - User Guide *(V2 p. 297)*
- Working with MIKE ECO Lab in MIKE SHE - User Guide *(V2 p. 307)*

16 Advection Dispersion - Reference

16.1 Simulation control

In the MIKE SHE water quality module, you can calculate solute transport in the different parts of the hydrological cycle. In the present version of MIKE SHE AD only three combinations are legal:

- groundwater transport can run as a stand-alone module,
- groundwater transport can be run in combination with the overland transport module, and
- all modules in combination can run.

Thus, a simulation with only overland or only the unsaturated zone is not possible and that combinations of the unsaturated zone with only the overland or groundwater component are not possible.

16.1.1 Flow Storing Requirements

The transport calculations are based on the water flow, water contents, hydraulic heads and water levels calculated in a MIKE SHE water movement simulation. Depending on the complexity of the advection/dispersion simulation, the water movement output must be stored with different storing time steps. The selected storing frequency should be sufficient to reflect the dynamics of the flow processes. However, the following two restrictions must be observed:

- The SZ head and SZ flow storing time steps must be equal, and
- The SZ storing time step must be an integer multiple of the UZ storing time step, which must be an integer multiple of the overland storing time step.

The last restriction above is controlled in the user interface.

16.1.2 Internal Boundary Conditions

If a simulation with MIKE SHE AD includes more than one part of the hydrological cycle, the solute fluxes between the different hydrologic components must be kept track of. In principle, the solute fluxes between the components follow the water flow between the components. Multiplying the flow rate with the solute concentration produces a source/sink term for the relevant components. Table 16.1 lists the solute exchange possibilities between the compo-

nents, in particular when one or more component is not included in the flow simulation.

Table 16.1 Internal boundary source/sinks between hydrologic components in MIKE SHE AD

Solutes from:	Primary solute sink:	Alternative solute sink (if primary sink unavailable):
Precipitation Fluxes	Overland flow	Unsaturated zone or Groundwater
Overland Fluxes Infiltration Overland flow	 Unsaturated zone MIKE Hydro River	 Groundwater External boundary (none)
Unsaturated Zone Fluxes Infiltration Bypass flow	Groundwater	External Boundary
Groundwater Fluxes SZ Drainage Upward flux to overland Upward flux to UZ Baseflow to streams	 MIKE Hydro River Overland flow Unsaturated Zone MIKE Hydro River	 Overland flow or External boundary (none) Overland flow (none)
MIKE Hydro River Baseflow to groundwater	 Groundwater	 (none)

An sketch of the different internal boundary conditions is shown in Figure 16.1. Each of these exchanges is detailed in the respective sub-section for each hydrologic component.

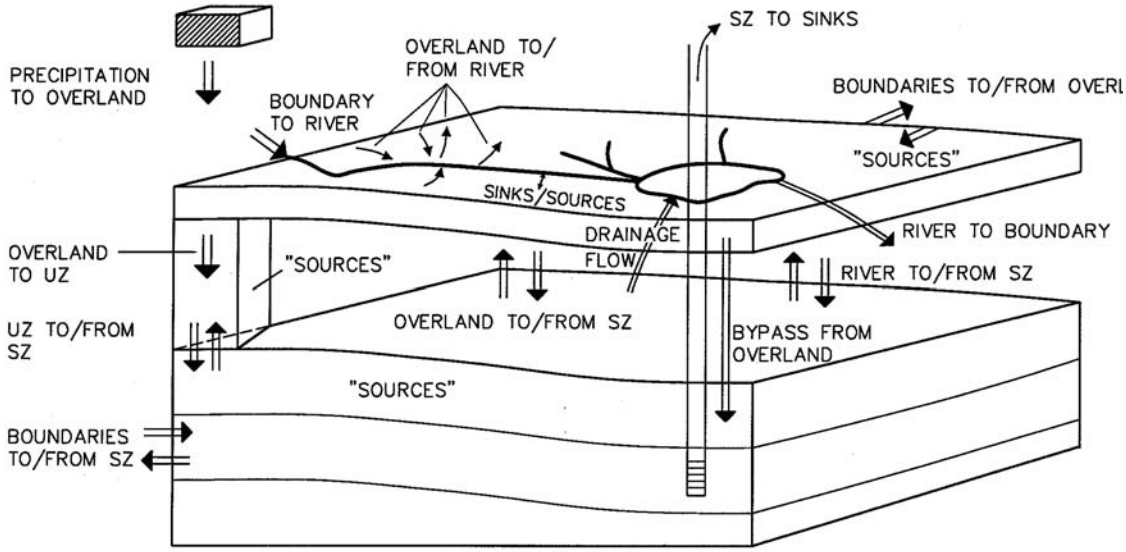


Figure 16.1 Outline of the different transport possibilities between components and boundaries.

16.1.3 Time step calculations

The time scales of the various transport processes are different. For example, the transport of solutes in a river is much faster than transport in the groundwater. The optimal time step is different for each component, where 'optimal' can be defined as the largest possible time step without introducing numerical errors. In addition, the optimal time step varies in time as a consequence of changing conditions in the hydrological regime within the catchment.

Different time steps are allowed for the different components. However, an explicit solution method is used, which sometimes requires very small time steps to avoid numerical errors. The Courant and Peclet numbers play an important role in the determination of the optimal time step.

The user can specify the maximum time step in each of the components. However, the actual simulation time step is controlled by the stability criterions, with respect to advective and dispersive transport, as well as the timing of the sources and sinks, and the simulation and storing time steps in the WM simulation.

In Figure 16.2, you can see an example of the sequence of calling each components in the MIKE SHE advection-dispersion module. The time step in the river transport calculation is usually the smallest, whereas time step for groundwater transport is always the largest.

A transport simulation begins with the overland component followed by MIKE Hydro River, the unsaturated zone component and the groundwater component. Figure 16.2 shows how the simulation time steps can be controlled solely by the storing time step in the flow simulation.

Solute sources and the storing of data in the different components influences the time step. For example, in Figure 16.2, a SZ source requires that all components have a break when the source starts.

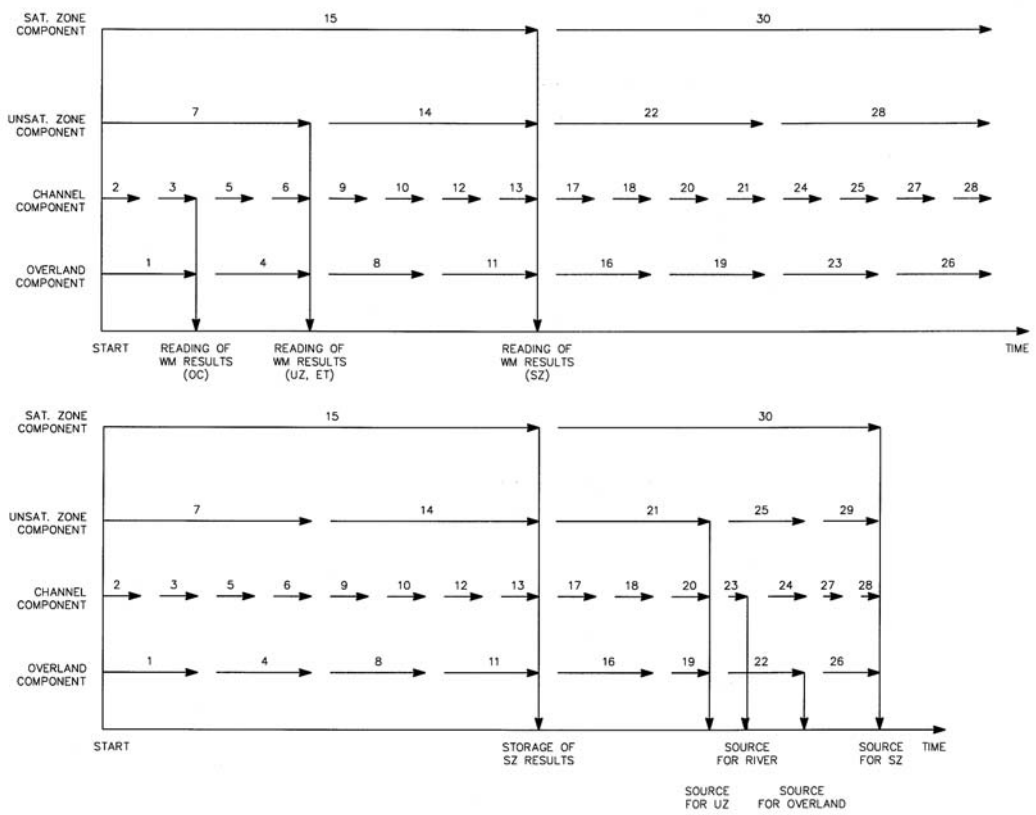


Figure 16.2 An example calculation sequence for solute transport in MIKE SHE.

Time step limitations

For each component the maximum allowable time step is determined by the advective and dispersive Courant number.

The advective Courant number in the *x*-direction, σ_x, is defined as:

$$\sigma_x = \frac{v_x \Delta t}{\Delta x} \tag{16.1}$$

and the dispersive Courant number, Γ_x, is defined as:

$$\Gamma = \frac{D_{11}}{\Delta x^2}\Delta t \tag{16.2}$$

The limitations are different in each flow component and will be described in more detail under the respective flow component sections.

You can also specify a maximum time step for each component, as well as a limiting solute flux per time step.

16.2 Solute Transport in the Saturated Zone

The solute transport module for the saturated zone in MIKE SHE allows you to calculate transport in 3D, 2D, layer 2D, or even 1D. However, the transport formulation is controlled by the water movement discretisation. If the vertical discretisation is uniform (except for the top and bottom layer) the transport scheme is described in a fully three-dimensional numerical formulation. If the numerical layers have different thicknesses a multi-layered 2D approach is used, where each layer exchanges flows with other layers as sources and sinks. If you specify a 1D or 2D flow simulation, the transport formulation is further simplified.

Temporal and spatial variations of the solute concentration in the soil matrix are described mathematically by the advection-dispersion equation and solved numerically by an explicit, third-order accurate solution scheme.

The forcing function for advective transport is the cell-by-cell groundwater flow, as well as groundwater head, boundary, drain and exchange flows, which are all read from the WM results files. Solute exchange between the other hydrologic components is generally simulated by means of explicit sources and sinks.

16.2.1 Governing Equations

The transport of solutes in the saturated zone is governed by the advection-dispersion equation, which for a porous medium with uniform porosity is

$$\frac{\partial c}{\partial t} = -\frac{\partial}{\partial x_i}(cv_i) + \frac{\partial}{\partial x_i}\left(D_{ij}\frac{\partial c}{\partial x_j}\right) + R_c \quad i,j = 1,2,3 \tag{16.3}$$

where c is the concentration of the solute, Rc is the sum of the sources and sinks, D_{ij} is the dispersion coefficient tensor and v_i is the velocity tensor.

The advective transport is determined by the water fluxes (Darcy velocities) calculated during a MIKE SHE WM simulation. To determine the groundwater velocity, the Darcy velocity is divided by the effective porosity

$$v_i = \frac{q_i}{\theta} \tag{16.4}$$

where q_i is the Darcy velocity vector and θ is the effective porosity of the medium.

The mathematical formulation of the dispersion of the solutes follows the traditional formulations generalised to three dimensions. This formula was developed assuming that the dispersion coefficient is a linear function of the mean velocity of the solutes. In the three-dimensional case of arbitrary flow-direction in an anisotropic aquifer, the dispersion tensor, D_{ij}, contains nine elements, giving a total of 36 dispersivities. The general formulation of the dispersion tensor is derived in Scheidegger (1961) and can be written as

$$D_{ij} = a_{ijmn}\frac{v_n v_m}{U} \tag{16.5}$$

where a_{ijmn} is the dispersivity of the porous medium (a fourth order tensor), v_n and v_m are the velocity components, and U is the magnitude of the velocity vector.

The derivation of D_{ij} and a_{ijmn} in MIKE SHE follows the work of Bear and Verruijt (1987). Two simplifications have been introduced with respect to dispersivity

- isotropy, and
- anisotropy with axial symmetry around the z-axis.

These simplifications are reflected in the number of non-zero dispersivities to be specified. Under isotropic conditions the dispersivity tensor, a_{ijmn}, solely depends on the longitudinal dispersivity, α_L, and the transversal dispersivity, α_T, in the following manner:

$$a_{ijmn} = \alpha_T \delta_{ij} \delta_{mn} + \frac{\alpha_L - \alpha_T}{2}\left(\delta_{im}\delta_{jn} + \delta_{in}\delta_{jm}\right) \tag{16.6}$$

where δ_{ij} is the Kronecker delta (with δ_{ij}=0 for i≠j and δij=1 for i=j). In the Cartesian co-ordinate system applied in MIKE SHE, the velocity components in

the coordinate directions are denoted V_x, V_y and V_z. Thus, we obtain the following expressions for the dispersion coefficients:

$$
\begin{aligned}
D_{xx} &= \left[\alpha_T\left(V_y^2+V_z^2\right)+\alpha_L V_x^2\right]/U \\
D_{yy} &= \left[\alpha_T\left(V_x^2+V_z^2\right)+\alpha_L V_y^2\right]/U \\
D_{zz} &= \left[\alpha_T\left(V_x^2+V_y^2\right)+\alpha_L V_z^2\right]/U \\
D_{xy} &= \left(\alpha_L-\alpha_T\right)V_x V_y/U = D_{yx} \\
D_{xz} &= \left(\alpha_L-\alpha_T\right)V_x V_z/U = D_{zx} \\
D_{yz} &= \left(\alpha_L-\alpha_T\right)V_y V_z/U = D_{zy}
\end{aligned}
\qquad (16.7)
$$

This is the general equation for the dispersion coefficients in an isotropic medium for an arbitrary mean flow direction. If the mean flow direction coincides with one of the axis of the Cartesian coordinate system the expression for the dispersion coefficients simplifies even further (e.g. if V_y and V_z are equal to zero then D_{xy}, D_{xz} and D_{yz} will also be zero).

Under fully anisotropic conditions, the dispersion coefficients depend on 36 dispersivities which is impractical to handle and estimate in practice. Thus, if we assume that the porous medium is symmetric around one of the axis, the number of non-zero dispersivities can be limited to five. This assumption is true if the medium is made up of layers normal to the axis of symmetry, which is the case for some geological deposits. Under these conditions the following expression for the a_{ijmn} terms (Bear and Verruijt (1987)) have been derived:

$$
\begin{aligned}
a_{ijmn} = {} & a_I\,\delta_{ij}\,\delta_{mn} \\
& + a_{II}\left(\delta_{im}\,\delta_{jn} + \delta_{in}\,\delta_{jm}\right) \\
& + a_{III}\left(\delta_{ij}\,h_m\,h_n + \delta_{mn}\,h_i\,h_j\right) \\
& + a_{IV}\left(\delta_{im}\,h_j\,h_n + \delta_{jm}\,h_i\,h_n + \delta_{in}\,h_j\,h_m + \delta_{jn}\,h_i\,h_m\right) \\
& + a_V\,h_i\,h_j\,h_m\,h_n
\end{aligned}
\qquad (16.8)
$$

where a_I, a_{II}, a_{III}, a_{IV} and a_V are independent parameters and h is a unit vector directed along the axis of symmetry. In MIKE SHE AD it is assumed that the axis of symmetry always coincides with the z-axis and *h* becomes equal to (0,0,1). Five dispersivities are then introduced

- α_{LHH} - the longitudinal dispersivity in the horizontal direction for horizontal flow
- α_{THH} - the transversal dispersivity in the horizontal direction for horizontal flow

- α_{LVV} - the longitudinal dispersivity in the vertical direction for vertical flow
- α_{TVH} - the transversal dispersivity in the vertical direction for horizontal flow
- α_{THV} - the transversal dispersivity in the horizontal direction for vertical flow

Thus, the dispersion coefficients can be written explicitly by combining Eq. (16.5) and Eq. (16.8) as follows:

$$
\begin{aligned}
D_{xx} &= \alpha_{LHH}\frac{V_x^2}{U} + \alpha_{THH}\frac{V_y^2}{U} + \alpha_{THV}\frac{V_z^2}{U} \\
D_{yy} &= \alpha_{THH}\frac{V_x^2}{U} + \alpha_{LHH}\frac{V_y^2}{U} + \alpha_{THV}\frac{V_z^2}{U} \\
D_{zz} &= \alpha_{TVH}\frac{V_x^2}{U} + \alpha_{TVH}\frac{V_y^2}{U} + \alpha_{LVV}\frac{V_z^2}{U} \\
D_{xy} &= (\alpha_{LHH} - \alpha_{THH})\frac{V_x V_z}{U} \\
D_{xz} &= \left(\frac{\alpha_{LVV}+\alpha_{LHH}}{2} - \frac{\alpha_{TVH}+\alpha_{THV}}{2}\right)\frac{V_x V_y}{U} \\
D_{yz} &= \left(\frac{\alpha_{LVV}+\alpha_{LHH}}{2} - \frac{\alpha_{TVH}+\alpha_{THV}}{2}\right)\frac{V_y V_z}{U}
\end{aligned}
\tag{16.9}
$$

and for symmetrical reasons $D_{xy} = D_{yx}$, $D_{xz} = D_{zx}$ and $D_{yz} = D_{zy}$.

Note that Eq. (16.9) can simplify to Eq. (16.7) if $\alpha_{LHH}=\alpha_{LVV}=\alpha_L$ and $\alpha_{THH}=\alpha_{TVH}=\alpha_{THV}=\alpha_T$

Burnett and Frind /9/ suggest that the dispersion should at least allow for the use of two transverse dispersivities - a horizontal transverse dispersivity and a vertical transverse dispersivity - to describe the difference in transverse spreading which is greater in the horizontal plane than in the vertical plane. In comparison, MIKE SHE uses all five dispersivities.

The determination of the five dispersivities is always difficult so often one has to rely on experience or on empirically derived values.

The dispersion term in the advection-dispersion equation accounts for the spreading of solutes that is not accounted for by the simulated mean flow velocities (the advection). Therefore, it is obvious that the more accurate you describe the spatial variability in the hydrogeologic regime and if the grid is sufficiently fine (i.e. the variations in the advective velocity) the smaller the dispersivities you need to apply in the model. Recent laboratory and field research have shown a relationship between the spatial variability of hydrogeologic parameters and the dispersivities. However, it is still difficult to

obtain sufficient knowledge about the spatial variability of, for example the hydraulic conductivity, to determine macro dispersivities applicable for solute transport models.

16.2.2 Solution Scheme

Regular Grid

The numerical solution to the advection-dispersion equation in MIKE SHE AD is based on the QUICKEST method. Leonard /36/ originally introduced this method, which was further developed by Vested et. al. /51/. It is a fully explicit scheme, which applies upstream differencing for the advection term and central differencing for the dispersion term. The equations are developed to third order and the scheme is mass conservative.

When the vertical discretisation is defined in a regular grid with uniform thickness of all layers except the upper and the lower ones the numerical scheme follows the fully three-dimensional formulation below.

Neglecting the dispersion terms and the source/sink term and assuming that the flow field satisfies the equation of continuity and varies uniformly within a grid cell the advection-dispersion equation may be written in mass conservation form as:

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial x}\left(v_x c\right) + \frac{\partial}{\partial y}\left(v_y c\right) + \frac{\partial}{\partial z}\left(v_z c\right) = 0 \tag{16.10}$$

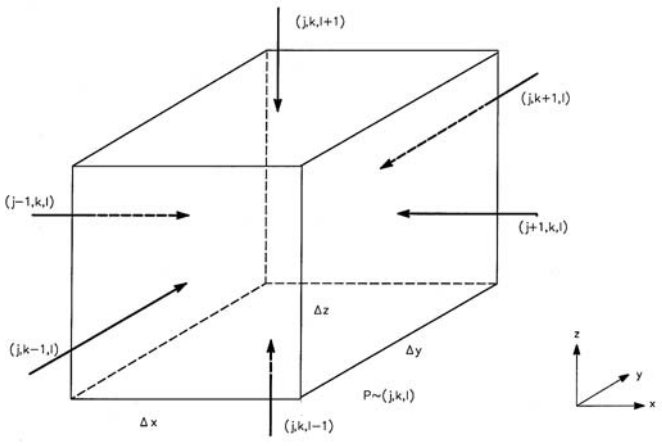


Figure 16.3 Control volume defining an internal SZ grid.

For the control volume shown in Figure 16.3 this equation is written in finite difference form as:

$$c_{j,k,l}^{n+1} - c_{j,k,l}^{n} + \sigma_x \left(c^{*}_{j+\gamma_x,k,l} - c^{*}_{j-\gamma_x,k,l} \right) + \sigma_y \left(c^{*}_{j,k+\gamma_y,l} - c^{*}_{j,k-\gamma_y,l} \right) + \sigma_z \left(c^{*}_{j,k,l+\gamma_z} - c^{*}_{j,k,l-\gamma_z} \right) = 0 \qquad (16.11)$$

where *n* denotes the time index.

In Eq. (16.11) σ_x, σ_y and σ_z are the directional Courant numbers defined by

$$\sigma_x = \frac{V_x \, \Delta t}{\Delta x}, \quad \sigma_y = \frac{V_y \, \Delta t}{\Delta y}, \quad \sigma_z = \frac{V_z \, \Delta t}{\Delta z} \qquad (16.12)$$

and the *c**-terms are the concentrations at the surface of the control volume at time *n*. As these terms are not located at nodal points they have to be interpolated from known concentration values:

$$c^{*}_{j+\gamma_x,k,l} = \sum \delta_i c_i$$
$$c^{*}_{j,k+\gamma_y,l} = \sum \gamma_i c_i$$
$$c^{*}_{j,k,l+\gamma_z} = \sum \beta_i c_i \qquad (16.13)$$

The concentration c_i is the concentration around the actual point, for example (*j-1,k-1,l*) and the weights δ_i, γ_i and β_i are determined in such a way that the scheme becomes third-order accurate. The determination of the weights is demonstrated in Vested et al. (1992) and their values are listed in Table 16.2.

A number of 8 weights has proven to be an adequate choice and their location for the determination of the $c^{*}_{j+½,k,l}$ is shown in Figure 16.4. The other "boundary" concentrations are found in a similar way

.

Table 16.2 Weight functions for advective transport

L_i	K_i	J_i
1	$\sigma_x\left(\frac{1}{6}\sigma_x^2-\frac{1}{2}\sigma_x+\frac{1}{3}\right)$	$\sigma_y\left(\frac{1}{6}\sigma_y^2-\frac{1}{2}\sigma_y+\frac{1}{3}\right)$	$:\sigma_z\left(\frac{1}{6}\sigma_z^2-\frac{1}{2}\sigma_z+\frac{1}{3}\right)$
2	$\sigma_x-\Sigma\delta_i, i=2$	$\sigma_y-\Sigma\gamma_i, i=2$	$\sigma_z-\Sigma\beta_i, i=2$
3	$\sigma_x\left(-\frac{1}{6}+\frac{1}{6}\sigma_x^2\right)$	$\sigma_y\left(-\frac{1}{6}+\frac{1}{6}\sigma_y^2\right)$	$\sigma_z\left(-\frac{1}{6}+\frac{1}{6}\sigma_z^2\right)$
4	$\sigma_x\left(-\frac{1}{2}\sigma_y+\frac{1}{2}\sigma_y^2\right)$	$\sigma_y\left(-\frac{1}{2}\sigma_x+\frac{1}{2}\sigma_x^2\right)$	$\sigma_z\left(-\frac{1}{2}\sigma_x+\frac{1}{2}\sigma_x^2\right)$
5	$\sigma_x\left(\frac{1}{2}\sigma_x\sigma_y-\frac{1}{3}\sigma_y\sigma_z\right)$	$\sigma_y\left(\frac{1}{2}\sigma_x\sigma_y-\frac{1}{3}\sigma_x\sigma_z\right)$	$\sigma_z\left(\frac{1}{2}\sigma_x\sigma_z-\frac{1}{3}\sigma_x\sigma_y\right)$
6	$\sigma_x\left(-\frac{1}{2}\sigma_z+\frac{1}{2}\sigma_z^2\right)$	$\sigma_y\left(-\frac{1}{2}\sigma_z+\frac{1}{2}\sigma_z^2\right)$	$\sigma_z\left(-\frac{1}{2}\sigma_y+\frac{1}{2}\sigma_y^2\right)$
7	$\sigma_x\left(\frac{1}{2}\sigma_x\sigma_z-\frac{1}{3}\sigma_y\sigma_z\right)$	$\sigma_y\left(\frac{1}{2}\sigma_y\sigma_z-\frac{1}{3}\sigma_x\sigma_z\right)$	$\sigma_z\left(\frac{1}{2}\sigma_y\sigma_z-\frac{1}{3}\sigma_x\sigma_y\right)$
8	$\sigma_x\left(\frac{1}{3}\sigma_y\sigma_z\right)$	$\sigma_y\left(\frac{1}{3}\sigma_x\sigma_z\right)$	$\sigma_z\left(\frac{1}{3}\sigma_x\sigma_y\right)$

The locations of the weights are determined by the points that enter into the discretisation and because the scheme is upstream centred the weights are positioned relative to the actual direction of the flow.

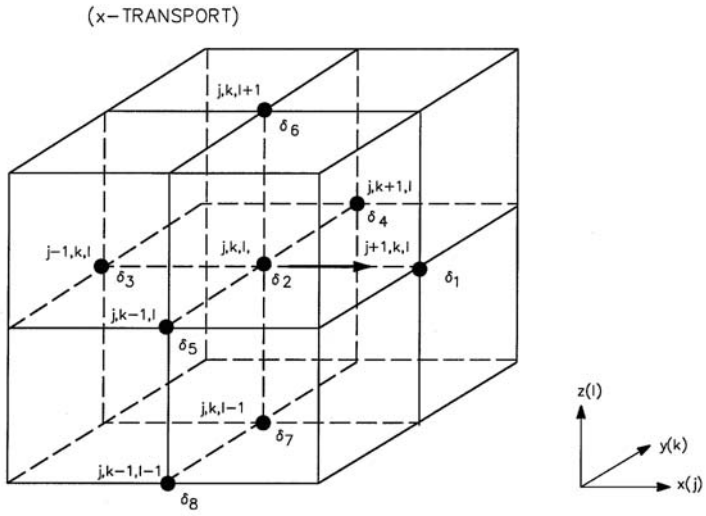


Figure 16.4 Location of interpolation weights for determination of concentrations at the location (j+½,k,l) - a grid boundary.

The dispersive transport can be derived in a similar way. With the finite difference formulation of the dispersive transport components based on upstream differencing in concentrations and central differencing in dispersion coefficients the transport in the x-direction can be expressed in the following manner:

$$
\begin{aligned}
TD_{xx\,j,k,l} = & -\tfrac{1}{2}\left(D_{xx\,j+1,k,l} + D_{xx\,j,k,l}\right)\left(c_{j+1,k,l} - c_{j,k,l}\right) / \Delta x^2 \\
& -\tfrac{1}{2}\left(D_{xy\,j+1,k,l} + D_{xy\,j,k,l}\right) \cdot \\
& \left(c_{j,k+1,l} + c_{j+1,k+1,l} - c_{j,k-1,l} - c_{j+1,k-1,l}\right) / 4\Delta y\Delta x \\
& -\tfrac{1}{2}\left(D_{xz\,j+1,k,l} + D_{xz\,j,k,l}\right) \cdot \\
& \left(c_{j,k,l+1} + c_{j+1,k,l+1} - c_{j,k,l-1} - c_{j+1,k,l-1}\right) / 4\Delta z\Delta x
\end{aligned}
\qquad (16.14)
$$

The dispersive transports in the other directions are expressed in a similar way. The dispersive transports are incorporated in the weight functions so that the mass transports can be calculated in one step.

Irregular Grid

In general the flow simulation may use varying layer thickness for the vertical discretisation of the saturated zone domain. In this case, the code checks each layer to see whether its thickness is identical with the thickness of the layer above and the layer below in each of the grid cells. If this is the case this layer is handled as described for “regular grid” above (3D). If this is not the

case a different approach is followed where a 2D regular grid solution based on the QUICKEST scheme is used for the horizontal transport and the vertical transport is taken into account as an explicit sink/source term.

16.2.3 Initial Conditions

The initial concentration is a fully distributed concentration field, which can be entirely uniform or constant by layer.

A boundary water flux into the model, results in a constant concentration boundary at the initial concentration value, unless explicitly specified as a time varying concentration boundary.

16.2.4 Source/Sinks, Boundary Conditions and other Exchanges

Boundary conditions for the groundwater transport component can be either

- a prescribed concentration (Dirichlet's condition), or
- prescribed flux concentration (Neumann's condition).

Catchment boundary cells with a specified head are treat as fixed concentration cells with a concentration equal to the initial concentration. A prescribed, time-varying concentration boundary can be specified at any internal node.

Prescribed flux concentration can be specified on the catchment boundary, as well as in any cell inside the model area. The flux concentration at the catchment boundary can be constant or time-varying.

Sinks can either extract pure water (concentration equal to zero) or water with the current concentration. Soil evaporation is the only sink, which removes water with a concentration equal to zero. Sinks where the concentration is equal to the actual solute concentration in the grids include pumping wells, drains and MIKE Hydro River nodes.

Referring back to Figure 16.1, you should note that

- If UZ transport is included, the upper boundary for the groundwater transport is the exchange of mass with the UZ component. Infiltration from (and to) the unsaturated zone is treated as a source (or sink) term. However, if the UZ flow had a shorter storing time step than the SZ flow, the concentration in the top layer of the SZ transport is updated at the UZ time step.
- If UZ bypass flow was specified, mass is transferred directly from the overland to the groundwater, with the flux equal to bypass flow multiplied with the concentration on the overland.

- Direct exchange between OL flow and SZ flow occurs when the soil is completely saturated. In this case, the infiltration from OL goes directly to the SZ and the mass flux is equal to the infiltration multiplied with the concentration on the overland.
- Exchanges with MIKE Hydro River are also treated explicitly as exchange flows. Inflow and outflow respectively are multiplied by the concentration in the river or the adjacent grids to the groundwater.
- SZ Drainage to the overland or MIKE Hydro River (or the boundary) is also treated as an SZ sink and the mass receiving component and is again calculated by multiplying the exchange flux with the concentration.

16.2.5 Additional options

Disable SZ solute flux to dummy UZ

The following Extra Parameter is useful, if you are using an alternative UZ model, such as DAISY, in MIKE SHE and you are trying to couple it to the WQ.

In this case, you will be typically using the Negative Precipitation *(V2 p. 329)* option. If you use this option, then you will not use a MIKE SHE UZ, and the UZ-SZ exchange will pass through a “dummy UZ” layer. When this is coupled to the water quality, solutes will also be passed to this dummy UZ layer and removed from the SZ domain and the model.

To prevent the upflow of solutes from SZ to the dummy UZ, you must specify the following Extra Parameter..

Parameter Name	Type	Value
disable sz transport to dummy uz	Boolean	On

SZ boundary dispersion

A detailed test of the MIKE SHE WQ engine comparing an SZ model with fixed concentration at an inflow boundary with an analytical solution for a fixed concentration source, showed that MIKE SHE under-estimates the mass flux into the model when the model includes longitudinal dispersion.

The problem is that the SZ transport scheeme (QUICKEST) doesn't include dispersive transport to/from open boundary cells. This is as designed, but apparently not correct. After including the boundary dispersion, the mass input to the model is within 2 % of the analytical solution.

From Release 2011 and onwards, the boundary dispersion has been made optional for backwards compatibility and is activated with the extra-parameter: .

Parameter Name	Type	Value
enable sz boundary dispersion	Boolean	On

However, the SZ boundary dispersion option (above) does not calculate dispersive transport to an inflow boundary correctly. Again, this problem was identified in the tests of MShe_WQ with MIKE ECO Lab vs analytical solution. For example:

- Species 1 enters the model via an inflow (flux) boundary with fixed concentration - including dispersive transport due to the new sz boundary dispersion option.
- Species 1 decays to Species 2 which again decays to Species 3.
- The concentrations of Sp2 & Sp3 are too high, especially close to the inflow boundary.

The analytical solution includes dispersive transport of Sp2 & Sp3 against the flow direction because the concentration of these species are 0 at the boundary. However, this dispersive mass flux to the boundary is not included in the SZ solver due to an old check in the code. When mass flux to/from a boundary point is reversed compared to the flow direction, the mass flux is simply reset to 0.

This made sense before the boundary dispersion was implemented because advective transport against the flow direction would be wrong. But, now, when the boundary dispersion is active, this situation is allowed.

16.2.6 Transport in Fractured Media

MIKE SHE AD is able to simulate solute transport in fractured media under some simplifying conditions. If we assume that water flows only in the frac-

tures and that solutes can enter into the soil matrix as immobile solutes the advection-dispersion equation changes to:

$$\theta_{im}\frac{\partial c_{im}}{\partial t} + \theta_m\frac{\partial c_m}{\partial t} =$$

$$-\frac{\partial}{\partial x_i}\left(\theta_m c_m v_i\right) + \frac{\partial}{\partial x_i}\left(D_{ij}\frac{\partial \theta_m c_m}{\partial x_j}\right) + R_c \qquad (16.15)$$

where c is the concentration of the solute, subscripts m and im are for mobile and immobile, respectively, R_c is the sources or sinks, D_{ij} is the dispersion tensor and v_i is the velocity tensor determined from fracture porosity.

The exchange of mass between the mobile water phase in the fractures and immobile water phase is described by the traditional diffusion equation:

$$\theta_{im}\frac{\partial c_{im}}{\partial t} = \beta\left(c_m - c_{im}\right) \qquad (16.16)$$

where β is the diffusion coefficient.

Diffusive exchange is included as a distributed source/sink term in the basic advection-dispersion equation.

16.3 Solute Transport in the Unsaturated Zone

The Solute Transport in the Unsaturated Zone links the transport in the overland flow and transport in the saturated zone together.

Solute transport in the unsaturated zone be simulated in both the soil matrix and macropores. Solute transport in the soil matrix is described by a 1D, unsaturated formulation of the advection-dispersion equation, which is considerably simpler than the 3D formulation in the saturated zone.Although, the unsaturated water movement calculations can be lumped together to save computational time, solute transport in the unsaturated zone is always calculated in every column. The solute transport boundary conditions and initial conditions are specified independent of any column lumping that was done in the water movement simulation.

16.3.1 Governing Equations

Soil Matrix Transport

For unsaturated solute transport in the soil matrix the advection-dispersion equation is

$$\frac{\partial c}{\partial t} = -\frac{\partial}{\partial z}(cv_z) + \frac{\partial}{\partial z}\left(D\frac{\partial c}{\partial z}\right) + R_c \tag{16.17}$$

where c is the concentration of the solute, R_c is sum of sources and sinks, D is the dispersion coefficient, and v_z is the vertical velocity.

The advective transport is determined by the water flux calculated during a MIKE SHE WM simulation. As the water flow is assumed strictly vertical, this restriction applies also to the advective transport of the dissolved solutes.

To determine the velocity, v_z, the flux is divided by the moisture content:

$$v_z = \frac{q}{\theta} \tag{16.18}$$

The mathematical formulation of the dispersion of the solutes follows the formulation derived for groundwater flow with a linear relation between the dispersion coefficient and the seepage velocity but limited to one dimension. In this case the dispersion coefficient can be written as:

$$D = \alpha_L v_z \tag{16.19}$$

where α_L is the longitudinal dispersivity of the porous medium which represents the heterogeneity of the soil hydraulic parameters. α_L is allowed to vary vertically to account for different degrees of inhomogeneity in the soil. For unsaturated flow, the dispersivity is dependent on the water content, however, this relationship is neglected. .

16.3.2 Solute transport in macropores

Internally in the macropores, solute transport is assumed to be dominated by advection, neglecting the influence of dispersion and diffusion.

The source/sink term, R_c, describing solute exchange between matrix and macropores is given by a combination of a diffusion component and a mass flow component

$$R_c = \beta_c \cdot \theta_{matrix} \cdot \frac{\theta_{mp}}{e_{mp}} \cdot (c_{mp} - c_{matrix}) + S_{mp} \cdot c' \tag{16.20}$$

where b_c is a mass transfer coefficient, c_{mp} and c_{matrix} are the solute concentrations in the macropores and matrix, respectively, and c' is the concentration in either matrix or macropores depending on the direction of the exchange flow, S_{mp}.

The mass transfer coefficient, b_c, can be derived from

$$\beta_c = C_k \cdot \frac{3 D_0 \cdot f}{d^2} \tag{16.21}$$

where C_k is the dimensionless contact factor to take care of an eventual coating at the interior walls of the macropores. The contact factor ranges from 0.0 (no contact) to 1.0 (full contact). D_0 [m^2/s] is the diffusion coefficient in free water of the solute species. f is a dimensionless impedance factor that represents and decreases with the tortuosity of the macropores. f ranges from 0.0 (zero diffusivity) to 1.0 (full diffusivity). Thereby, b_c depends on both solute species and soil type.

Even though the applied macropore description can be regarded as mechanistic with parameters having physical meaning, some of the parameters required to characterise the macropore system are either difficult or impossible to measure. This is particularly the case for the parameters regulating exchange between matrix and macropores. Field observations of soil structure and the occurrence of biotic macropores can give indications of the mass exchange parameters (Jarvis et al., 1997; Jarvis, 1998), though recent experiences reveal that parameters obtained from such macroscopic observations often need adjustments towards longer diffusion lengths when applied to field measurements (Saxena et al., 1994; Larsson and Jarvis, 1999). The main reason for this is expected to be organic and clay coatings on the aggregate surfaces which reduce mass exchange rates between the two domains (Thoma et al., 1992; Vinther et al., 1999).

16.3.3 Solution Scheme

Similar to the saturated zone, the unsaturated solute transport is solved explicitly, using upstream differencing for the advection term and central differencing for the dispersion term.

Neglecting the dispersion terms and the source/sink term and assuming that the flow field satisfies the equation of continuity and varies uniformly within a grid cell, the advection component is

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial z}(v_z c) = 0 \tag{16.22}$$

:

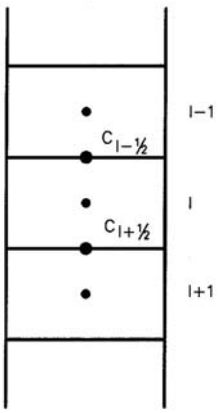


Figure 16.5 Control volume defining an internal grid.

For the control volume shown in Figure 16.5, this equation is written in finite difference form as:

$$c_l^{n+1} - c_l^n + \sigma_z \left(c^*_{l+½} - c^*_{l-½} \right) = 0 \tag{16.23}$$

where n denotes the time index, c_l is the concentration in the computational node, c^* is the interpolated concentration at the edges of the grid at time n, and σ_z is the directional Courant number defined by

$$\sigma_z = \frac{v_z \Delta t}{\Delta z} \tag{16.24}$$

As the c^*-terms are not located at the nodal points, they have to be interpolated from known concentration values. The equation for this follows the one derived for the saturated zone

$$c_{l+½} = \sum \beta_i c_i \tag{16.25}$$

However, in the unsaturated zone only three weights need to be determined

$$\beta_1 = \frac{\sigma_z^2}{6} - \frac{\sigma_z}{2} + \frac{1}{3}$$
$$\beta_2 = 1 - \beta_1 - \beta_3$$
$$\beta_3 = \frac{\sigma_z^2}{6} - \frac{1}{6} \qquad (16.26)$$

where the weights are positioned relative to the actual flow direction.

Dispersive transport can be derived in a similar way. If the finite difference formulation of the dispersive transport is based on upstream differencing in concentration and central differencing in the dispersion coefficients, the dispersive transport is

$$TD_{z,l} = -\frac{1}{2}(D_{z,l+1} + D_{z,l})(c_{l+1} - c_l)/\Delta z^2 \qquad (16.27)$$

The dispersive transport is incorporated into the weights.

The above solution is strictly speaking only valid for a regular discretisation but if the resolution varies slowly the error introduced is small.

16.3.4 Initial Conditions

The initial concentration in the unsaturated zone is given as average concentrations. The unit for the concentration is [mass/volume]. This way of giving the initial conditions has the advantage that you does not have to worry about the vertical discretisation and the water content. On the other hand, you do not know the exact amount of mass introduced into the unsaturated zone, since this depends on the initial water content determined in the water movement simulation.

16.3.5 Source/Sinks, Boundary Conditions and other Exchanges

Sources for solute input into the unsaturated zone can be given in a number of ways both as point or line sources at specific depth intervals or as area sources at specific locations.

Dissolved matter can enter the unsaturated zone in three different ways. It can either be added to the precipitation as a so-called precipitation source (if the overland part is not included in the simulation) or it can be a UZ-source introduced at a certain depth or it can enter from the saturated zone. The pre-

cipitation source option is described in the section about Overland Solute Transport.

The unsaturated zone transport component exchanges mass with the overland and the groundwater transport components as indicated in Figure 2. The transport from the overland component is a one way transport from the overland to the unsaturated zone whereas both transport to and from the groundwater can occur.

Point and line sources can be included with units of [mass/time]. Spatially distributed sources can be included with units of [mass/area/time]. In each calculation time step the solute mass in all grids nodes is updated with mass from the source.

It is not possible to introduce external sinks in the unsaturated zone. However, water can be removed by the roots or via soil evaporation, which can consequently increase solute concentrations.

16.4 Solute Transport in Overland Flow

MIKE SHE calculates the movement of solutes in overland flow, when ever ponded water exists. In surface water the mixing and spreading of solutes is mainly due to turbulence, which appears when the flow velocity exceeds a certain level. This process is known as turbulent diffusion and is generally far more important than molecular diffusion.Although this process is physically different from the spreading of solutes in groundwater, solute transport in surface water is usually still described using the advection-dispersion equation. Similar to the saturated zone, the 2D advection-dispersion equation for overland transport is solved using the explicit, third-order accurate QUICKEST scheme.

As for solute transport in the saturated zone, the dispersion coefficients depend on the spatial and temporal scale of averaging. However, dispersion in surface water depends on the homogeneity of the velocity distribution in the flow cross-section. To some extent, the dispersion depends on the mean flow velocity. However, there is no general dependence between the dispersion coefficient and the mean flow velocity. Therefore, in surface water models, the dispersion coefficient is usually specified directly. In MIKE SHE, the dispersion coefficients are assumed constant in time but may vary in space.

16.4.1 Governing Equations

The transport of solutes on the ground surface is governed by the two-dimensional advection-dispersion equation

$$\frac{\partial c}{\partial t} = - \frac{\partial}{\partial x_i}(cv_i) + \frac{\partial}{\partial x_i}\left(D_{ij}\frac{\partial c}{\partial x_j}\right) + R_c \quad i,j = 1,2 \tag{16.28}$$

where c is the concentration of the solute, R_c is sum of sources and sinks, D_{ij} is the dispersion tensor and v_i is the velocity tensor.

The velocity of water is determined from the water flux and water depth calculated during the WM simulation.

For overland transport the longitudinal and transverse dispersion coefficients (D_L and D) are specified directly and the dispersion coefficients applied in Eq. (16.28) are determined as for isotropic conditions in groundwater as

$$\begin{aligned} D_{xx} &= D_{11} = D_L\frac{V_x^2}{U^2} + D_T\frac{V_y^2}{U^2} \\ D_{yy} &= D_{22} = D_T\frac{V_x^2}{U^2} + D_L\frac{V_y^2}{U^2} \\ D_{xy} &= D_{yx} = D_{12} = (D_L - D_T)\frac{V_xV_y}{U^2} \end{aligned} \tag{16.29}$$

The water depth on the ground surface varies in space and time, due to variations in topography, as well as variations in precipitation, evaporation, infiltration etc. Since evaporation can concentrate a solute beyond its solubility, a mass balance of precipitated solute is maintained, where the solute will redissolve if additional water becomes available. The precipitation and dissolution of the solute is controlled by its solubility.

16.4.2 Solution scheme

The solution scheme applied for overland transport uses the same QUICKEST scheme as in the saturated zone.It is a fully explicit scheme that using upstream differencing.

Neglecting the dispersion terms and the source/sink term and assuming that the flow field satisfies the equation of continuity and varies uniformly within a grid cell, the advection-dispersion equation can be written as

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial x}\left(v_x c\right) + \frac{\partial}{\partial y}\left(v_y c\right) = 0 \tag{16.30}$$

and when written in finite difference form becomes

$$c_{j,k}^{n+1} - c_{j,k}^{n} + \sigma_x \left(c^{*}_{j+\gamma_x,k} - c^{*}_{j-\gamma_x,k}\right) + \sigma_y \left(c^{*}_{j,k+\gamma_y} - c^{*}_{j,k-\gamma_y}\right) = 0 \tag{16.31}$$

where *n* denotes the time index.

In Eq. (16.31), σ_x and σ_y are the directional Courant numbers defined by

$$\sigma_x = \frac{v_x \, \Delta t}{\Delta x}, \quad \sigma_y = \frac{v_y \, \Delta t}{\Delta y} \tag{16.32}$$

and the c^{*}-terms are the concentrations at the surface of the control volume at time *n*. As these terms are not located at nodal points, they are interpolated from known concentration values by

$$c^{*}_{j+\gamma_x,k} = \sum \alpha_i c_i$$
$$c^{*}_{j,k+\gamma_y} = \sum \beta_i c_i \tag{16.33}$$

The concentration c_i is the concentration around the actual point, for example (*j-1,k*) and the weights δ_i and δ_i are determined in such a way that the scheme becomes third-order accurate. The determination of the weights is demonstrated in Vested et al. (1992) and listed in Table 16.3. The other “boundary” concentrations are found in a similar way.

Table 16.3 Weight functions for advective transport

I	α	β
1	$\sigma_x(\sigma_x^2/6 - \sigma_x/2 + 1/3)$	$\sigma_y(\sigma_y^2/6 - \sigma_y/2 + 1/3)$
2	$\sigma_x - (\alpha_1 + \sigma_3 + \alpha_4 + \alpha 5)$	$\sigma_y - (\beta_1 + \beta_3 + \beta_4 + \beta_5)$
3	$\sigma_x(-1/6 + \sigma_x^2/6)$	$\sigma_y(-1/6 + \sigma_y 2/6)$

Table 16.3 Weight functions for advective transport

l	α	β
4	$\sigma_x(-\sigma_y/2 + \sigma_y^2/2)$	$\sigma_y(-\sigma_x/2 + \sigma_x^2/2)$
5	$\sigma_x(\sigma_{xsy}/2)$	$\sigma_y(\sigma_x\sigma_y/2)$

The locations of the weights are determined by the points that enter into the discretisation. Since the scheme is upstream centred, the weights are positioned relative to the actual direction of the flow. This is outlined in more detail for in the saturated zone Solution Scheme *(V2 p. 263)* section.

The dispersive transport can be derived in a similar way. With the finite difference formulation of the dispersive transport components based on upstream differencing in concentrations and central differencing in dispersion coefficients, the transport in the x-direction can be written as

$$TD_{xi,j,k} = -\tfrac{1}{2}\left(D_{xx,j+1,k} + D_{xi,j,k}\right)\left(c_{j+1,k} - c_{j,k}\right)/\Delta x^2 - \tfrac{1}{2}\left(D_{xy,j+1,k} + D_{xy,j,k}\right) * \left(c_{j,k+1} + c_{j+1,k+1} - c_{j,k-1} - c_{j+1,k-1}\right)/4\Delta y\Delta x \quad (16.34)$$

The dispersive transport in the *y* direction is done in a similar way. The dispersive transports are incorporated in the weight functions, so that the mass transports can be calculated in one step.

16.4.3 Initial Conditions

The initial concentration can be a fully distributed concentration field (e.g. measured or simulated concentrations at a certain time). The unit for the overland concentration is [mass/area].

If there is a flux of water into the model area from boundary points the flux concentration in these points will be constant in time and equal to the initial concentration, if the flux concentration at any of these points is not specified as a time-varying source concentration.

16.4.4 Source/Sinks, Boundary Conditions and other Exchanges

Dissolved solutes can be added to the overland flow via precipitation, or from discharging groundwater. Alternatively, the solute can be added directly as a spatially distributed source on the land surface.

As indicated in Figure 16.2 the overland transport component exchanges mass with the MIKE Hydro River, the unsaturated zone and the saturated zone. In the case of exchange to MIKE Hydro River, the solute mass is simply

added as a source term to MIKE Hydro River. Similarly, infiltration is added as a source in the unsaturated zone. Exchange directly to the saturated zone can occur, if by-pass flow is allowed, in which case, the bypass flow concentration is the same as the concentration in the overland flow. Exchange to overland flow from the saturated zone occurs if the water table rises above the ground surface.

A spatially distributed source is specified using a dfs2 file, where the source strength is given in units of [mass/area/time].

It is not possible to introduce external sinks for overland transport. However, solute concentrations can increase due to evaporation.

16.5 Solute Transport in MIKE Hydro River

In MIKE SHE, the solute transport in the river channels is handled by the MIKE Hydro River Advection-Dispersion (AD) module.

In MIKE Hydro River, the 1D advection-dispersion equation is solved using an implicit finite difference scheme that is, in principle, unconditionally stable with negligible numerical dispersion. A correction term has been added to reduce the third-order truncation error, making it possible to simulate very steep concentration gradients.

Longitudinal dispersion in channels is largely controlled by the non-uniform velocity distribution both spatially and temporally. In rivers the dispersion coefficient is normally on the order of 5 to 10 m2/s increasing to between 30 and 100 m2/s when 2D processes, such as secondary currents and wind induced turbulence begin to dominate.

MIKE Hydro River exchanges solutes with MIKE SHE's overland and saturated zone flow components.

Detailed information on this module is available as part of the MIKE Hydro River technical documentation, which can be found in .pdf form in your installation directory.

17 Reactive Transport - Reference

Several reaction processes can be added to the solute transport calculations including

- Sorption and desorption,
- Degradation, and
- Plant uptake.

In the saturated and unsaturated all three of these processes are available. However, in the overland flow only degradation is available, but in MIKE Hydro River advanced reactions are possible using MIKE ECO Lab, which is a general equation solver for any kinetic reaction process.

The conservative transport of solutes in the unsaturated zone and in the groundwater is governed by the normal advection-dispersion equation described in Equation (16.3). When processes, such as sorption and decay, are included, the equation is extended to

$$\frac{\delta c}{\delta t} = -\frac{\delta}{\delta x_i}(c v_i) + \frac{\delta}{\delta x_i}\left(D_{ij}\frac{\delta c}{\delta x_j}\right) - \frac{\rho_b}{\theta}\frac{\delta c^*}{\delta t} + \left(\frac{\delta c}{\delta t}\right)_{reac} \tag{17.1}$$

where ρ_b is the bulk density of the porous medium, θ is the porosity of the porous medium, c^* is the mass of solutes sorbed per dry unit weight of solid and the term $\delta c/\delta t$ on the right hand side is a term indicating a biological or chemical reaction of the solute.

This way of describing reaction processes is very simplified and, in some cases, may give incorrect results. Nevertheless, it is a very common way of describing the reaction processes in hydrologic systems.

17.1 Sorption

Sorption includes a number of geochemical and chemical reactions, such as adsorption of solutes to the aquifer material surface by electrostatic forces (called cation exchange). If these processes occur sufficiently fast compared with the water flow velocity they can be described by an equilibrium sorption isotherm.

Different equilibrium sorption isotherms have been identified from experimental results with different sediments, soils and rock types, see, for example,

Fetter, 1993. MIKE SHE AD includes three of the most commonly applied isotherms - namely the linear, Freundlich and Langmuir equilibrium sorption isotherms.

Sorption processes that are slow compared with the water flow velocity must be described by a kinetic sorption isotherm. In MIKE SHE AD the three equilibrium sorption isotherms have been extended to include a kinetically controlled sorption process so that a certain part of the sorbed matter is "transferred" to another part of the soil material.

17.1.1 Equilibrium Sorption Isotherms

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:

$$c' = K_d \cdot c \tag{17.2}$$

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.

$$K_d = f_{oc} \cdot K_{oc} \tag{17.3}$$

where f_{oc} is the fraction of organic carbon.

The retardation factor, R, 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

$$R = \frac{v}{v_c} = 1 + \frac{\rho_b}{\theta} \cdot K_d \tag{17.4}$$

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:

$$c' = K_f \cdot c^N \tag{17.5}$$

where K_f and N are constants. Note that the units of Kf is a function of the units of c.

The relationship between K and N is shown in Figure 17.1.

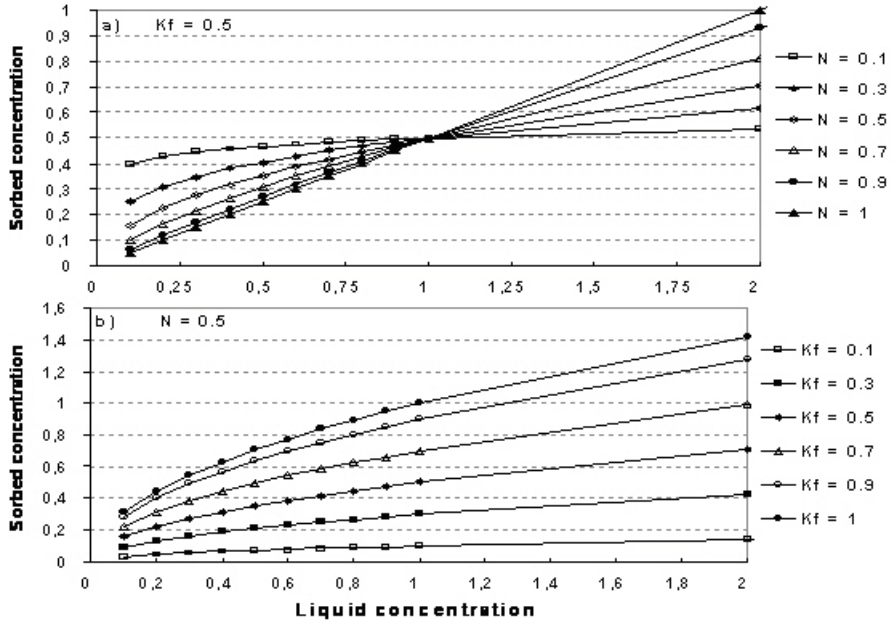


Figure 17.1 Illustration of the Freundlich isotherm. a) effect of change in N, b) effect of change in Kf

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

$$\frac{c}{c^*} = \frac{1}{\alpha\beta} + \frac{c}{\beta} \tag{17.6}$$

or

$$c^* = \frac{c\alpha\beta}{1+\alpha c} \tag{17.7}$$

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.

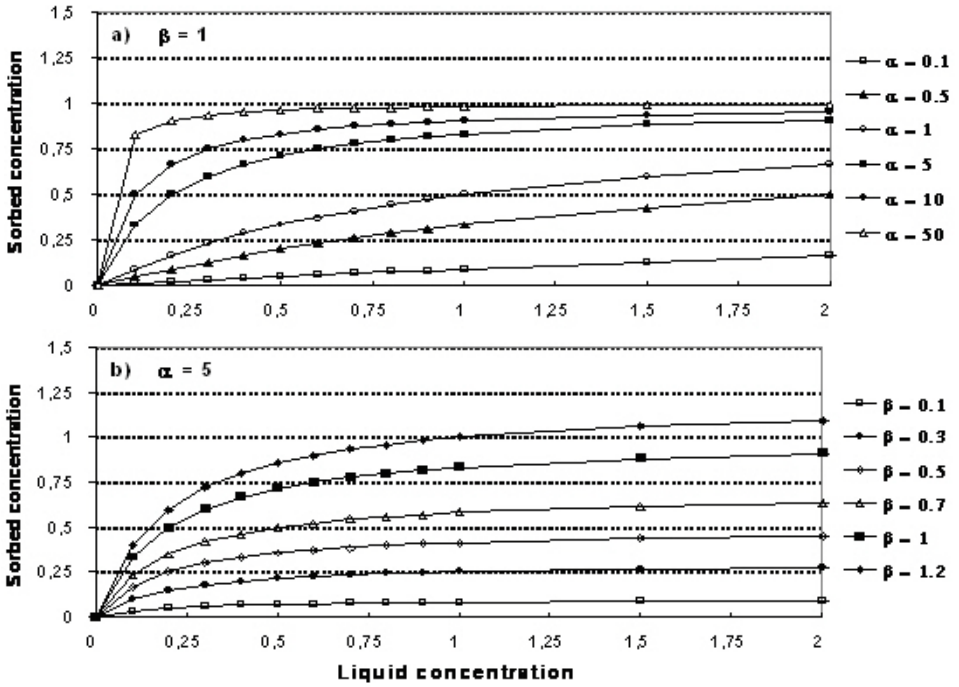


Figure 17.2 Illustration of the Langmuir isotherm. a) effect of change in α, b) effect of change in β.

17.1.2 Kinetic Sorption Isotherms

The three equilibrium sorption isotherms described so far 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. A chemical formulation of this approach is given by:

$$c \leftrightarrow c^{*} \leftrightarrow c^{**} \tag{17.8}$$

where c* is the amount of the solutes sorbed (described by the equilibrium sorption model), c** is the amount of the sorbed matter that is converted to the kinetically controlled sorption domain. Mathematically the work of Brusseau (1995) is implemented in MIKE SHE AD as

$$\frac{\partial c^{*}}{\partial t} = K_1 \left(c^{**} - c^{*} \right) \tag{17.9}$$

where K_1 is the constant defining the rate of kinetic sorption. The formula is generalised so that effects of hysteresis can be taken into account, by specifying a K_1 value for adsorption (c*>c**) and another value for desorption (c**>c*).

The formula is generally applicable for all the equilibrium sorption models. All constants appearing in the sorption models are assumed constant in time but may vary in space.

17.1.3 Sorption in Dual Porosity Systems

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 has included a 'sorption bias factor', F_b. This allows you to explicitly control the sorption distribution between the fractures and the matrix.

Mathematically, the bulk mass available for sorption in the macro pores, ρ_{ma}, is described by:

$$\rho_{ma} = \frac{\theta_{ma} + F_b \theta_{mi}}{\theta_{ma} + \theta_{mi}} \rho_b \qquad for\ 0 \le F_b \le 1$$

$$\rho_{ma} = \frac{\theta_{ma} (1 + F_b)}{\theta_{ma} + \theta_{mi}} \rho_b \qquad for\ -1 \le F_b < 0 \tag{17.10}$$

where F_b is the sorption bias factor, ρ_b is the bulk mass, θ_{ma} and θ_{mi} are the macro pore and the matrix porosity respectively. The available bulk mass for sorption in the macropores is "the remainder" mass in the soil. If $F_b = 0$ the distribution of sorption sites between macro pores and matrix is assumed to be proportional to the distribution of porosities. If $F_b = 1$ sorption is assumed to occur in macro pores only and if $F_b = -1$ sorption is only occurring in the matrix region. The nature of Eq. (17.10) is illustrated in Figure 17.3.

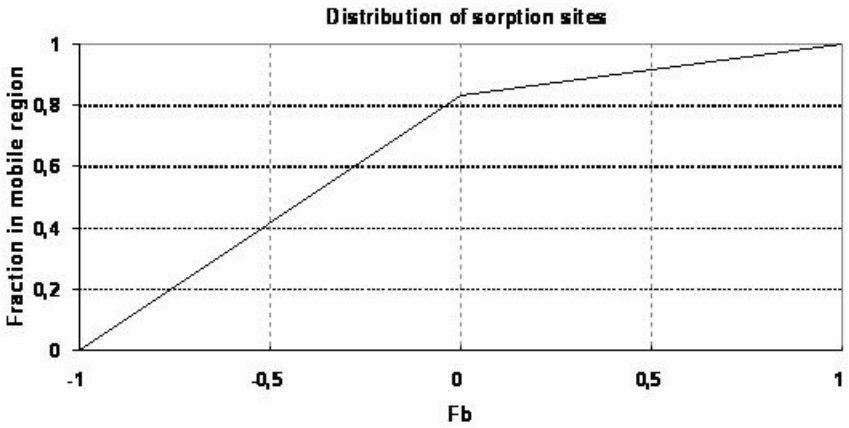


Figure 17.3 Illustration of the fraction of sorption sites located in the macro pore region (ρ_{ma}/ρ_b) as a function of F_b. Mobile porosity = 0.25, immobile porosity=0.05.

17.2 Decay

Biological degradation, radioactive decay or other kinds of attenuation of solutes can often be described as a first-order degradation process, with an exponential decrease of concentration over a half-life. This is described in MIKE SHE as

$$\left(\frac{\partial c}{\partial t}\right)_{\text{reac}} = \mu_{\text{ref}} c \tag{17.11}$$

where μ_{ref} is the reference degradation rate coefficient calculated by

$$\mu_{\text{ref}} = \frac{\ln 2}{\lambda} \tag{17.12}$$

where λ is the half-life of the species.

Following the work of Boesten and van der Linden (1991), to overcome some of the difficulties in simplifying complex biological and chemical reactions, the decay in MIKE SHE is dependent of the soil moisture content and soil temperature as

$$\mu = \mu_{\text{ref}} F_w F_t \tag{17.13}$$

where the F_w is the water content function given as

$$F_w = \left(\frac{\theta}{\theta_S}\right)^B \tag{17.14}$$

and where θ is actual soil moisture, θ_S is saturated moisture content and B is an empirical constant. F_t is the soil temperature function given as:

$$F_t = 0 \qquad for(T_s < 0^oC) \tag{17.15}$$

$$F_t = \frac{T_5}{5} e^{\alpha(5 - T_{ref})} \qquad for(0 \le T_s \le 5^oC) \tag{17.16}$$

$$F_t = e^{\alpha(T_s - T_{ref})} \qquad for(T_s > 5^oC) \tag{17.17}$$

where T_s is the actual temperature of the soil, T_{ref} is the reference temperature at which μ_{ref} is measured and α is a constant depending on T, T_{ref}, the gas constant and the molar activation.

For simplicity the soil temperature over depth is calculated as a function of the air temperature by an experimentally derived formula given by Klein (1995)

$$T_s = T_{sy} + 0.346(T_{air} - T_{sy})e^{-2.7028z} \tag{17.18}$$

where T_{sy} is the mean daily soil temperature from yesterday, T_{air} is the mean daily air temperature and z is depth.

Note that for large depths, this function responds very slowly to variations in air temperature. Therefore, long simulations may be necessary to achieve the required initial temperature distribution. An example of simulated soil temperature distributions as a function soil depth is shown in Figure 17.4.

MIKE SHE allows you to specify separate half-lives for the matrix and fractions in dual porosity models, since degradation is likely to be faster in the fractures where higher oxygen contents are more likely.

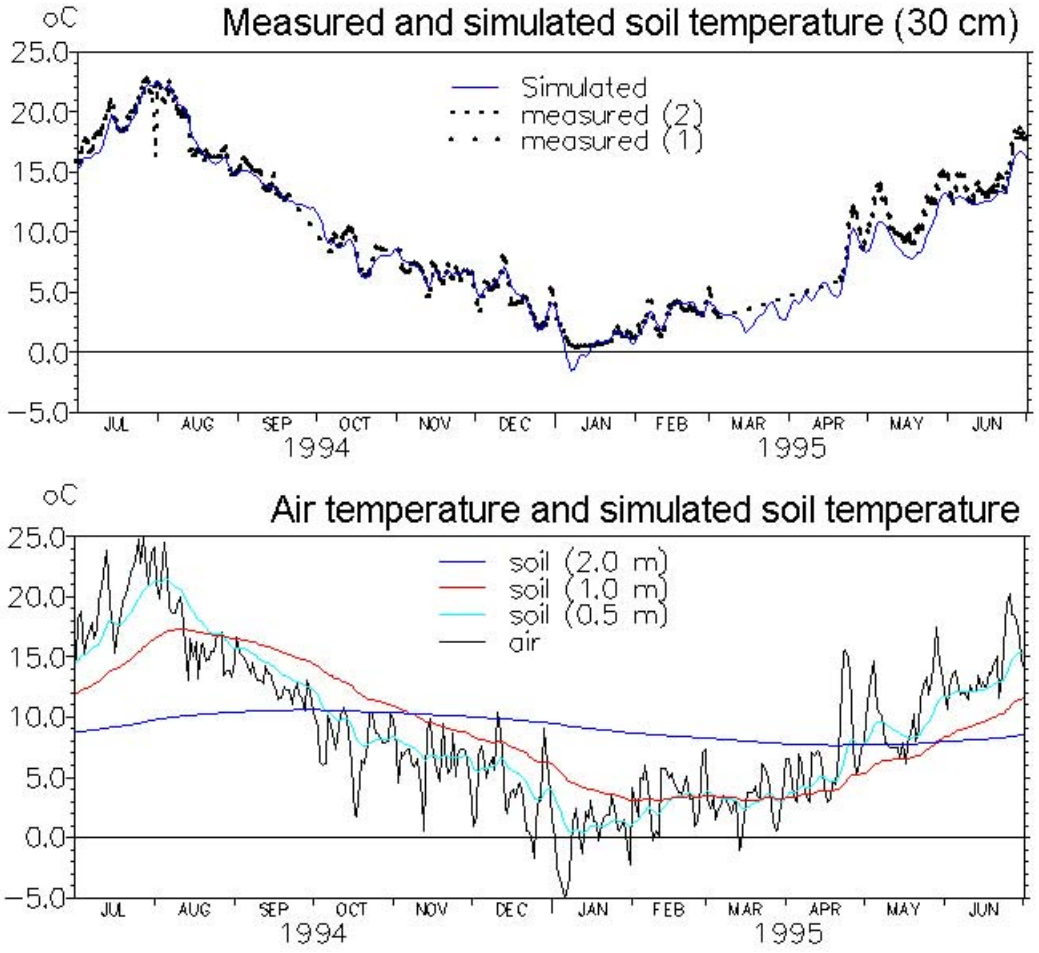


Figure 17.4 Application of the soil temperature function. Top: measured and simulated soil temperature in 30 cm depth (measurements were based on two replicates). Bottom: soil temperatures simulated in different depths based on measured air temperatures.

17.3 Plant Uptake

Plant uptake of solutes is described as passive transport, along with the transpiration stream as a function of the solute concentration in the liquid phase. Different roots have different capabilities when it comes to filtering out various solutes. Thus, an empirical concentration factor determines to what extent the available solute is taken up by the plants.

$$R_r = f_c S_r c \tag{17.19}$$

where R_r is the sink term in the advection-dispersion equation, f_c is the concentration factor, S_r is the root water uptake and c is the liquid concentration.

17.4 Process verification

The performance of MIKE SHE's basic reactive transport module, with equilibrium and non-equilibrium sorption and degradation, has been verified against analytical solutions calculated with CXTFIT (Toride et al., 1995). The verification tests were conducted using steady-state saturated water flow through a 1 m deep column discretised in 5 cm elements. The simulations were run for one month with maximum time step equal to 15 min. Pore flow velocity was 25 cm/day, the dispersivity was 1 cm and the bulk density was 1.5 g/cm^3. Furthermore, the diffusion process in fractured media with a fracture porosity of 0.25 and a matrix porosity of 0.05 is verified both without and with sorption.

The verification results confirm that the numerical solutions are satisfactory, since the calculated solute breakthrough and mass recovery curves are very similar to the analytical solutions.

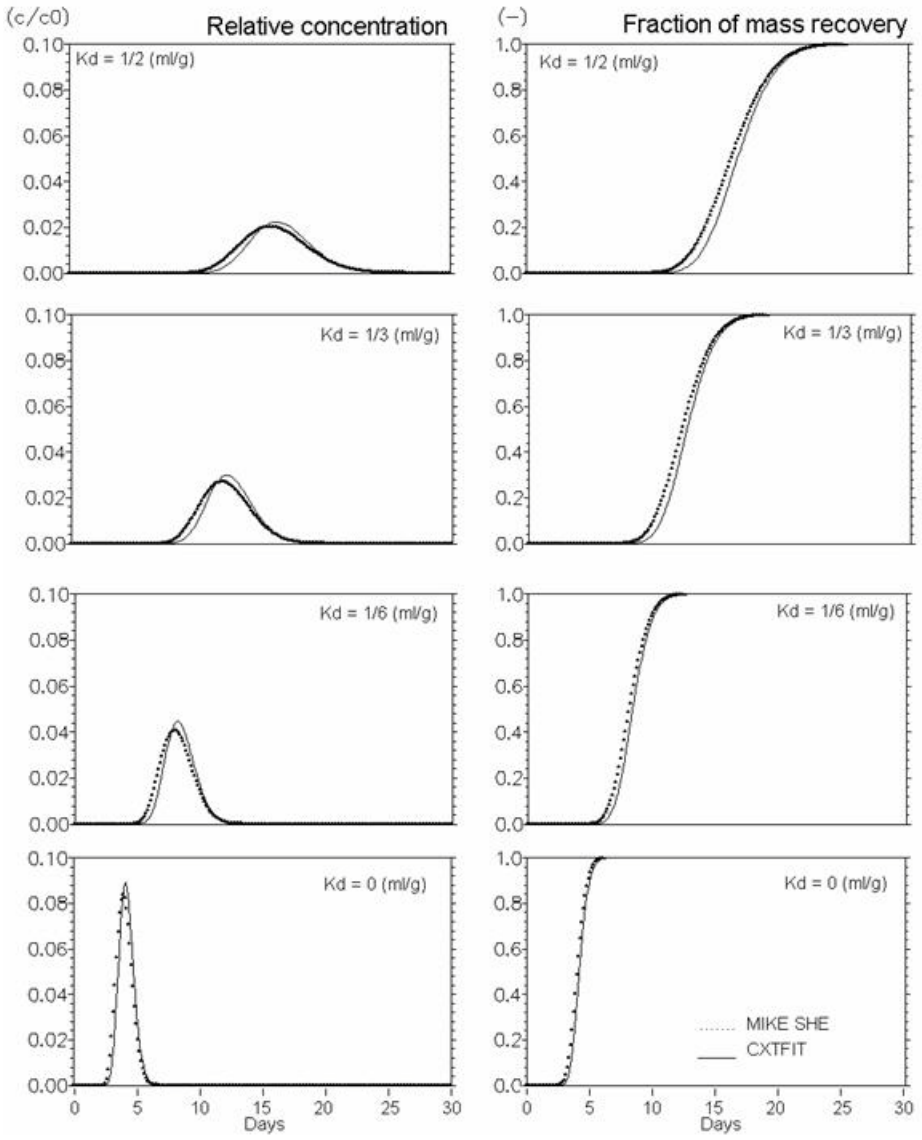


Figure 17.5 Linear equilibrium sorption. Effect of K_d (ml/g)

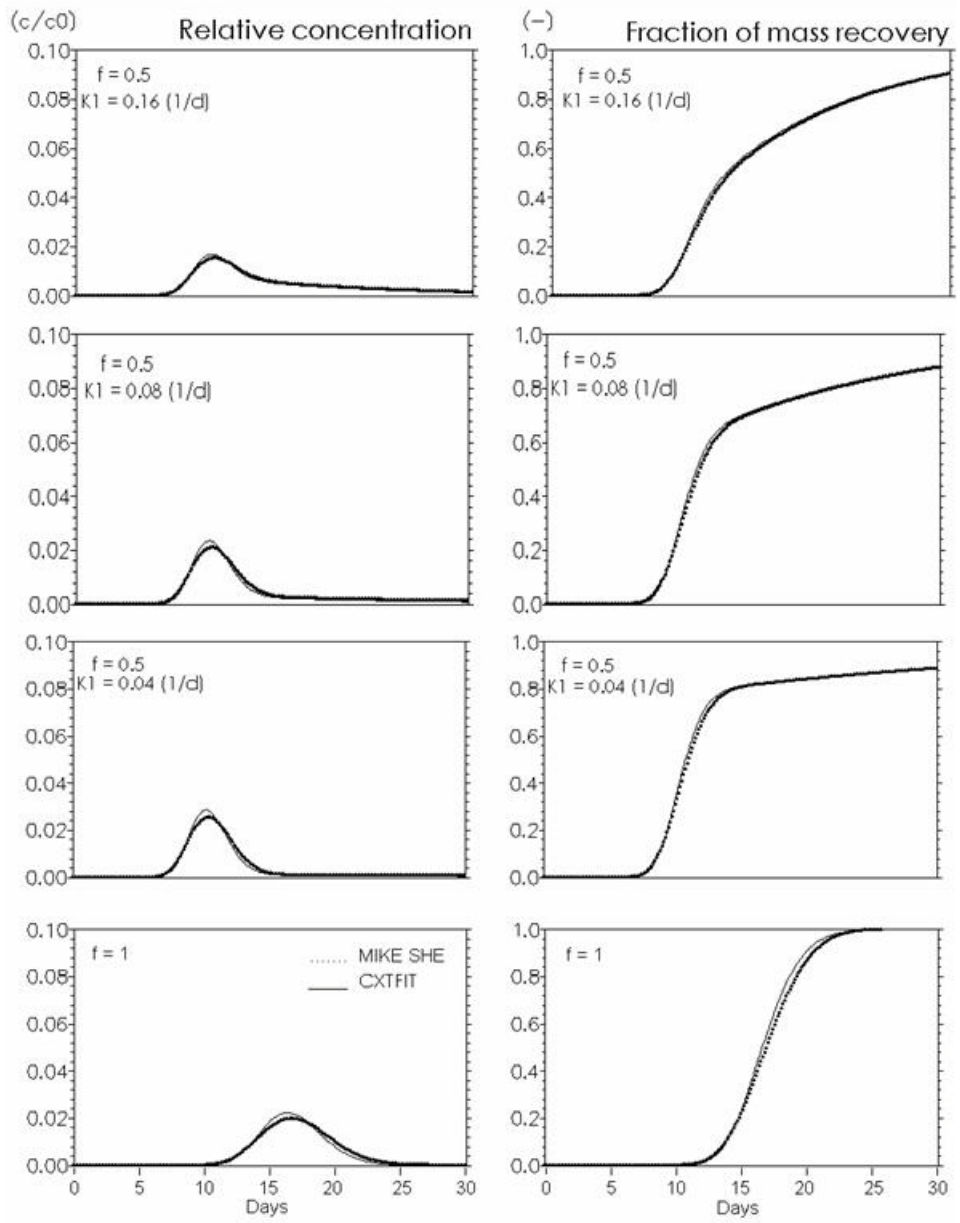


Figure 17.6 Kinetic sorption . Effects of rate constant K1 (d^{-1}) in Eq.(17.9). Input to CXTFIT: f (fraction of equilibrium sorption sites). K_d = 0.5 ml/g. Input to MIKE SHE: Kd = f*K_d (CXTFIT). At f=1 the function is reduced to equilibrium sorption with K_d = 0.5 ml/g.

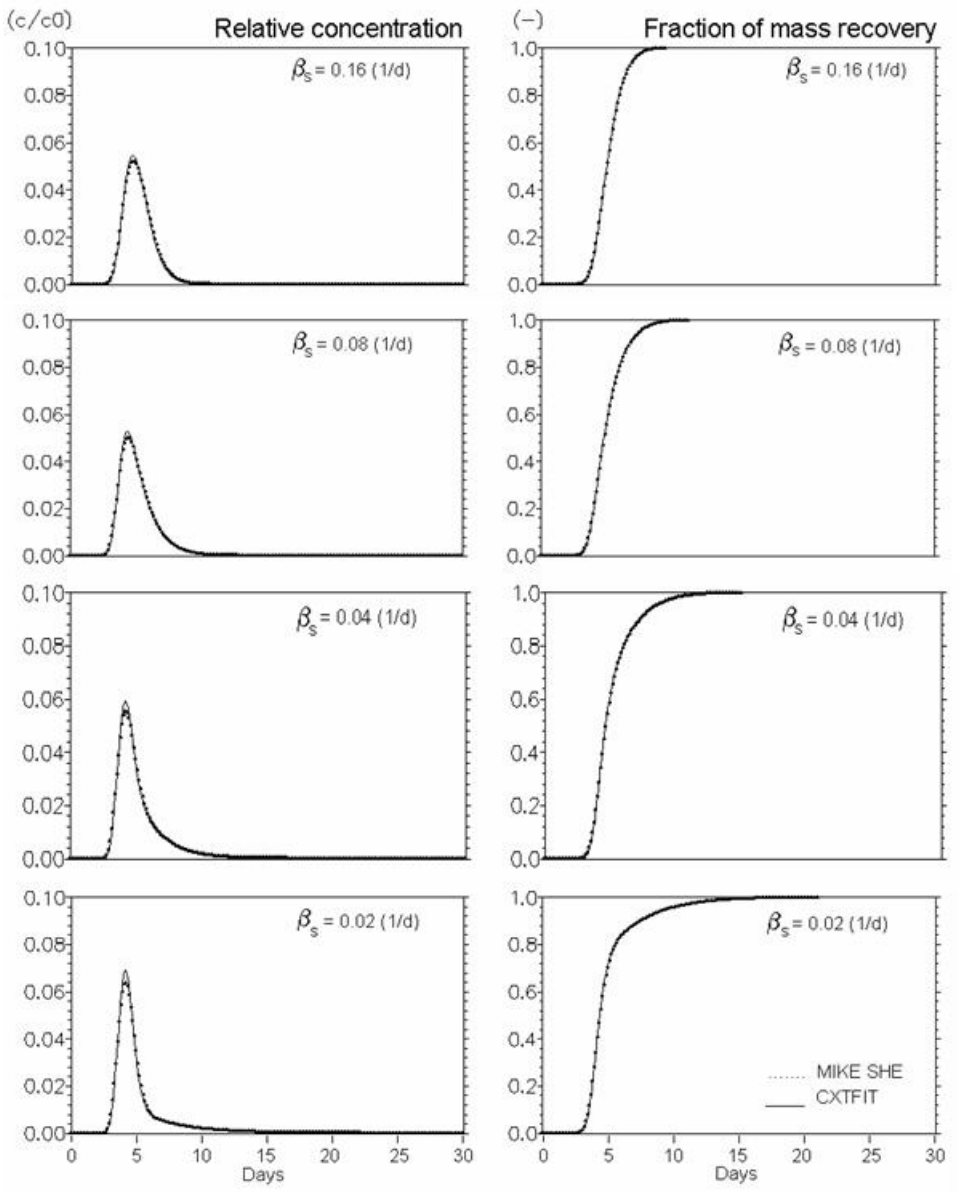


Figure 17.7 Conservative solute transport in dual porosity systems. Mobile porosity = 0.25, immobile porosity = 0.05. Effect of mass transfer coefficient (βs in Eq. (16.16) (d^{-1})).

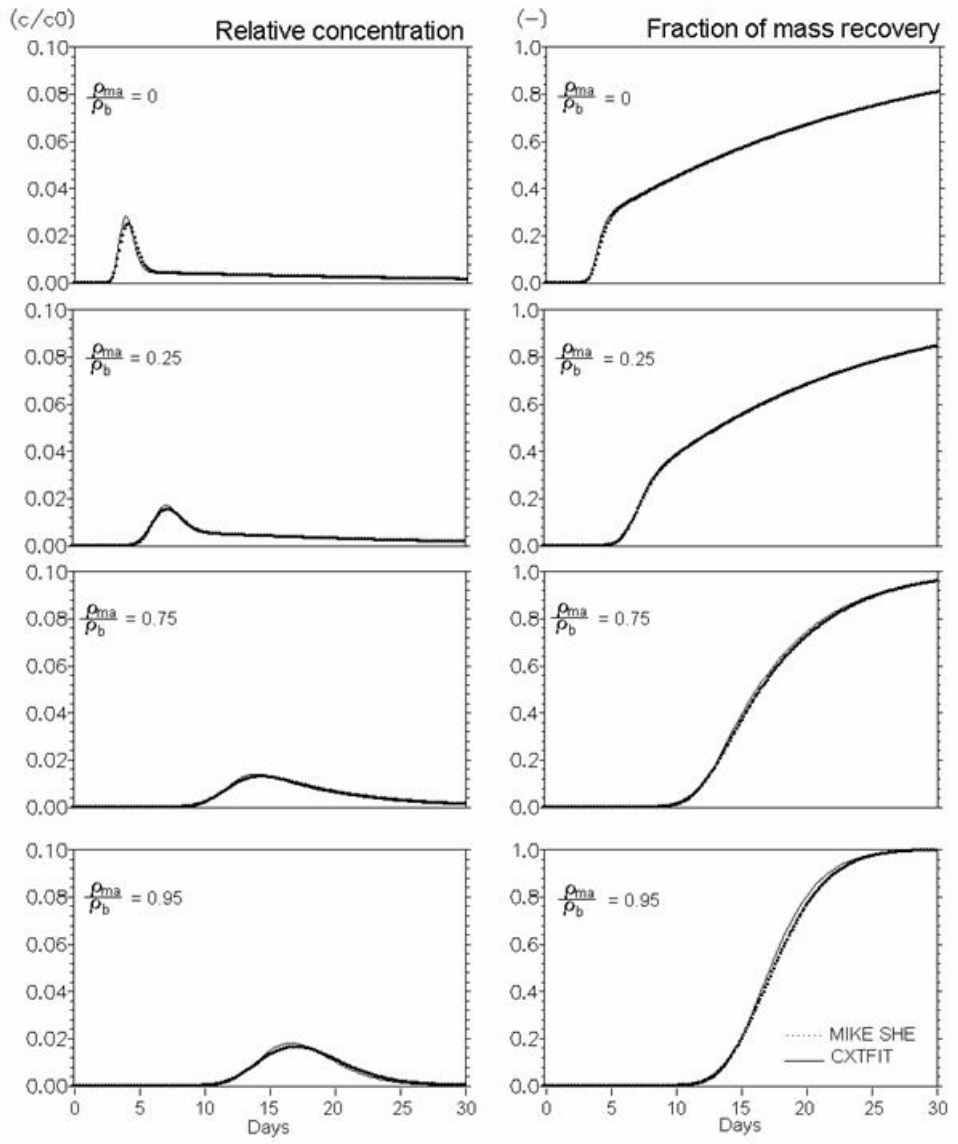


Figure 17.8 Verification of reactive solute transport in dual porosity systems. Mobile porosity = 0.25, immobile porosity = 0.05. Effect of distribution of sorption sites between mobile and immobile regions (ρ_{ma}/ρ_b Eq. 47) (Kd = 0.5 ml/g, mass transfer coefficient (β_s) = 0.08 d^{-1}).

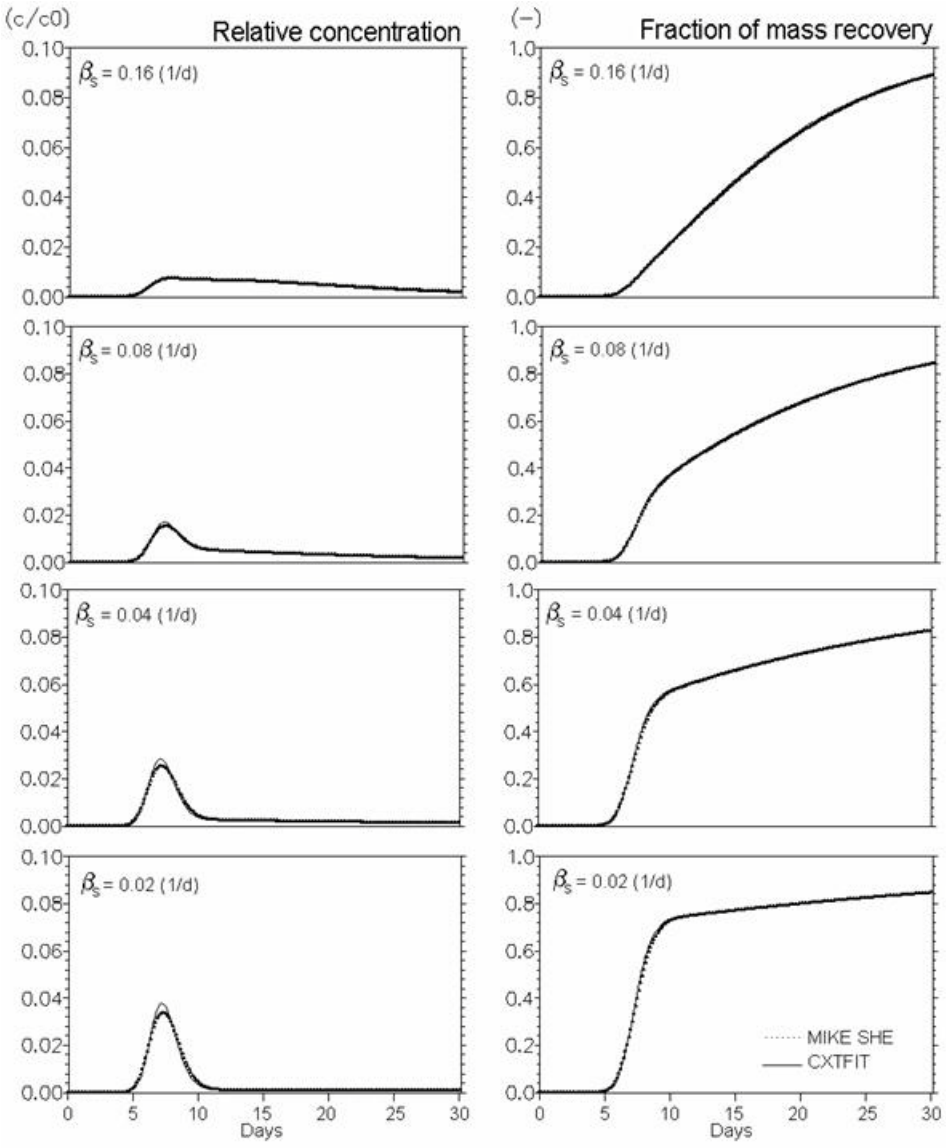


Figure 17.9 Verification of reactive solute transport in dual porosity systems. Mobile porosity = 0.25, immobile porosity = 0.05. Effect of mass transfer coefficient (βs in Eq. (16.16) (d^{-1})). Kd = 0.5 ml/g, ρ_{ma}/ρ_b = 0.25 (Eq. (17.10)).

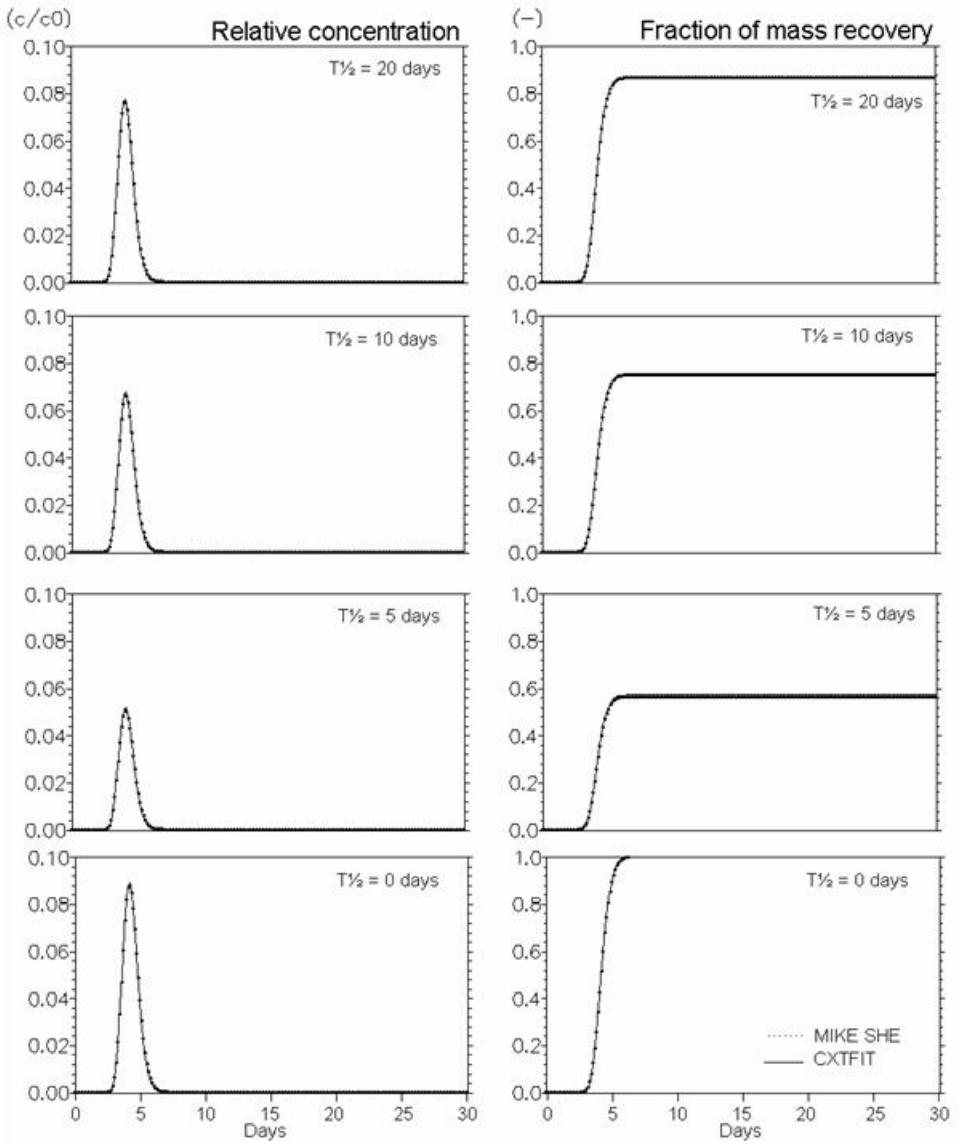


Figure 17.10 Verification of the description of first order degradation. Effect of half life time - T½ (Λ) (days) (Eq. (17.12)).

17.4.1 Other Processes - Simulation Examples

Some of the process descriptions included in MIKE SHE are too complex to be verified against analytical solutions, including transport of “reactive” solutes in the unsaturated zone under different soil hydrological conditions, plant uptake of solutes, transport in macropores in the unsaturated zone and the influence of temperature and soil moisture content on degradation processes.

To illustrate one of these cases, plant uptake of solutes is simulated in Figure 17.11. If plant uptake is included a “concentration factor” (see Eq. (17.19))is needed, indicating to which extent solutes are taken up by the plants.

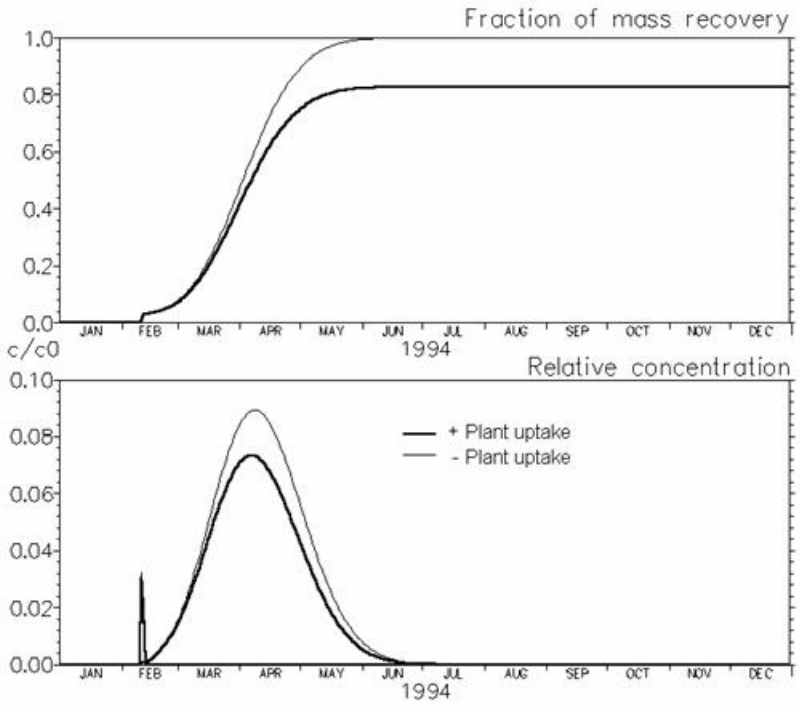


Figure 17.11 Illustration of the effect of plant uptake on solute breakthrough. Plant uptake was simulated with $f_c = 0.5$. The early peak concentrations arise from macro pore transport and were almost alike in the two simulations.

18 Working with Solute Transport - User Guide

The complete MIKE SHE advection-dispersion (AD) module is comprised of four independent components, each describing the transport processes in one of the parts of the hydrological cycle. Used in combination they describe solute transport in the entire hydrological cycle. The four components are:

- Overland Transport
- Channel Transport (MIKE Hydro River)
- Unsaturated Zone Transport
- Groundwater Transport

A number of processes relevant for simulating reactive solute transport are included in MIKE SHE including

- Water and solute transport in macro pores,
- Sorption of solutes described by either equilibrium sorption isotherms (Linear, Freundlich or Langmuir) or kinetic sorption isotherms, which include effects of hysteresis in the sorption process,
- Attenuation of solutes described by an exponential decay, and
- Plant uptake of solutes.

Current Limitations

The solute transport module in MIKE SHE currently does not support

- exchange **from** MIKE Hydro River **to** Overland flow,
- any solute transfer via irrigation,
- any solute transfer via flood codes, and
- solute migration **from** UZ **to** OL.

In the first three cases, the solutes will remain in the source location and only the water will be transfered. This will lead to increasing concentrations at the source.

In the last case, there is no mechanism in MIKE SHE to transfer water from UZ to OL, so there is also no means to move solutes from the UZ cells onto the ground surface. This has implications for salinity modelling, as there is no way for runoff to remove surface salts that migrate upwards due to capillarity and concentrate on the ground surface due to evaporation.

18.1 Flow Storing Requirements

Solute transport calculations in MIKE SHE AD are based on the water fluxes from a MIKE SHE Water Movement (WM) simulation. To ensure that all the needed WM result data types are stored, you have to specify that results should be stored for an AD simulation. See Storing of Results *(V1 p. 325)*.

The WM data should be stored frequently enough to describe the dynamics of the flow. The selected storing frequencies of flow results will usually be a compromise between limitations in disk space and resolution of the flow dynamics. The maximum computational time steps in a transport simulation are often restricted by advective and dispersive stability criterions. However, the transport time step cannot be greater than the flow storing time step in each component.

18.2 Storing of Results

The simulated concentration distribution in each component as well as the mass balances and fluxes will be stored in dfs2 and dfs3 files with different time steps. Besides these result files, the program also writes output to the error log, which describes errors encountered during execution and a print log which contains execution step information, statistics on the run and a mass balance (if requested).

Normally, the results from the saturated zone (species concentration in each grid) is by far the most disk consuming parameter. So, be careful with the storing time step. Mass balances, which includes time series of mass storage and fluxes between components (and sources, drains, boundaries, etc.) can be stored at smaller time steps.

When you select the time step you should also be aware of the time scale of the process. The time scale for transport processes in groundwater is usually much larger than the time scale for transport in a river.

Enter the desired time steps - notice that the unit is hours - in each of the edit fields. There are no limitations on this time step but if you select a time step less than the simulation time step, the storing time step will be the new simulation time step.

18.3 Simulation and Time Step Control

Simulation time steps are to some extent controlled by the user. Several possibilities for time step control exist to make the execution as fast as possible with no numerical dispersion and instabilities.

The first possibility for controlling the simulation time steps in the different components is simply to define the maximum time step in each component. Note that time steps should be given in increasing order i.e. $dt_{RIVER} \leq dt_{OVER-}$

$_{LAND} \leq dt_{UZ} \leq dt_{SZ}$. Also note that this is the MAXIMUM time step. That is, the actual simulation time step is controlled by the stability criterions with respect to advective and dispersive transport given below. Furthermore, time steps for transport cannot exceed the storing time step for the relevant data in the flow result file from a MIKE SHE flow simulation.

Enter the maximum allowable Courant number for each component. The Courant number is defined by *V x dt/dx* (velocity times time step divided by “grid size”). This number should normally not exceed 1.0 for one- and two-dimensional transport (UZ, Overland and Channel Flow) and 0.8 for three-dimensional transport (SZ). The maximum time step will be calculated accordingly.

Enter the maximum allowable dispersive Courant number for each component. The dispersive Courant number is defined by *D x dt/dx2* (Dispersion coefficient times time step divided by “grid size” squared). This number should normally not exceed 0.5. The maximum time step will be calculated accordingly.

The transport limits are used to avoid negative concentrations in cases with extreme gradients (e.g. close to sources) or in areas with highly irregular velocity fields. Enter the maximum allowable transport from a node or grid as a fraction of the storage in the node or grid. A recommended value for all components is 0.9, which ensures that this option is in use (the value 0 determines that this option is not in use).

18.3.1 Calibrating and verifying the model

The advection-dispersion of solutes depends largely on the simulated flows and fluxes calculated by the MIKE SHE flow model. After your first AD simulations, you will usually have to go back and improve the calibration of your flow model. Rarely, can the simulated concentrations and mass fluxes be calibrated to the measured concentrations by tuning only the solute transport model.

It is important to recognise that a transport model must be calibrated. This is true for all applications larger than the laboratory scale since model output cannot necessarily be compared directly to measured values. Measurements are mostly point measurements at a certain time whereas results often are mean values over larger volumes and longer times.

The purpose of the calibration is to tune the model so that it is able to reproduce measured conditions for a particular period in a satisfactory way. This period - known as the calibration period - should be chosen long enough to include events of similar kind as the ones you are going to investigate.

A satisfactory calibration is reached when the model is able to reproduce the measured values taking the following conditions into account:

- uncertainty in the measurements (time, space, equipment)
- representativeness of measurements (point/average grid values)
- differences between your conceptual model and nature
- uncertainty in other model parameters and data (source description etc.)

In general, it is impossible to specify an exact level of divergence between measured data and computed results before the model is satisfactorily calibrated. In each application you have to consider all factors influencing your result.

After the calibration, you should verify your model by running one or more simulations for which measurements are available without changing your model parameters. If the model is able to reproduce the validation measurements you can consider your calibration to be successful. This ensures that simulations can be made for any period similar to the calibration and the verification period with satisfactory results.

18.4 Executing MIKE SHE WQ

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

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

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

MIKE SHE WQ can also be launched from a batch file with or without the MZLaunch function. For more information on this, see Using Batch Files *(V1 p. 80)*

18.5 Output

The output of the MIKE SHE AD is stored to several dfs0, dfs2 and dfs3 files which can be viewed and processed with the different tools available for these files in MIKE ZERO.

For each species, a concentration file is created for each hydrologic process - a dfs2 file for OL, and a dfs3 file each for UZ and SZ.

For each species, the total WQ mass balance is stored in two dfs0 files. The xx_species_AllItems.dfs0 includes all of the possible water quality mass bal-

ance items. The xx_species.dfs0 is a copy of the _AllItems.dfs0 file with all of the non-zero items removed.

The first 20 items in the _AllItems.dfs0 file define the global mass balance (see Table 18.2). There are four items: Storage, Input, Output and Error. Each of these is calculated for each of the five storage items: SZ, Immob(SZ), UZ, MP(UZ), and OL.

The item Immob(SZ) is for solutes stored in the SZ matrix porosity when the dual porosity option in SZ is turned on. In this case, water and solutes move in the fractures and solutes diffuse into the rock matrix. The fractures are then the primary porosity.

The item MP(UZ) is for solutes stored in the UZ macropores when the macropore option in UZ is turned on. In this case, water and solutes move in both the matrix and the macropores, but the volume of water in the macropores is generally much less than the volume of water in the matrix. So, the primary porosity is the UZ matrix.

The Error is calculated for each of the five items as:

$$Error = Output - Input + \Delta Storage$$

However, the mass in the SZ and UZ items includes the mass in both the primary and secondary porosities.

The Output, Input and Storage are all displayed as positive values in the dfs0 file and the WQ log file. A positive change in storage denotes an increase in mass.

Table 18.1 WQ mass balance items in the _AllItems.dfs0 file

Mass balance item	Component
Storage	SZ, Immob(SZ), UZ, MP(UZ), OL
Input	SZ, Immob(SZ), UZ, MP(UZ), OL
Output	SZ, Immob(SZ), UZ, MP(UZ), OL
Error	SZ, Immob(SZ), UZ, MP(UZ), OL

The rest of the items in the _AllItems.dfs0 file are only non-zero if the item is relevant for the current WQ simulation. Table 18.2 lists all of the rest of the items in the _AllItems.dfs0 organized by the source of the solute.

Table 18.2 Available WQ mass balance items in the _AllItems.dfs0 file

From	To	Comment
Sources	SZ, UZ, OL, River	Note that sources can be specified as positive or negative.
Ext. Sources (OpenMI)	SZ, UZ, OL	This is non-zero only if a model is linked to MIKE SHE by OpenMI that adds mass to the component.
Ext. Input (OpenMI)	SZ Drain	This is non-zero only if a model is linked to MIKE SHE by OpenMI that adds mass to the component. However, this is a special case because you can add mass directly to the SZ drain without it actually becoming part of the SZ model. The mass is then added to the model at the location were the drain discharges (i.e. river link, SZ boundary, or local SZ depression)
Boundary	SZ, UZ, OL, River	The (Boundary to River) item is typically zero, but can be non-zero in a couple of rare cases: If you have 1) a fixed head SZ boundary, or 2) a fixed concentration boundary, next to a river link, then mass from this cell to the river will be added to (Boundary to River).
Precip	UZ, OL	Mass from precipitation is always added to either OL or UZ, even though precipitation can be added to SZ in the WM module. If you have a SZ-only simulation, then mass from precipitation is included in the (Precip to OL) and then (OL to SZ)
Decay	SZ, Immob(SZ), UZ, MP(UZ), OL	This is the mass that has decayed in each of the processes
Sorp/DeSorp	SZ, Immob(SZ), UZ, MP(UZ), OL	This is the net Sorption and Desorption to the soil matrix in each of the processes. If mass is sorbed to the soil matrix it is removed from solution and this value will be negative. If mass desorbs from the soil matrix it is added to solution and this value will be positive.

Table 18.2 Available WQ mass balance items in the _AllItems.dfs0 file

From	To	Comment
Colloid-Sorp/DeSorp	SZ, Immob(SZ), UZ, MP(UZ), OL	This is the net Sorption and Desorption to colloids in each of the processes. If mass is sorbed to the colloids it is removed from solution and this value will be negative. If mass desorbs from the colloids it is added to solution and this value will be positive. However, this is normally zero, because colloid transport is not available in the commercial version of MIKE SHE, only a research version.
EcoLab	SZ, Immob(SZ), UZ, MP(UZ), OL	This is the mass change resulting from passing solutes to and from MIKE ECO Lab - positive if MIKE ECO Lab causes the mass to increase and negative if mass decreases.
SZ	UZ, MP(UZ), OL, Sinks, Sources, Ext.Sinks(OpenMI), Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab, Plant Uptake	Note that there is no mass transfer to SZ drains because the SZ drains have not storage. Mass that discharges to SZ drains passes straight through the drain and is added to the end recipient (i.e. a river, boundary, or local depression.
Immob(SZ)	UZ, Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab	
SZ Baseflow	River	This is a special item that includes only the mass from SZ to rivers.
SZ Drain	River, SZ (Local Dep), Boundary	This is a special item that divides up the SZ mass discharge to drains by the end recipient.
SZ Flow	Boundary	This is a special item to distinguish between SZ mass discharge to boundaries via drains and direct discharge to the boundary.
SZ(Fract)	Immob(SZ)	This is a special item that includes only the mass exchange between fractures and the matrix when the dual porosity option is selected in the SZ.

Table 18.2 Available WQ mass balance items in the _AllItems.dfs0 file

From	To	Comment
UZ	SZ, Immob(SZ), OL, Sinks, Sources, Ext.Sinks(OpenMI), Boundary, Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab, Plant Uptake	Note that the (UZ to Boundary) item refers to mass discharge from UZ to SZ boundaries. This can arise, for example, when a UZ column discharges into an SZ cell that contains an internal boundary condition, such as a constant head. Note that the (UZ to Immob(SZ)) item will only be non-zero when the dual porosity option in SZ is turned on. In this case, as the water table increases, there will be a transfer of UZ mass to water in both the SZ fractures and SZ immobile matrix based on the ratios of their porosities. Related to the above, if macropores are active, then mass in the UZ macropores will be distributed to both the SZ matrix and the SZ fractures.
MP(UZ)	SZ, Immob(SZ), UZ(Matr), Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab	
UZ(Matr)	MP(UZ)	This is a special item that includes only the mass exchange between macropores and the matrix when the macropore option is selected in the UZ.
OL	SZ, UZ, MP(UZ), River, Sinks, Sources, Ext.Sinks(OpenMI), Boundary, Decay, Sorp/DeSorp, Colloid-Sorp/DeSorp, EcoLab	
River	SZ, OL, Boundary	(River to OL) is always zero because this exchange has not yet been implemented.

18.6 Coupling Water Quality in MIKE SHE and Rivers

18.6.1 with MIKE Hydro River

Detailed information on the MIKE Hydro River Water Quality modules are found in the MIKE Hydro River documentation.

18.6.2 with MIKE 11

Detailed information on the MIKE 11 Water Quality modules are found in the MIKE 11 documentation.

The coupling between MIKE 11 and the rest of MIKE SHE's hydrologic processes is relatively automatic. You must set up a MIKE 11 WQ model independent of MIKE SHE and specify this .sim11 file in the Rivers and Lakes dialogue. This .sim11 file must only have the same network geometry as the WM .sim11 file. It does not have to be the same .sim11 file.

The MIKE 11 WQ model can also include MIKE ECO Lab, which will allow you simulate eutrophication, etc. in the surface water.

There are a few caveats/limitations that you need to be aware of:

- Species names must be identical in MIKE SHE and MIKE 11. If they are not identical, then the solutes will be transferred to the river as an infinite sink, but will not be transported in MIKE 11.
- The overland WQ must be included if you want to simulate water quality coupled to MIKE 11.
- Recycling of WM results is not supported in MIKE 11. This means that if you want to simulate the coupling between MIKE 11 and the rest of MIKE SHE, your WQ simulation must be continuous.

18.6.3 WQ and MIKE 11 River Structures

Normally, MIKE 11 will transport solutes without trouble, but this can be complicated during the coupling of WQ to MIKE SHE.

In particular, solutes will not be transported through MIKE 11 structures, such as wiers, unless the checkbox "structures" in the HD parameters file, tab "Add Output" is activated.

The reason is that MIKE SHE takes the HD result file and uses it to compute the WQ transport in the river model too. Therefore it requires the HD discharge at all structures, which is only provided in the additional outputs of the MIKE 11 simulation.

19 Working with MIKE ECO Lab in MIKE SHE - User Guide

MIKE ECO Lab is an open and generic equation solver. MIKE ECO Lab is mostly used for water quality and ecosystem modelling, such as modelling eutrophication of lakes, calculating the fate and transport of heavy metals and determining ecology indicators (e.g. distribution of sea grass). Originally, MIKE ECO Lab was developed as a tool to simulate water quality reactions in surface water, but has been expanded to include agent based modelling and other more complex reactions and components.

In MIKE SHE, MIKE ECO Lab can be used from basic water quality kinetic reactions in surface and groundwater, to complex coupled feedback interactions between nutrients, plant growth and hydrology.

MIKE ECO Lab in MIKE SHE depends on MIKE SHE's WQ module You must run a WQ simulation after the WM simulation and the WQ module calls MIKE ECO Lab.

When running MIKE SHE + MIKE ECO Lab, MIKE ECO Lab reads concentrations (state variables) from MIKE SHE's WQ module, reads other necessary input data files, generates additional output and passes modified concentrations (state variables) back to MIKE SHE.

MIKE ECO Lab acts on a cell basis. It is called for each cell in the model. By default it is called at every time step in the MIKE SHE WQ simulation, but optionally can be called less frequently.

Using MIKE ECO Lab is a four step process:

1. Create a MIKE ECO Lab template file that defines the equations to be solved.
2. Specify the name of the template file in the MIKE SHE.
3. Define the links between template variables and MIKE SHE parameters.
4. Run the MIKE SHE Water Quality model.

During the simulation, MIKE SHE passes the State Variables (eg concentrations) to MIKE ECO Lab. MIKE ECO Lab updates the State Variable values and passes them back to MIKE SHE. At the same time, MIKE ECO Lab will write to any specified output files.

The output files are standard dfs2 or dfs3 output files. These files can be used as input in subsequent WM or other WQ simulations, or viewed in the Results Viewer, etc.

19.1 MIKE ECO Lab Templates

A MIKE ECO Lab Template contains the mathematical description of the ordinary differential equations to be solved. These could, for example, describe an ecosystem including the processes affecting the ecosystem.

A MIKE ECO Lab template contains six components:

- State Variables - State variables represent the state that the user wants to predict (at least one state variable must be specified).
- Constants - Constants are used as arguments in the mathematical expressions of processes in a MIKE ECO Lab model. They are constant in time, but can vary in space.
- Forcings - Forcings are used as arguments in the mathematical expression of processes in a MIKE ECO Lab model. They can vary in time and space. They basically represent variables of an external nature that can affect the ecosystem.
- Auxiliary Variables - Auxiliary Variables are arguments in the mathematical expression in a MIKE ECO Lab model. They can be used as intermediate calculations that can include any state variable, constant or forcing. They can also be used to specify results directly.
- Processes - Processes describe the transformations that affect the state variables. That means processes are used as arguments in the differential equations that MIKE ECO Lab solves to determine the state of the State Variables.
- Derived Outputs - Derived Outputs are output files that are derived based on the model results.

Additional details on developing MIKE ECO Lab templates is available in the MIKE ZERO ECO Lab manual.

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].

19.1.1 Developing a template

Creating and developing a MIKE ECO Lab template involves several steps.

1. Create a MIKE ECO Lab template

First you must create a MIKE ECO Lab template from the File/New menu or the New File icon. In both cases, you will chose MIKE Zero and then MIKE ECO Lab (.ecolab) in the New File dialogue. This will create a new blank MIKE ECO Lab template file.

Alternatively, you can copy and edit an existing MIKE ECO Lab template.

A few tips will be useful before you start.

- You should try to keep the names of the Constants, Forcings, etc. as short as practical. The names are used when defining Processes, Auxiliary Variables, and Derived Outputs.
- The names used in the definitions are case-sensitive.
- The names must be unique within the list of Constants, Forcings etc.
- To add a new Constant, Forcing, etc. right click on the item and chose the appropriate option.

2. Add State Variables to the Template

The only available State Variables are species concentrations. You must add one State Variable item for each Species in MIKE SHE that you want MIKE ECO Lab to modify during the WQ simulation.

The MIKE ECO Lab template must include at least one State Variable.

Note: The State Variable name must be exactly the same as the Species name in MIKE SHE.

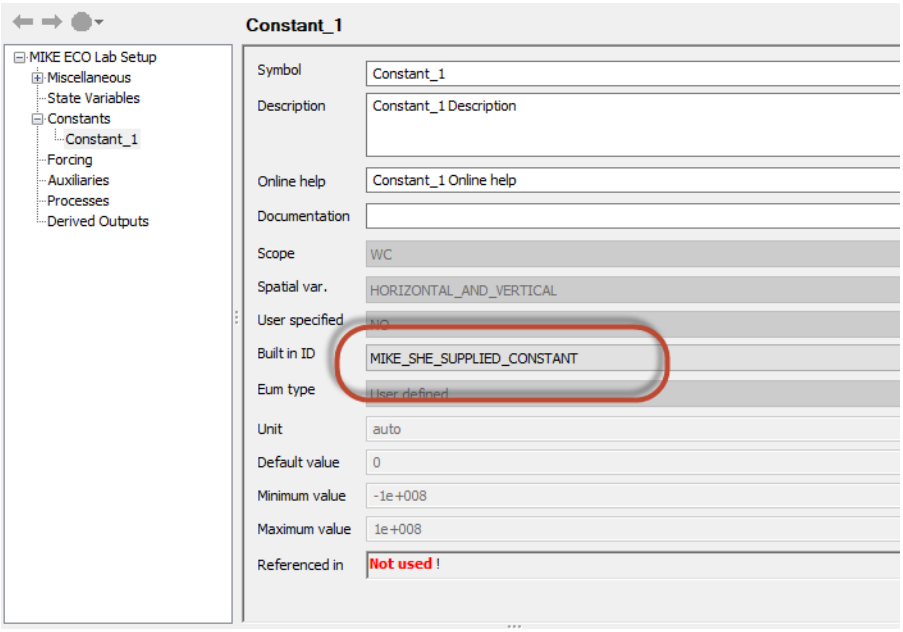
The exception to the exact naming rule is when simulating Dual domain mass transfer. In this case, the State Variable name must use the reserved suffix "_2" for the solute in the secondary porosity. For example, OXYGEN and OXYGEN_2 would be the State Variable names for the species OXYGEN in MIKE SHE.

3. Add one or more Constants

Constants are spatially distributed values that are constant in time. Each constant is either pre-defined by the MIKE SHE setup (MIKE_SHE_SUPPLIED_CONSTANT), or it is User Defined in the MIKE SHE model. All other values will be ignored.

Pre-defined MIKE SHE Supplied Constants

If the Constant is defined by the MIKE SHE model setup, then the Built in ID must be MIKE_SHE_SUPPLIED_CONSTANT.



The available pre-defined MIKE SHE Supplied Constants are different depending on hydrological context (ie Overland Flow, UZ or SZ)..

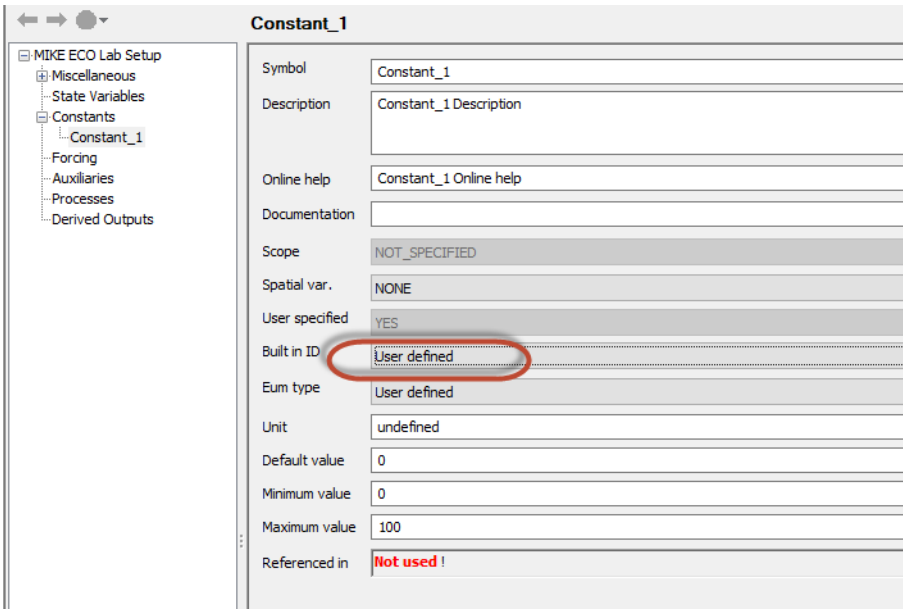
Table 19.1 Available MIKE SHE Supplied Constants

Overland Flow	Unsaturated Zone	Saturated Zone
Cell size	Cell size	Cell size
Topography	Cell volume	Cell volume
Detention storage	Topography	Topography
	Depth to cell top	Depth to cell top
	Depth to cell bottom	Depth to cell bottom
	Saturated water content	Primary porosity
	Macropore porosity	Secondary porosity
	Primary bulk density	Primary bulk density
	Secondary bulk density	Secondary bulk density

The Symbol name that you define is carried over to the MIKE SHE list of Constants. In the MIKE SHE list of constants, you will define the actual value that will be supplied by MIKE SHE to MIKE ECO Lab.

User Defined Constants

If your Constant is not a pre-defined MIKE SHE constant, then you must define a value for the constant. In this case, you must define it as User Defined Constant.



The actual values and spatial distribution of the Constant will be defined in the MIKE SHE Setup Editor. Any User defined Constants will be added to the list of constants under each of the domains (ie OL, UZ and SZ).

In the template, the only thing you need to specify is the name of the Constant. The name is then used when defining the Processes, Derived Outputs, etc.

If you have already defined the Constant in the Setup Editor, you may need to define it again in the list of Constants in the Setup Editor. This is most easily done by re-loading template with the browse button.

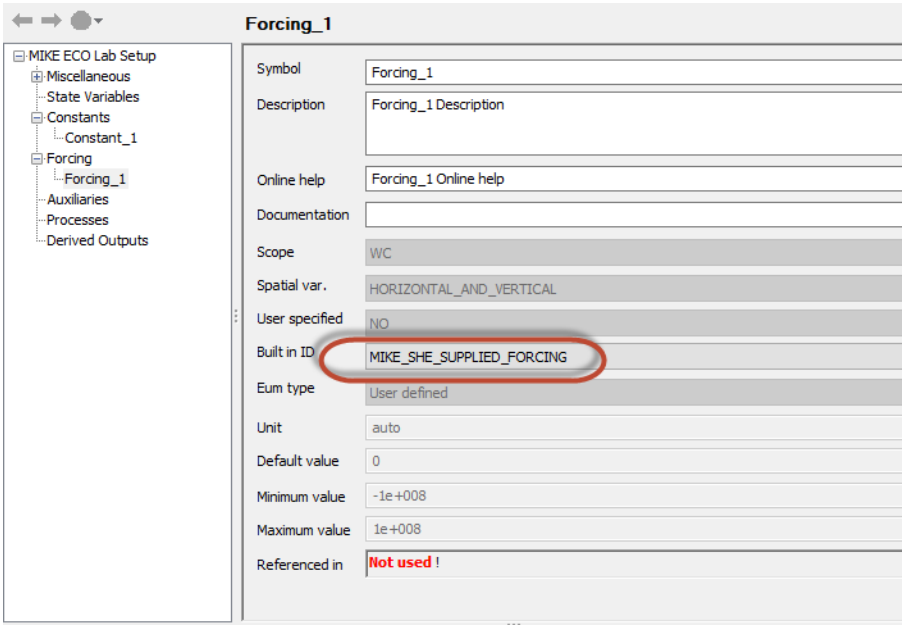
4. Add one or more Forcings

A Forcing is any spatially distributed value that is time varying. You can think of it as a value that is affecting the State Variable during the simulation. For example, air temperature will affect the degradation rate of a solute.

Similar to the Constants, each forcing is either pre-defined by the MIKE SHE setup (MIKE_SHE_SUPPLIED_FORCING), or it is User Defined in the MIKE SHE model. All other values will be ignored.

Pre-defined MIKE SHE Supplied Forcings

If the Constant is defined by the MIKE SHE model setup, then the Built in ID must be MIKE_SHE_SUPPLIED_FORCING.



The available pre-defined MIKE SHE Supplied Forcings are different depending on hydrological context (ie Overland Flow, UZ or SZ)..

Table 19.2 Available MIKE SHE Supplied Constants

Overland Flow	Unsaturated Zone	Saturated Zone
Time step length	Time step length	Time step length
Precipitation rate	Precipitation rate	Precipitation rate
Air temperature	Air temperature	Air temperature
SW solar radiation	SW solar radiation	SW solar radiation
Depth of overland water	UZ Water content	Depth to phreatic surface (positive)
Depth to phreatic surface (positive)	Macropore water content	SZ potential head
SZ potential head	Depth to phreatic surface (positive)	SZ saturated thickness
	SZ potential head	Soil temperature
	Soil temperature	

The Symbol name that you define is carried over to the MIKE SHE list of Forcings. In the MIKE SHE list of forcings, you will define the actual value that will be supplied by MIKE SHE to MIKE ECO Lab.

Note: The various other “MIKE_SHE_” forcings are used by MIKE Hydro River to define concentrations and mass of solute entering the river.

5. Create Auxiliary Variables, Processes and Derived Outputs

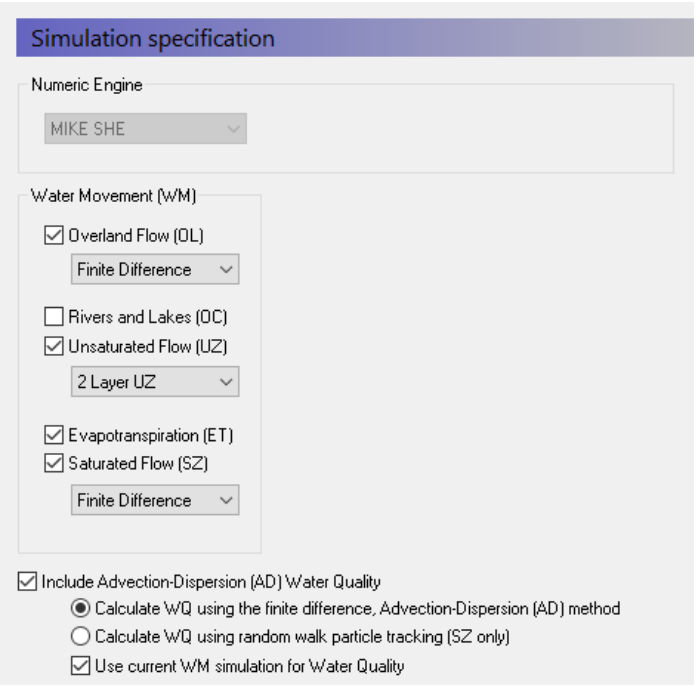
Auxiliary variables are used for intermediate calculations and must be defined as 3D (UZ and SZ) or 2D variables.

Processes transform a State Variable or calculate another result. Spatial variation and type must be defined for each process. Each process can be included in the results file by choosing "YES" in the "Output" box.

Derived Outputs allow the user to define output files based on the process results.

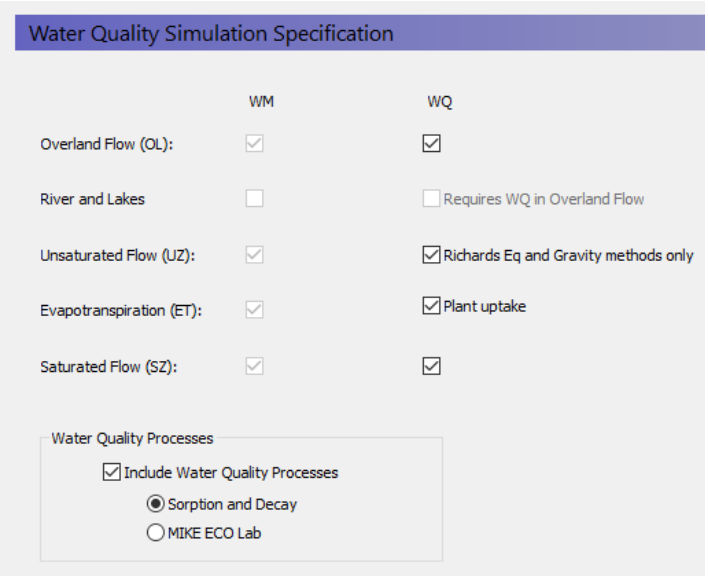
19.1.2 MIKE ECO Lab templates in MIKE SHE

MIKE ECO Lab is only available when Water Quality is selected. Thus, to be able to use MIKE ECO Lab, the Water Quality option in the main Simulation Specification dialogue must be selected

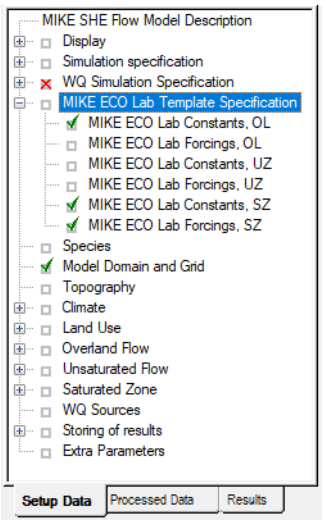


Note also that MIKE ECO Lab will only work with the Finite Difference AD method.

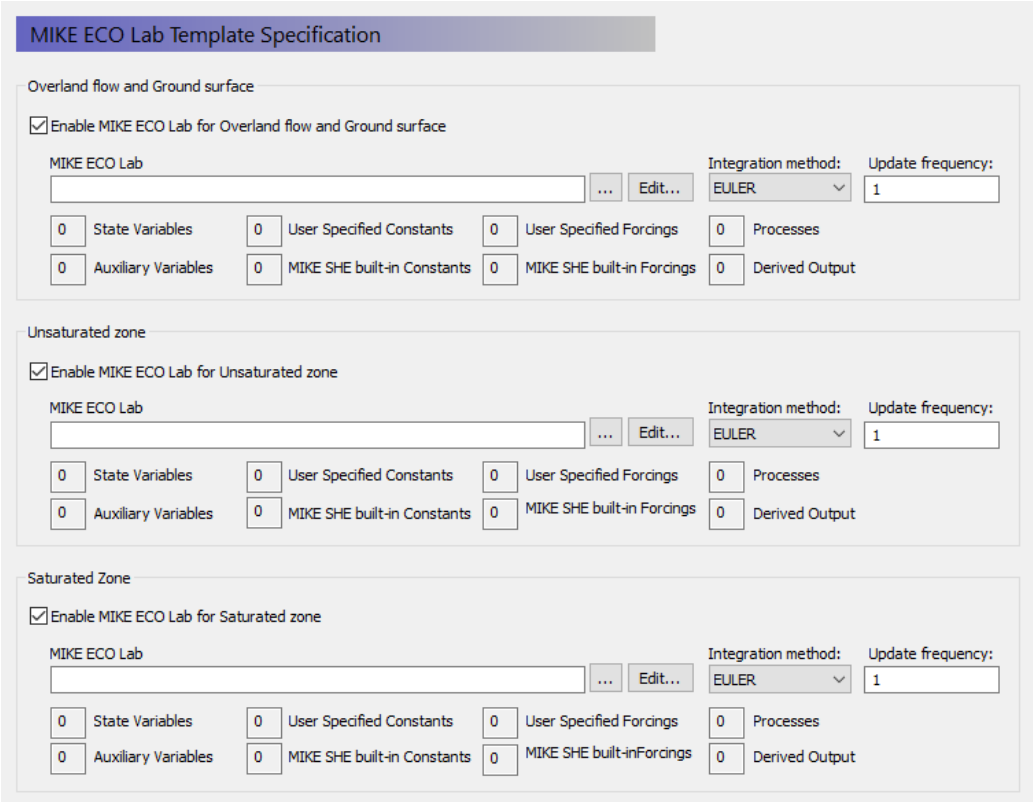
After activating the Water Quality module, the MIKE ECO Lab option must be selected in the WQ Simulation Specification dialogue.



A new data tree branch will appear, where MIKE ECO Lab templates can be specified for each of the hydrologic processes - Overland Flow, the Unsaturated Zone, and the Saturated Zone. Separate templates are required for each of these zones because the processes in each of these domains are very different



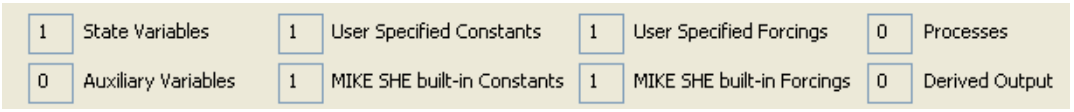
In the MIKE ECO Lab Template Specification dialogue, there is a checkbox for each of the processes. These checkboxes are active if the Water Quality is activated for the process in the WQ Simulation Specification dialogue



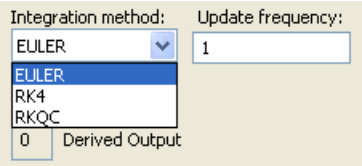
.

When you enable 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.



When calculating the concentrations (State Variables) for the next time step, an explicit time-integration of the transport equations is made. Depending on the desired accuracy of this numerical integration, you can chose one of three different integration methods. The methods are in increasing level of accuracy (and numerical effort), starting with the Euler method and finishing with the Runge Kutta 5th Order method).



Finally, for each template you can specify an update frequency (see above). The update frequency tells MIKE SHE how frequently to call 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.

19.1.3 State Variables in MIKE SHE

The State Variables defined in the MIKE ECO Lab template are passed to MIKE SHE as Species. This is the only way to pass information from MIKE ECO Lab to MIKE SHE.

The Species Name in MIKE SHE 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").

There are four species types in MIKE SHE. Species can be either:

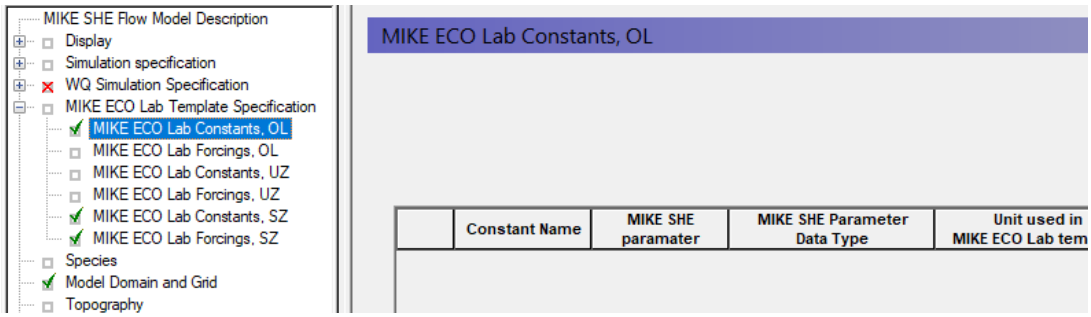
- **Dissolved**
  Dissolved species are mobile in the water. They are active in the subsurface and surface water. Disolved species have a default concentration of [$\mu g/m^3$].
- **Sorbed**
  Sorbed species are only available in the subsurface. They are fixed to the soil matrix and do not move with 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 [$\mu g/m^3$].
- **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.

In particular, the Fixed species is especially interesting in MIKE SHE. It is a species type without pre-defined units of concentration. The non-dimensional

species cannot be transported with the flow and can be used as a book-keeping mechanism for resulting processes. MIKE ECO Lab itself can read a value from any file, update the value and write it back to the file. However, MIKE ECO Lab cannot append a new value to a file. That is, MIKE ECO Lab cannot read a value, update the value and add the new value as a new time step in file. The Fixed species type gives you a mechanism for handling various auxiliary user defined quantities during the simulation. In other words, MIKE ECO Lab can read the current Fixed species value, and return a new value to MIKE SHE. MIKE SHE moves forward in time, and MIKE ECO Lab starts over again. In the mean time, MIKE SHE has saved the value from the previous time step. This mechanism greatly increases the flexibility of the MIKE ECO Lab coupling in MIKE SHE. It allows you for example, to create things like ecological indexes that map changing ecohydrologic conditions over time.

19.1.4 Specifying Constants and Forcings in MIKE SHE

When you browse to the MIKE ECO Lab template, the template is read by MIKE SHE's Setup Editor to find the MIKE_SHE_SUPPLIED_FORCINGS and MIKE_SHE_SUPPLIED_CONTANTS used in the template. These are separated into Built-in and User Specified Forcings and Constants. In both cases, the Forcing or Constant is added to the appropriate list of Forcings and Constants in the MIKE SHE data tree - separated into the different domains (OL, UZ, and SZ).



Built-in Forcings and Constants

If the Forcing or Constant is not user defined (User Defined = "NO"), then the Forcing or Constant is an internal value in MIKE SHE and will be passed automatically to 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).

After selecting the parameter from the list of available parameters in the combo box, select the units that are being used in the template. The list of units is taken from the standard available units in the EUM database for the particular item. The Constant or Forcing can be used in various equations in the template. However, there is no check on the units being used. So, it is

expected that the Forcing or Constant will used one of the options from the list of units in the EUM database. Otherwise, and appropriate conversion must be done in the template equation.

User Specified Forcings and Constants

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

In this case, there is nothing to specify on the main list of Forcings and Constants, but a Forcing or Constant item is added to the data tree down under the appropriate branch in the data tree. In this branch you will find a table of user specified Forcings and Constants, plus individual sections for each item. Each item can be spatially defined similarly to other constants or time varying values in MIKE SHE.

Note: The Forcings and Constants are defined by Water Quality layer in the Saturated Zone. Thus, you have to define at least one Water Quality Layer in the Saturated Zone.

The list of user defined Forcings and Constants is initially taken from the template. However, the list is not fully dynamic. Thus, if you add items to the template, these will be added to the list. However, if you remove items from the template, or change the name in the template, the item will not be removed from the list. This allows you maintain your data inputs while you are developing your template. Any data that is not used in the template will be pre-processed like any other data, but will not be used in MIKE ECO Lab. If you don’t need the items any more, you can delete them from the list.

Tip: The fact the list of user specified Forcings and Constants is not permanently linked to the template, allows you to pre-process any static or time-varying data and map it to the numerical grid. You can then use this pre-processed data in other grid operations in MIKE SHE.

19.1.5 Running MIKE ECO Lab with MIKE SHE

To run MIKE ECO Lab with MIKE SHE, simply pre-process and run the model normally. You can view the user specified Constants and Forcing in the Pre-processed tab. The Derived Outputs will be written to the default output directly along with all of the regular output from MIE SHE. The Auxiliary Variables and Processes will also be written to this directory, if the Output option is on.

All output can be viewed or processed normally with all of the regular MIKE Zero tools.

20 Particle Tracking-Reference

The MIKE SHE Particle Tracking (PT) module is an alternative description of solute transport. PT calculates the location of a number of particles at every time step. The particles are displaced individually in the three-dimensional, saturated groundwater zone (SZ). The movement of each particle is composed of a deterministic part, where the particle is moved according to the local groundwater velocity calculated by the MIKE SHE, and a stochastic part, where the particle is moved randomly based on the local dispersion coefficients.

The PT module is a part of the MIKE SHE Advection Dispersion module and many of the same equations are used. For example, basically the same governing equation and the exchange of data with the MIKE SHE water movement simulation is similar in both modules. Therefore, you may find MIKE SHE AD Technical Reference a good source of additional information.

The PT module is typically used to delineate capture zones, upstream zones, and to determine groundwater age

20.1 Governing equations

The transport of solutes and particle tracking in the saturated zone is governed by the advection-dispersion equation, which for a porous medium with uniform porosity distribution is formulated as:

$$\frac{\partial c}{\partial t} + \nabla \cdot (\underline{u} c) - \nabla \cdot (\underline{D} \cdot \nabla c) = 0 \tag{20.1}$$

where c is the solute concentration, t is time, $\underline{u}$ is the groundwater pore velocity, and $\underline{D}$ is the dispersions tensor. In the particle model a large number of particles are moved individually in a number of time steps according to contributions from advective and dispersive transport. A particle mass is associated to each particle, which means that the number of particles in a cell corresponds to a solute concentration.

For isotropic conditions in the soil matrix the displacement of a particle is described by the following equation [Thompson et al., 1987].

$$\underline{X}_p(t_{n+1}) = \underline{X}_p(t_n) + [\underline{u}(\underline{X}_{p,n}, t_n) + \nabla \cdot \underline{D}(\underline{X}_{p,n}, t_n)]\Delta t + \underline{B}(\underline{X}_{p,n}, t_n) \cdot \underline{Z}_{p,n+1} \sqrt{\Delta t} \tag{20.2}$$

where $\underline{X}_p$ is the particle co-ordinates, $\Delta t = t_{n+1} - t_n$ is the time step length, $\underline{Z}_{p,n+1}$ is a vector containing three independent random numbers equally distributed in the interval *[-1, +1]* and

$$\underline{B} = \underline{R} \cdot \underline{B}^* \tag{20.3}$$

where

$$\underline{R} = \begin{bmatrix} \frac{u_x}{|\underline{u}|} & \frac{-u_y}{\beta} & \frac{-(u_y^2 + u_z^2 + u_x u_z)}{\beta|\underline{u}|} \\ \frac{u_y}{|\underline{u}|} & \frac{u_x + u_z}{\beta} & \frac{u_y(u_x - u_z)}{\beta|\underline{u}|} \\ \frac{u_z}{|\underline{u}|} & \frac{-u_y}{\beta} & \frac{u_x^2 + u_y^2 + u_x u_z}{\beta|\underline{u}|} \end{bmatrix} \tag{20.4}$$

$$|\underline{u}| = \sqrt{u_x^2 + u_y^2 + u_z^2} \tag{20.5}$$

$$\beta = \sqrt{|\underline{u}|^2 + 2u_x u_z + u_y^2} \tag{20.6}$$

$$\underline{B}^* = \begin{bmatrix} \sqrt{2(\alpha_L|\underline{u}| + D_m)} & 0 & 0 \\ 0 & \sqrt{2(\alpha_T|\underline{u}| + D_m)} & 0 \\ 0 & 0 & \sqrt{2(\alpha_T|\underline{u}| + D_m)} \end{bmatrix} \tag{20.7}$$

α_L and α_T are the longitudinal and transversal dispersion coefficients, respectively and D_m is the neutral dispersion. Using (20.2) repeatedly, the location of a particle at time $t_N = N\Delta t$ can be determined:

$$\underline{X}_p(t_N) = \underline{X}_p(t_o) + \sum_{n=0}^{N} ([\underline{u}(\underline{X}_{p,n}, t_n) + \nabla \cdot \underline{D}(\underline{X}_{p,n}, t_n)]\Delta t + \underline{B}(\underline{X}_{p,n}, t_n) \cdot \underline{Z}_{p,n}\sqrt{\Delta t} \tag{20.8}$$

After applying (20.8) for a large number of particles, (i.e. N_p), the average solute concentration for an arbitrary volume can be calculated using (20.9)

$$c_{V,N} = \frac{1}{V} \sum_{p=1}^{N_p} m_p \delta; \delta = \begin{cases} 1; \underline{X}_{p,N} \in V \\ 0; \underline{X}_{p,N} \notin V \end{cases} \tag{20.9}$$

where m_p is the particle mass. Using this procedure an accurate solution of the advection-dispersion equation (20.1) can be obtained [Thompson et al., 1987; Thompson and Dougherty, 1988; Kitanidis, 1994].

The term $\nabla \cdot \underline{D}(\underline{X}_{p,n}, t_n)$ in (20.2) and (20.8) is assumed to be much smaller than the remaining term and is omitted for the benefit of the computational speed. This may, however in some situations result in an accumulation of

particles near boundaries or stagnation points. [Kinzelbach and Uffink, 1989; Uffink, 1988; Kitanidis, 1994].

Prior to the particle tracking calculations the transient three-dimensional ground water flow field must be calculated. The groundwater velocities are used by the particle model to calculate $\underline{u}(\underline{X}_{p,n}, t_n)$ using linear interpolation for the spatial interpolation in the three directions in the grid cells. For time integration, simple Eulerian integration is used. The numerical input used by the water movement calculations is reused in the particle model as control volumes (see (20.9)) and for the specification of initial and boundary conditions.

The vertical position of the particles is corrected for changes in cell thickness when a particle moves horizontally from one cell to the next. The correction uses the relative vertical location at the old location to determine the new vertical location:

$$z_{new} = \left(\frac{z_{old} - Bottom_{old}}{Top_{old} - Bottom_{old}}\right) \times (Top_{new} - Bottom_{new}) + Bottom_{new} \quad (20.10)$$

where *old* indicates the previous cell and *new* the current cell. The correction is only applied when moving horizontally from one cell to the next i.e. there is no interpolation of layer thickness during the movement within a single cell. This results in sudden changes in the vertical location at cell boundaries.

The following particle sinks can remove particles from model cells:

- constant concentration boundary receiving particles
- well
- river
- drain connected to a river or the boundary
- exchange to the unsaturated zone
- constant concentration source with a lower concentration than the calculated concentration

The following particle sources can add particles to model cells:

- constant concentration boundaries
- a solute concentration in precipitation
- a source in the saturated zone with a specified mass inflow rate
- a constant concentration source with a higher concentration than the calculated concentration

The PT module only calculates particle movements in the saturated zone. However, the volume of water removed by the wells, rivers, drains and the

unsaturated zone is known. This volume of water is used to calculate the number of particles that are removed by each of the sinks using the formula:

$$n_i = n \times \frac{V_i}{V_{\text{sink}}} \times \frac{V_{\text{sink}}}{V_{\text{sink}} + V_{tot}} \tag{20.11}$$

where n_i is the number of particles removed by sink i, n is the number of particles in the saturated zone, V_i is the volume of water exchanged with sink i, V_{sink} is the volume of water exchanged with all sinks, V_{tot} is the total volume of water in the saturated zone.

Equation (20.11) is used to calculate the number of particles, which should be removed by each sink at each time step. This is, however, not necessarily a whole number of particles. The PT module takes care of this by retaining all the fractions of particles from previous time steps until it can remove a whole particle. Particles are always assigned one by one to the sinks, with preference given to the sink in need of most particles. In case there is more than one sink in a cell, with each of these sinks requiring the same number of particles, there is a random assignment of one particle to one of these sinks. If there are any more particles left after this assignment, the next particle will then go to one of the other sinks.

Constant concentration sources and sinks at boundaries or inside the model domain are handled by calculating the number of particles that corresponds to the concentration and truncating this value to a whole number. For the mass flux source and the precipitation source the concentration is again converted to a number of particles. The whole number obtained by truncating this value is added to the compartment containing the source. The fractions that are left over after truncation are accumulated until a whole number of particles has been attained in one of the next time steps at which time an additional particle is added to the compartment in which the source is located.

Drains remove particles to rivers or to the boundary out of the model domain. Drains can also transfer particles internally in SZ. If this occurs the particles are moved from one compartment to another by the drain. Note that there is no time lag in this process.

To trace the particles, calculate transport times, capture zones, groundwater age, etc. each particle is associated with a particle ID, model time and location at which the particle was introduced in the model (time and co-ordinates of 'birth'). When particles enter sinks or are introduced into the model domain by a source, this information is registered together with the source/sink type and the registration time and location before removing or adding the particle. The registration process is also used for keeping track of particles that enter registration cells. A particle is only registered the first time it enters a registration cell. This avoids repeated registration of particles that have entered a registration cell and which are not (immediately) removed by a sink.

ADDITIONAL OPTIONS

21 Extra Parameters

The Extra Parameters section is a special section of the Setup data tree that allows you to input parameters for options that have not yet been included in the MIKE SHE user interface.

The Extra Parameters are only recognized if the Name (e.g. “sheet piling module”) are spelled exactly correct. After the initial run, you should check in the Preprocessor_print.log file to ensure that the module has actually been activated.

Available Extra Parameters include:

Climate

- Temperature coupling between MIKE Hydro River and MIKE SHE *(V2 p. 326)*
- Negative Precipitation *(V2 p. 329)*
- Precipitation multiplier *(V2 p. 330)*

Rivers

- Activating the MIKE 1D Engine with MIKE 11 *(V2 p. 330)*
- Irrigation River Source Factors *(V2 p. 331)*
- Validation of HD Res file *(V2 p. 332)*

Overland Flow

- Time-varying Overland Flow Boundary Conditions *(V2 p. 332)*
- Time varying surface infiltration (frozen soils) *(V2 p. 334)*
- Time varying surface roughness (Manning’s M) *(V2 p. 335)*
- Simplified Overland Flow options *(V2 p. 335)*
- Explicit Overland Flow Output *(V2 p. 336)*
- Alternative low gradient damping function *(V2 p. 337)*
- OL Drainage Options *(V2 p. 338)*
- OL Drainage to Specified MIKE Hydro River H-points *(V2 p. 339)*
- Time varying OL drainage parameters *(V2 p. 341)*

Unsaturated Zone

- UZ-SZ exchange *(V2 p. 343)*
- Transpiration during ponding *(V2 p. 343)*
- Threshold depth for infiltration (2-Layer UZ) *(V2 p. 344)*

Saturated Zone

- Sheet Pile Module *(V2 p. 345)*
- SZ Drainage to Specified MIKE Hydro River H-points *(V2 p. 350)*

- SZ Drainage Downstream Water Level Check *(V2 p. 353)*
- Time varying SZ drainage parameters *(V2 p. 353)*
- Time varying hydraulic conductivity *(V2 p. 354)*
- Canyon exchange option for deep narrow channels *(V2 p. 356)*
- 2-norm reduction-criteria in the inner iteration loop *(V2 p. 356)*

Water Quality

- Disable SZ solute flux to dummy UZ *(V2 p. 357)*
- SZ boundary dispersion *(V2 p. 357)*

Miscellaneous

- Maximum number of threads *(V2 p. 358)*
- Including OpenMI *(V2 p. 359)*
- Plot control for Detailed Time Series Output *(V2 p. 359)*
- Extra Pre-Processing output *(V2 p. 360)*
- GeoViewer Output *(V2 p. 360)*

21.1 Climate

21.1.1 Temperature coupling between MIKE Hydro River and MIKE SHE

Temperature is not a feature that can currently be modelled in MIKE SHE though MIKE Hydro River has the possibility of modelling a heat balance in the stream. The heat balance in the stream is depended on the temperature of the lateral sources in the river. This option does not introduce temperature modelling in general in MIKE SHE, but rather it give the user the possibility to introduce a temperature forcing on the water which is passed from MIKE SHE to MIKE Hydro River either as base flow, overland or drain flow. The temperature is in turn received by MIKE Hydro River as though it came from MIKE SHE which makes it versatile for a later potential full implementation of temperature modelling in MIKE SHE. The functionality allows you to

- Define temperature as a spatially distributed, time varying property in MIKE SHE for ponded water and groundwater.
- Pass temperature as a property of the baseflow, overland flow and drain flow
- Add a temperature property to the baseflow and drainflow based on the temperature of the SZ cell were the flow originates from.
- Add a defined temperature property to the ponded water.
- Use a volume based mixing function for paved area drainage (from ponded water) and SZ drainage.

- Make cell temperature an OpenMI exchange item, so that you can access cell temperature during the simulation.

The above functionality will allow users to specify the temperature of the groundwater and ponded water. However, it will not provide the functionality to transport temperature as a property in MIKE SHE. In other words, the temperature is a defined property and there is no internal mixing or recalculation of cell temperature as a function of cell-by-cell flow. The temperature distributions are supplied through the use of extra parameters in the Extra Parameters table.

Option setup

To initiate the temperature module, you need to specify the following:

Parameter Name	Type	Value
user specified temperature	Boolean	On
If the above is true, then you must also specify:		
distributed ol temperature	Boolean	On/Off
distributed drain temperature	Boolean	On/Off
distributed sz temperature	Boolean	On/Off

Temperature of ponded water

The temperature of the ponded water is generally controlled by the air temperature, but it is not linked to the air temperature specified in the user interface. The temperature of the ponded water also controls the temperature of the water in the OL-drains.

Parameter Name	Type	Value
If distributed OL temperature is OFF (then dfs0 input):		
ol temperature dfs0 file name	file name	.dfs0 file
ol temperature item number	integer	item number in dfs0 file, greater than zero
If distributed OL temperature is ON (then dfs2 input):		
ol temperature dfs2 file name	file name	.dfs2 file
ol temperature item number	integer	item number in dfs2 file, greater than zero

Temperature of OL drainage water

The OL drainage water is the same temperature as the source ponded water. There is no change in temperature while the water is in the drain. This is con-

sistent with the fact that the OL drain water is controlled by the same air temperature as the ponded water in general.

Temperature of SZ drainage water

The SZ drainage water is treated separately from the SZ water because it does not travel directly to the river, but via surface water channels.

Parameter Name	Type	Value
If distributed drainage temperature is OFF (then dfs0 input):		
drain temperature dfs0 file name	file name	.dfs0 file
drain temperature item number	integer	item number in dfs0 file, greater than zero
If distributed drainage temperature is ON (then dfs2 input):		
drain temperature dfs2 file name	file name	.dfs2 file
drain temperature item number	integer	item number in dfs2 file, greater than zero

Temperature of SZ water

The SZ water is sent to the River directly via the baseflow component.

The SZ temperatures have to be specified for every layer as either all dfs0 files or all dfs2 files, depending on the initialization above.

Parameter Name	Type	Value
If distributed SZ temperature is OFF (then dfs0 input):		
sz temperature layer 1 dfs0 file name	file name	.dfs0 file
sz temperature layer 1 item number	integer	item number in dfs0 file, greater than zero
sz temperature layer n dfs0 file name	file name	.dfs0 file
sz temperature layer n item number	integer	item number in dfs0 file, greater than zero

Parameter Name	Type	Value
If distributed SZ temperature is ON (then dfs2 input):		
sz temperature layer 1 dfs2 file name	file name	.dfs2 file
sz temperature layer 1 item number	integer	item number in dfs2 file, greater than zero
sz temperature layer n dfs2 file name	file name	.dfs2 file
sz temperature layer n item number	integer	item number in dfs2 file, greater than zero

21.1.2 Negative Precipitation

Negative precipitation is sometimes required when net groundwater recharge has been calculated using an external program, such as DAISY GIS. In this case, the evapotranspiration may exceed infiltration leading to a net upward flux of water from the groundwater table. However, the standard precipitation module in MIKE SHE does not recognize negative rainfall. In this case, you must specify the negative rainfall using the following Extra Parameters options:

Parameter Name	Type	Value
use negative precipitation	Boolean	On
If the negative precipitation is uniformly distributed:		
negative precipitation max depth	float	greater than zero
negative precipitation max layer	integer	greater than zero
If the negative precipitation is spatially distributed:		
negative precipitation max depth dfs2 file	file name	.dfs2 file
negative precipitation max depth dfs2 item	integer	item number in dfs2 file, greater than zero
negative precipitation max layer dfs2 file	file name	dfs2 file
negative precipitation max layer dfs2 item	integer	item number in dfs2 file, greater than zero

- **Max depth** This represents the depth of the root zone plus the thickness of the capillary fringe and is the maximum depth from which negative precipitation can be extracted.
- **Max layer** This is the maximum layer depth from which negative precipitation can be extracted.

Note: The negative precipitation option will only work if there is no UZ model active.

21.1.3 Precipitation multiplier

To facilitate calibration and sensitivity analysis of recharge, in models where measured precipitation is not being used, a multiplication factor has been implemented.

Parameter Name	Type	Value
precipitation factor	float	greater than zero

If this extra parameter is used, then all precipitation values are multiplied by the factor prior to being used in MIKE SHE.

21.2 Rivers

21.2.1 Activating the MIKE 1D Engine with MIKE 11

To activate the MIKE 1D engine when running MIKE 11 you need to specify the following Extra Parameter:

Parameter Name	Type	Value
use mike 1d	boolean	on

21.2.2 Irrigation River Source Factors

A global “river source volume factor” and “river source discharge factor” are available as extra parameters for increased control of river sources during irrigation.

Parameter Name	Type	Value
river source volume factor	float	positive
river source discharge factor	float	0 or positive

None, one, or both can be specified. If the factor is not specified, then a Volume factor of 0.99 and a Discharge factor of 0.0 will be used.

The factors are used in the calculation of the available water (depth) of a river source:

$$Depth = MIN\left(\frac{C_s \cdot \Delta t}{A}, \frac{F_V \cdot V_L}{A} + \frac{F_D \cdot D_L \cdot \Delta t}{A}\right) \quad (21.1)$$

where *Depth* is the available water depth in the river link, C_s is the source capacity, Δt is the time step length, F_V is the specified volume factor, V_L is the volume of water in the link, F_D is the specified volume discharge, D_L is the river link discharge, and A is the cell area.

The river link discharge is the same as used when checking with the threshold discharge for switching on/off the source. It is the absolute discharge in the middle of the MIKE SHE river link, interpolated between two MIKE Hydro River H-points.

MIKE SHE prints the following message in the xxx_WM_Print.log file when the parameters are specified:

```
Extra-parameter specified:

river source volume factor

value = 1.500000


Extra-parameter specified:

river source discharge factor

value = 1.000000
```

MIKE SHE also prints the following warnings in the xxx_WM_Init_Messages.log file if one or both of the factors may result in water balance errors or numerical instabilities

```
WARNING: Specified value for river source volume factor
is greater than 1 : 1.500000.

There is a risk of water balance errors and/or instabili-
ties in the coupling between MIKE SHE and MIKE 11.


WARNING: Specified value for river source discharge fac-
tor is greater than 0 : 1.000000.

There is a risk of water balance errors and/or instabili-
ties in the coupling between MIKE SHE and MIKE 11.
```

Note: This option is less useful now that River Sources are defined by both an Upstream and Downstream chainage. The option is maintained for backward compatibility.

21.2.3 Validation of HD Res file

In some cases you may want to turn off the time stamp validation for the MIKE 11 or MIKE Hydro River output files. For example, this may be useful if you are linking older existing MIKE 11 models as input for the MIKE SHE WQ.

Parameter Name	Type	Value
disable validation of HD res file	Boolean	On

21.3 Overland Flow

21.3.1 Time-varying Overland Flow Boundary Conditions

The default boundary condition for Overland Flow in MIKE SHE is a constant water level on the outer boundary. The value of this boundary condition is determined by the initial water depth on the boundary. In most models the recommended value is a water depth of zero. In this case, if the water level adjacent to the boundary increases, water will discharge across the boundary and out of the model. If you want to prevent overland outflow then you can use the Separated Flow Areas option to restrict lateral flow out of the model.

If you specify a non-zero value for initial water depth on the boundary, then this value becomes a constant for the entire simulation. If the water level

inside the model decreases below this value, the boundary will act as an infinite source of inflow to the model.

However, in many models - especially those with significant wetland areas - the constant water level condition on the boundary is too restrictive.

The following extra parameter options allow you to specify a time varying condition for the outer boundary of the overland flow. If you initialize this option, then you must supply a dfs2 integer grid code file that defines the locations at which you want a time varying boundary. The input requirements have been set up such that you can re-use the model domain dfs2 output file from the pre-processor. In the model domain pre-processed output, the outer cells are defined by a value of 2 and the inner cells are defined by a value of 1.

If the grid code value on the boundary is:

- 2 - the cell is a time varying boundary node, or
- 1 - the cell will have a constant water depth equal to the initial water depth.

The second required file is the actual time-varying water level values. These can be obtained from any MIKE SHE simulation, where the overland water elevation has been stored as a grid series output. There is no requirement that they be stored on the same grid. Internally, the actual boundary condition values will be interpolated from the nearest input values. Thus, the OL boundary conditions can be taken from a coarse regional model and applied to a local scale model.

Finally, each file name must be accompanied by an integer item number that defines which item in the dfs2 file should be used.

Parameter Name	Type	Value
time varying ol boundary	Boolean	On
ol boundary code file name	filename	.dfs2 file
ol boundary code item number	integer	item number in dfs2 file, greater than zero
ol boundary head file name	filename	.dfs2 file
ol boundary head item number	integer	item number in dfs2 file, greater than zero

The Hot Start function is not impacted by the time varying OL boundary. If the continuing simulation includes the time varying OL function then it will be

used. If the continuing simulation does not include the time varying OL function the head from the hot start time point.

21.3.2 Time varying surface infiltration (frozen soils)

A common characteristic in cold climates is that infiltration is reduced during the winter months. When the air temperature is cold enough to maintain precipitation as snow, then infiltration will be limited in any case. However, in the spring, when snow storage is melting, then infiltration may still be limited for some period of time.

Although this function was conceived as a way to support reduced infiltration in winter, it can be used any time a time varying leakage is required.

The time varying infiltration function is a modification of the Surface-Subsurface Leakage Coefficient *(V1 p. 260)* to allow it to be time varying.

The time varying dfs2 file must cover the model domain and the model simulation period.

Parameter Name	Type	Value
time varying ol leakage coefficient	Boolean	On
leakage coefficient dfs2 file name	filename	.dfs2 file
leakage coefficient item number	integer	item number in dfs2 file, greater than zero
Optional		
mean step accumulated leakage coefficient	Boolean	On

By default, the Time Series Types *(V1 p. 143)* is Instantaneous, but their is an option that allows you to use Mean Step Accumulated values. The use of Mean Step Accumulated does not change the meaning of the item, but changes the way the values are interpolated.

Note: The code does not check for the time series type.

Note The areas in which these values will be applied has not changed. The areas are defined in the original Surface-Subsurface Leakage Coefficient *(V1 p. 260)* dialogue. That is, the values specified in the User Interface act as a mask, where the leakage coefficient is active if a non-delete value is specified in the user interface.

21.3.3 Time varying surface roughness (Manning's M)

The time varying roughness function can be used any time the roughness changes over the simulation period. For example, the roughness can be correlated to changes in vegetation type over time, or climatic conditions (winter vs. summer).

The time varying dfs2 file must cover the model domain and the model simulation period.

Parameter Name	Type	Value
time varying ol manning number	Boolean	On
manning number dfs2 file name	filename	.dfs2 file, Item Type: Manning's M
manning number item number	integer	item number in dfs2 file, greater than zero
Optional		
mean step accumulated manning number	Boolean	On

By default, the Time Series Types *(V1 p. 143)* is Instantaneous, but their is an option that allows you to use Mean Step Accumulated values. The use of Mean Step Accumulated does not change the meaning of the item, but changes the way the values are interpolated.

Note The code does not check for the time series type.

21.3.4 Simplified Overland Flow options

Avoiding the redistribution of ponded water

In the standard version of the Simplified Overland Flow solver, the solver calculates a mean water depth for the entire flow zone using the available overland water from all of the cells in the flow zone. During the Overland flow time step, ET and infiltration are calculated for each cell and lateral flows to and from the zone are calculated. At the end of the time step, a new average water depth is calculated, which is assigned to all cells in the flow zone.

In practice, this results in a redistribution of water from cells with ponded water (e.g. due to high rainfall or low infiltration) to the rest of the flow zone where cells potentially have a higher infiltration capacity. To avoid this redistribution, an option has been added where the solver only calculates overland

flow for the cells that can potentially produce runoff, that is, only in the cells for which the water depth exceeds the detention storage depth.

Parameter Name	Type	Value
only simple OL from ponded	Boolean	On

Routing simple overland flow directly to the river

In the standard version of the Simplified Overland Flow solver, the water is routed from 'higher' zones to 'lower' zones within a subcatchment. Thus, overland flow generated in the upper zone is routed to the next lowest flow zone based on the integer code values of the two zones. In other words, at the beginning of the time step the overland flow leaving the upper zone (calculated in the previous time step) is distributed evenly across all of the cells in the receiving zone. In practice, this results in a distribution of water from cells in the upstream zone with ponded water (e.g. due to high rainfall or low infiltration) to all of the cells in the downstream zone with potentially a large number of those cells having a higher infiltration capacity. In this case, then, overland flow generated in the upper flow zone may never reach the stream network because it is distributed thinly across the entire downstream zone.

To avoid excess infiltration or evaporation in the downstream zone, an option was added that allows you to route overland flow directly to the stream network. In this case, overland flow generated in any of the overland flow zones is not distributed across the downstream zone, but rather it is added directly to the MIKE Hydro River stream network as lateral inflow.

Parameter Name	Type	Value
no simple OL routing	Boolean	On

21.3.5 Explicit Overland Flow Output

If you are using the explicit overland flow solver, the time step depends on the location in the model with the critical Courant criteria. The grid series output allows you to save the Courant criteria, so that you can see where the critical locations are. However, the grid series output is an average Courant number over the storing time step, where there can be hundreds of OL time steps in a storing time step. If you are experiencing very short time steps due to short duration rainfall events, for example, the critical information can be difficult to distill from the dfs2 grid series output.

To make it easier to find the critical locations, an extra parameter option was added that writes out the critical locations at every time step, if the time step is reduced below a user-defined fraction of the storing time step.

Parameter Name	Type	Value
adaptive OL time step info threshold fraction	Float	between 0.0 and 1.0 Default = 0.01

The default value is 0.01. This means that if the reduced time step is less than 0.1 times the Max OL time step, then a message will be printed in the _WM.log file. Such as:

Adaptive time step info from Explictit OL solver:

OL step no: 59: ... Final time step = 1.8108 seconds

with the following four reasons:

Critical: OL Wave Courant number. Cell (8,21)...

Critical: Net outflow from OL cell to River. Cell (8,21) ...

Critical: Net OL outflow from cell. Cell: (17,19) ...

Critical: Net outflow from River to OL. River link between ...

If you experience frequent severe reductions in the OL time step when using the explicit OL solver, then this threshold can cause very large log files to be created. If you are not interested in this information, then you can reduce this threshold to reduce the frequency of the output.

21.3.6 Alternative low gradient damping function

In flat areas with ponded water, the head gradient between grid cells will be zero or nearly zero, which means that as the gradient goes to zero Δt also goes to zero. To allow the simulation to run with longer time steps and dampen any numerical instabilities in areas with low lateral gradients, the calculated intercell flows are multiplied by a damping factor when the gradients are close to zero.

Compared to the default damping function, an alternative damping function is available as an Extra Parameter that goes to zero more quickly and is consistent with the function used in MIKE FLOOD.

To activate the alternate function, you must specify the following boolean parameter in the Extra Parameters *(V1 p. 334)* dialogue:.

Parameter Name	Type	Value
Enable Alternative Damping Function	Boolean	On

The alternative function is a single parabolic function. For more detail, see the section Low gradient damping function *(V2 p. 59)* in the Reference manual.

21.3.7 OL Drainage Options

By default, the OL Drainage function routes the available ponded water to the OL drainage network. However, the available ponded depth does not include the detention storage. If you want to route all of the ponded water in a cell - including the water in detention storage - to the OL Drainage network, then you can define the following Extra Parameter:

Parameter Name	Type	Value
allow paved routing of detention storage	Boolean	On

By default the OL Drainage will occur no matter the elevation of the destination cell. In some cases this could lead to unrealistic drainages. For example, OL drainage on a flood plain mapped to a river will always drain to the river, even if the river is higher than the drain level. To counter such unrealistic conditions, this Extra Parameter will prevent OL drainage in the case where the drain level is below the bed level of the river.

Note: This check is between the Drain level and the Bed level - not between the dynamic water levels.

Parameter Name	Type	Value
check OL-drain level against bed level'	Boolean	On

The interaction between the multi-cell OL solution and the OL drainage is complex and could lead to some unexpected results. Further, the multi-cell OL solution is more likely to be used as a more physical representation of the

drainage rendering the OL drainage obsolete in this case. This option allows you to disable OL drainage when using the multi-cell OL approach.

Parameter Name	Type	Value
disable multi-cell OL-drainage	Boolean	On

21.3.8 OL Drainage to Specified MIKE Hydro River H-points

The Distributed Drainage option allows you to route drainage from the OL drainage directly to MIKE Hydro River H-points. This is different from the normal drainage function, which routes OL drainage to river links rather than directly to H-points. Further, this option can route drainage to MIKE Hydro River branches that are not defined in the MIKE SHE coupling section of the MIKE Hydro River network file.

The following steps are required to activate this option:

1. Create a pfs file containing information for each specified drainage area to be routed to the specific MIKE Hydro River H-points..

Line item	Comment
[MIKESHE_MIKE11DrainageReach_File]	
[SpecifiedMIKE11ReachesForDrainage] NrOfReaches = 1 RiverChainageUnit = 'meter'	NrOfReaches is the number of items specified in the section below
[Reach_1] DrainCode = 1 BranchName = 'Lammehavebækken' Upstream_Chainage = 6000. Downstream_Chainage = 8459 EndSect // Reach_1	For each specified reach, you must include a section specifying the MIKE SHE drain code, and the MIKE Hydro River branch name and the upstream and downstream chainage.
EndSect // SpecifiedMIKE11ReachesForDrainage	
EndSect // MIKESHE_MIKE11DrainageReach_File	

Note: The pfs files line text refers to M11, but refers to either MIKE 11 or MIKE Hydro River.

The drain code references the area that OL drainage is routed to the specified MIKE Hydro River branch and chainage. The drain code must be greater than or equal to zero. Drain code values equal to zero (0) are not included in the reference drainage system. Furthermore, an error condition will occur if the specified drain code does not exist in the drainage code file used in MIKE SHE

The branch name must be spelled correctly and include all spaces contained in the name, if any. The branch name should be enclosed in quotes. An error condition will occur if the specified branch is not present in the MIKE Hydro River network.

The chainages refer to the starting and ending chainage of the specified branch which drainage and/or paved area discharge is routed to. The interval does not have to correspond exactly to specific MIKE Hydro River H-points because the MIKE SHE pre-processor finds the closest H-points to the specified interval. If the upstream and downstream chainages are the same, the drainage and/or paved area discharge is routed to the closest H-point.

2. Add the following items to the Extra Parameters list

Parameter Name	Type	Value
use specified reaches for OL-drainage	Boolean	On
specified reaches for OL-drainage	file name	the pfs file name, including the path

3. In the OL Drainage item under the Overland flow, select **distributed drainage options**. See Ponded Drainage *(V1 p. 264)*.
4. Specify drain codes is the same manner as usual. Remember that all drain codes in the pfs file must exist in the active domain of the model or you will get an error.
5. Specify where the Distributed drainage option should be used in Drainage Option Distribution *(V1 p. 270)* item in the data tree under the Saturated Zone. Distributed drainage will be used in all cells with a value of 3. If a combination of the original drainage method and the Distributed drainage option is going to be used, 2 should be used for areas using the original drainage option and 3 should be used where you want the Distributed drainage option to be used.
6. Pre-process and run your MIKE SHE model normally.

If the MIKE SHE setup does not successfully pre-process you should review the above steps to see if you have any error in the setup. The *_PP_Print.log*

file in your simulation subdirectory should help you identify why the MIKE SHE setup failed to pre-process.

If the MIKE SHE setup successfully pre-processes you should also look at the pre-processed data (on the Processed data tab) and the *_PP_Print.log* file in your simulation subdirectory to make sure you are comfortable with how the preprocessor has set up the drainage reference system. You can search for *Making setup of Specified MIKE 11 Reaches For Drainage* in the *_PP_Print.log* file to find the start of the section that details the drainage reference system.

Water balance

The water balance utility can be used to look at differences between drainage discharges from areas using the original drainage option and the Distributed Drainage option. The MIKE SHE water balance configuration file (MSHE_Wbl_Config.pfs in the installation directory) should be reviewed to see which water balance types segregate standard drainage flow (data type ol.qoldrtorivin) and Distributed drainage flow (data type ol.qoldrtoM11Hpoint) (see Using the Water Balance Tool *(V1 p. 91)*)

21.3.9 Time varying OL drainage parameters

In projects where you want to simulate the build out of an OL Drainage network over time, or changes in the OL Drainage time constants over time, then you can use this set of extra parameters. Without this set of extra parameters you would have to hot start your simulation at regular time intervals with the new OL Drainage parameters.

Note: If you specify time varying OL Drainage parameters, you will not be able to use any of the drainage routing methods that depend on the OL drain level. The preprocessor checks this and gives an error if you have specified

- option 1 (routing based on levels), or
- option 3 (distributed options) AND any of the distributed option codes are 1 (routing based on levels in these cells).

To activate time varying OL Drainage parameter options, you must specify the following extra parameters

Parameter Name	Type	Value
time varying OL-drainage levels	Boolean	On
time varying OL-drainage time constants	Boolean	On
OL-drainage level dfs2 file name	file name	.dfs2 file
OL-drainage level item number	integer	item number in dfs2 file, greater than zero
OL-drainage time constant dfs2 file name	file name	.dfs2 file
OL-drainage time constant item number	integer	item number in dfs2 file, greater than zero
Optional:		
mean step accumulated OL-drainage levels	Boolean	On
mean step accumulated OL-drainage time constants	Boolean	On

Note: The time constants above refer to Inflow Time Constants. There is no option for time varying Outflow Time Constants

The dfs2 Drain Level is an elevation that can be specified using the following three EUM Data Units *(V1 p. 131)*:

- Elevation
- Depth Below Ground (i.e. positive values)
- Height Above Ground (i.e. negative values)

By default, the Time Series Types *(V1 p. 143)* is Instantaneous, but there is an extra option that allows you to used Mean Step Accumulated values if you want. The use of Mean Step Accumulated does not change the meaning of the item, but changes the way the values are interpolated.

Note: The code does not check for the time series type.

All specifications are printed to the *projectname*_PP_Print.log and *projectname*_WM_Print.log files.

21.4 Unsaturated Zone

21.4.1 UZ-SZ exchange

In some cases, a local water balance error is generated during the UZ-SZ exchange. This can occur when the water table is rising faster than the solution scheme can keep up. There is an Extra Parameter that can help if you notice such condition:

Parameter Name	Type	Value
max iterations UZ-coupling	Integer	greater than or equal to 10

The internal default value in MIKE SHE is 10, but you may find a value of between 200 and 500 yields better results if you are seeing some local water balance issues. The impact on run-time is marginal, as the extra iterations are not used if the solution converges earlier.

21.4.2 Transpiration during ponding

In general, plants are not very tolerant of saturated soil in their root zone. Saturated soil is quickly depleted of oxygen and the roots will soon die. MIKE SHE normally takes care of this automatically by removing ET from ponded water before calculating transpiration from the unsaturated or saturated zones. If there is sufficient ponded water then the entire ET will be satisfied from the ponded water.

However, some plants, such as rice, are more tolerant of saturated soils and still extract ET from saturated soils, although normally at a reduced rate. If ET from the soil zone is ignored, then the distribution of water supplied to ET will be incorrect.

The transpiration during ponding option changes the order in which the ET is calculated. In this case, the ET rate is multiplied by an anaerobic tolerance

factor and ET is removed from the soil before being removed from the ponded water.

Parameter Name	Type	Value
allow transpiration during ponding	Boolean	On
global anaerobic tolerance factor	Float	Greater than or equal to 0 Less than or equal to 1
optional (instead of global value)		
anaerobic tolerance factor dfs2 file name	file name	.dfs2 file
anaerobic tolerance factor item number	integer	item number in dfs2 file, greater than zero

21.4.3 Threshold depth for infiltration (2-Layer UZ)

The 2-Layer water balance method for the unsaturated zone does not include evapotranspiration from the soil surface. Thus, even a small amount of water on the ground surface will infiltrate. If you use this extra parameter, then you can define a depth of overland water that must be exceeded before infiltration will occur. This keeps small amounts of precipitation from infiltrating and allows them to evaporate instead.

The calculated infiltration is simply reduced if the remaining overland water depth will be smaller than the specified threshold value.

Parameter Name	Type	Value
use threshold depth for infiltration	Boolean	On
threshold depth for infiltration meter	Float	Greater than zero

Note: This option is less useful with the ET Deficit Factor introduced in the 2008 Release, which maintains ET at the full rate until the specified deficit is reached. The option is maintained for backward compatibility.

21.5 Saturated Zone

21.5.1 Sheet Pile Module

The Sheet Piling module is not yet included in the MIKE SHE GUI. However, the input for the module is fairly simple and is handled via the Extra Parameters options

The Sheet Piling module is activated by including the following two parameters in the Extra Parameters section of the data tree, and creating the required module input file:

Parameter Name	Type	Value
sheet piling module	Boolean	On
sheet piling file	file name	the file name of the Sheet Pile input file

Sheet Pile Location

The location of the sheet piles is defined using a dfs2 file with integer grid codes. One file (or item) is required for each computational layer with sheet piling. Each file must have the same grid size as the MIKE SHE model. The grid codes are “composed” of simple sums of 100, 10, 1, 0 where:

100 = a N-S sheet piling “link” between the actual cell and the next cell in positive x-direction,

10 = a E-W sheet piling “link” between the actual cell and the next cell in the positive y-direction,

1 = a Horizontal sheet-piling “surface” between the actual layer and the layer above (ground surface if actual layer is 1), and

0 = no sheet piling.

Thus, for example, a cell containing the code “110” defines the existence of sheet piling along the Eastern and Northern cell boundaries. A cell containing the code “11” defines a sheet piling along the Northern cell boundary and at the top of the layer.

Leakage Coefficient

The Leakage Coefficient is required for flow in the x-, y-, and z-direction for each layer containing sheet piling. The Leakage Coefficient is required in the x-direction if any cell contains a “100” value, in the y-direction if any cell contains a “10” value, and in the z-direction if any cell contains a “1” value.

The leakage coefficients can be specified as a global value (per layer) or as a distribution in a dfs2 file. In the case of a dfs2 file, the values must be specified in the cells where the grid codes are specified. The EUM type (unit) of the

dfs2 files must be “Leakage coefficient/Drain time constant” with the unit 1/Time.

Top and bottom levels (optional)

This option can be used when the vertical sheet piling only extends across part of a layer. The levels are specified in the same cells as the leakage coefficients in the x- and y-direction, one set of top and bottom levels for each direction.

The levels can be specified as global values (per layer) or as a distribution in a dfs2 file. Both can be absolute levels or relative to ground. The EUM type of the dfs2 files must be “elevation” for absolute levels, and “depth below ground” (positive values) or “height above ground” (negative values) when specified relative to the ground surface. The type and unit of the global value is “elevation” (m) when absolute, and “height above ground” (m) (negative value) when relative.

In cells where the sheet pile extends across the entire layer, the top and bottom levels should simply be set to large positive and negative values respectively (e.g. 1.0E+30 and -1.0E+30).

Input File for the Sheet Pile Module

The name of the input file is specified in the Extra Parameters section described above. The file has the general MIKEZero parameter file (pfs) format. The exact format of the file is given below, along with a description of the different data items.

Note: The pfs format must be adhered to exactly. There is a small utility (pfsEditor.exe) in the installation \bin directly that you can use for editing and testing pfs files that you create.

.

Line item	Comment
[MIKESHE_SheetPiling_File] FileVersion = 2 [SheetPiling]	FileVersion can be 1 or 2, but must be 2, if you want to check for the SpecifiedXYLevels option
NrOfLayers = 1	Total number of SZ layers with sheet piling
SpecifiedXYLevels = 1	0: not specified. 1: top and bottom levels specified for each layer Note: only checked when FileVersion > 1
[Layer_1]	This section must be repeated for each -NrOfLayers- sheet piling layer. The sections must be named Layer_1, Layer_2, etc.
LayerNumber = 1	The MIKE SHE SZ layer number of the actual sheet piling layer (1 = top layer).
[GridCodes] Type = 1 FixedValue = 0 [DFS_2D_DATA_FILE] FILE_NAME = |.\SPGrid_1.dfs2| ITEM_COUNT = 1 ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // GridCodes	**[GridCodes] section** Specification of grid codes for the current layer. **Type** Normally 1 because a dfs2 file is required. 0 means global value. **FILE_NAME** Name of the dfs2 file with grid codes. The file name is enclosed in "|" which tells the system that the name is relative to the location of this module input file. **ITEM_NUMBERS**: One number (because ITEM_COUNT must be 1) defining the item of the dfs2 file to be used.

Line item	Comment
[X_Leakage] Type = 0 FixedValue = 1.0E-7 [DFS_2D_DATA_FILE] FILE_NAME = |.\maps\SPLeakX_1.dfs2| ITEM_COUNT = 1 ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // X_Leakage	**[X_Leakage] section:** Required if there are any cells with N-S sheet piling affecting the flow in the x-direction (codes containing 100). **Type** Set to 0 if a global value is specified and 1 if using a dfs2 file. **FixedValue** The global value (1/s) which is read if Type = 0. **FILE_NAME** and **ITEM_NUMBERS** Dfs2 file name and item number if Type = 1 (relative file name as explained under Grid Codes).
[Y_Leakage] Type = 0 //(0:Fixed value,1:DFS2 file) FixedValue = 2.0E-7 [DFS_2D_DATA_FILE] FILE_NAME = |.\maps\SPLeakY_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 1 1 EndSect // DFS_2D_DATA_FILE EndSect // Y_Leakage	**Y_Leakage] section**: Required if there are any cells with E-W sheet piling affecting the flow in the y-direction (codes containing 10).
[Z_Leakage] Type = 0 //(0:Fixed value,1:DFS2 file) FixedValue = 3.0E-7 [DFS_2D_DATA_FILE] FILE_NAME = |.\maps\SPLeakZ_1.dfs2| ITEM_COUNT = 1 ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // Z_Leakage	**[Z_Leakage] section**: Required if there are any cells with horizontal sheet piling affecting the vertical flow (codes containing 1).

Line item	Comment
[X_TopLevel] RelativeToGround = 0 // 0: no, 1: yes Type = 1 //(0:Fixed value,1:DFS2 file) FixedValue = 0.0 [DFS_2D_DATA_FILE] FILE_NAME = |.\YLevels_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // Y_TopLevel	**[X_TopLevel] section**: Required if SpecifiedXYLevels=1 and there are any codes containing 100.
[X_BottomLevel] RelativeToGround = 0 // 0: no, 1: yes Type = 1 //(0:Fixed value,1:DFS2 file) FixedValue = 0.0 [DFS_2D_DATA_FILE] FILE_NAME = |.\YLevels_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 2 EndSect // DFS_2D_DATA_FILE EndSect // Y_BottomLevel	**[X_BottomLevel] section**: Required if SpecifiedXYLevels=1 and there are any codes containing 100.

Line item	Comment
[Y_TopLevel] RelativeToGround = 0 // 0: no, 1: yes Type = 1 //(0:Fixed value,1:DFS2 file) FixedValue = 0.0 [DFS_2D_DATA_FILE] FILE_NAME = |.\YLevels_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 1 EndSect // DFS_2D_DATA_FILE EndSect // Y_TopLevel	**[Y_TopLevel] section**: Required if SpecifiedXYLevels=1 and there are any codes containing 10.
[Y_BottomLevel] RelativeToGround = 0 // 0: no, 1: yes Type = 1 //(0:Fixed value,1:DFS2 file) FixedValue = 0.0 [DFS_2D_DATA_FILE] FILE_NAME = |.\YLevels_1.dfs2| ITEM_COUNT = 1 //(must be 1) ITEM_NUMBERS = 2 EndSect // DFS_2D_DATA_FILE EndSect // Y_BottomLevel	**[Y_BottomLevel] section**: Required if SpecifiedXYLevels=1 and there are any codes containing 10.
EndSect // Layer_1	
EndSect // SheetPiling	
EndSect // MIKESHE_SheetPiling_File	

21.5.2 SZ Drainage to Specified MIKE Hydro River H-points

The Distributed Drainage option allows you to route drainage from the SZ drains directly to MIKE Hydro River H-points. This is different from the normal drainage function, which routes drainage and paved area discharges to river links rather than directly to H-points. Further, this option can route drainage to MIKE Hydro River branches that are not defined in the MIKE SHE coupling section of the MIKE Hydro River network file.

The following steps are required to activate the Distributed Drainage option:

1. Create a pfs file containing information for each specified drainage area to be routed to the specific MIKE Hydro River H-points..

Line item	Comment
[MIKESHE_MIKE11DrainageReach_File]	
[SpecifiedMIKE11ReachesForDrainage] NrOfReaches = 1 RiverChainageUnit = 'meter'	NrOfReaches is the number of items specified in the section below
[Reach_1] DrainCode = 1 BranchName = 'Lammehavebækken' Upstream_Chainage = 6000. Downstream_Chainage = 8459 EndSect // Reach_1	For each specified reach, you must include a section specifying the MIKE SHE drain code, and the MIKE 11 branch name and the upstream and downstream chainage.
EndSect // SpecifiedMIKE11ReachesForDrainage	
EndSect // MIKESHE_MIKE11DrainageReach_File	

Note: The pfs files line text refers to M11, but refers to either MIKE 11 or MIKE Hydro River.

The drain code references the area that drainage is routed to the specified MIKE Hydro River branch and chainage. The drain code must be greater than or equal to zero. Drain code values equal to zero are not included in the reference drainage system. Furthermore, an error condition will occur if the specified drain code does not exist in the drainage code file used in MIKE SHE

The branch name must be spelled correctly and include all spaces contained in the name, if any. The branch name should be enclosed in quotes. An error condition will occur if the specified branch is not present in the MIKE Hydro River network.

The chainages refer to the starting and ending chainage of the specified branch which drainage is routed to. The interval does not have to correspond exactly to specific MIKE Hydro River H-points because the MIKE SHE pre-processor finds the closest H-points to the specified interval. If the upstream and downstream chainages are the same, the drainage is routed to the closest H-point.

2. Add the following items to the Extra Parameters list

Parameter Name	Type	Value
use specified reaches for drainage	Boolean	On
specified reaches for drainage	file name	the pfs file name, including the path

3. In the Drainage item under the Saturated Zone, select **distributed drainage options**. See Drainage *(V1 p. 315)*.
4. Specify drain codes is the same manner as usual. Remember that all drain codes in the Distributed Drainage option pfs file must exist in the active domain of the model or you will get an error.
5. Specify where the Distributed Drainage option should be used in Drainage Distribution item in the data tree under the Saturated Zone. The option will be used in all cells with a value of 3. If a combination of the original drainage method and the Distributed Drainage option is going to be used, 2 should be used for areas using the original drainage option and 3 should be used where you want the Distributed Drainage option to be used.
6. Pre-process and run your MIKE SHE model normally.

If the MIKE SHE setup does not successfully pre-process you should review the above steps to see if you have any error in the setup. The *_PP_Print.log* file in your simulation subdirectory should help you identify why the MIKE SHE setup failed to pre-process.

If the MIKE SHE setup successfully pre-processes you should also look at the pre-processed data (on the Processed data tab) and the *_PP_Print.log* file in your simulation subdirectory to make sure you are comfortable with how the preprocessor has set up the drainage reference system. You can search for *Making setup of Specified MIKE 11 Reaches For Drainage* in the *_PP_Print.log* file to find the start of the section that details the drainage reference system.

Water balance

The water balance utility (e.g., Saturated zone - detailed) can be used to look at differences between drainage discharges from areas using the original drainage option and the RFD option. The MIKE SHE water balance configuration file (MSHE_Wbl_Config.pfs in the installation directory) should be reviewed to see which water balance types segregate standard drainage flow (data type sz.qszdrtorivin) and RFD drainage flow (data type sz.qszdrtoM11Hpoint) (see Using the Water Balance Tool *(V1 p. 91)*)

21.5.3 SZ Drainage Downstream Water Level Check

You can optionally check the downstream water level before calculating SZ drainage. This prevents drainage from being added to rivers during a flood, for example. It also prevents recirculation of SZ drainage water when using Flood Codes.

Testing has shown that the test on drainage to local depression can negatively impact runtime because the number of outer iterations in the PCG solver may increase. Thus, the downstream check has been separated into two Extra Parameters.

Parameter Name	Type	Value
check gradient for drainage to river or mouse	Boolean	On
check gradient for drainage to local depression	Boolean	On

21.5.4 Time varying SZ drainage parameters

In projects where you want to simulate the build out of an SZ drainage network over time, or changes in the SZ drainage time constants over time, then you can use this set of extra parameters. Without this set of extra parameters you would have to hot start your simulation at regular time intervals with the new SZ drainage parameters.

The time varying SZ drains are also allowed to shift between layers. However, if the SZ drainage level goes above or below the model, the level will be adjusted and a warning is issued to the log file.

Note: The SOR solver does not allow drainage in any layer except the top layer and the drain level will be adjusted accordingly.

Note: If you specify time varying SZ drainage parameters, you will not be able to use any of the drainage routing methods that depend on the drain level. The preprocessor checks this and gives an error if you have specified

- option 1 (routing based on levels), or
- option 3 (distributed options) AND any of the distributed option codes are 1 (routing based on levels in these cells).

To activate time varying SZ drainage parameter options, you must specify the following extra parameters

Parameter Name	Type	Value
time varying drainage levels	Boolean	On
time varying drainage time constants	Boolean	On
drainage level dfs2 file name	file name	.dfs2 file
drainage level item number	integer	item number in dfs2 file, greater than zero
drainage time constant dfs2 file name	file name	.dfs2 file
drainage time constant item number	integer	item number in dfs2 file, greater than zero
Optional:		
mean step accumulated drainage levels	Boolean	On
mean step accumulated drainage time constants	Boolean	On

The dfs2 Drain Level is an elevation that can be specified using the following three EUM Data Units *(V1 p. 131)*:

- Elevation
- Depth Below Ground (i.e. positive values)
- Height Above Ground (i.e. negative values)

By default, the Time Series Types *(V1 p. 143)* is Instantaneous, but there is an extra option that allows you to used Mean Step Accumulated values if you want. The use of Mean Step Accumulated does not change the meaning of the item, but changes the way the values are interpolated.

Note: The code does not check for the time series type.

All specifications are printed to the *projectname*_PP_Print.log and *projectname*_WM_Print.log files.

21.5.5 Time varying hydraulic conductivity

This set of extra parameters allows you to define the horizontal and vertical hydraulic conducivity as time varying parameters. This is especially useful in permafrost conditions or in mining applications.

The time varying hydraulic conductivity can be specified for any or all SZ layers. For each layer with a time varying value, a time varying dfs2 file must be specified with an Item Type = “Conductivity”.

Each time varying layer, requires all three parameters: a Boolean “on”, the file name, and the item number in the file.

Layers are number from the top down, starting with 1 at the top.

Parameter Name	Type	Value
time varying horizontal conductivity	Boolean	On
One set of 3 parameters for each SZ layer that has a time varying hydraulic conductivity		
time varying horizontal conductivity layer 1	Boolean	On
horizontal conductivity layer 1 dfs2 file name	file name	.dfs2 file Item type: Conductivity
horizontal conductivity layer 1 item number	integer	item number in dfs2 file, greater than zero
Optional:		
mean step accumulated horizontal conductivity	Boolean	On

Parameter Name	Type	Value
time varying vertical conductivity	Boolean	On
One set of these 3 parameters for each SZ layer that has a time varying hydraulic conductivity		
time varying vertical conductivity layer 1	Boolean	On
vertical conductivity layer 1 dfs2 file name	file name	.dfs2 file Item type: Conductivity
vertical conductivity layer 1 item number	integer	item number in dfs2 file, greater than zero
Optional:		
mean step accumulated vertical conductivity	Boolean	On

By default, the Time Series Types *(V1 p. 143)* are Instantaneous, but there is an extra option that allows you to used Mean Step Accumulated values if you want. The use of Mean Step Accumulated does not change the meaning of the item, but changes the way the values are interpolated.

Note: The code does not check for the time series type.

21.5.6 Canyon exchange option for deep narrow channels

In the case of a deep, narrow channel crossing multiple model layers, the head difference can optionally be limited by the bottom elevation of the layer. Thus,

$$\Delta h = h_{grid} - max(h_{riv}, z) \tag{21.2}$$

where z is the bottom of the current layer.

The above formulation reduces the infiltration from upper layers by reducing the available gradient. Without the 'Canyon' option, MIKE SHE effectively assumes that the river is hydraulically connected to the upper most model layer, since MIKE SHE calculates the exchange flow with all layers that intersect the river based on the difference between the river level and the water table.

Currently, this option is only available for steady-state models.

Parameter Name	Type	Value
enable canyon exchange	Boolean	On

21.5.7 2-norm reduction-criteria in the inner iteration loop

When the 2-norm option is active, the inner iteration loop of the PCG solver ends when the specified reduction of the 2-norm value is reached. Thus, if the 2-norm reduction criteria is set to 0.01, the inner iteration residual must be reduced by 99% before the inner iteration loop will exit.

This option is sometimes efficient in achieving convergence in the linear matrix solution before updating the non-linear terms in the outer iteration loop. It may thus improve the convergence rate of the solver. Continued iterations to meet user-defined criteria in the inner loop may not be feasible before the changes in the outer iteration loop have been minimised. On the other hand, very few iterations in the inner loop may not be sufficient. The 2-norm may be used to achieve a more optimal balance between the computational efforts spent in the respective solver loops.

Convergence is, however, not assumed until the user defined head and water balance criteria are fulfilled. A reasonable value for the 2-norm reduction criteria has been found to be 0.01. .

Parameter Name	Type	Value
2-norm reduction fraction for sz solver	Float	Greater than 0

21.6 Water Quality

21.6.1 Disable SZ solute flux to dummy UZ

The following Extra Parameter is useful, if you are using an alternative UZ model, such as DAISY, in MIKE SHE and you are trying to couple it to the WQ.

In this case, you will be typically using the Negative Precipitation *(V2 p. 329)* option. If you use this option, then you will not use a MIKE SHE UZ, and the UZ-SZ exchange will pass through a “dummy UZ” layer. When this is coupled to the water quality, solutes will also be passed to this dummy UZ layer and removed from the SZ domain and the model.

To prevent the upflow of solutes from SZ to the dummy UZ, you must specify the following Extra Parameter..

Parameter Name	Type	Value
disable sz transport to dummy uz	Boolean	On

21.6.2 SZ boundary dispersion

A detailed test of the MIKE SHE WQ engine comparing an SZ model with fixed concentration at an inflow boundary with an analytical solution for a fixed concentration source, showed that MIKE SHE under-estimates the mass flux into the model when the model includes longitudinal dispersion.

The problem is that the SZ transport scheme (QUICKEST) doesn't include dispersive transport to/from open boundary cells. This is as designed, but apparently not correct. After including the boundary dispersion, the mass input to the model is within 2% of the analytical solution.

From Release 2011 and onwards, the boundary dispersion has been made optional for backwards compatibility and is activated with the extra-parameter: .

Parameter Name	Type	Value
enable sz boundary dispersion	Boolean	On

However, the SZ boundary dispersion option (above) does not calculate dispersive transport to an inflow boundary correctly. Again, this problem was identified in the tests of MShe_WQ with MIKE ECO Lab vs analytical solution. For example:

- Species 1 enters the model via an inflow (flux) boundary with fixed concentration - including dispersive transport due to the new sz boundary dispersion option.
- Species 1 decays to Species 2 which again decays to Species 3.
- The concentrations of Sp2 & Sp3 are too high, especially close to the inflow boundary.

The analytical solution includes dispersive transport of Sp2 & Sp3 against the flow direction because the concentration of these species are 0 at the boundary. However, this dispersive mass flux to the boundary is not included in the SZ solver due to an old check in the code. When mass flux to/from a boundary point is reversed compared to the flow direction, the mass flux is simply reset to 0.

This made sense before the boundary dispersion was implemented because advective transport against the flow direction would be wrong. But, now, when the boundary dispersion is active, this situation is allowed.

21.7 Miscellaneous

21.7.1 Maximum number of threads

MIKE SHE has been parallelized, but it uses a “shared memory” approach. This means that the efficiency starts dropping off even after the second core. After about 4 cores, you won't see much speed improvement. It depends on the model.

MSHE 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. The following extra parameter option allows you to control the parallelization of each simulation.

Parameter Name	Type	Value
max number of threads	Integer	Greater than or equal to 1

21.7.2 Including OpenMI

If you want to link a program to MIKE SHE using OpenMI then you must specify the following Extra Parameter..

Parameter Name	Type	Value
make omi file	Boolean	On

When enabled, an *.omi file for WM is created called MIKESHE_WM_Setup-Name.omi.

If Water Quality is included, a second *.omi file is created called MIKE-SHE_WQ_SetupName.omi.

These omi files are to be used in the OpenMI configuration editor.

21.7.3 Plot control for Detailed Time Series Output

On the Results Tab, the Detailed Time Series plots are created in a set of .html files. The default file length is 5 plots per file. However, you can control the number of plots per html file by using the following Extra Parameter. .

Parameter Name	Type	Value
max number of detailed ts plots per html file	Integer	Greater than or equal to 1

Note: If the loading of the html file can become very slow if the simulation is long and there are many plots in the file.

21.7.4 Extra Pre-Processing output

The pre-processing log file, *_PP_Print.log, can be very long. To improve the readability of the file, some long tables and unnecessary output has been removed. To include these tables in the log file, use the following extra parameter:

Parameter Name	Type	Value
extended PP print	Boolean	On

21.7.5 GeoViewer Output

The GeoViewer is a MIKE Zero tool that is used in the MIKE GeoModel product for viewing geologic cross-sections in your conceptual model.

The GeoViewer Output extra parameters will create a set of dfs2 output files during the pre-processing that will allow you to look at your pre-processed model in the GeoViewer.

The GeoViewer Output is activated by

Parameter Name	Type	Value
make SZ level dfs2 files	Boolean	On
Optional		
adjust dfs2 levels	Boolean	On

If this option is active, then the following files will be created:

- *setupname**setupname*_GeoLayers.dfs2 - containing the top and bottom of each geologic layer

If there are lenses:

- *setupname**setupname*_GeoLenses.dfs2 - containing the top and bottom of each geologic lense and delete values where there are no lenses

If the computational layers are not defined by geologic layers:

- *setupname**setupname*_CompLayers.dfs2 - containing the top and bottom of each computational layer

If the optional second parameter is used, then the top and bottom elevations that are written to the files will be adjusted to be confined between the topography and the lowest computational layer.

REFERENCES

/1/ Abbott, M.B., Bathurst, J.C., Cunge, J.A., O'Connell, P.E., and Rasmussen, J. (1986): An introduction to the European Hydrological System - Système Hydrologique Européen, SHE, 2: Structure of a physically-based, distributed modelling system. J. Hydrol., 87, 61-77.

/2/ Abbott, M.B. and Cunge, J.A.(1982): Engineering Applications of Computational Hydraulics, Vol. I. Pitman Advanced Publ. Program, London.

/3/ Ammentorp, H.C., A. Refsgaard (1991): Operationalizing a three-dimensional solute transport model (in Danish). Report M4/7. Landfill project. Danish Environmental Protection Agency.

/4/ Andersen, J., Refsgaard, J.C. and Jensen, K.H. (2001): Distributed hydrological modelling of the Senegal River Basin - model construction and validation, in Journal of Hydrology, v247, p200-214.

/5/ Bear, J (1979): Hydraulics of groundwater. McGraw-Hill Inc., 567 pp.

/6/ Bear, J. and A. Verruijt (1987): Modeling Groundwater Flow and Transport. D. Reidel Pub. Com., Dordrecht, Holland.

/7/ Boesten, J.J.T.I. & van der Linden, A.M.A., 1991. Modelling the influence of sorption and transformation on pesticide leaching and persistence. J. Environ. Qual., 20, 425-435.

/8/ Brusseau, M. L., 1995: The effect of nonlinear sorption on the transformation of contaminants during transport in porous media. Journal of Contaminant Hydrology, 17, p 277-291.

/9/ Burnett, R.D. and E.O. Frind (1987): Simulation of contaminant transport in three dimensions. 2. Dimensionality Effects. Water Resour. Res., 23(4), 695-705.

/10/ Butts, M. B., Payne, J.T., Kristensen, M. and Madsen, H. (2004): An evaluation of the impact of model structure on hydrological modelling uncertainty for streamflow prediction, in Journal of Hydrology, v298, p242-266.

/11/ Camp Dresser & McKee Inc. (2001): Evaluation of Integrated Surface Water and Groundwater Modeling Tools, Internal report, pp35

/12/ Christiansen, J.S., M. Thorsen, T. Clausen, S. Hansen , J.C. Refsgaard (2004), Modelling of macropore flow and transport processes at catchment scale, Journal of Hydrology 299, 136–158

/13/ Chow, T. C., Maidment, D. R. and Mays, L.W., 1988. Applied

Hydrology, McGraw-Hill Series in Water Resources and Environmental Engineering, pp.201-210, 257-263

/14/ Crawford, N.H., and Linsley, R.K., 1966, Digital simulation in hydrology: the Stanford Watershed Simulation Model IV: Technical Report no. 39, Department of Civil Engineering, Stanford University, Stanford, CA, 210p

/15/ Ekebjærg, L. and Justesen, P. (1991): An explicit scheme for advection-diffusion modelling in two dimensions. Computer Methods in Applied Mechanics and Engineering, 88 (3), 287-297.

/16/ FAO, 1979. Yield response to water. FAO irrigation and drainage paper no. 33. FAO, Rome Italy.

/17/ Fetter, C.W., 1992: Contaminant Hydrogeology. Macmillan Pub. Com., New York, New York.

/18/ Fleming G., 1975. *Computer Simulation Techniques in Hydrology,* Elsevier Environmental Science Series

/19/ Freeze, R.A., Cherry, J.A. (1979): Groundwater, Prentice-Hall, Inc., Englewood Cliffs, N.J.

/20/ Freeze, R.A., and Harlan, R.L. (1969): Blueprint of a physically-based, digitially simulated hydrologic response model, in Journal of Hydrology v9, p237-258.

/21/ Graham, D.N. and M. B. Butts (2005): Flexible, integrated watershed modelling with MIKE SHE. In Watershed Models, Eds. V.P. Singh & D.K. Frevert Pages 245-272, CRC Press.

/22/ Hill, M.C. (1990): Pre-conditioned conjugate gradient 2 (PCG2), a computer program for solving groundwater flow equations: U.S. Geological Survey Water Resources Investigations Report 90-4048, 43 p.

/23/ Jarvis, N.J., (1991) MACRO - A model of water movement and solute transport in macroporous soil. Monograph, Reports and disserta-tions, 9. Dept. of Soil Science, Swedish Univ. of Agrig. Sci., Uppsala, Sweden.

/24/ Jarvis, N.J., (1998) Modelling the impact of preferential flow on non-point source pollution, In: Physical Nonequilibrium Processes in Soils. Modeling and Application, Ann Arbor Press, Inc, Chelsea, Michigan, pp. 195–217.

/25/ Jarvis, N.J., Hollis, J.M., Nicholls, T.H., Mayr, T., Evans, S.P., (1997) MACRO_DB: a decision-support tool for assessing pesticide fate and mobility in soils. Environmental Modelling and Software 12,

251–265.

/26/ Jensen, K.H., 1983. Simulation of water flow in the unsaturated zone including the root zone. Institute of Hydrodynamics and hydraulic engineering. Technical University of Denmark. Series Paper no. 33, 259pp.

/27/ Jensen, K.H., H.C. Ammentorp and T. Sevel (1984): Modelling solute transport in the unsaturated zone. Nordic Hydrology 15(4/5), 223-234.

/28/ Kaiser-Hill Company (2001): Model Code and Scenario Selection Report Site Wide Water Balance Rocky Flats Environmental Technology Site, Internal Report, pp66

/29/ Kimley-Horn & Assoc. Inc., US Army Corps of Engineers, Jacksonville District, and South Florida Water Management District (2002): B.2 Hydraulics - Final Model Evaluation Report - EAA Storage Reservoirs - Phase 1, public report to the South Florida Water Management District, pp40

/30/ Klein, M., 1995. PELMO - Pesticide Leaching Model, version 2.01. Users Manual. Frauenhofer-Institut für Umweltche-mie und Ökotox-iko-logie, 57392 Schmallenberg

/31/ Knudsen, J., 1985a. WATBAL User's Guide. Danish Hydraulic Institute, Denmark

/32/ Knudsen, J., 1985b. WATBAL Hydrological Modelling System A Short Description. Danish Hydraulic Institute, Denmark

/33/ Konikow, L.F. and D.B. Grove (1977): Derivation of Equations describing Solute Transport in Groundwater, US.G.S. Water Resour. Invest. 77-19.

/34/ Kristensen, K. J. and S.E. Jensen, 1975. A model for estimating actual evapotranspiration from potential evapotranspiration. Royal Veterinary and Agricultural University, Nordic Hydrology 6, pp. 170-188

/35/ Larsson, M.H., Jarvis, N.J., (1999) Evaluation of a dual-porosity model to predict field-scale solute transport in a macroporous soil. Journal of Hydrology 215, 153–171.

/36/ Leonard, B.P. (1979): A stable and accurate convective modelling procedure based on quadratic upstream interpolation. Comput. Methods Appl. Mech. Engrg., Vol. 19, pp. 59-98.

/37/ Leonard, B.P. (1988): Simple high-accuracy resolution program for convective modelling of discontinuities, Int. Journal for Numerical

Methods in Fluids, Vol. 8, pp. 1291-1318.

/38/ McDonald, M.C., and A.W. Harbaugh (1988): A modular three-dimensional finite difference groundwater flow model: U.S. Geological Survey Techniques of Water Resources Investigations, book 6, chap.A.1, 586 p.

/39/ Middlemis, H. (2004): Benchmarking Best Practice for Groundwater Flow Modelling, report to The Winston Churchill Memorial Trust of Australia, pp45

/40/ Miljøstyrelsen, 1981. The surface water component of an integrated hydrological model. The Danish committee for hydrology. Suså report no. 12. The Danish Environmental Protection Agency, Copenhagen.

/41/ Plauborg, F. and J.E. Olesen, 1991. Development and validation of the model MARKVAND for irrigation scheduling in agriculture. Tidsskrift for Planteavls Specialserie, Beretning S 2113. Danish Institute of Plant and Soil Science. Foulum, Denmark. (In Danish)

/42/ Refsgaard, A., Refsgaard, J.C., Clausen, T. (1993): A three-dimensional module for groundwater flow and solute transport in SHE. Danish Hydraulic Institut - Internal Note.

/43/ Refsgaard, J.C. and J. Knudsen, 1996. Operational validation and intercomparison of different types of hy7drological models, *Water Resources Research*, **32**(7): pp2189-2202

/44/ Refsgaard, J.C., T.H. Christensen and H.C. Ammentorp (1991): A Model for oxygen transport and consumption in the unsaturated zone. Journal of Hydrology, 129, 349-369.

/45/ Refsgaard, J.C. and Storm, B. (1995): MIKE SHE, in Computer Models of Watershed Hydrology, Singh, V.P., Ed., Water Resources Publications, Colorado, USA, p809-846.

/46/ Scheidegger, A.E. (1961): General theory of dispersion in porous media. Jour. of Geophys. Research, Vol. 66, no. 10.

/47/ Thoma, S.G., Gallepgos, D.P., Smith, D.M., (1992) Impact of fracture coatings on fracture/matrix flow interactions in unsaturated porous media. Water Resources Research 28, 1357–1367.

/48/ Thomas, R.G. (1973): Groundwater models. Irrigation and drainage. Spec. Pap. Food Agricultural Organis. No. 21, U.N., Rome.

/49/ Thorsen, M., Jørgensen, P.R., Felding, G., Jacobsen, O.H., Spliid, N.H., Refsgaard, J.C., (1998) Evaluation of a stepwise procedure for comparative validation of pesticide leaching models. Journal of

Environmental Quality 27(5), 1183–1193.

/50/ Toride, N., F.J. Leij and M. Th. van Genuchten (1995). The CXTFIT code for estimating transport parameters from laboratory or field tracer experiments. Version 2.0. Research report No. 137. U.S. Salinity Lab-oratory. Agricultural Research Service. USDA, Riverside, California.

/51/ Vested, H.J., Justesen, P. and Ekebjærg, L. (1992): Advection-dispersion modelling in three dimensions. Appl. Math. Modelling, Vol. 16, pp 506-519.

/52/ Villholth, K., 1994. Field and Numerical Investigation of Macropore Flow and Transport Processes. Series Paper no. 57. Institute of Hydrodynamics and Hydraulic Engineering, Technical University of Denmark, 230 pp.

/53/ Vinther, F.P., Eiland, F., Lind, A.-M., Elsgaard, L., (1999) Microbial biomass and numbers of denitrifies related to macropore channels in agricultural and forest soils. Soil Biology and Biochemistry 31, 603–611.

/54/ West Consultants Inc., Gartner Lee Ltd., and Aqua Terra Consultants (2001): Scientific Review of the Integrated Hydrologic Model, IGSW/CNTB121, report prepared for Tampa Bay Water, pp197

/55/ Yan J.J. and K.R. Smith, 1994: Simulation of Integrated Surface Water and Ground Water Systems - Model Formulation. Water Resources Bulletin, Vol. 30, No. 5, pp 1-12.

INDEX