MIKE
Powered by DHI

MIKE SHE

User Guide and Reference Manual



PLEASE NOTE

COPYRIGHT

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI A/S (hereinafter referred to as “DHI”). For details please refer to your 'DHI Software Licence Agreement'.

LIMITED LIABILITY

The liability of DHI is limited as specified in your DHI Software Licence Agreement:

In no event shall DHI or its representatives (agents and suppliers) be liable for any damages whatsoever including, without limitation, special, indirect, incidental or consequential damages or damages for loss of business profits or savings, business interruption, loss of business information or other pecuniary loss arising in connection with the Agreement, e.g. out of Licensee's use of or the inability to use the Software, even if DHI has been advised of the possibility of such damages.

This limitation shall apply to claims of personal injury to the extent permitted by law. Some jurisdictions do not allow the exclusion or limitation of liability for consequential, special, indirect, incidental damages and, accordingly, some portions of these limitations may not apply.

Notwithstanding the above, DHI's total liability (whether in contract, tort, including negligence, or otherwise) under or in connection with the Agreement shall in aggregate during the term not exceed the lesser of EUR 10.000 or the fees paid by Licensee under the Agreement during the 12 months' period previous to the event giving rise to a claim.

Licensee acknowledge that the liability limitations and exclusions set out in the Agreement reflect the allocation of risk negotiated and agreed by the parties and that DHI would not enter into the Agreement without these limitations and exclusions on its liability. These limitations and exclusions will apply notwithstanding any failure of essential purpose of any limited remedy.



CONTENTS

Getting Started 21

1 Introduction 23
1.1 Process models 26
1.2 Requirements 28
1.2.1 Input requirements 28
1.2.2 Model limits 30
1.2.3 MIKE SHE demo model limits 31
1.2.4 Hardware requirements 32
1.3 Getting Help 33
1.4 Service and Maintenance 33
1.4.1 Service packs 34

2 Building a MIKE SHE Model 35
2.1 MIKE Zero 35
2.1.1 MIKE Zero editors 36
2.2 The MIKE SHE User Interface 37
2.2.1 The Setup Editor 38
2.2.2 The Setup Data Tree 40
2.2.3 Background maps 40
2.2.4 Initial model setup 41
2.2.5 Simulation parameters 43
2.2.6 Hot starting from a previous simulation 44
2.3 Model Domain and Grid 44
2.4 Topography 46
2.5 Climate 47
2.5.1 Precipitation 47
2.5.2 Evapotranspiration 47
2.5.3 Freezing and Snow melt 48
2.6 Land Use 49
2.6.1 Vegetation 50
2.7 Rivers, Lakes and Sewers 51
2.8 Overland Flow 52
2.8.1 Overland Drainage and Paving 55
2.8.2 Multi-cell overland flow 56
2.8.3 Overland Flow Performance 57
2.8.4 MIKE+ 2D Overland 57
2.9 Unsaturated Flow 57
2.9.1 Soil profiles 58
2.9.2 Initial conditions 59
2.9.3 Macropore flow 59
2.9.4 Green and Ampt infiltration 60
2.9.5 UZ column classification 60
2.9.6 Coupling between unsaturated and saturated zone 61
2.10 Saturated Groundwater Flow 62
2.10.1 Conceptual geologic model for the finite difference approach 63
2.10.2 Specific yield of upper SZ layer 64

2.10.3 Numerical layers . . . 65
2.10.4 Groundwater drainage . . . 65
2.10.5 Groundwater wells . . . 66
2.10.6 Linear reservoir groundwater method . . . 66
2.11 Storing of results . . . 67
2.11.1 Detailed time series output . . . 69
2.11.2 Detailed River Time Series output . . . 69
2.11.3 Grid series output . . . 70

3 Running Your Model . . . 71
3.1 Preprocessing your model . . . 71
3.1.1 Viewing the pre-processed data . . . 72
3.1.2 Editing the pre-processed data . . . 72
3.2 Pre-processed data items . . . 73
3.2.1 MIKE 1D coupling . . . 73
3.2.2 Land Use . . . 73
3.2.3 Unsaturated Flow . . . 74
3.2.4 Saturated Flow . . . 74
3.3 The Results Tab . . . 76
3.3.1 Detailed Time Series Results . . . 77
3.3.2 Gridded Results . . . 78
3.3.3 Detailed River Time Series Results . . . 80
3.3.4 Run Statistics . . . 80
3.4 Controlling Your Simulation . . . 82
3.4.1 Model limits . . . 82
3.4.2 Speeding up your simulation . . . 82
3.4.3 Controlling the Time Steps . . . 83
3.5 Using Batch Files . . . 86
3.6 OpenMI . . . 88
3.7 Parallelization of MIKE SHE . . . 88

4 MIKE SHE Results . . . 91
4.1 Output Files . . . 91
4.1.1 Log files . . . 92
4.2 Multiple Simulations . . . 92
4.3 Output Items . . . 92
4.3.1 Overland flow . . . 93
4.3.2 Recharge . . . 94

5 The Results Viewer . . . 97
5.1 Saturated Zone Cross-section Plots . . . 97
5.1.1 Saving and loading profiles . . . 98
5.2 UZ Specific Plots . . . 98
5.2.1 UZ Scatter and Filled Plots . . . 98
5.2.2 Transient UZ Column Plot . . . 98

6 Using the Water Balance Tool . . . 99
6.1 Creating a Water Balance . . . 99

6.1.1 Create a new water balance document . . . 100
6.1.2 Extract the water balance data . . . 101
6.1.3 Specify your water balance . . . 102
6.1.4 Calculate and View the Water Balance . . . 105
6.2 Display and auto-run of water balances in the MIKE SHE Results tab . . . 106
6.3 Calculating Water Balances in Batch Mode . . . 106
6.4 Available Water Balance Items . . . 108
6.4.1 Snow Storage . . . 109
6.4.2 Canopy interception storage . . . 111
6.4.3 Ponded water storage . . . 113
6.4.4 Unsaturated Zone storage . . . 117
6.4.5 Saturated Zone storage . . . 119
6.4.6 Limitations for Linear Reservoir and Sub-catchment OL Water Balance . 130
6.5 Standard Water Balance Types . . . 131
6.5.1 Total water balance . . . 133
6.6 Making Custom Water Balances . . . 136
6.6.1 Customizing the chart output . . . 138

MIKE ZERO Options . . . 141

7 EUM Data Units . . . 143
7.1 Changing from SI to Imperial (American) data units. . . . 145
7.2 Restoring the default units . . . 145
7.3 Changing the EUM data type of a Parameter . . . 146
7.3.1 Changing the EUM Type of a .dfs0 Parameter . . . 147
7.3.2 Changing the EUM Type of a .dfs2 Parameter . . . 148

Working with Data . . . 151

8 Time Series Data . . . 153
8.1 Creating Time Series in MIKE SHE . . . 153
8.1.1 Import from ASCII or Excel . . . 155
8.2 Working with Spatial Time Series . . . 155
8.3 Time Series Types . . . 155

9 Using MIKE SHE with ArcGIS . . . 159
9.1 Converting ArcGIS ASCII Grids to dfs . . . 159
9.2 Converting dfs files to ASCII Grids and shape files . . . 159

10 Spatial Data . . . 161
10.1 The Grid Editor . . . 161
10.2 Gridded Data Types . . . 161
10.3 Integer Grid Codes . . . 162
10.4 Gridded (.dfs2) Data . . . 163
10.4.1 Stationary Real Data . . . 164
10.4.2 Time-varying Real Data . . . 166

10.4.3 Integer Grid Codes . . . 167
10.5 Interpolation Methods . . . 169
10.5.1 Bilinear Interpolation . . . 169
10.5.2 Triangular Interpolation . . . 172
10.5.3 Inverse Distance Interpolation . . . 173
10.6 Performing simple math on multiple grids . . . 174
10.7 Performing Complex Operations on Multiple Grids . . . 176
11 Setup Data Tab . . . 179
11.1 Display . . . 180
11.1.1 Foreground/Background . . . 181
11.1.2 Image Overlays . . . 182
11.1.3 Grid Overlays . . . 183
11.1.4 Shape Overlays . . . 184
11.1.5 River Overlays . . . 185
11.1.6 MIKE SHE Well Database Overlays . . . 185
11.1.7 Current Layer . . . 186
11.2 Simulation Specification . . . 186
11.2.1 Simulation Title . . . 188
11.2.2 Simulation Period . . . 188
11.2.3 Time Step Control . . . 190
11.2.4 OL Computational Control Parameters . . . 195
11.2.5 UZ Computational Control Parameters . . . 200
11.2.6 SZ Computational Control Parameters . . . 201
11.3 Python Plugins . . . 206
11.4 Water Quality Simulation Specification . . . 207
11.4.1 WQ Simulation Title . . . 208
11.4.2 WQ Simulation Period . . . 209
11.4.3 Water Quality Time Step Control . . . 211
11.5 Particle Tracking Specification . . . 213
11.5.1 Particle Tracking Simulation Title . . . 215
11.5.2 Particle Tracking Simulation Period . . . 216
11.5.3 Particle Tracking Control . . . 218
11.6 MIKE ECO Lab Template Specification . . . 220
11.6.1 MIKE ECO Lab Overview Tables . . . 221
11.7 User Specified MIKE ECO Lab Constants and Forcings . . . 222
11.8 Species . . . 223
11.9 Water Quality Sorption and Decay . . . 225
11.10 Model Domain and Grid . . . 228
11.11 Sub-catchments . . . 230
11.11.1 River Links . . . 231
11.12 Topography . . . 232
11.13 Climate . . . 233
11.13.1 Precipitation Rate . . . 234
11.13.2 Precipitation Lapse Rate . . . 236
11.13.3 Net Rainfall Fraction . . . 236
11.13.4 Infiltration Fraction . . . 237

11.13.5 Reference Evapotranspiration . . . 238
11.13.6 Short Wave Solar Radiation . . . 239
11.13.7 Air Temperature . . . 240
11.13.8 Temperature Lapse Rate . . . 241
11.13.9 Wet Lapse Rate . . . 242
11.13.10 Snow Melt . . . 243
11.13.11 Threshold Melting Temperature . . . 244
11.13.12 Degree-day Melting or Freezing Coefficient . . . 245
11.13.13 Melting Coefficient for Solar Radiation . . . 245
11.13.14 Minimum Snow Storage for Full Area Coverage . . . 246
11.13.15 Maximum Wet Snow Fraction in Snow Storage . . . 246
11.13.16 Initial Total Snow Storage . . . 247
11.13.17 Initial Wet Snow Fraction . . . 248
11.14 Land Use . . . 248
11.14.1 Vegetation . . . 249
11.14.2 Vegetation Properties . . . 256
11.14.3 Irrigation Command Areas . . . 258
11.14.4 Sheet Application Area . . . 264
11.14.5 Irrigation Demand . . . 265
11.14.6 Irrigation Priorities . . . 267
11.14.7 Irrigation Log Areas . . . 268
11.15 Rivers, Lakes and Sewers . . . 268
11.15.1 Flood codes . . . 270
11.15.2 Bathymetry . . . 271
11.16 Overland Flow . . . 271
11.16.1 Manning number . . . 273
11.16.2 Detention Storage . . . 274
11.16.3 Initial Water Depth . . . 275
11.16.4 Surface-Subsurface Leakage Coefficient . . . 276
11.16.5 Sub-areas for reduced ponded exchange . . . 277
11.16.6 Paved Area Fraction . . . 278
11.16.7 Separated Flow Areas . . . 279
11.16.8 Overland Drainage . . . 280
11.16.9 Overland Flow Zones . . . 286
11.16.10 Initial Mass (per Species) . . . 287
11.16.11 Dispersion coefficient along columns/rows . . . 288
11.17 Unsaturated Zone . . . 288
11.17.1 Soil Profile Definitions . . . 293
11.17.2 Mass Transfer Coefficient to Unsaturated Macropores . . . 296
11.17.3 Groundwater Depths used for UZ Classification . . . 297
11.17.4 Ground Water Table for UZ classification . . . 298
11.17.5 Partial automatic classification . . . 298
11.17.6 Specified classification . . . 299
11.17.7 2-Layer UZ soil properties . . . 300
11.17.8 ET Surface Depth . . . 302
11.17.9 Initial Conditions (UZ) . . . 303
11.18 Groundwater table for lower UZ boundary . . . 304
11.19 Saturated Zone . . . 304

11.19.1 Linear Reservoir Method . . . 305
11.19.2 3D Finite Difference Method . . . 306
11.19.3 Interflow Reservoirs . . . 308
11.19.4 Baseflow Reservoirs . . . 310
11.19.5 Geological Units . . . 312
11.19.6 Geological Layers . . . 313
11.19.7 Geological Lenses . . . 314
11.19.8 Geological Unit Distribution . . . 315
11.19.9 Horizontal Hydraulic Conductivity . . . 315
11.19.10 Vertical Hydraulic Conductivity . . . 316
11.19.11 Specific Yield . . . 316
11.19.12 Specific Storage . . . 317
11.19.13 Porosity . . . 318
11.19.14 Secondary Porosity . . . 318
11.19.15 Bulk Density . . . 319
11.19.16 Dispersion Coefficients LHH, THH, TVH, LVV, THV . . . 320
11.19.17 Water Quality Layers . . . 321
11.19.18 Dual Porosity transfer coefficient . . . 322
11.19.19 Computational Layers . . . 323
11.19.20 Initial Potential Head . . . 324
11.19.21 Initial Soil Temperature . . . 324
11.19.22 Outer boundary conditions . . . 325
11.19.23 Internal boundary conditions . . . 327
11.19.24 Initial (secondary) concentration . . . 329
11.19.25 Initial number of particles per cell . . . 330
11.19.26 PT Registration Codes/Lenses . . . 330
11.19.27 Drainage . . . 331
11.19.28 Drain Level . . . 334
11.19.29 Drain Time Constant . . . 335
11.19.30 Drain Codes . . . 335
11.19.31 Option Distribution . . . 337
11.19.32 Pumping Wells . . . 338
11.20 Sources . . . 339
11.20.1 Strength . . . 340
11.21 Storing of Results . . . 341
11.21.1 Detailed WM time series output . . . 344
11.21.2 Detailed WQ time series output . . . 347
11.21.3 Detailed River Time Series Output . . . 350
11.21.4 Grid series output . . . 352
11.22 Extra Parameters . . . 353
11.23 Data description . . . 354
11.23.1 Layer description . . . 354
11.23.2 Item description . . . 354
11.24 Geometry . . . 354
11.24.1 Upper Level . . . 355
11.24.2 Lower Level . . . 355
11.24.3 Horizontal Extent . . . 356
11.25 Water Quality Decay Processes . . . 356

11.25.1 Half Life . . . 357
11.25.2 Macropore/Secondary Half Life . . . 357
11.26 Water Quality Sorption processes . . . 358
11.26.1 Linear Kd . . . 358
11.26.2 Freundlich Kf and N . . . 359
11.26.3 Langmuir alpha and beta . . . 360
11.26.4 Kinetic Sorption/Desorption Rate . . . 361

12 Preprocessed Data Tab . . . 363
12.1 Processed Data . . . 363
12.1.1 Viewing the pre-processed data . . . 364
12.1.2 Editing the pre-processed data . . . 364
12.1.3 Rivers . . . 365
12.1.4 Land Use . . . 365
12.1.5 Unsaturated Flow . . . 365
12.1.6 Saturated Flow . . . 366
12.2 Log Files . . . 367

13 Results Tab . . . 369
13.1 Results of Simulation . . . 369
13.1.1 MIKE SHE Detailed Time Series . . . 370
13.1.2 Gridded Data Results Viewer . . . 371
13.1.3 Detailed River Time Series . . . 373
13.2 Results Post-processing . . . 374
13.2.1 Run Statistics . . . 374
13.2.2 Water Balances . . . 376
13.2.3 PT Registration Extraction . . . 377
13.2.4 PT Pathline Extraction . . . 380
13.3 Log Files . . . 382

14 Python Scripting in MIKE SHE . . . 383
14.1 Python Plugins . . . 384
14.1.1 How to enable Python plugins in the MIKE SHE user interface . . . 384
14.1.2 Creating a Python plugin . . . 385
14.1.3 Functions available in the MShePy library . . . 386
14.1.4 How to exchange MIKE SHE items with Python . . . 386
14.1.5 How to create and handle MShePy datasets . . . 387
14.2 Running MIKE SHE simulations from Python scripts . . . 390
14.2.1 Initializing a MIKE SHE model . . . 390
14.2.2 Running MIKE SHE water movement . . . 391
14.2.3 Termination and postprocessing . . . 391
14.2.4 Python script example . . . 391
14.3 Plugins application example . . . 392
14.3.1 Calculation of the reference evapotranspiration . . . 392
14.3.2 Reference evapotranspiration adjusted for the altitude . . . 393
14.3.3 Managing irrigation as function of precipitation in the previous months . . . 394
14.4 Summary tables of exchangeable items between MIKE SHE and Python . . . 395

MIKE SHE Editors . . . 399
15 Well editor . . . 401
15.1 Interactive Map . . . 401
15.2 Well Locations . . . 402
15.3 Well Filters . . . 403
15.4 Layers Display . . . 404
15.5 Importing Data . . . 404
15.5.1 Importing TAB delimited text file . . . 405
16 Vegetation Properties Editor . . . 407
16.1 Vegetation Database Items . . . 407
16.1.1 Specifying Vegetation Properties in a Database . . . 407
16.1.2 Vegetation stages . . . 408
16.1.3 Evapotranspiration Parameters . . . 409
16.1.4 Vegetation Development Table . . . 409
16.1.5 Irrigation Parameters . . . 410
16.2 Example database . . . 412
17 UZ Soil Properties Editor . . . 413
17.1 Definitions of soil properties . . . 413
17.2 Soils Database . . . 414
17.3 Soil Moisture Retention Curve . . . 415
17.4 Hydraulic Conductivity Function . . . 418
17.5 Macropore Parameters . . . 421
18 Water Balance Editor . . . 423
18.1 Extraction . . . 424
18.2 Postprocessings . . . 425
18.2.1 Postprocessing Detail . . . 426
18.3 Results . . . 428

The MIKE SHE Reference Guide . . . 429
19 Reference Guide Overview . . . 431

Reference Manual for Water Movement . . . 433
20 Water Movement Overview . . . 435
21 Evapotranspiration - Technical Reference . . . 437
21.1 ET from Snow . . . 437
21.2 ET from Canopy Interception . . . 438
21.2.1 Evaporation from the Canopy . . . 438
21.3 Kristensen and Jensen method . . . 439
21.3.1 Plant Transpiration . . . 439
21.3.2 Soil Evaporation . . . 444
21.3.3 Evapotranspiration Coefficients C1, C2 and C3 . . . 445

21.4 The 2-Layer Water Balance Method . . . 447
21.4.1 Explicit ET calculation . . . 448
21.4.2 Soil Moisture . . . 449
21.4.3 Canopy Interception . . . 451
21.4.4 Infiltration . . . 452
21.4.5 Evapotranspiration . . . 453
21.4.6 Coupling to the Saturated Zone . . . 456

22 Working with Evapotranspiration - User Guide . . . 457
22.1 Some ET Definitions . . . 457
22.2 ET from Canopy Interception . . . 459

23 Snow Melt - Technical Reference . . . 461
23.1 Air temperature correction . . . 461
23.2 Precipitation correction . . . 462
23.3 Snow melting and freezing . . . 462
23.3.1 Snow pack area coverage . . . 464
23.3.2 ET from snow . . . 464

24 Working with Freezing and Melting - User Guide . . . 467
24.1 Elevation lapse rates . . . 467
24.1.1 Air temperature lapse rate . . . 467
24.1.2 Precipitation lapse rate . . . 467
24.2 Evapotranspiration . . . 468
24.3 Time varying infiltration . . . 468
24.4 Snow water equivalent . . . 469

25 Channel Flow - Technical Reference . . . 471
25.1 Surface Water Exchange Mechanisms . . . 471

26 Working with Rivers and Streams - User Guide . . . 473
26.1 Channel Flow . . . 473
26.1.1 MIKE 1D Overview . . . 473
26.2 Building a MIKE 1D river model . . . 474
26.2.1 MIKE 1D River Model Network Limitations . . . 475
26.2.2 River Cross-sections . . . 476
26.3 Coupling of MIKE SHE and a MIKE 1D River Model . . . 477
26.3.1 MIKE SHE Branches vs. MIKE 1D River Branches . . . 479
26.3.2 The River-Link Cross-section . . . 480
26.3.3 Connecting Rivers to MIKE SHE . . . 481
26.3.4 Evaluating your river links . . . 482
26.3.5 Groundwater exchange with Rivers . . . 485
26.3.6 Steady-state groundwater simulations . . . 489
26.4 Overland Flow Exchange with Rivers . . . 490
26.4.1 Lateral inflow to rivers from MIKE SHE overland flow . . . 491
26.4.2 Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes . . . 492
26.4.3 Direct Overbank Spilling to and from Rivers . . . 493
26.5 Unsaturated Flow Exchange with Rivers . . . 494

26.6 Water balance with Rivers . . . 494
26.7 MIKE+ User Interface . . . 494
26.7.1 MIKE SHE Coupling Reaches . . . 494
26.8 Common MIKE 1D River Error Messages . . . 501

27 Sewers - Technical Reference . . . 503

28 Working with Sewers - User Guide . . . 509
28.1 Output Files . . . 512
28.2 Warning Messages . . . 512
28.3 Water Balance Limitations . . . 513

29 Overland Flow - Technical Reference . . . 515
29.1 Finite Difference Method . . . 515
29.1.1 Diffusive Wave Approximation . . . 515
29.1.2 Finite Difference Formulation . . . 518
29.1.3 Successive Over-Relaxation (SOR) Numerical Solution . . . 519
29.1.4 Explict Numerical Solution . . . 520
29.1.5 Boundary conditions . . . 522
29.1.6 Low gradient damping function . . . 523
29.1.7 Overland drainage . . . 525
29.1.8 Multi-cell Overland Flow Method . . . 529
29.2 Simplified Overland Flow Routing . . . 533
29.2.1 Theoretical basis . . . 534
29.2.2 Implementation in MIKE SHE . . . 537
29.2.3 Coupling to other processes . . . 537
29.2.4 Avoiding the redistribution of ponded water . . . 537
29.2.5 Routing simple overland flow directly to the river . . . 538

30 Working with Overland Flow and Ponding- User Guide . . . 541
30.1 Overland Flow Control . . . 541
30.1.1 Principal parameters . . . 541
30.1.2 Separated Flow areas . . . 543
30.1.3 Output: Overland Flow Velocities . . . 543
30.2 Overland Flow Performance . . . 544
30.2.1 Stagnant or slow moving flow . . . 545
30.2.2 Threshold gradient for overland flow . . . 546
30.3 Multi-cell Overland Flow . . . 547
30.3.1 Evaporation . . . 549
30.3.2 Infiltration to SZ and UZ with the Multi-Grid OL . . . 550
30.3.3 Multi-cell Overland Flow + Saturated Zone drainage . . . 552
30.3.4 Test example for impact on simulation time . . . 552
30.3.5 Limitations of the Multi-cell Overland Flow Method . . . 556
30.3.6 Setting up and evaluating the multi-grid OL . . . 556
30.4 Overland Drainage . . . 558
30.4.1 Specifying Overland Drainage . . . 560
30.4.2 Runoff Coefficient . . . 562
30.4.3 Time constants . . . 563
30.4.4 OL Drainage Reference System . . . 563

30.4.5 Preprocessed data . . . 567
30.4.6 Time varying OL Drainage parameters . . . 568
30.5 Reduced OL leakage to UZ and to/from SZ . . . 569
30.5.1 Surface-Subsurface Leakage Coefficient . . . 570
30.5.2 Simulation of paved areas . . . 570
30.5.3 Backwards compatibility of Paving prior to Release 2017 . . . 570
30.5.4 Reduced Leakage with Multi-cell OL . . . 571
30.5.5 Reduced leakage in ponded areas only . . . 572
30.6 Simulating surface water infrastructure . . . 573
31 Unsaturated Zone - Technical Reference . . . 575
31.1 Richards Equation . . . 576
31.1.1 Numerical Solution of Richards Equation . . . 577
31.1.2 Boundary Conditions . . . 581
31.1.3 Initial Conditions . . . 585
31.1.4 Sources and sinks . . . 586
31.2 Gravity Flow . . . 587
31.2.1 Solution method . . . 587
31.2.2 Initial Conditions . . . 588
31.2.3 Sources and sinks . . . 589
31.3 Two-Layer Water Balance . . . 589
31.4 Simplified Macropore Flow (bypass flow) . . . 590
31.4.1 Adjustment for the 2-Layer Water Balance method . . . 591
31.5 Full Macropore Flow . . . 591
31.5.1 Macropore flow solution method . . . 593
31.5.2 Verification of the Macropore Algorithm . . . 595
31.6 Green and Ampt Infiltration . . . 596
31.6.1 Implementation of Green and Ampt in the 2LWB method . . . 598
31.6.2 Implementation in the Gravity Flow method . . . 599
31.7 Coupling the Unsaturated Zone to the Saturated Zone . . . 601
31.7.1 Steps in the Coupling Procedure . . . 602
32 Working with Unsaturated Flow - User Guide . . . 607
32.1 Physics of Unsaturated Zone Flow . . . 607
32.1.1 The saturation-pressure function . . . 609
32.1.2 Hydraulic conductivity as a function of saturation . . . 610
32.1.3 Typical soil parameters . . . 611
32.1.4 UZ Soils Editor . . . 611
32.2 Overview of the UZ User Interface . . . 611
32.2.1 Definition of the soil profile . . . 612
32.2.2 Definition of vertical UZ numerical grid . . . 613
32.2.3 Results . . . 615
32.3 Overview of Numerical Methods . . . 616
32.3.1 UZ-SZ coupling . . . 617
32.4 Options for the UZ model . . . 619
32.4.1 Bypass flow . . . 619
32.4.2 Full macropore flow . . . 619

32.4.3 Green and Ampt infiltration . . . 620
32.4.4 Lumped UZ Calculations (Column Classification) . . . 620

33 Saturated Flow - Technical Reference . . . 623
33.1 3D Finite Difference Method . . . 623
33.1.1 The Pre-conditioned Conjugate Gradient (PCG) Solver . . . 623
33.1.2 The Successive Overrelaxation (SOR) Solver . . . 628
33.1.3 PCG Steady State Solver . . . 632
33.1.4 Boundary Conditions . . . 632
33.1.5 SZ exchange with ponded water . . . 634
33.1.6 Saturated Zone Drainage . . . 635
33.1.7 Initial Conditions . . . 637
33.1.8 The Sheet Pile Module . . . 638
33.2 Linear Reservoir Method . . . 638
33.2.1 Linear Reservoir Theory . . . 639
33.2.2 General Description . . . 640
33.2.3 Subcatchments and Linear Reservoirs . . . 643
33.2.4 Calculation of Interflow . . . 644
33.2.5 Calculation of Interflow Percolation and Dead Zone Storage . . . 646
33.2.6 Calculation of Baseflow . . . 647
33.2.7 UZ Feedback . . . 648
33.2.8 Coupling to Rivers . . . 649
33.2.9 Limitations of the Linear Reservoir Method . . . 649

34 Working with Groundwater - User Guide . . . 651
34.1 Conceptualization of the Saturated Zone Geology . . . 651
34.1.1 Lenses . . . 653
34.2 Numerical Layers . . . 654
34.2.1 Specific Yield of the upper SZ numerical layer . . . 654
34.2.2 SZ Boundary Conditions . . . 655
34.2.3 Groundwater Drainage . . . 655
34.3 MIKE SHE versus MODFLOW . . . 663
34.3.1 Importing a MODFLOW 96 or MODFLOW 2000 Model . . . 664

Reference Manual for Water Quality . . . 669

35 Water Quality Overview . . . 671

36 Advection Dispersion - Reference . . . 673
36.1 Simulation control . . . 673
36.1.1 Flow Storing Requirements . . . 673
36.1.2 Internal Boundary Conditions . . . 673
36.1.3 Time step calculations . . . 675
36.2 Solute Transport in the Saturated Zone . . . 677
36.2.1 Governing Equations . . . 677
36.2.2 Solution Scheme . . . 681
36.2.3 Initial Conditions . . . 685
36.2.4 Source/Sinks, Boundary Conditions and other Exchanges . . . 685

36.2.5 Additional options . . . 686
36.2.6 Transport in Fractured Media . . . 687
36.3 Solute Transport in the Unsaturated Zone . . . 688
36.3.1 Governing Equations . . . 689
36.3.2 Solute transport in macropores . . . 689
36.3.3 Solution Scheme . . . 690
36.3.4 Initial Conditions . . . 692
36.3.5 Source/Sinks, Boundary Conditions and other Exchanges . . . 692
36.4 Solute Transport in Overland Flow . . . 693
36.4.1 Governing Equations . . . 694
36.4.2 Solution scheme . . . 694
36.4.3 Initial Conditions . . . 696
36.4.4 Source/Sinks, Boundary Conditions and other Exchanges . . . 696
36.5 Solute Transport in MIKE 1D . . . 697

37 Reactive Transport - Reference . . . 699
37.1 Sorption . . . 699
37.1.1 Equilibrium Sorption Isotherms . . . 700
37.1.2 Kinetic Sorption Isotherms . . . 702
37.1.3 Sorption in Dual Porosity Systems . . . 703
37.2 Decay . . . 704
37.3 Plant Uptake . . . 706
37.4 Process verification . . . 707
37.4.1 Other Processes - Simulation Examples . . . 713

38 Working with Solute Transport - User Guide . . . 715
38.1 Flow Storing Requirements . . . 715
38.2 Storing of Results . . . 716
38.3 Simulation and Time Step Control . . . 716
38.3.1 Calibrating and verifying the model . . . 717
38.4 Executing MIKE SHE WQ . . . 718
38.5 Output . . . 718
38.6 Coupling Water Quality in MIKE SHE and Rivers . . . 722

39 Working with MIKE ECO Lab in MIKE SHE - User Guide . . . 725
39.1 MIKE ECO Lab Templates . . . 726
39.1.1 Developing a template . . . 726
39.1.2 MIKE ECO Lab templates in MIKE SHE . . . 731
39.1.3 State Variables in MIKE SHE . . . 734
39.1.4 Specifying Constants and Forcings in MIKE SHE . . . 735
39.1.5 Running MIKE ECO Lab with MIKE SHE . . . 736

40 Particle Tracking-Reference . . . 737
40.1 Governing equations . . . 737

41 Particle Tracking (PT) . . . 741
41.1 Requirements in MIKE SHE WM . . . 741
41.1.1 Flow Storing Requirements . . . 741
41.1.2 Specification of Well Fields . . . 742

41.2 Output from the PT simulations . . . 742
41.3 Extraction of particle registrations . . . 744
41.3.1 Running from a batch file . . . 745
41.3.2 Limitations with the PT registration method . . . 745
41.4 Extraction of Particle Path Lines . . . 746

Additional Options . . . 747

42 Extra Parameters . . . 749
42.1 Climate . . . 750
42.1.1 Temperature coupling between Rivers and MIKE SHE . . . 750
42.1.2 Negative Precipitation . . . 753
42.1.3 Precipitation multiplier . . . 754
42.2 Rivers . . . 754
42.2.1 Order of creating river links . . . 754
42.2.2 Irrigation River Source Factors . . . 755
42.2.3 Validation of HD Res file . . . 756
42.3 Overland Flow . . . 756
42.3.1 Time-varying Overland Flow Boundary Conditions . . . 756
42.3.2 Simplified Overland Flow options . . . 758
42.3.3 Explicit Overland Flow Output . . . 759
42.3.4 Alternative low gradient damping function . . . 759
42.3.5 OL Drainage Options . . . 760
42.3.6 OL Drainage to Specified River H-points . . . 761
42.4 Unsaturated Zone . . . 763
42.4.1 UZ-SZ exchange . . . 763
42.4.2 Transpiration during ponding . . . 763
42.4.3 Threshold depth for infiltration (2-Layer UZ) . . . 764
42.5 Saturated Zone . . . 765
42.5.1 Sheet Pile Module . . . 765
42.5.2 SZ Drainage to Specified River H-points . . . 770
42.5.3 SZ Drainage Downstream Water Level Check . . . 773
42.5.4 Time varying hydraulic conductivity . . . 773
42.5.5 2-norm reduction-criteria in the inner iteration loop . . . 774
42.6 Water Quality . . . 775
42.6.1 Disable SZ solute flux to dummy UZ . . . 775
42.6.2 SZ boundary dispersion . . . 775
42.7 Miscellaneous . . . 776
42.7.1 Maximum number of threads . . . 776
42.7.2 Including OpenMI . . . 776
42.7.3 Plot control for Detailed Time Series Output . . . 777
42.7.4 Extra Pre-Processing output . . . 777
42.7.5 Mean-Step-Accumulated time varying parameters . . . 777

References . . . 779

Appendix A 785

A.1 MIKE SHE OUTPUT ITEMS 787

Index 803

GETTING STARTED

1 Introduction

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

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

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

Since the mid-1980's, MIKE SHE has been further developed and extended by DHI Water & Environment. Today, MIKE SHE is an advanced, flexible framework for hydrologic modelling. It includes a full suite of pre- and post-processing tools, plus a flexible mix of advanced and simple solution techniques for each of the hydrologic processes. MIKE SHE covers the major processes in the hydrologic cycle and includes process models for evapotranspiration, overland flow, unsaturated flow, groundwater flow, and channel flow and their interactions. Each of these processes can be represented at different levels of spatial distribution and complexity, according to the goals of the modelling study, the availability of field data and the modeller’s choices, (Butts et al. 2004). The MIKE SHE user interface allows the user to intuitively build the model description based on the user's conceptual model of the watershed. The model data is specified in a variety of formats independent of the model domain and grid, including native GIS formats. At run time, the spatial data is mapped onto the numerical grid, which makes it easy to change the spatial discretisation.

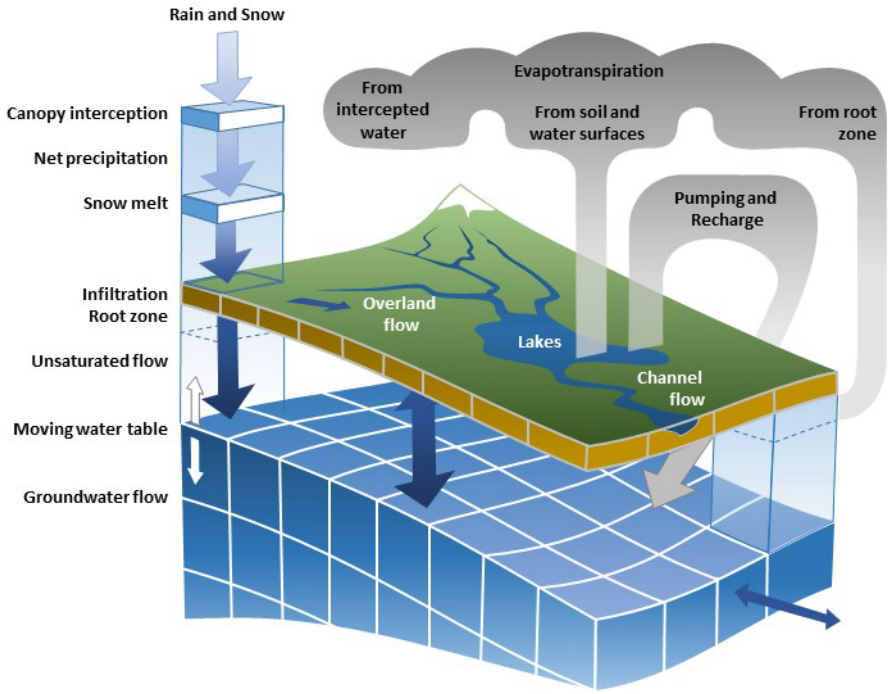


Figure 1.1 Hydrologic processes simulated by MIKE SHE

MIKE SHE uses MIKE 1D to simulate channel flow. MIKE 1D includes comprehensive facilities for modelling complex channel networks, lakes and reservoirs, and river structures, such as gates, sluices, and weirs. In many highly managed river systems, accurate representation of the river structures and their operation rules is essential. In a similar manner, MIKE SHE can also be linked to MIKE 1D sewer models, which can be used to simulate the interaction between urban storm water and sanitary sewer networks and groundwater. MIKE SHE is applicable at spatial scales ranging from a single soil profile, for evaluating crop water requirements, to large regions including several river catchments, such as the 80,000 km^2 Senegal Basin (e.g. Andersen et al., 2001). MIKE SHE has proven valuable in hundreds of research and consultancy projects covering a wide range of climatological and hydrological regimes, many of which are referenced in Graham and Butts (2006).

The need for fully integrated surface and groundwater models, like MIKE SHE, has been highlighted in many studies (e.g. Camp Dresser & McKee Inc., 2001; Kaiser-Hill, 2001; West Consultants Inc. et al., 2001; Kimbley-Horn & Assoc. Inc. et al., 2002; Middlemis, 2004, which can all be downloaded from the MIKE SHE web site). These studies compare and contrast available integrated groundwater/surface water codes. They also show that few codes exist that have been designed and developed to fully integrate surface water and groundwater. Further, few of these have been applied outside of the academic community (Kaiser-Hill, 2001).

Applications around the world

MIKE SHE has been used in a broad range of applications. It is being used operationally in many countries around the world by organizations ranging from universities and research centres to consulting engineers companies (Refsgaard & Storm, 1995). MIKE SHE has been used for the analysis, planning and management of a wide range of water resources and environmental and ecological problems related to surface water and groundwater, such as:

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

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

User interface

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

1. Develop a GUI that promotes a logical and intuitive workflow, which is why it includes
   - A dynamic navigation tree that depends on simple and logical choices
   - A conceptual model approach that is translated at run-time into the mathematical model
   - Object oriented "thinking" (geo-objects with attached properties)
   - Full, context-sensitive, on-line help
   - Customized input/output units to support local needs
2. Strengthen the calibration and result analysis processes, which is why it includes

- Default HTML outputs (calibration hydrographs, goodness of fit, water balances, etc.)
- User-defined HTML outputs
- A Result Viewer that integrates 1D, 2D and 3D data for viewing and animation
- Water balance, auto-calibration and parameter estimation tools.

3. Develop a flexible, unstructured GUI suitable for different modelling approaches, which is why it includes
   - Flexible data format (gridded data, .shp files, etc.) that is easy to update for new data formats
   - Flexible time series module for manipulating time-varying data
   - Flexible engine structure that can be easily updated with new numerical engines

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

1.1 Process models

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

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

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

Therefore, it is often practical to use simplified process descriptions. Similarly, in most watershed problems one or two hydrologic processes dominate the watershed behaviour. For example, flood forecasting is dominated by river flows and surface runoff, while wetland restoration depends mostly on satu-

rated groundwater flow and overland flow. Thus, a complete, physics-based flow description for all processes in one model is rarely necessary. A sensible way forward is to use physics-based flow descriptions for only the processes that are important, and simpler, faster, less data demanding methods for the less important processes. The downside is that the parameters in the simpler methods are usually no longer physics meaningful, but must be calibrated-based on experience.

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

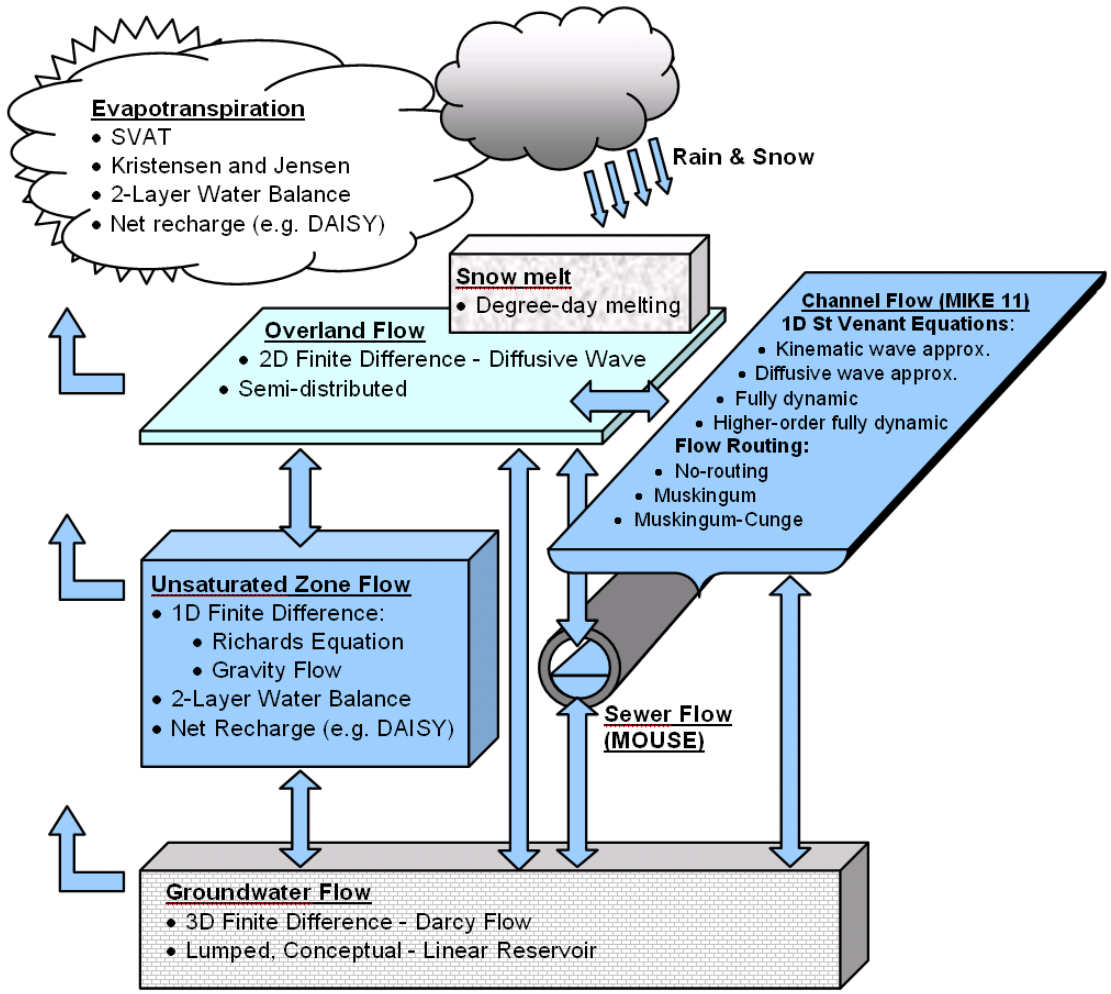


Figure 1.2 Schematic view of the process in MIKE SHE, including the available numeric engines for each process. The arrows show the available exchange pathways for water between the process models. Note: the SVAT evapotranspiration model is not yet available in the commercial version of MIKE SHE

1.2 Requirements

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

1.2.1 Input requirements

The flexibility of MIKE SHE means that there is no predefined list of required input data. The required data depends on the hydrologic process included and the process model selected, which, in turn, depend on what problem you

are trying to solve with MIKE SHE. However, the following basic model parameters are required for nearly every MIKE SHE model:

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

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

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

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

- Species to be simulated, and
- Source locations

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

Table 1.1 Principle parameters for MIKE SHE

	Principle calibration parameters	**Other parameters**
Overland flow (finite difference)	Surface roughness	Detention storage
Overland flow (sub-catchment-based)	Surface roughness	Detention storage Slope parameters
River flow	River bed roughness River bed leakage coefficient	

Table 1.1 Principle parameters for MIKE SHE

	Principle calibration parameters	Other parameters
Unsaturated flow (finite difference)	Saturated hydraulic conductivity	Soil water contents at saturation, field capacity, and wilting point Soil function parameters
Unsaturated flow (2-layer method)	Saturated hydraulic conductivity	Soil water contents at saturation, field capacity, and wilting point Capillary thickness
Actual Evapotranspiration	Leaf Area Index Root depth	Canopy Interception FAO Crop coefficient Kristensen and Jensen ET parameters
Groundwater flow (finite difference)	Hydraulic conductivity Specific yield Specific storage	Drain level Drain time constants
Groundwater flow (linear reservoir)	Reservoir time constants Reservoir volumes (specific yield, depths)	Interbasin transfers (dead zone storage)
Water quality	Porosity Soil bulk density Dispersivities Sorption and degradation rate constants	Source strength

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

1.2.2 Model limits

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

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

The physical limits are generally related to memory size. If you model requires more memory than is physically installed on the computer, then the computer will start to swap data to the hard disk. This will vastly slow down your simulation.

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

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

1.2.3 MIKE SHE demo model limits

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

The current demo restrictions are as follows:

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

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

1.2.4 Hardware requirements

The hardware requirements for MIKE SHE depend on the model that you are trying to simulate. As a rule of thumb, any good quality, new computer should be sufficient for an average MIKE SHE model. Thus, a typical machine for an average MIKE SHE model will have at least a 2GHz CPU, 8-16GB of RAM, and 100-500 GB of free disk space.

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



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

64-bit CPU

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

Multi-core/processor computers

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

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

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

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

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

RAM

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

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

CPU Speed

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

1.3 Getting Help

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

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

1.4 Service and Maintenance

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

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

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

1.4.1 Service packs

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

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

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

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

2 Building a MIKE SHE Model

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

1. The MIKE SHE User Interface
2. Background maps
3. Initial model setup
4. Simulation parameters
5. Model Domain and Grid
6. Topography
7. Climate
8. Rivers, Lakes and Sewers
9. Overland Flow
10. Unsaturated Flow
11. Saturated Groundwater Flow
12. Storing of results
13. Preprocessing your model
14. Running Your Model

2.1 MIKE Zero

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

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

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

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

2.1.1 MIKE Zero editors

- The MIKE Zero also includes general tools for data editing, analysis and manipulation. Some of these have their own file types, or documents. The MIKE Zero documents include (with the tools commonly used for MIKE SHE in bold):
- **The Time Series Editor (.dfs0)**- for time series data
- The Profile Series Editor (.dfs1)- for time varying 1D data (profiles are not used in MIKE SHE)
- **The Grid Editor (.dfs2 and .dfs3)** - for time varying 2D and 3D data
- Data Manager - for finite element data
- **The Plot Composer (.plc)** - for creating standard report plots
- **Result Viewer (.rev)** - for results presentation
- Bathymetry (.batsf) - for sea bed elevations
- Animator (.mza) - for 3D visualization of 2D surface water and waves
- **MIKE ECO Lab (.ecolab)** - for detailed water quality processes
- **AUTOCAL (.auc)** - for autocalibration, sensitivity analysis and scenario management
- EVA Editor (.eva) - for extreme value analysis of surface water flows
- Mesh Generator (.mdf) - for creating meshes for the finite element versions of MIKE 21 and MIKE 3
- Data Extraction FM (.dxfm) - for extracting data from finite element results files
- **MIKE Zero Toolbox (.mzt)** - various tools for data manipulation

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

In addition to the MIKE Zero document-based tools, there are a number of other important MIKE Zero utilities, including:

- **MIKE View** - a results evaluation utility for MIKE 1D river and sewer flow results.

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

2.2 The MIKE SHE User Interface

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

1. Set up the model,
2. Run the model, and
3. Assess the results.

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

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

The following sections provide a quick overview of the main hydrologic processes in MIKE SHE. For more detailed information on the individual parameters, see Setup Data Tab.

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

- Working with Evapotranspiration - User Guide *(p. 457)*
- Working with Freezing and Melting - User Guide *(p. 467)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Working with Rivers and Streams - User Guide *(p. 473)*
- Working with Unsaturated Flow - User Guide *(p. 607)*
- Working with Groundwater - User Guide *(p. 651)*

2.2.1 The Setup Editor

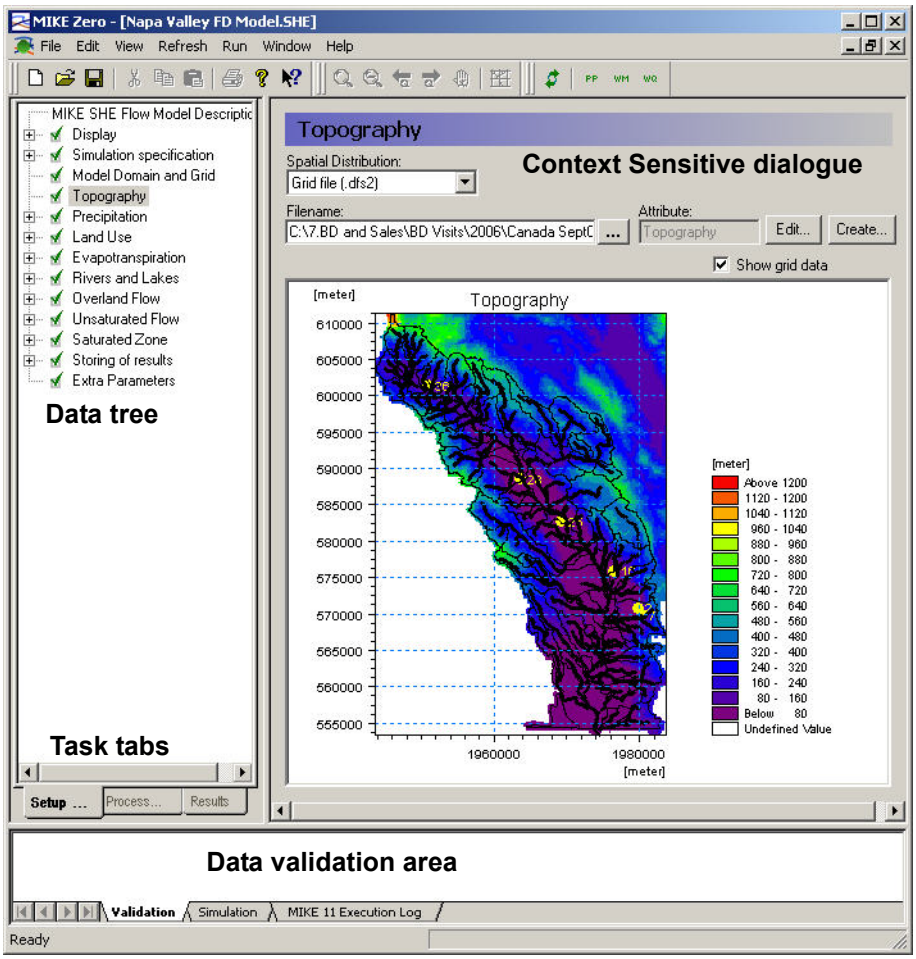


Figure 2.1 Graphical overview of the topography in the MIKE SHE GUI, without the Project Explorer.

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

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

The context sensitive dialogue on the right allows you to input the required data associated with your current location in the data tree. The dialogues vary with the type of data, which can be any combination of static and dynamic data, as well as spatial and non-spatial data. In the case of spatial and time varying data, the actual data is not input to the GUI. Rather, a file name must

be specified and the link to the file is stored in the GUI. Furthermore, the distribution of the data in time and space need not correspond between the various entries. For example, rainfall data may be entered as hourly values and pumping rates as weekly values, while the model may be run with daily time steps.

In addition to the data tree the main MIKE SHE menu also provides you with an overview of some of the standard parameters specific to your model. The parameters displayed depend on the options selected in the Simulation specification in the data tree. The table not only provides an overview of the key parameters but also provides a quick way of modifying the input parameters without having to go through the data tree. Overviews of module specific parameters are also available in the main menus of the Land Use, Overland Flow and the Saturated Zone modules. The model parameters can either be displayed in the same order as in the data tree (the default option) or by parameter. The Sort by Parameter option groups layered items by parameter, for example the horizontal conductivity or lower levels.

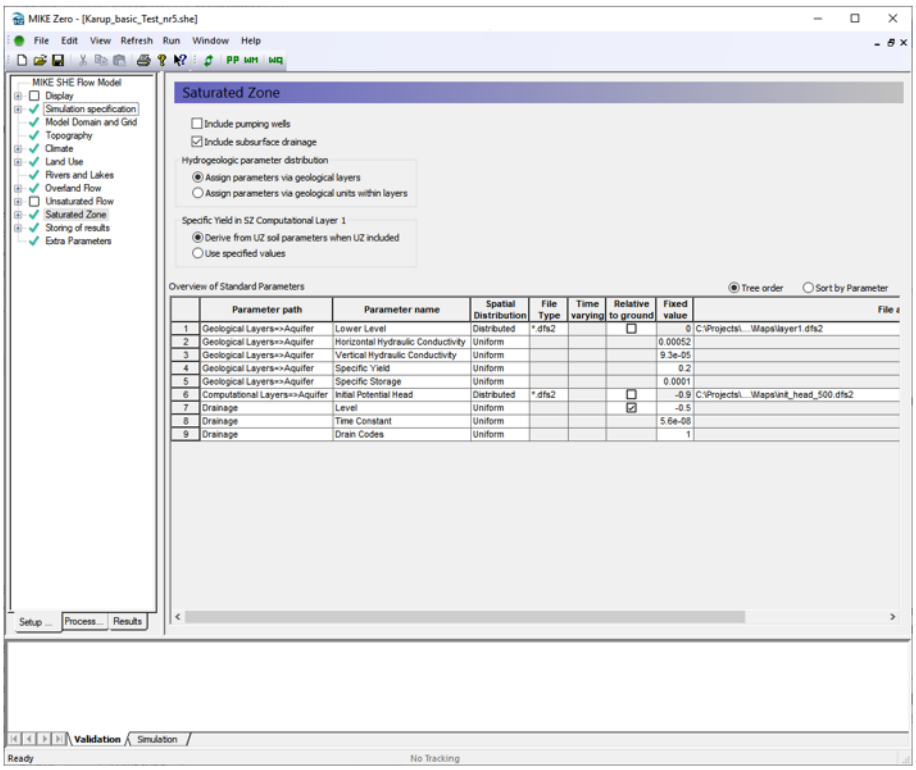


Figure 2.2 Example of Overview of Standard Parameters in the Saturated Zone dialogue in the MIKE SHE GUI.

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

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

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

2.2.2 The Setup Data Tree

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

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

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

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

2.2.3 Background maps

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

The Display item is located at the top of the data tree to make it easy to add and edit your background maps. In the Display item, you can add any number

of images to your model setup, in a variety of formats. The images are carried over to the various editors, so you can keep a consistent display between the set up editor and, for example, the Grid Editor and the Results Viewer.

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

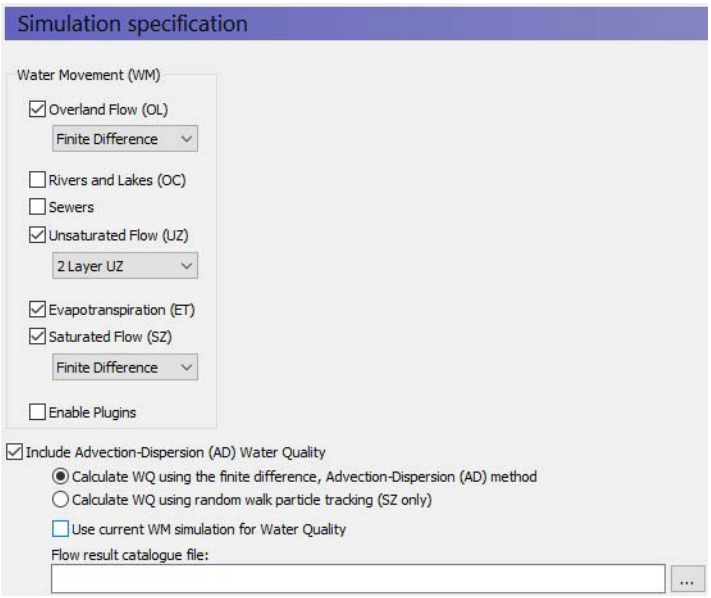


Note: The display of the river network is not carried over to the Results Viewer, but can be added to the view in the Results Viewer by adding a MIKE 1D results file.

2.2.4 Initial model setup

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

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



Water Quality

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

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

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

- Overland Flow - Technical Reference *(p. 515)*
- Channel Flow - Technical Reference *(p. 471)*
- Evapotranspiration - Technical Reference *(p. 437)*
- Unsaturated Zone - Technical Reference *(p. 575)*
- Saturated Flow - Technical Reference *(p. 623)*
- Particle Tracking-Reference *(p. 737)*
- Advection Dispersion - Reference *(p. 673)*

2.2.5 Simulation parameters

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

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

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

Solver parameters

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

- OL Computational Control Parameters *(p. 195)*
- UZ Computational Control Parameters *(p. 200)*
- SZ Computational Control Parameters *(p. 201)*

Time step control

Likewise, the time step control is important, but the default values are usually reasonable to get your model up and running. Then, you should go back to the Time Step Control *(p. 190)* dialogue to optimize your simulation time stepping. For more information on time step control, you can see the Controlling the Time Steps *(p. 83)* section.



Note: Although the different hydrologic processes can run on different time steps, the processes exchange water explicitly. There are restrictions on the relationship between the time steps in the processes. In particular, the longer time steps must be even multiples of the shorter time steps. In other words, a 24 hour groundwater time step can included four 6-hour unsaturated flow time steps, which can each include three 2-hour overland flow time steps. See Time Step Control *(p. 190)* for more information.

2.2.6 Hot starting from a previous simulation

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

To start a model from a previous model run, you must first save the hot start data, in the Storing of Results *(p. 341)* dialogue. In this dialogue, you specify the storing interval for hot start data. Then in the Simulation Period *(p. 188)* dialogue, you can specify the hot start file and then select from the available stored hot start times.

Hot start limitations

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

- MKE 11 and MIKE 1D require an independent hot-start.
- The Water Quality simulations cannot be started from a hot start file.
- There is no append function for the hot start results, so your simulation will generate an independent set of results.
- The pre-processed data does not reflect the hot-start information. The pre-processed data is based on the specified input data, not the results file from which the simulation will be started. This primarily affects initial conditions.

2.3 Model Domain and Grid

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

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

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

Using a dfs2 file

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

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

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

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

The Model Domain and Grid does not have to have the same dimensions (size and spacing) as other specified dfs2 files (e.g. Topography). However, if the other dfs2 input files are coincident, that is if the rows and columns align with one another, then an average of the cell values is used. If the dfs2 files are not coincident, then the Bilinear Interpolation *(p. 169)* method is used to determine the cell value.



Note: The dfs2 files for **integer grid codes must be coincident with the model grid.** For more information on this see Integer Grid Codes *(p. 167)*.

Using an polygon shape (shp) file

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

An important advantage of using a polygon for the model domain, is that the number of rows and columns can be easily adjusted. See Using MIKE SHE with ArcGIS *(p. 159)* for more information.

Creating dfs2 files

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

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

Geographic projections

MIKE SHE supports all available geographic projections. If you have defined the domain using a dfs2 file, then the geographic projection is defined in the dfs2 file. If you use polygon shape file, then you must defined the projection in

the Model Domain and Grid *(p. 228)* dialogue. See Using MIKE SHE with ArcGIS *(p. 159)* for more information.

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

2.4 Topography

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

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

- Lower Level *(p. 355)*,
- Upper Level *(p. 355)*,
- Initial Potential Head *(p. 324)*, and
- Drain Level *(p. 334)*.

Depth parameters, such as ET Surface Depth *(p. 302)*, are also measured from the topography.

File Formats

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

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

The Bilinear Interpolation *(p. 169)* method is useful for interpolating previously gridded DEM data. Whereas, the Triangular Interpolation *(p. 172)* method is useful for contour data digitized from a DEM. Inverse Distance Interpolation *(p. 173)* is usually used for sparse or irregularly spaced data.

ArcGIS Grid Files If you have an ArcGIS Grid DEM, this can be converted to a dfs2 file using the MIKE Zero Toolbox. For more information see the Using MIKE SHE with ArcGIS *(p. 159)* section. Alternatively, a dfs2 plug-in is available for ArcGIS, that allows you to read and write dfs2 files directly in ArcGIS.

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

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

2.5 Climate

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

2.5.1 Precipitation

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

If you use a rate, then the EUM Data Units *(p. 143)* must be “Precipitation” and the time series must be Mean Step Accumulated *(p. 158)*.

If you use an amount, then the EUM Data Units must be “Rainfall” and the time series must be Step Accumulated *(p. 157)*.

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

2.5.2 Evapotranspiration

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

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

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

In addition to the Kristensen and Jensen model, MIKE SHE also includes a simplified ET model that is used in the Two-Layer UZ/ET model. The Two-Layer UZ/ET model divides the unsaturated zone into a root zone, from which ET can occur and a zone below the root zone, where ET does not occur. The Two-Layer UZ/ET module is based on a formulation presented in Yan and Smith (1994). Its main purpose is to provide an estimate of the actual evapo-

transpiration and the amount of water that recharges the saturated zone. It is primarily suited for areas where the water table is shallow, such as in wetland areas.

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

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

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

2.5.3 Freezing and Snow melt

MIKE SHE includes a comprehensive snow melt module based on a modified degree-day method. If the Include snow melt *(p. 233)* checkbox is checked then rain accumulates as snow if the Air Temperature *(p. 240)* is below the Threshold Melting Temperature *(p. 244)* (the temperature at which the snow starts to melt - usually 0 C). If the air temperature is above the threshold, then the snow will melt at the rate specified by the Degree-day Melting or Freezing Coefficient *(p. 245)*.

Dry snow acts like a sponge and does not immediately release melting snow. Thus, melting snow is added to wet snow storage. When the amount of wet snow exceeds the Maximum Wet Snow Fraction in Snow Storage *(p. 246)*, the excess is added to ponded water, which is then free to infiltrate or runoff.

More detailed information on the snow melt process can be found in the online help for the individual dialogues and in the Snow Melt - Technical Reference *(p. 461)* section.

Air Temperature

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

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

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

For more information on the snow melt parameters, see the specific snow melt dialogue information in the Climate *(p. 233)* section of the on-line help and User Interface manual

Frozen Ground

Frozen ground is not directly simulated in MIKE SHE, but can be approximated by using a time varying infiltration rate, where you manually reduce the infiltration rate during the winter.

2.6 Land Use

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

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

Topography - The topography is a physical property of the land surface that defines the hydraulics of both the overland flow and the unsaturated flow. See Topography *(p. 46)*. Related to topography is the definition of Sub-catchments *(p. 230)*, which is needed when you are using the Linear Reservoir method for groundwater or the simple, catchment based overland flow method.

Flood zones - In MIKE SHE, flood zones can be defined relative to the River model branches using Flood codes. For details on how use Flood codes see the chapter on Surface Water *(p. 97)*.

Hydraulic properties - The properties related directly to overland sheet flow are found under Overland Flow *(p. 52)*. This includes the Manning number *(p. 273)* or surface roughness and the Detention Storage *(p. 274)*, both of which are influenced or even defined by the vegetation.

Hydraulic flow - Areas of the land surface can be hydraulically divided by man-made structures, such as road ways and embankments, which can be defined by Separated Flow Areas *(p. 279)*.

Infiltration properties - The infiltration rate is a property of the soil type, which may be modified by the land use. Related to the gross infiltration rate is the presence or absence of macropores and other soil features leading to rapid infiltration. Both of these properties are found in the Unsaturated Flow *(p. 57)* section. However, land surface sealing and compaction can be defined as a reduced contact between ponded water and the subsurface. This is defined in the Overland flow section as a Surface-Subsurface Leakage Coefficient *(p. 276)*.

Groundwater drainage - As the groundwater table rises, it intersects low lying topographic features, such as ditches, or other man-made drainage features, such as buried farm drains. These features are related to land use, but are specified as Groundwater drainage *(p. 65)*

Overland drainage and Paving - Heavy rainfall will generate ponded areas in the landscape and paving will inhibit infiltration. However, 2D overland flow does not usually travel far. It typically drains into local, small-scale ditches and channels. The overland drainage function is used to route local ponded water and rainfall to urban and natural drainage features.

2.6.1 Vegetation

The vegetation properties are used to calculate the Actual evapotranspiration from the Reference evapotranspiration defined under Climate.

The primary vegetation properties are Leaf Area Index (LAI) and Root Depth (RD). The vegetation parameters can be specified in two different ways. They can either be specified directly as uniform values, time series, sub areas, gridded values or time varying grids, or, they can be defined as a crop rotation in the Vegetation Properties Editor *(p. 407)*.

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

Leaf Area Index

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

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

Root Depth

Root depth is defined as the depth below ground in **millimetres** to which roots extend. The root depth is not necessarily the average root depth. In

some cases it may be the maximum root depth. The root depth defines the depth at which water can be extracted from the unsaturated zone. If the root depth is deeper than the depth of the capillary zone, then the roots will be able to extract water from the saturated zone.

The thickness of the capillary zone is defined by the soils function in the soil properties for the Richards and Gravity flow methods. In the 2Layer UZ method, the thickness of the capillary zone is defined by the ET Surface Depth *(p. 302)*.

If you are using the Richards or Gravity Flow UZ methods, then you will also be able to use the Root Shape factor (AROOT) for each vegetation type. This allows you, for example, to extract more water from the upper UZ cells than the lower cells, which is typical of grasses in semi-arid climate zones.

2.7 Rivers, Lakes and Sewers

In the Rivers, Lakes and Sewers dialogue (below) you can link MIKE SHE to a MIKE 1D model. The MIKE 1D model can include rivers, a collection system (sewers), or both.



The check boxes under the heading "Coupling" are used to inform MIKE SHE about what is included in the MIKE 1D model.

MIKE 1D is the numerical engine used for hydraulic modeling in MIKE+. Therefore, MIKE 1D models are built using MIKE+. Models built in MIKE+ must be exported to the m1dx format to be used in MIKE SHE. This is done using the Export functionality in MIKE+.

If the MIKE 1D model includes a collection system model that will be coupled to MIKE SHE, the .m1dx file must be accompanied by a .couple file exported from the same MIKE+ model. The .couple file must have the same name as the .m1dx file and must be located in the same directory.

Older MIKE 1D river models, developed using the decommissioned products MIKE HYDRO River or MIKE 11 can still be used in MIKE SHE. The .mhydro

and .sim11 extensions are still supported in the Rivers, Lakes and Sewers dialogue.

When linking to a sim11 file, the directory in which the sim11 file is located should include the other setup files, including:

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

In the Rivers, Lakes and Sewers dialogue, there are two Inundation Areas options These options are always available for input, but are only used if you have made accompanying specifications in the MIKE+ Groundwater couplings. These options are:

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

More information about coupling to rivers can be found in Section 26.3 Coupling of MIKE SHE and a MIKE 1D River Model and in the MIKE+ documentation.

Details for specifying linkages between MIKE SHE and a MIKE+ collection system model are provided in section 28 Working with Sewers - User Guide and in the MIKE+ documentation.

2.8 Overland Flow

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

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

The Simplified Overland Flow Routing *(p. 533)* method can be used for regional applications when detailed flow is not required. This method assumes that ponded water in the upland areas of a subcatchment flows into the flood plain areas of the subcatchment, which in turn discharges uniformly into the stream network located in the subcatchment.

The Finite Difference Method *(p. 515)* uses the diffusive wave approximation and should be used when you are interested in calculating local overland flow and runoff. There are two solution methods available.

- Successive Over-Relaxation (SOR) Numerical Solution *(p. 519)*
- Explict Numerical Solution *(p. 520)*

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

The Explicit method is generally used when the river is allowed to spill from the river onto the flood plain. Alternatively, you can use Flood codes *(p. 270)* to inundated flood plain areas based on the water level in the river.

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

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

Detailed information on Overland Flow, the coupling between MIKE 1D and MIKE SHE and the overbank spilling options, and ways to optimize the calculation of overland flow can be found in the chapters:

- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Working with Rivers and Streams - User Guide *(p. 473)*

Manning's M

The Manning M is equivalent to the Stickler roughness coefficient, the use of which is described in Overland Flow - Technical Reference *(p. 515)*.

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

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

Detention Storage

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

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

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

Initial and Boundary Conditions

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

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

If you need to specify time varying overland flow boundary conditions, you can use the Extra Parameter option Time-varying Overland Flow Boundary Conditions *(p. 756)*.

Separated flow areas

The Separated Flow Areas *(p. 279)* are typically used to prevent overland flow from flowing between cells that are separated by topographic features, such as dikes, that cannot be resolved within a the grid cell.

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

2.8.1 Overland Drainage and Paving

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

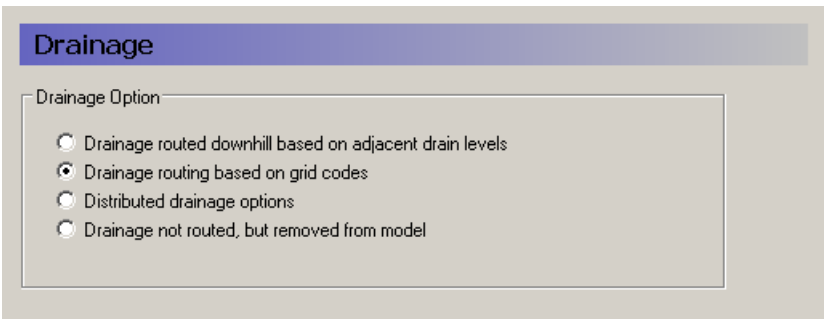
OL Drainage is taken from rainfall and ponded water on the cell. Water that is removed is routed to local surface water bodies, local topographic depressions, or out of the model. The amount of drainage is calculated based on the depth of ponding, rainfall and the drain level using a linear reservoir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

When water is removed from a drain, it is immediately moved to the OL Drain Storage. In other words, the drain module assumes that the time step is longer than the time required for the drainage water to move to the storage. From the OL Drain Storage, the water is released to the recipient at a rate based on the discharge time constant.

Each cell requires a drain level, inflow time constant (which is the same as a leakage factor, optional), and an outflow time constant. Both drain levels and time constants can be spatially distributed. The default drainage level is the topography. A typical time constant may be between 0.01 and 0.00001 1/s.

Drainage reference system

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



- **Drain Levels** The drainage recipient is calculated based on the drain levels in all the down gradient cells. That is, the location of the recipient cell is calculated as if the drain water was flowing downhill (based on the drain levels). This is the most common method of specifying drainage routing and the default setting.
- **Drain Codes** The drainage recipient is specified by the user based on a distribution map of integer code values.

- **Distributed option** With this option there are several different drainage possibilities, including a combination of Codes and Levels. The Distributed option can also be used to define a specific river H-point or sewer manhole as a recipient.
- **Removed** The fourth option is simply a head dependent boundary that removes the drainage water from the model.

2.8.2 Multi-cell overland flow

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

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

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

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

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

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

During the simulation, the cross-sectional area available for flow between the grid cells is an average of the available flow area in each direction across the cell. This adjusted cross-sectional area is factored into the diffusive wave approximation used in the 2D OL solver. For numerical details see Multi-cell Overland Flow Method *(p. 529)* in the Reference manual.

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

of flooding and the area of flooding in an area is more important than the rate and direction of ingress.

The multi-grid option is described in more detail in the chapter Multi-cell Overland Flow *(p. 108)*.

2.8.3 Overland Flow Performance

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

The chapter Surface Water in MIKE SHE *(p. 99)* contains many more details on simulating overland flow and the coupling to MIKE 1D. In particular the section Overland Flow Performance *(p. 105)* contains detailed information on improving the performance of the overland flow in your model.

2.8.4 MIKE+ 2D Overland

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

The MIKE+ 2D Overland module enables coupling of the 2D MIKE 21FM surface water model for detailed, accurate flow on the flood plain with MIKE 1D for channel flow. MIKE+ allows you to define flood plain structures such as embankments and culverts that can have very significant impacts on flow velocity and direction. The MIKE 21FM engine can also more accurately simulate flood wave propagation on a surface simply because of the higher order numerical method used.

2.9 Unsaturated Flow

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

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

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

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

More detailed information on the setup and calculation of unsaturated flow is found in the Chapters:

- Working with Evapotranspiration - User Guide *(p. 457)*
- Working with Freezing and Melting - User Guide *(p. 467)*
- Working with Unsaturated Flow - User Guide *(p. 607)*

The Technical Reference manual includes detailed information on the calculation methods - Unsaturated Zone - Technical Reference *(p. 575)*.

2.9.1 Soil profiles

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

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

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

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

Soil properties database

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

Vertical Grid Discretisation

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

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

2.9.2 Initial conditions

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

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

2.9.3 Macropore flow

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

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

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

Simple bypass flow is commonly used to provide some rapid recharge to the groundwater table. In many applications, if all the rainfall is infiltrated normally, the actual evapotranspiration is too high and very little infiltration reaches the groundwater table. In reality some infiltration recharges the groundwater system due to macropores and sub-grid variability of the soil profile. In other words, there is usually sub-areas in a grid cell with much

higher infiltration rates or where the unsaturated zone thickness is much less than that defined by the average topography in the cell.

Simple bypass flow is described in the Reference section under Simplified Macropore Flow (bypass flow) *(p. 590)*.

Full Macropore Flow - Macropores are defined as a secondary, additional continuous pore domain in the unsaturated zone. Full macropore flow is generally reserved for very detailed unsaturated root-zone models, especially in water quality models where solute transformations are occurring in the macropores. Full bypass flow is described in the Reference section under Full Macropore Flow *(p. 591)*.

2.9.4 Green and Ampt infiltration

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

For more information on the Green and Ampt method, see the section Green and Ampt Infiltration *(p. 596)* in the Reference Guide.

2.9.5 UZ column classification

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

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

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

- the water table is very dynamic and spatially variable because the classification is not dynamic,
- if the 2 layer UZ method is used because the method is fast and the benefit would be limited,

- if irrigation is used in the model because irrigation zones are not a classification parameter, and
- if flooding and flood codes are used, since the depth of ponded water is not a classification parameter

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

- **Automatic classification** With automatic option, the UZ columns are divided up based on the internal classification rules. The depth to the water table, Groundwater Depths used for UZ Classification *(p. 297)*, is the lower UZ boundary condition.
- **Specified classification** With the specified option, you must supply a list of grid codes, Specified classification *(p. 299)*, that defines the computational column and the columns to which the results will be applied.
- **Calculated in all Grid points (default)** In many models the classification system is not feasible or recommended. In this case, the UZ flow will be calculated in all soil columns.
- **Partial Automatic** Finally a combination of the Automatic classification and the Specified classification is available, where an. Integer Grid Code file must be provide (see Partial automatic classification *(p. 298)*) to define the different areas.

2.9.6 Coupling between unsaturated and saturated zone

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

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

Error in the mass balance originates from two sources:

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

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

In the second case above, the MIKE SHE forces the specific yield of the top SZ layer to be equal to the “specific yield” of the UZ zone as defined by the difference between the specified moisture contents at saturation, θ_s, and field capacity, θ_{fc}.This correction is calculated from the UZ values in the UZ cell in which the initial SZ water table is located. For more information see Specific Yield of the upper SZ numerical layer *(p. 222)*.

UZ - SZ limitations

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

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

For more detailed information on the UZ-SZ coupling see Unsaturated Zone - Technical Reference *(p. 575)*. The chapter, Working with Unsaturated Flow - User Guide *(p. 607)*, also contains more detailed information on the setup and evaluation of the unsaturated model.

2.10 Saturated Groundwater Flow

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

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

Finite Difference Method

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

Linear Reservoir Method

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

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

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

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

2.10.1 Conceptual geologic model for the finite difference approach

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

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

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

You also have the option to set up your conceptual model

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

Lenses

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

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

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

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

2.10.2 Specific yield of upper SZ layer

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

For more information on the SZ-UZ specific yield see Specific Yield of the upper SZ numerical layer *(p. 222)*.

2.10.3 Numerical layers

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

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

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

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

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

2.10.4 Groundwater drainage

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

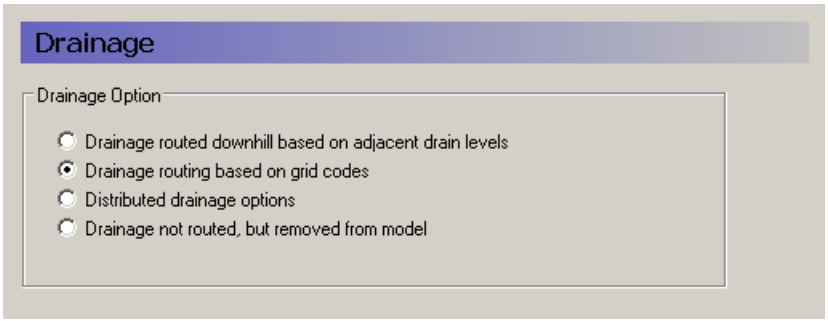
Saturated zone drainage is removed from the layer of the SZ layer containing the drain level. Water that is removed from the saturated zone by drains is routed to local surface water bodies, local topographic depressions, or out of the model. The amount of drainage is calculated based on the groundwater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

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

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

Drainage reference system

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



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

2.10.5 Groundwater wells

Groundwater wells can be included in your SZ simulation. The groundwater well locations, filter depth, pumping rates etc. are stored in a .wel file that is edited using the Well editor *(p. 401)*.

2.10.6 Linear reservoir groundwater method

In the linear reservoir method, the entire catchment is subdivided into a number of subcatchments and within each subcatchment the saturated zone is represented by a series of interdependent, shallow interflow reservoirs, plus a

number of separate, deep groundwater reservoirs that contribute to stream baseflow.

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

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

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

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

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

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

For more detailed information, see the section Linear Reservoir Method *(p. 638)*.

2.11 Storing of results

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

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

from a grid series output file, but the time resolution will be the same as the specified saving interval.

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

Water balance output

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

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

Water balances are calculated using a separate water balance utility, which is described in detail in the chapter Using the Water Balance Tool *(p. 99)*.

Hot start output

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

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

Using the hot start involves:

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

Water quality output

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

Storing intervals

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

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

2.11.1 Detailed time series output

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

Each item in the Detailed time series is displayed automatically in a plot on the Run tab while the simulation is running.

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

A full list of available output items, as well as more detail on the individual items is found in the section Output Items *(p. 92)*.

Importing ASCII data

See the section Storing of Results *(p. 341)* for how to import a detailed time series setup from a text file.

2.11.2 Detailed River Time Series output

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

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

Importing ASCII data

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

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

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

Branch_name - The name of the River Branch (check your spelling!)

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

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

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

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

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

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

2.11.3 Grid series output

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

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

A full list of available output items, as well as more detail on the individual items is found in the section Output Items *(p. 92)*.

3 Running Your Model

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



PP - The PP button starts the preprocessing. You must first **P**re**P**rocess your model data to create the numerical model from your grid independent data. See Preprocessing your model *(p. 71)*.

WM - The WM button starts the **W**ater **M**ovement simulation. You can only run your water movement simulation after you have preprocessed your data.

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

 - The *Execute*... button runs PP, WM and WQ in sequence. Only the items checked in the *Run* menu will be executed. The *Execute* option can also be launched using the Alt - R - E hot-key sequence.

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 does the same as the *Execute* button.

When you start your model a Launch Settings dialog will appear. If you want to skip this dialog hold the shift key pressed while clicking one of the above buttons.

3.1 Preprocessing your model

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

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

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

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

After you have successfully preprocessed your model, the pre-processed data will be automatically loaded when you expand the data tree. The data tree reflects all the spatial data defined in the model set up tab. In other words, if the overland flow is not included in the Simulation Specification *(p. 186)* dialogue, then the Overland item will not be included in the pre-processed data tree.



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

3.1.1 Viewing the pre-processed data

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

3.1.2 Editing the pre-processed data

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

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

Option 1

1. Right click on the map view and save the data to a new dfs2 file,
2. open the new dfs2 file in the Grid Editor, and
3. make the changes in the new dfs2 file and save the file.

Option 2

1. Use the View button to open the dfs2 or dfs3 pre-processed file in the Grid Editor,
2. make your changes in the file, and
3. save the file with a new name.

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

3.2 Pre-processed data items

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

3.2.1 MIKE 1D coupling

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

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

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

Related Items:

- Surface Water in MIKE SHE *(p. 99)*
- Working with Rivers and Streams - User Guide *(p. 473)*
- Channel Flow - Technical Reference *(p. 471)*

3.2.2 Land Use

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

Shape files

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

3.2.3 Unsaturated Flow

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

Soil profiles

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



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

UZ Classification Codes

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

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

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

Related Items:

- Unsaturated Zone *(p. 288)*
- Soil Profile Definitions *(p. 293)*
- Partial automatic classification *(p. 298)*
- Specified classification *(p. 299)*

3.2.4 Saturated Flow

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

Specific Yield of upper SZ layer

MIKE SHE forces the specific yield of the top SZ layer to be equal to the “specific yield” of the UZ zone as defined by the difference between the specified

moisture contents at saturation, θ_s, and field capacity, θ_{fc}.This correction is calculated from the UZ values in the UZ cell in which the initial SZ water table is located. This is reflected in the pre-processed data.

For more information on the SZ-UZ specific yield see Specific Yield of the upper SZ numerical layer *(p. 222)*.

Saturated Zone Drainage

The rate of saturated zone drainage is controlled by the drain elevation and the drain time constant. However, the destination of the drainage water is controlled by the drain levels and the drain codes, which determine if the water flows to a river, a boundary, or a local depression. The algorithm for determining the drainage source-recipient reference system is described in Groundwater drainage *(p. 65)*.

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

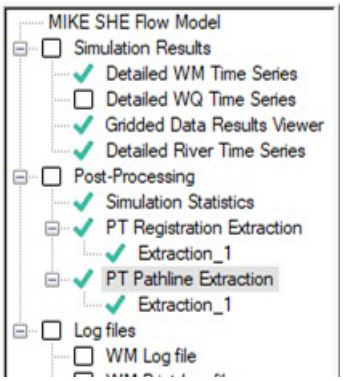
Drainage to local depressions and boundary - This grid displays all the cells that drain to local depressions or to the outer boundaries. All drainage from cells with the same negative value are drained to the cell with the corresponding positive code. If there is no corresponding positive code, then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

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

Related Items:

- Groundwater drainage *(p. 65)*
- Drainage *(p. 331)*
- Drain Level *(p. 334)*
- Drain Time Constant *(p. 335)*
- Drain Codes *(p. 335)*
- Option Distribution *(p. 337)*

3.3 The Results Tab

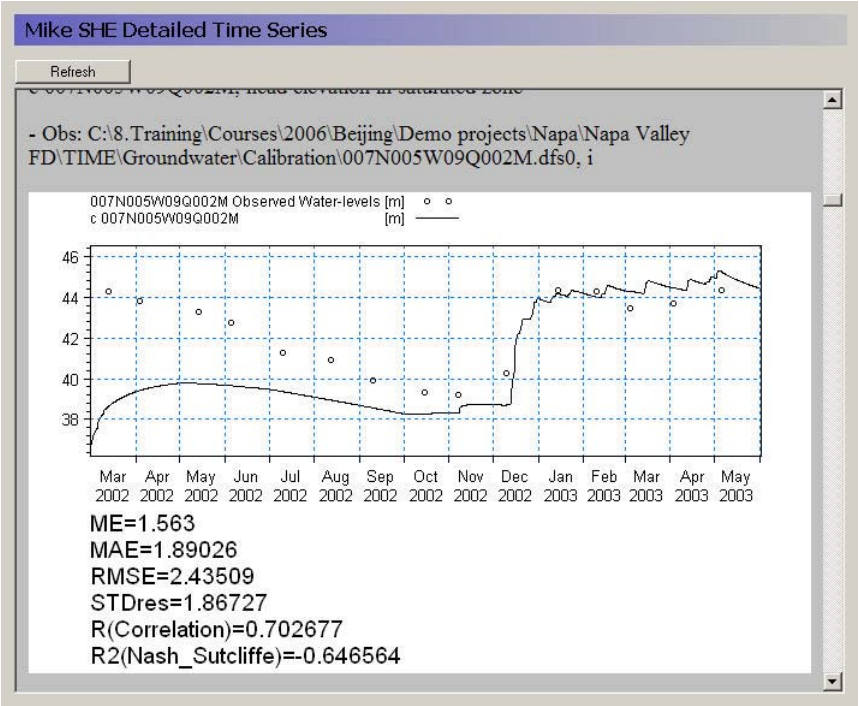


All the simulation results are collected in the Results tab. This includes Detailed time series output for both MIKE SHE and the river model, as well as Grid series output for MIKE SHE.

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

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

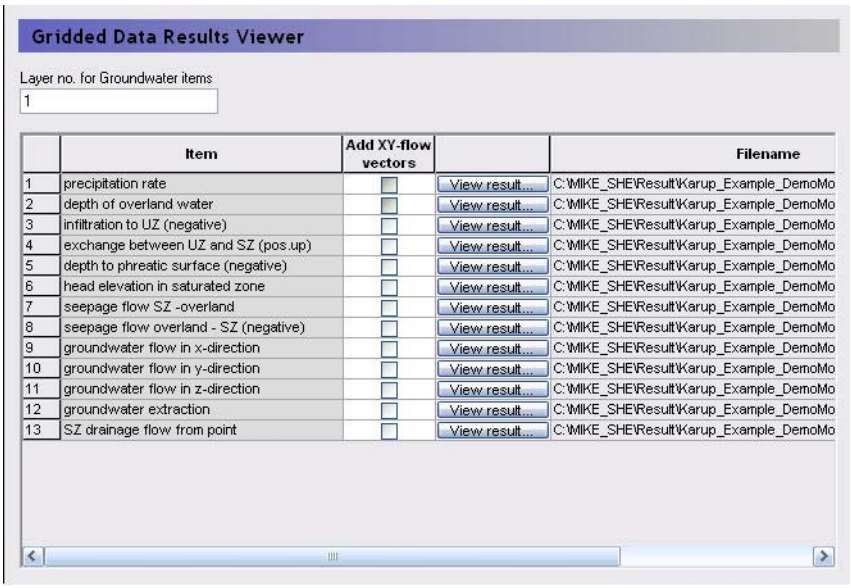
3.3.1 Detailed Time Series Results



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

For information on the statistics see Statistic Calculations *(p. 375)*.

3.3.2 Gridded Results



Gridded data results for MIKE SHE can be viewed by selecting the Gridded Data Results Viewer item on the Results tab. The table is a list of all gridded data saved during a MIKE SHE simulation. The items in this list originate from the list of items selected in the Grid series output *(p. 352)* dialogue from the Setup tab.

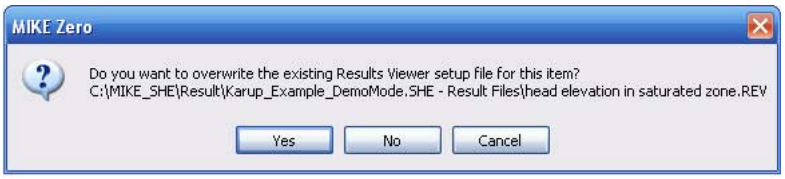
Clicking on the View result button will open the Results Viewer to the current item. All overlays from MIKE SHE (e.g. shape files, images, and grid files) will be transferred as overlays to the result view. However, the River Network is not transferred as an overlay.

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

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

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

The “Overwrite existing file” warning

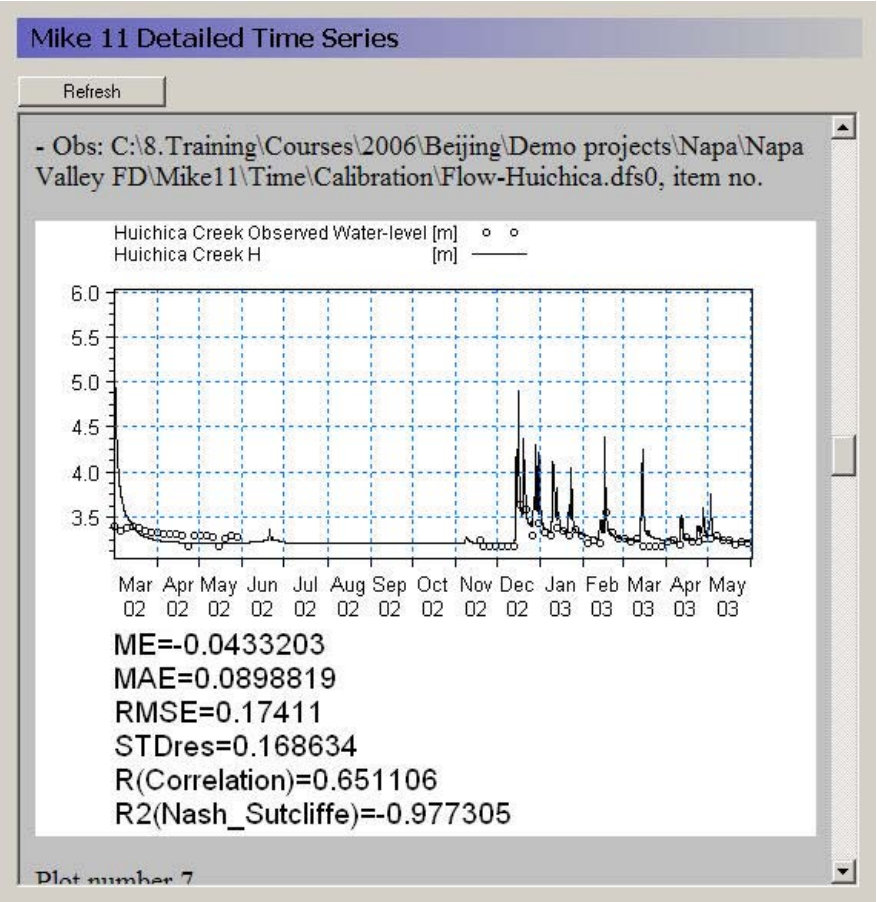


When the Result Viewer opens one of the items in the table, it creates a setup file for the particular view with the extension .rev. The name of the current .rev file is displayed in the title bar of the Results Viewer.

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

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

3.3.3 Detailed River Time Series Results



The Detailed River Time Series page includes an HTML plot of each point selected in the Setup Editor. The HTML plots are updated during the simulation whenever you enter the view. Alternatively, you can select the Refresh button to refresh the plot.

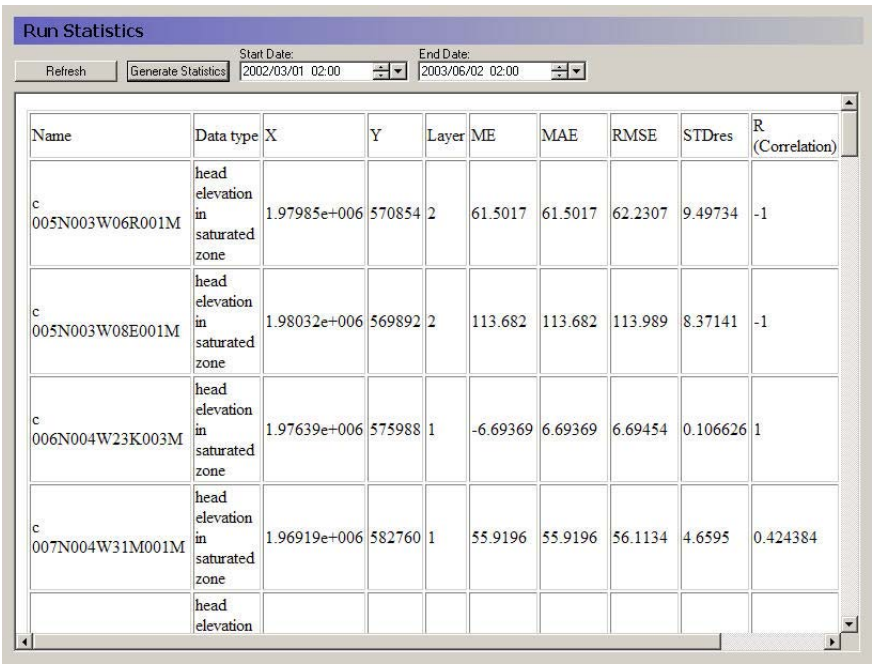
For information on the statistics see Statistic Calculations *(p. 375)*.

3.3.4 Run Statistics

Run statistics can be generated in HTML format for a MIKE SHE simulation. The run statistics table information can be copied and pasted directly into any word processing program, such as Microsoft Word, or spread-sheet, such as Microsoft Excel. The Run Statistics HTML document includes MIKE SHE and river model results for all Detailed Time Series items that have observation data.

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

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



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

.

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

For information on the statistics see Statistic Calculations *(p. 375)*.

Shape file output for run statistics

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

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

3.4 Controlling Your Simulation

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

3.4.1 Model limits

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

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

The physical limits are generally related to memory size. If you model requires more memory than is physically installed on the computer, then the computer will start to swap data to the hard disk. This will vastly slow down your simulation. The section, Hardware requirements *(p. 32)*, outlines some hardware considerations when using MIKE SHE.

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

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

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

3.4.2 Speeding up your simulation

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

- **Do you really need a fine discretisation during calibration?** A coarser grid may allow you to do many more calibration runs. Then when the model is calibrated, you can refine the grid for the final simulations - but remember to check you calibration first.
- **Do you really need the Richards equation for unsaturated flow?** For regional models, the two layer water balance method may be sufficient, which is very fast. The gravity flow method is also, typically 2-5 times faster than the Richards equation method. Again during the calibration it can be a good idea to use one of the simpler methods and the more detailed method for the final simulations. However, switching between methods will likely invalidate your UZ calibration, and require additional calibration adjustments.
- **Is your river model simulation too detailed?** If your river cross-sections are too close together, the river model will run with a very short time step. Regional river models can often be run with the simple routing methods, which are very fast.

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

- Hardware requirements *(p. 32)* contains information on different hardware configurations,
- Controlling the Time Steps *(p. 83)* contains information on how the dynamic time step control works,
- Overland Flow Performance *(p. 105)* contains information on how to improve the efficiency of the overland flow solution, which can be very time consuming if you have permanently ponded water,
- Parallelization of MIKE SHE *(p. 88)* contains information on the using MIKE SHE with multi-core PCs and 64-bit operating systems.

3.4.3 Controlling the Time Steps

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

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

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

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

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

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

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

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

Precipitation-dependent time step control

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

The parameters controlling the time step adjustment are in the Time Step Control *(p. 190)* dialogue. In particular, the following three parameters control the time step during rainfall events:

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

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

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

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

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

Actual time step for the different components

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

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

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

until the maximum allowed time step is reached.

Relationship to Storing Time Steps

The Storing Time Step specified in the Detailed WM time series output *(p. 344)* dialogue, must also match up with maximum time steps. Thus,

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

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

3.5 Using Batch Files

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

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

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

Setup the different models

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

Create the batch file

To create a batch file, you must create a text file with the extension .BAT. Then add the DOS commands in the order that you would like them executed. But, before you can run the MIKE SHE executables, you must add the

MIKE SHE installation directory to your PATH variable. The default installation directory depends on your operating system. For example, for Windows 10 the default directory is:

```
C:\Program Files(x86)\DHI\MIKE Zero\2026\bin\x64
```

The path for the actual installation directory is saved in an environment variable called DHI_MIKE_2026, so the DOS command to add the default path to the PATH variable can be written as:

```
Set PATH=%PATH%;%DHI_MIKE_2026%
```

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

```
MSHE_PreProcessor MyModel.she

MSHE_watermovement MyModel.she
```

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

```
MSHE_Simulation MyModel.she
```

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

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

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

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

will close the MzLaunch utility when the simulation finishes.

Analyse the Results

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

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

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

MSHE_Wbl_Post.exe //apv My_WB_areas.WBL 1
```

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

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

3.6 OpenMI

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

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

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

Linking MIKE SHE with OpenMI

If you want to link MIKE SHE to another program using OpenMI, then you will need to initialize MIKE SHE to produce the required OpenMI linkages. This is done using the Extra Parameter option: Including OpenMI *(p. 776)*.

3.7 Parallelization of MIKE SHE

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

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

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

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

4 MIKE SHE Results

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

4.1 Output Files

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

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

.hot.sheres - This is a reduced version of the .sheres file for hot starting only. It does not reference all simulations outputs, but just the minimum set required for hot starting, so it cannot be used for any other purposes like creating a water balance. This file will only be created if the simulation does store hot start data.

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

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

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

Viewing Output Files

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

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

All three of these are MIKE Zero tools and are described in the MIKE Zero documentation. See the section on The Results Viewer *(p. 97)* for more details on the Results Viewer.

4.1.1 Log files

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

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

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

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

4.2 Multiple Simulations

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

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

4.3 Output Items

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

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

In the absence of an explicit remark, the sign convention for MIKE SHE's output is positive in the positive direction. In other words, all flows in the direction

of increasing X, Y and Z coordinates are positive. Thus, vertical downward flows, such as infiltration are negative.

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

Also important to remember is that the output items related to flow are accumulated over the storing time step. In many cases, these values are required for the Water Balance program described in the section Using the Water Balance Tool *(p. 99)*. The values that are part of the water balance are automatically turned on when the water balance option is selected.

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

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

The available output items for gridded data and time series data are listed in Appendix A.1 MIKE SHE Output Items *(p. 787)*.



Note The **Code** listed in the tables in Appendix A.1 MIKE SHE Output Items *(p. 787)* is the **Data Type Code** that is needed when importing time series items into the Detailed WM time series output *(p. 344)* dialogue.

4.3.1 Overland flow

The overland flow velocities are discussed in more detail in the section, Output: Overland Flow Velocities *(p. 543)*.

Overland flow in the x- and y-direction

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

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

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

H Water Depth, P flux and Q flux

The P and Q fluxes are instantaneous fluxes across the positive cell faces of the cells. These are found in a separate *_flood.dfs2* results file, along with the H Water Depth. This file is the same format as the MIKE 21 output files generated by MIKE+. 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.

4.3.2 Recharge

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

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

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

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

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

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



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

5 The Results Viewer

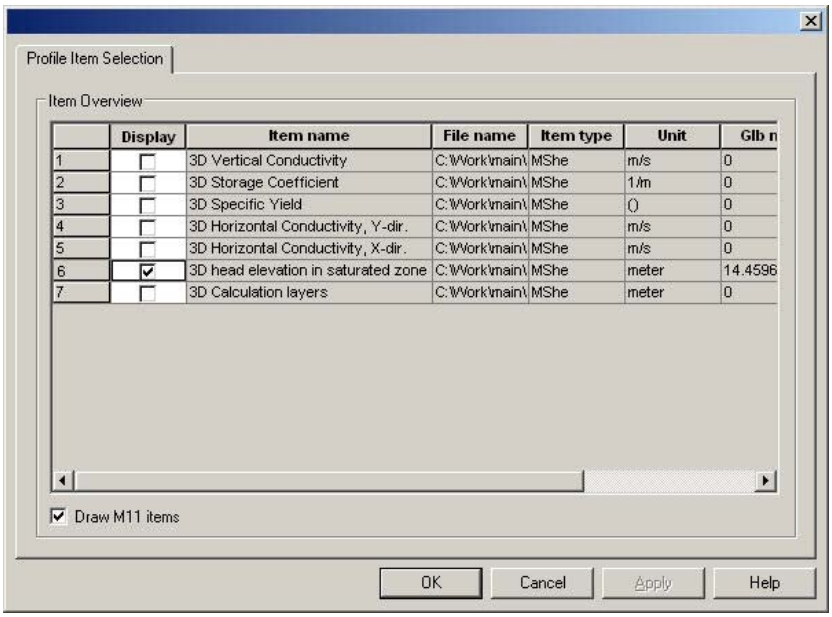
The Results Viewer is a generic MIKE Zero tool for displaying both gridded and unstructured spatial data. For more information on the Results Viewer, see the separate MIKE Zero documentation.

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

5.1 Saturated Zone Cross-section Plots

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

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



.

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

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

5.1.1 Saving and loading profiles

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

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

5.2 UZ Specific Plots

5.2.1 UZ Scatter and Filled Plots

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

This option is no longer recommended and is essentially obsolete.

5.2.2 Transient UZ Column Plot

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

After selecting the UZ Plot extractor tool move the cursor over the column you want to extract the results from and double-click. The UZ Plot shows either water content, the pressure or the saturation for each node in the column (y-axis) for the entire simulation period (x-axis).

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

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

6 Using the Water Balance Tool

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

A water balance can be generated at a variety of spatial and temporal scales and in a number of different formats, including dfs0 time series files, dfs2 grid series files, and ASCII text output suitable for importing to Microsoft Excel. You can also automatically create a picture that visualizes the interrelationships between the various water balance components (see Figure 6.1).

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

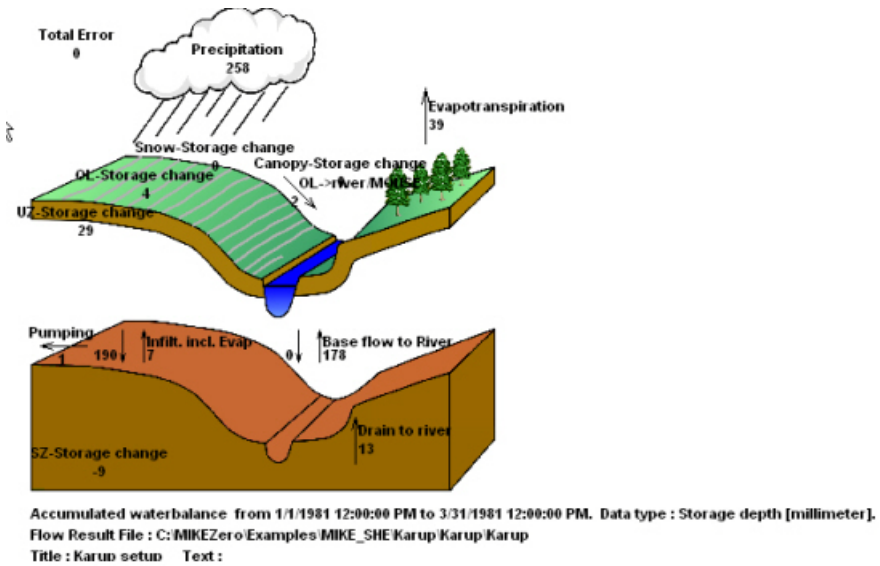
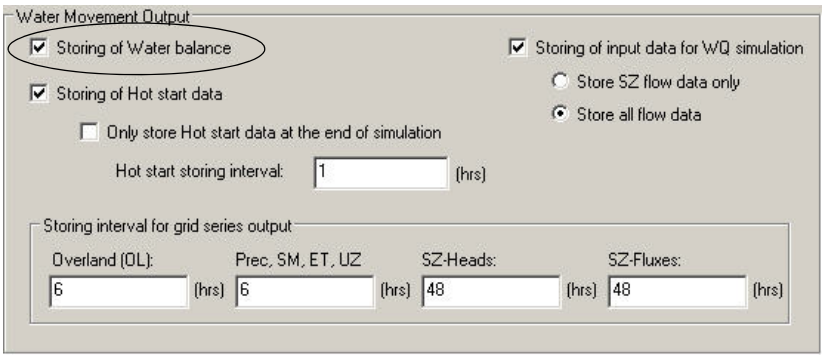


Figure 6.1 Graphical water balance output example

6.1 Creating a Water Balance

Before you can create a water balance for a MIKE SHE WM simulation, you must have saved the water balance data during the simulation. Saving of the water balance data is specified in the Storing of Results *(p. 341)* dialogue. If you have forgotten to save the water balance data, then you will need to re-run your simulation.



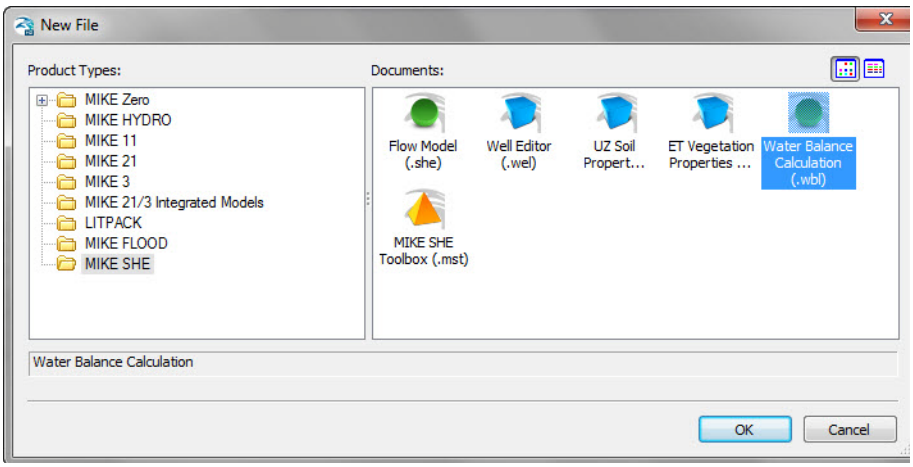
Note: the available time step resolution of the water balance will be the same as the Storing Interval for Grid Series Output. If the storing intervals are different, then it will be the largest Storing Interval.

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

1. Create a new water balance document *(p. 100)*,
2. Extract the water balance data *(p. 101)*
3. Specify your water balance *(p. 102)*, and
4. Calculate and View the Water Balance *(p. 105)*.

6.1.1 Create a new water balance document

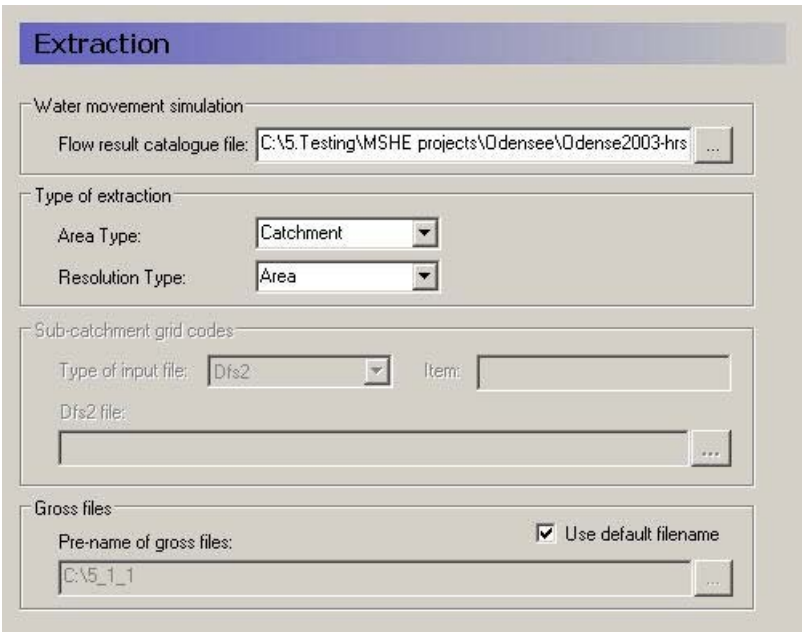
The new water balance document is created by selecting the File/New item in the top menu, or clicking on the New icon in the top menu bar. In the dialogue that appears, select MIKE SHE and Water Balance Calculations in the right hand box, as shown below.



6.1.2 Extract the water balance data

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

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



Flow result catalogue file

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

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



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

Type of Extraction

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

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

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

Sub-catchment grid codes

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

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

Gross files

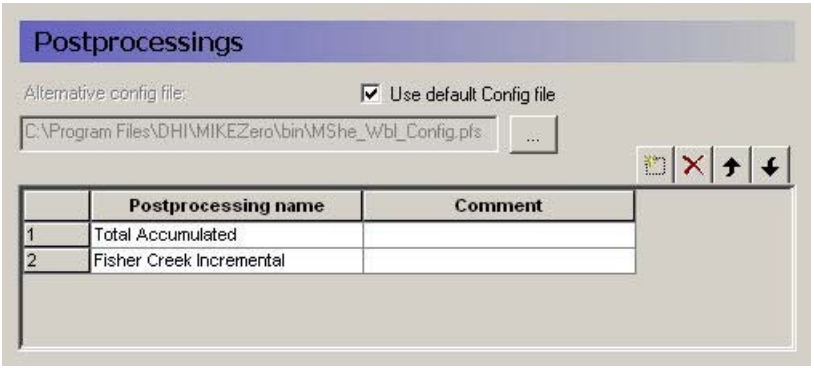
The pre-processor extracts the water balance data from the standard MIKE SHE output files and saves the data in a set of “gross” files. The file names of the gross files is built up from the project name and prefix specified here. The default value is normally fine.

Run the extraction

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

6.1.3 Specify your water balance

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

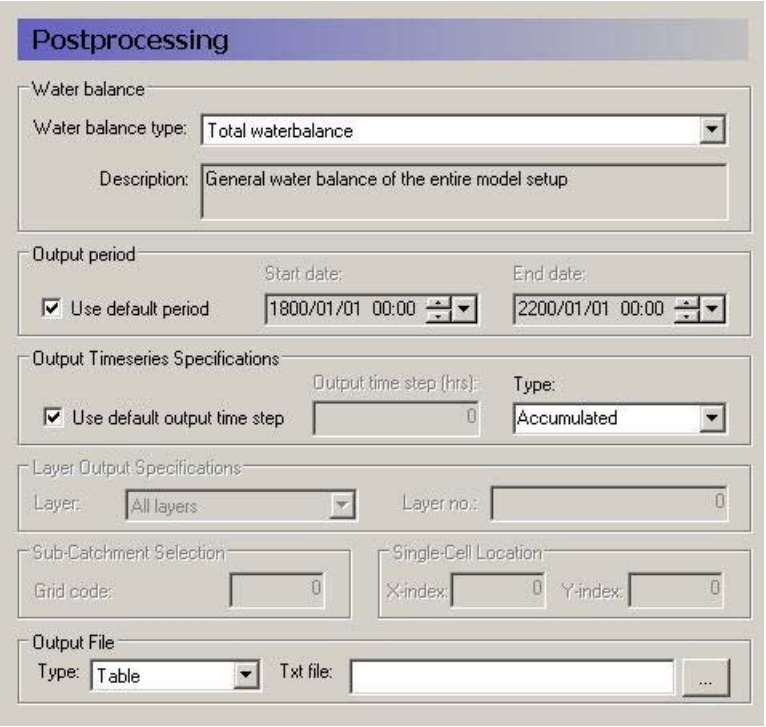


A single Postprocessing item is created by default when the water balance file is created. The default Postprocessing name can be change to a more appropriate name. Postprocessing items that are no longer needed can be deleted using the Delete button.

Use default Config file

Unchecking the Use default Config file checkbox, allows you to specify the location of a custom water balance Config file. Development of custom water balance configuration files is described in detail in Making Custom Water Balances *(p. 136)*.

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



Water Balance

Multiple postprocessings can be run on each water balance extraction. More detail on the types of available water balances data are discussed in the Available Water Balance Items *(p. 108)* section. In brief, the available types include

- The total water balance of the entire model catchment or sub-catchments in an ASCII table, a dfs0 file, a dfs2 map file,
- A graphical cartoon (“Chart” type) of the total water balance (such as Figure 6.1),
- Model errors for each hydrologic component (overland, unsaturated zone, etc.) in an ASCII table, a dfs0 file, or a dfs2 map file (also by layer),
- The snow melt and canopy/interception water balance in an ASCII table, or a dfs0 file,
- An abbreviated or detailed water balance for overland or unsaturated flow in an ASCII table, or a dfs0 file, and
- An abbreviated or detailed water balance by layer for saturated flow in an ASCII table, or a dfs0 file.

Output Period

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

Output Time Series Specification

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

Layer Output Specifications

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

Sub-catchment Selection

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

Single Cell Location

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

Output File

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

Run the Post Processing

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

6.1.4 Calculate and View the Water Balance

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

or Grid Editor will be opened. For the chart output, the graphic will be displayed by the program WblChart.

Units for the water balance

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

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

6.2 Display and auto-run of water balances in the MIKE SHE Results tab

It's possible to add one or more water balances to the Results tab of MIKE SHE, enabling viewing water balance results in the same way as in the Result tab of the Water Balance tool. In addition, if a water balance has been added to the MIKE SHE Results tab, then the water balance is recalculated automatically after each MIKE SHE simulation.

Details about adding water balances to the MIKE SHE Results tab are provided in 13.2.2 Water Balances *(p. 376)*.

6.3 Calculating Water Balances in Batch Mode

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

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

The water balance utility stores all of its information in a .wbl file. The .wbl file is an ASCII file that can be edited with Notepad or other text editor, but the format of the water balance file must be preserved. For more information on editing the .wbl file and creating custom water balances, see Making Custom Water Balances *(p. 136)*.

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

To perform the water balance extraction use:

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

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

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

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

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

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

The number in the Postprocessing command must be consistent with the water balance utility file (i.e., the number cannot be greater than the number of Postprocessing items in the file). Otherwise, the program will terminate with an error. The Postprocessing step cannot be executed before an Extraction step but only one Extraction step needs to be run for a single water balance utility file.

To run the water balance utility in batch mode, the .wbl file must be created prior to executing the water balance and all file names in the .wbl file need to be valid.

If during calibration the same MIKE SHE file name is used for each simulation then the same .wbl file can be used for all calibration runs. If the MIKE SHE simulation to be evaluated is different from the MIKE SHE simulation used to set up the water balance file, you will have to edit the .wbl file.

To run the Extraction and Postprocessing steps in batch mode, the PATH statement needs to include the directory where MIKE SHE was installed. For the 2026 Release, the default directory is

```
C:\Program Files(x86)\DHI\MIKE Zero\2026\bin\x64
```

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

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

```
rem ----------------------------------------

MSHE_Wbl_Ex.exe WaterConservationAreas.WBL

MSHE_Wbl_Post.exe WaterConservationAreas.WBL 1
```

```
MSHE_Wbl_Post.exe WaterConservationAreas.WBL 2

MSHE_Wbl_Post.exe WaterConservationAreas.WBL 3
```

6.4 Available Water Balance Items

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

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

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

- Table 6.1 SM - Precipitation and snowmelt items *(p. 110)*
- Table 6.2 CI - Canopy interception water balance items *(p. 112)*
- Table 6.3 OL - Overland flow items *(p. 115)*
- Table 6.4 UZ - Unsaturated Zone items *(p. 118)*
- Table 6.5 SZ - Saturated Zone - all layers *(p. 122)*
- Table 6.6 SZ - Saturated Zone - specified by layers *(p. 126)*
- Table 6.7 SZ - Saturated Zone - Linear Reservoir all layers *(p. 128)*

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

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

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

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

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

6.4.1 Snow Storage

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

The items listed in Table 6.1 are those found in the "*Snow Melt component - detailed*" water balance output in the water balance configuration file:

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

```
    Group = 'Error(sm.smwblerr)'

EndSect  // WblTypeDefinition
```

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

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

Table 6.1 SM - Precipitation and snowmelt items

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

Table 6.1 SM - Precipitation and snowmelt items

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

6.4.2 Canopy interception storage

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

The items listed in Table 6.2 are those found in the "*Canopy Interception component*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
    Name = 'CI'
```

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

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

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

Table 6.2 CI - Canopy interception water balance items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ci.qPad	Total precipitation reaching the canopy (Precipitation + sprinkler irrigation + snowmelt to OL - precipitation converted to snow)	Inflow - negative	yes
-ci.qPnet	Canopy throughfall to ponded water	Outflow - positive (Note sign change in water balance definition)	yes
ci.qEInt	Evaporation from intercepted storage	Outflow - positive	yes
ci.dIntSto	Change in interception storage	Positive when interception storage increases	yes
ci.ciWblErr	Interception storage water balance error	Positive if water generated (Δstorage + Outflow > Inflow)	

6.4.3 Ponded water storage

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

The items listed in Table 6.3 are those found in the "*Overland flow - detailed*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
    Name = 'OL_DETAIL'
    DisplayName = 'Overland flow - detailed'
    Description = 'Detailed Overland component water balance'
    NoGroups = 20
    Group = 'qpnet(ol.qpnet)'
    Group = 'qirrdrip(ol.qirrdrip)'
    Group = 'qeol(ol.qeol)'
    Group = 'qh(ol.qh+ol.qhmp)'
    Group = 'qolszpos(-ol.qolszpos)'
    Group = 'qolszneg(-ol.qolszneg)
    Group = 'qolin(ol.qolin)'
    Group = 'qolout(ol.qolout)'
    Group = 'qolrivpos(ol.qolrivpos)'
    Group = 'qolrivneg(ol.qolrivneg)'
    Group = 'qoldrstoinfl(ol.qoldrstoinfl)'
    Group = 'qoldrin(ol.qoldrin)'
    Group = 'qfloodtorivin(ol.qfloodtorivin)'
    Group = 'qfloodtorivex(ol.qfloodtorivex)'
    Group = 'qolMousepos(ol.qolMousepos)'
    Group = 'qolMouseneg(ol.qolMouseneg)'
    Group = 'qOlExtSink(ol.qOlExtSink)'
```

```
    Group = 'qOlExtSource(ol.qOlExtSource)'

    Group = 'dolsto(ol.dolsto)'

    Group = 'olwblerr(ol.olwblerr)'

EndSect  // WblTypeDefinition[WblTypeDefinition]
```

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

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

Special considerations for water balances in Flood Code cells

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

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

When the groundwater table is at or above the land surface, water can exchange directly between ponded water and the saturated zone. The unsaturated zone does not exist.This exchange is being calculated by the SZ solver, so it will not be updated during the UZ or OL time step. If the land surface is an active flood code cell, then then the water is added to or removed from the storage available for exchange with the river and the two terms qSZFloodPos and qSZFloodNeg may be non-zero.

The exchange between ponded water and the river in active flood code cells is calculated based on the change of storage due to the various source/sink terms over the MIKE SHE overland time step. This includes overland flow between flooded and non-flooded cells, rainfall, evaporation and infiltration to UZ. Direct flow between SZ and flooded cells when the groundwater table is above ground will not be included and treated separately. Thus, in a flood code cell

1. At the beginning of the overland time step, the ponded water level is set equal to the corresponding water level in the river (if this is above the MIKE SHE ground level) and the status of the cell is set to active.
2. At the end of the overland flow time step, MIKE SHE calculates the change in ponded water level and adds or subtracts this as lateral inflow to the river over the next river time step(s), covering the period of the MIKE SHE Overland time step.

qSzFlood is the contribution to lateral SZ exchange with the river in active flood cells.

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

Table 6.3 OL - Overland flow items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ol.qPnet	Canopy throughfall to ponded water. This is the same value as ci.qPnet, but with the opposite sign.	Inflow - negative	yes
ol.qIrrDrip	Irrigation added to ponded water. This includes both drip irrigation and sheet irrigation, since both are added directly to ponded storage.	Inflow - negative	yes
ol.qEOL	Direct evaporation from ponded water	Outflow - positive	yes
ol.qH	Infiltration from ponded water into the UZ	Outflow - positive	yes
ol.qHmp	Infiltration from ponded water into the UZ macropores	Outflow - positive	yes
-ol.qOLSZpos	Direct flow up from SZ to OL. This is a positive upwards flow in the MIKE SHE results files.	Inflow - negative (Note sign change in water balance definition)	yes
-ol.qOLSZneg	Direct flow down from OL to SZ. This is a negative downwards flow in the MIKE SHE results files.	Outflow - positive (Note sign change in water balance definition)	yes
ol.qOLin	Inflow to overland storage across the boundary of the model, or inflow across the boundary of the water balance sub-area	Inflow - negative	yes
ol.qOLout	Outflow from overland storage across the boundary of the model, or outflow across the boundary of the water balance sub-area	Outflow - positive	yes
ol.qOLRivPos	Overland outflow to the river	Outflow - positive	yes
ol.qOLRivNeg	Inflow from the river to overland storage	Inflow - negative	yes

Table 6.3 OL - Overland flow items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ol.qOlDrStoInfl	Overland storage that is added to the Overland drain storage.	Outflow - positive This is an outflow in the OL WB. It corresponds to the inflow to the OL Drain WB.	yes
ol.qOlDrIn	Inflow to Overland storage in a cell received from Overland drain storage.	Inflow - negative This is an inflow in the OL WB. It corresponds to the outflow in the OL Drain WB (ol.qOlDrOut) that is added to ponding in a cell.	yes
ol.qOlDrOut	Outflow of overland drain storage that is added to a cell or removed (i.e. to the boundary)	Outflow - positive This is an outflow in the OL Drain WB. In the OL WB, this is the part that is added to ponding in a cell. It appears in the ol.q.OlDrIn term.	no
ol.qOCDr	Overland drain storage that is added to a river link - excluding any flow to a specified river H-point or sewer node.	Outflow - positive This is an outflow in the OL Drain WB. This term does not appear in the OL WB.	no
ol.qOCDrToM11HPoint	Overland drain storage that is added to a specified river H-point.	Outflow - positive This is an outflow in the OL Drain WB. This term does not appear in the OL WB.	no
ol.qFloodToRivIn	Net lateral inflow exchange between active flood code cells OL and the river nodes that are inside the current water balance area	Inflow (negative) or Outflow (positive)	yes
ol.qFloodToRivEx	Net lateral inflow exchange between active flood code cells OL and the river nodes that are outside the current water balance area This is always zero unless the water balance is being calculated on a sub-area.	Inflow (negative) or Outflow (positive)	yes

Table 6.3 OL - Overland flow items

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ol.qOLMousePos	Outflow from overland storage to sewer model	Outflow - positive	yes
ol.qOLMouseNeg	Inflow from sewers to overland storage	Inflow - negative	yes
ol.qOLExtSink	Outflow to OpenMI sink	Outflow - positive	yes
ol.qOLExtSource	Inflow from OpenMI sink	Inflow - negative	yes
ol.dOLSto	Change in overland storage	Positive if storage increases	yes
ol.OLWblErr	OL water balance error	Positive if water generated (Δstorage + Outflow > Inflow)	

6.4.4 Unsaturated Zone storage

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

The items listed in Table 6.4 are those found in the "*Unsaturated Zone - detailed*" water balance output in the water balance configuration file:

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

```
    Group = 'duzdef(-uz.duzdef)'

    Group = 'uzszstocorr(uz.uzszstocorr)'

    Group = 'uzwblerr(uz.uzwblerr)'

EndSect  // WblTypeDefinition
```

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

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

Table 6.4 UZ - Unsaturated Zone items

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

Table 6.4 UZ - Unsaturated Zone items

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

6.4.5 Saturated Zone storage

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

The items listed in Table 6.5 are those found in the "*Saturated Zone - detailed*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
    Name = 'SZ_DETAIL'
    DisplayName = 'Saturated Zone - detailed'
    Description = 'Detailed Saturated ... balance (depth-integrated)'
    NoGroups = 29
    Group = 'qszprecip(sz.qszprecip)'
    Group = 'qrech(uz.qrech)'
    Group = 'qrechmp(uz.qrechmp)'
    Group = 'qolszpos(sz.qolszpos)'
    Group = 'qolszneg(sz.qolszneg)'
    Group = 'qetsz(sz.qetsz)'
    Group = 'qszin(sz.qszin)'
    Group = 'qszout(sz.qszout)'
    Group = 'dszsto(sz.dszsto)'
    Group = 'qszabsex(sz.qszabsex)'
    Group = 'qszdrin(sz.qszdrin)'
    Group = 'qszdrout(sz.qszdrout)'
    Group = 'qszdrtorivin(sz.qszdrtorivin)'
    Group = 'qszdrtorivex(sz.qszdrtorivex)'
    Group = 'qszdrtoM11HPoint(sz.qszdrtoM11HPoint)'
    Group = 'qszrivneg(sz.qszrivneg)'
    Group = 'qszrivpos(sz.qszrivpos)'
    Group = 'qszfloodneg(ol.qsztofloodneg)'
    Group = 'qszfloodpos(ol.qsztofloodpos)'
    Group = 'qgihbpos(sz.qgihbpos)'
```

```
        Group = 'qgihbneg(sz.qgihbneg)'
        Group = 'qirrwell(sz.qirrwell)'
        Group = 'qszdrtoMouse(sz.qszdrtoMouse)'
        Group = 'qszMousepos(sz.qszMousepos)'
        Group = 'qszMouseneg(sz.qszMouseneg)'
        Group = 'qSzExtSink(sz.qSzExtSink)'
        Group = 'qSzExtSource(sz.qSzExtSource)'
        Group = ''cross time step(sz.qolnextts)'
        Group = 'Error(sz.szwblerrtot)'
    EndSect  // WblTypeDefinition
```

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

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

water directly to the saturated zone, which is a negative downward flow and a negative water balance inflow to SZ.

Table 6.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sz.qSzPrecip	Precipitation added directly to the SZ layer. This can only be non-zero when the simulation does not include UZ. If UZ is included, but the groundwater table is at the ground surface (no UZ cells), the precipitation to SZ is included in the term sz.qOlSzNeg. Can be an outflow if the negative precipitation option specified in the Extra Parameters (Negative Precipitation *(p. 753)*). In this case, negative precipitation can be removed from multiple SZ layers.	Inflow - negative Can be positive (outflow) if negative precipitation option specified.	yes
uz.rech	Recharge out of the bottom of the UZ soil column to SZ via the UZ soil matrix. In the MIKE SHE results, recharge is a vertical downward flow, thus in the negative direction. This is the same sign as the water balance convention of negative inflow.	Inflow - negative	yes
uz.rechmp	Recharge out of the bottom of the UZ soil column to SZ via the UZ macropores or by-pass flow. In the MIKE SHE results, recharge is a vertical downward flow, thus in the negative direction. This is the same sign as the water balance convention of negative inflow.	Inflow - negative	yes
sz.qOlSzPos	Upward flow directly from SZ to ponded water. This is non-zero only when the groundwater table is at or above the ground surface. The sign is positive upwards which is the same as the positive outflow water balance sign convention.	Outflow - positive	yes

Table 6.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sz.qOlSzNeg	Downward flow directly from ponded water to SZ. This is non-zero only when the groundwater table is at or above the ground surface. The sign is positive upwards which is the same as the negative inflow water balance sign convention.	Inflow - negative	yes
sz.EtSz	Evapotranspiration directly from SZ.	Positive - outflow	yes
sz.qSzIn	Inflow to SZ storage across the boundary of the model, or inflow across the boundary of the water balance sub-area. Inflow from internal fixed head cells is also included in this term.	Inflow - negative	yes
sz.qSzOut	Outflow from SZ storage across the boundary of the model, or outflow across the boundary of the water balance sub-area. Outflow to internal fixed head cells is also included in this term, as well as drainage to local depressions that contain a fixed head boundary condition.	Outflow - positive	yes
sz.dSzSto	Change in SZ storage	Positive when storage increases	yes
sz.qSzAbsEx	Groundwater pumping from SZ. This does not include irrigation wells and shallow irrigation wells, but includes outflow to fixed head drain internal boundary conditions.	Outflow - positive Can be negative (Inflow) if injection specified for wells.	yes
sz.qSzDrIn	SZ drainage to local depressions in the current water balance area from areas outside of the current water balance sub-area. This term also includes inflow to the SZ drainage system added via OpenMI.	Inflow - negative	yes

Table 6.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
sz.qSzDrOut	SZ drainage to the model boundary, SZ drainage removed directly from the model. This term also includes SZ drainage to local depressions located outside of the current water balance sub-area.	Outflow - positive	yes
sz.qSzDrToRivIn	SZ drainage to MIKE SHE River Links inside of the water balance sub-area.	Outflow - positive	yes
sz.qSzDrToRivEx	SZ drainage to MIKE SHE River Links outside of the water balance sub-area. This can only be non-zero if the water balance is calculated for a sub-area.	Outflow - positive	yes
sz.qSzDrToM11HPoint	SZ drainage to specified river h-points. These are specified in the Extra Parameter option in SZ Drainage to Specified River H-points *(p. 770)*	Outflow - positive	yes
sz.qSzRivPos	Baseflow from SZ to MIKE SHE River Links	Outflow - positive	yes
sz.qSzRivNeg	Infiltration from MIKE SHE River Links to SZ	Inflow - negative	yes
ol.qSZToFloodPos	Direct flow upwards from SZ to an active flood code cell (active means that it is actually flooded and the water level is controlled by the water level in the river). This is a positive upwards flow in the MIKE SHE results files. Only non-zero when the groundwater table is at or above the ground surface.	Outflow - positive (Note sign change compared to detailed Ponded Storage water balance)	yes

Table 6.5 SZ - Saturated Zone - all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
ol.qSZToFloodNeg	Direct flow downwards from an active flood code cell to SZ. (active means that it is actually flooded and the water level is controlled by the water level in the river). This is a negative downwards flow in the MIKE SHE results files. Only non-zero when the groundwater table is at or above the ground surface.	Inflow - negative (Note sign change compared to detailed Ponded Storage water balance)	yes
sz.qGihbPos	Outflow from SZ storage to internal general head boundaries (GHB cells)	Outflow - positive	yes
sz.qGihbNeg	Inflow from internal general head boundaries (GHB cells) to SZ storage	Inflow - negative	yes
sz.qIrrWell	Groundwater pumping from irrigation wells. This includes both specified irrigation wells and shallow wells.	Outflow - positive	yes
sz.qSzDrToMouse	SZ drainage to specified sewer manholes.	Outflow - positive	yes
sz.qSzMousePos	Outflow from SZ storage to sewer pipes	Outflow - positive	yes
sz.qSzMouseNeg	Inflow from sewer pipes to SZ storage	Inflow - negative	yes
sz.qSzExtSink	Outflow to external sinks specified via OpenMI	Outflow - positive	yes
sz.qSzExtSource	Inflow from external sources specified via OpenMI	Inflow - negative	yes
sz.qolnextts	Flow from/to OL that could not be applied in one time step and has been transferred to the next. Net transfer between previous/next time step. This is considered a wbl error, so it is not being subtracted from the total error.	Net in transfer: negative Net out transfer: positive	no
sz.szWblErrTot	Aggregated SZ water balance error for all layers	Positive if water generated (Δstorage + Outflow > Inflow)	

Saturated Zone layers

The Saturated Zone water balance can also be calculated by numerical layer. This means that all of the items in Table 6.5 are repeated for each numerical layer. However, in this case the water balance error term, sz.szWblErrTot is replace by a water balance error for each layer.

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

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

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

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



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

Table 6.6 SZ - Saturated Zone - specified by layers

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

Saturated Zone Linear Reservoir water balance

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

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

The items listed in Table 6.7 are those found in the "*Saturated Zone - layers(Linear Reservoir)*" water balance output in the water balance configuration file:

```
[WblTypeDefinition]
    Name = 'SZ_LAYER_LR'
    DisplayName = 'Saturated Zone - layers(Linear Reservoir)'
    Description = 'Saturated zone water balance for linear reservoir'
    NoGroups = 13
    Group = 'recharge(uz.qrech+uz.qrechmp)'
    Group = 'evapotranspirationSZ(sz.qetsz)'
    Group = 'lateral IN(sz.qszin)'
    Group = 'lateral OUT(sz.qszout)'
    Group = 'percolation(sz.qszzneg)'
    Group = 'To river(sz.qszrivpos)'
    Group = 'From river(sz.qszrivneg)'
    Group = 'storagechange(sz.dszsto)'
    Group = 'deadzonestoragechange(sz.dszsto_dead)'
    Group = 'pumping(sz.qszabsex)'
    Group = 'Irr.pumping(sz.qirrwell)'
    Group = 'feedbackUZ(sz.qUZfeedback)'
    Group = 'Error(sz.szwblerr)'
EndSect  // WblTypeDefinition
```

Table 6.7 SZ - Saturated Zone - Linear Reservoir all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
recharge (uz.qrech+uz.qrechmp)	This is the total recharge into the inter-flow reservoirs. If UZ is not simulated, then uz.qrech is still calculated based on the infiltration from OL.	Inflow - negative	yes
evapotranspirationSZ (sz.qetsz)	This is the direct ET from the water table. In the LR SZ method, the water table is constant and fixed at the beginning of the simulation. If the root zone reaches the water table, then ET will be taken from the water table as an infinite sink when the reference ET is not satisfied by the other sources.	Outflow - postitive	yes
lateral IN (sz.qszin)	In the LR SZ model, infiltration to the interflow reservoirs and percolation to the baseflow reservoirs is distributed equally to the entire reservoir. When you calculate the water balance in a sub-area, sz.qszin is the amount of recharge/percolation that is distrib-uted into the sub-area. For example, if all your recharge occurs outside of your sub-area, this is the increase in groundwater storage that occurs inside your sub-area. This can only be non-zero for sub-area water balances.	Inflow - negative	yes
lateral OUT (sz.qszout)	In the LR SZ model, infiltration to the interflow reservoirs and percolation to the baseflow reservoirs is distributed equally to the entire reservoir. When you calculate the water balance in a sub-area, sz.qszout is the amount of recharge/percolation that is distrib-uted to areas outside of the sub-area. For example, if all your recharge occurs inside your sub-area, this is the increase in groundwater storage that occurs outside your sub-area. This can only be non-zero for sub-area water balances.	Outflow - positive	yes

Table 6.7 SZ - Saturated Zone - Linear Reservoir all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
percolation (sz.qszzneg)	Infiltration from interflow reservoirs to baseflow reservoirs. This is defined only for the lower (baseflow) layer in the water balance output, but is used in the water balance error calculation of the interflow reservoirs with the opposite sign. The term sz.qszzpos is not included here because the LR method does not allow any transfer of water from the baseflow reservoir upwards to the interflow reservoir.	Inflow - negative (to the baseflow reservoir)	yes
To river (sz.qszrivpos)	Outflow from interflow and baseflow reservoirs to MIKE SHE River Links.	Outflow - positive	yes
From river (sz.qszrivneg)	Inflow from MIKE SHE River Links to the baseflow reservoir. For the interflow reservoirs, this is always zero because the river model only discharges to the baseflow reservoirs.	Inflow - negative (to the baseflow reservoir)	yes
storagechange (sz.dszsto)	Change in storage in the interflow and baseflow reservoirs.	Positive if storage increases	yes
deadzonestorageschange (sz.dszsto_dead)	Change in storage in the deadzone storage. This is calculated as a change in storage, but it is equal to the outflow to dead zone storage because there is no option in MIKE SHE to reduce the dead zone storage.	Outflow - Positive	yes
pumping (sz.qszabsex)	Groundwater pumping from the baseflow reservoirs. This is always zero for the interflow reservoirs.	Outflow - positive But, can be negative if injection rates specified in wells	yes
Irr.pumping (sz.qirrwell)	This is the sum of groundwater pumping for irrigation - irrigation wells + shallow irrigation wells.	Outflow - positive	yes
feedbackUZ (sz.qUZfeedback)	This is a fraction of the discharge from the baseflow reservoirs to the river to account for discharge to riparian zones that is lost to ET.	Outflow - positive (from baseflow reservoirs only)	yes

Table 6.7 SZ - Saturated Zone - Linear Reservoir all layers

Item	Description	Sign Convention in the Water balance	Included in Wbl Error
Error (sz.szwblerr)	SZ water balance error for the current layer only available for LAYER water balances	Positive if water generated (Δstorage + Outflow > Inflow)	
Error (sz.szwblerrtot)	SZ water balance error for the both the interflow and baseflow reservoirs combined. This is only available for the total water balance option.	Positive if water generated (Δstorage + Outflow > Inflow)	

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

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

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

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

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

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

subcatchment. However, the baseflow reservoir only received the "average" over the area (total percolation/baseflow res. area).

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

6.5 Standard Water Balance Types

Table 6.8 summarizes the 35 standard water balance types defined in the water balance configuration file. Some of the water balances cannot be used in certain conditions and these restrictions are listed in the table.

Table 6.8 Water balance types available in the default configuration files

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

Table 6.8 Water balance types available in the default configuration files

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

Table 6.8 Water balance types available in the default configuration files

Water balance type	Description
Map output: Saturated Zone Storage	Distributed output: Saturated zone Storage (depth-integrated)
Map output: Saturated Zone Storage - layer(s)	Distributed output: Saturated zone Storage (each or specified layer)

6.5.1 Total water balance

This is the most commonly used water balance calculation. It is typically used to get an overview of the overall water balance of a MIKE SHE simulation.

In the description below, it is assumed that the water balance is for the entire model domain. However, you can specify the water balance as a sub-domain. In this case, the boundary flows etc. are calculated in and out of the sub-domain.

Table 6.9 Description of items in the Total Water balance

Parameter	Items	Description
Precip	sm.qp	Total rainfall. This includes “snow” has equivalent rainfall, when the air temperature is below the defined freezing point.
Canopy Stor. Change	ci.dintsto	Change in canopy storage. This is the difference in the total amount of canopy storage between storing time steps. Canopy storage is the amount of rainfall that is captured on the leaves per time step in time steps that are raining. It is a vegetation parameter.
Evapotrans	ci.qeint+ol.qeol+uz.qeuz+ uz.qtuz+ sz.qetsz+ sm.qesnow	Actual ET. This is the sum of calculated ET from all sources including: - evaporation from the canopy - evaporation from ponding - evaporation from the top UZ layer - transpiration from the root zone - evaporation from the SZ if there is no UZ layer - evaporation from snow
Snow Stor.Change	sm.dsnowsto	Change in snow storage. This is the difference in the total amount of snow storage between storing time steps.
OL Stor.Change	ol.dolsto	Change in ponded storage. This is the difference in the total amount of ponded water between storing time steps.

Table 6.9 Description of items in the Total Water balance

Parameter	Items	Description
OL Bou.Inflow	ol.qolin	Overland inflow.This is the amount of water that flows out of the model as overland flow across the model boundaries.
OL Bou.Outflow	ol.qolout	Overland outflow. This is the amount of water that flows out of the model as overland flow across the model boundaries.
OL->Ext.Sinks	ol.qOlExtSink+ ol.qOlExtSource	Net defined exchange if ponded water. There are several ways to add or remove water directly from ponded water in a cell (e.g. as function via OpenMI). This is the net of all these functions.
OL->River/Sewer	ol.qolrivpos+ol.qolrivneg+ ol.qfloodtorivin+ ol.qfloodtorivex+ ol.qolMousepos+ ol.qolMouseneg	Net overland exchange with the river. This is the sum of all exchanges between ponded water and the river.
OL Drain Stor. Change	ol.doldrsto	Change in storage in ponded drains. The MIKE SHE engine supports conceptual drainage of ponded water. This by-passes the classic 2D overland flow. The drainage is removed from the ponded storage (o.dolstor) and added to this storage before it is discharged.
OL Drain->River/Sewer	ol.qocdr+ ol.qocdrtoM11HPoint+ ol.qoldrtoMouse	This is the total discharge of the OL drain storage to the river and the sewer. The reference to “M11HPoint” refers to the option to discharge to a specific river model H-point.
OL Drain->OL IN	ol.qoldrin	Drainage to an internal cell. The ponded drainage can drain to an internal model cell. This item is used if the discharge cell is in the area selected for the water balance calculation.
OL Drain->OL OUT	ol.qoldrout	Drainage to an internal cell. The ponded drainage can drain to an internal model cell. This item is used if the discharge cell is NOT in the area selected for the water balance calculation.
Irrigation	sm.qirrsprinkler+ol.qirrdrip	Total irrigation. This is the sum of irrigation applied by sprinkler and irrigation added directly to the ponded storage (drip irrigation). Note that sprinkler irrigation is reduced by the canopy storage, whereas drip irrigation is not.

Table 6.9 Description of items in the Total Water balance

Parameter	Items	Description
SubSurf.Stor.Change	sz.dszsto+ sz.dszsto_dead+ uz.uzszstocorr+ sz.qUZfeedback- uz.qgwfeedbackuz- uz.duzdef	Total change in storage below the ground surface. This is the total change in the amount of water storage in the ground, including: - change in groundwater storage - change in deadzone storage (SZ Linear Reservoir option) - correction for water transfer to/from SZ and UZ - UZ feedback (SZ Linear Reservoir option)
SubSurf.Bou.Inflow	-sz.qszin+sz.qgihbneg	Groundwater boundary inflow. This is the total amount of water that is added to the groundwater, except via pumping injection.
SubSurf.Bou.Outflow	sz.qszout+sz.qgihbpos	Groundwater boundary outflow. This is the total amount of water that is removed from the groundwater, except via pumping and ET.
SubSurf->Ext.Sinks	sz.qSzExtSink+ sz.qSzExtSource+ uz.qUzExtSink+ uz.qUzExtSource	External sinks. This is the net amount of water added/substracted from the subsurface via functions (e.g. OpenMI).
Pumping	sz.qszabsex+sz.qirrwell	Total pumping. This is the sum of regular groundwater pumping and abstraction from irrigation wells. Note that these are specified separately in MIKE SHE.
SZ Drain->River	sz.qszdrtorivin+ sz.qszdrtoM11HPoint	Groundwater drainage to river. This is the amount of drainage to M1D rivers. The reference to “M11HPoint” refers to the option to discharge to a specific H-point in MIKE 1D.
SZ Drain->Ext.River	sz.qszdrtorivex	Groundwater drainage to rivers outside of the domain. MIKE SHE allows groundwater to drain to MIKE 1D nodes that are outside of the model domain or water balance area.
SZ Drain Inflow	sz.qszdrin+ sz.qszMouseneg	Groundwater drainage added to SZ. The groundwater drainage function can also discharge to internal cells. In this case the SZ drainage is added to the SZ in the target location. Drainage from a MIKE 1D sewer node can also be added to SZ.

Table 6.9 Description of items in the Total Water balance

Parameter	Items	Description
SZ Drain Outflow	sz.qszdrout+ sz.qszMousepos+ sz.qszdrtoMouse	Groundwater drainage removed from SZ that does not go to a river. The groundwater drainage function can also discharge to internal cells and boundaries. Drainage to a MIKE 1D sewer node is also included here.
Baseflow to river	sz.qszrivpos+ol.qsztofloodpos	Groundwater inflow to rivers. This is the amount of water that flows into the bottom of the rivers from groundwater.
Baseflow from river	sz.qszrivneg	River discharge to groundwater. This is the amount of water that infiltrates from the bottom of the rivers into the groundwater.
Cross time step	sz.qolnextts	Flow between OL-SZ that could not be applied in one time step and has been transferred to the next. Net transfer between previous/next time step. This is considered a wbl error, so it is not being subtracted from the total error.
Error	sm.smwblerr+ci.ciwblerr+ ol.olwblerr+ol.oldrwblerr+ uz.uzwblerr+sz.szwblerrtot	Total error. This is the sum of all the individual error terms from the different water balance components. A positive total error means that the model is generating water. A negative total error means that the model is losing water.

6.6 Making Custom Water Balances

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

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

To illustrate how you could add an additional water balance type, the table below describes the format for each line of the water balance type definition. The example is for an extra water balance type to calculate the net vertical

flow in a specified SZ layer. This water balance type can only be used with the single-cell resolution and specified output layers options.

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

Line item	Comment

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

[WblTypeDefinition] Name = 'TOTAL' ... Group = 'SZ Storage(sz.szsto)' EndSect // WblTypeDefinition	Existing water balance definitions
[WblTypeDefinition]	First line of the water balance definition
Name = 'SZ_LAYER_NET_VERT_FLOW_MAP'	Internal name. No spaces allowed
DisplayName = 'Map output: Net Vertical Saturated Zone Flow - layer(s)'	Name displayed in the combobox
Description = 'Distributed output: Saturated zone Storage (specified layer)'	Description displayed under the combobox
NoGroups = 1	Number of calculation groups in the output file
Group = 'SZ Vertical Flow(sz.qszzpos+ sz.qszzneg)'	Definition of the calculation group, consisting of a name and a sum of the particular water balance items (no spaces) from Table 6.1 to Table 6.7. Map items can only have one group (NoGroups = 1)
EndSect // WblTypeDefinition	
EndSect // MIKESHE_WaterBalance_ConfigFile	last line in the file

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

6.6.1 Customizing the chart output

The chart water balance is a special water balance function that creates an ASCII file that is read by another program to generate the graphic in Figure 6.1.

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

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

.

Table 6.11 MSHE_Wbl_Config.pfs - example section to edit to customize the chart water balance

Line item	Comment

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

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

Table 6.11 MSHE_Wbl_Config.pfs - example section to edit to customize the chart water balance

Line item	Comment
EndSect // WblTypeDefinition	
EndSect // MIKESHE_WaterBalance_ConfigFile	last line in the file

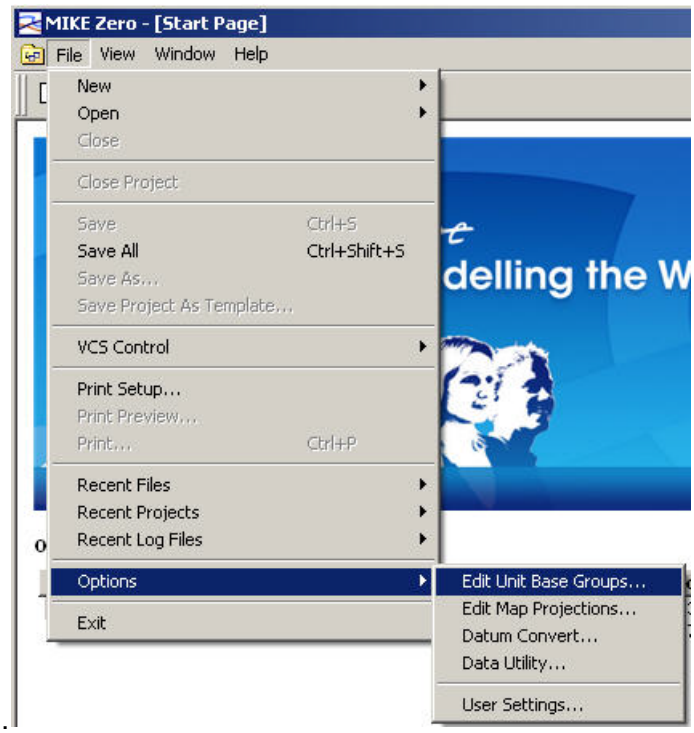
MIKE ZERO OPTIONS

7 EUM Data Units

All MIKE Zero products use a standard library of data units, called the Engineering Unit Management (EUM) library. This allows you to change the displayed units for any value that is included in the library.

Every parameter in MIKE SHE has been added to the EUM library and to change the displayed unit, you must know the EUM Data Type. In most cases, the EUM Data Type is displayed in the fly-over text when you put your mouse cursor in the text field. Alternatively, all items in the on-line help (F1) list the EUM Data Type in the table at the beginning of the section.

To change the display units of any EUM Data Type, you must close all open documents and then select 'Options/Edit Unit Base Groups...' from the File pull down menu



When you select this menu item, the Unit Base Group Editing dialogue appears. By default all of the data units for each active module are displayed. For a clearer overview of the data types, close all of the model engines that are not relevant.

Next select the data item that you want to change the units of. Then select the new units from the combobox list of available units.

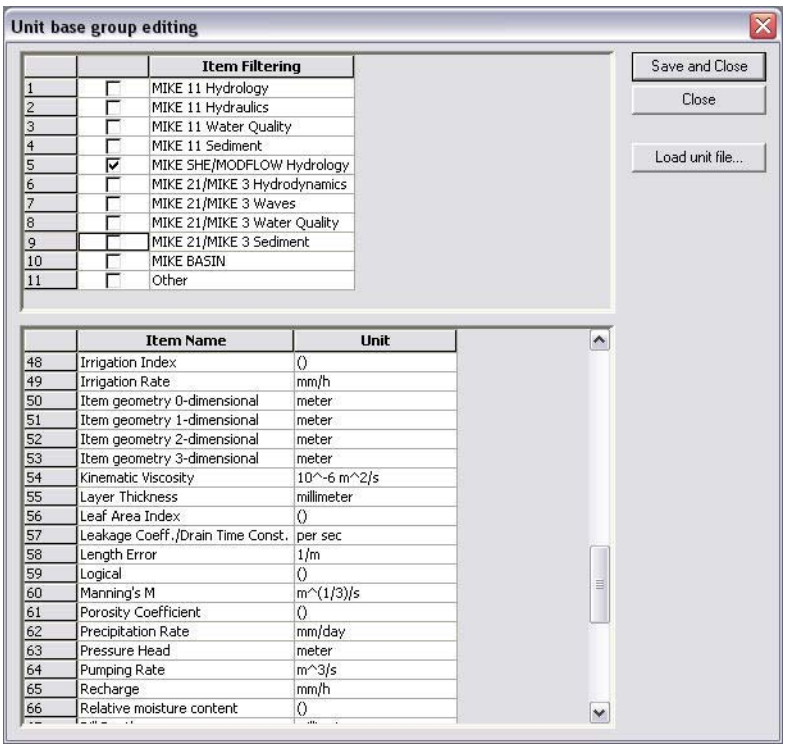
After you have changed the data units, click 'Save and Close'. This saves your changes to the default Unit Base Groups (.ubg) file:

C:\Program Files\Common Files\DHI\MIKEZero\MIKEZero.ubg

which is read every time you open a model.

Note: If you have already added data to your model, changing the Unit Base Group will not convert any of your data. This process simply changes the displayed units in the user interface and the conversion factors used to make the input files internally consistent.

In some cases the relevant data item name is not clear, as there may be several data items with similar names. This is more likely to occur if several modules are selected at the same time. To find out which data item is correct, close the dialogue and re-open your model. Then either move the mouse to the relevant text box, where a fly-over text box should appear telling you what is the relevant data type for this field. Alternatively, for gridded data, you can use the Create button to create a data file and then notice the data type that is displayed in the dialogue.



Finally, occasionally, you may find that the data unit that you are looking for is not available. In this case, contact your local Technical Support Centre, who should forward your request to the developer for inclusion in the next release.

7.1 Changing from SI to Imperial (American) data units.

The default Unit Base Groups (.ubg) file,

C:\Program Files\Common Files\DHI\MIKEZero\MIKEZero.ubg

is read every time you open a model.

In the same directory there are two standard Unit Base Group files:

MIKEZero_Default_Units.ubg

MIKEZero_US_Units.ubg

The first is the default file and contains standard SI units for all data items in all of the MIKE Zero products. The second contains standard Imperial (US) units for most data items in all of the MIKE Zero products.

To change the display units for all of your data items to Imperial units, load the MIKEZero_US_Units.ubg file, Save and Close the dialogue and then reopen your model.

If you want to change individual data items to SI or Imperial, you can change the items individually. Then use the Save and Close button to save your changes back to the MIKEZero.ubg file. If you want to create special unit versions, then you can copy the MIKEZero.ubg to a different file name and reload it.

7.2 Restoring the default units

You can return to your default unit specification at any time, by Loading either of the default .ubg files:

MIKEZero_Default_Units.ubg

MIKEZero_US_Units.ubg

which are found in the

C:\Program Files\Common Files\DHI\MIKEZero\

directory.

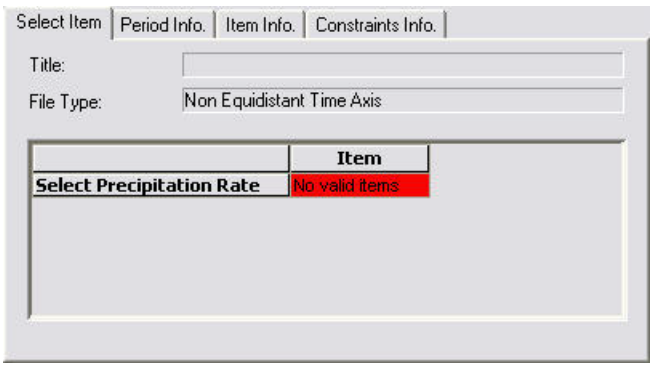
Note! If you want to save any of your model specific changes, then you should first save the MIKEZero.ubg to a new name.

7.3 Changing the EUM data type of a Parameter

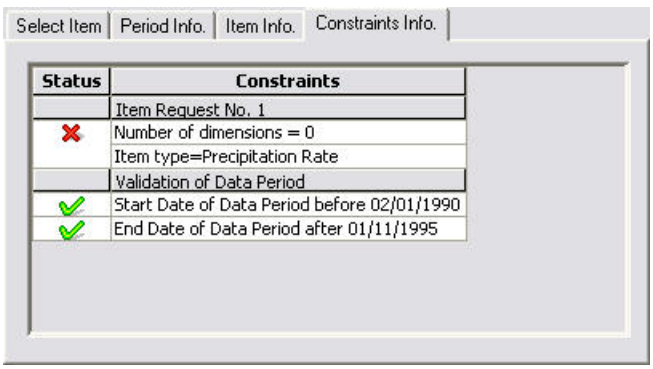
When you create a .dfs0 or .dfs2 parameter file, you must also define the EUM data type for each parameter in the file. When you assign a .dfs0 or a .dfs2 file to a parameter value, then MIKE SHE automatically verifies that the correct EUM data type is being used. If the wrong data type is present then you will not be able to select OK in the file browser dialogue.

For example, in the following set of dialogues, an Evapotranspiration time series was selected instead of the correct Precipitation time series file

The first error is in the Select Item tab, where there is a message that no Valid Items are found.

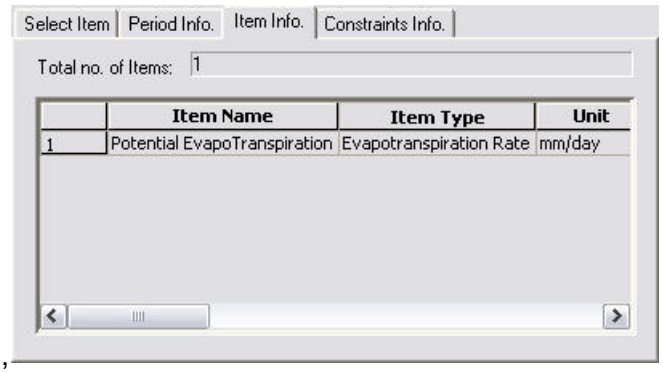


The find out why there is no valid items, you should look in the Constraints Info tab:



Here you can see that the Item type is supposed to be Precipitation Rate, but this constraint has failed.

To find out what the Item Type of the selected file is, look at the Item Info tab:



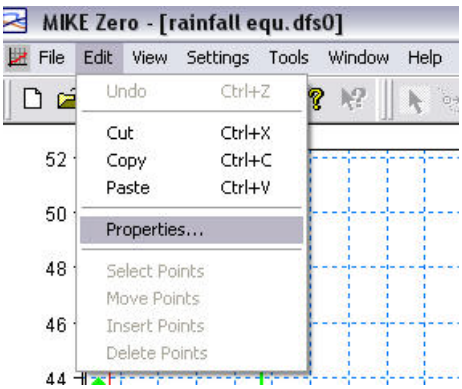
,

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

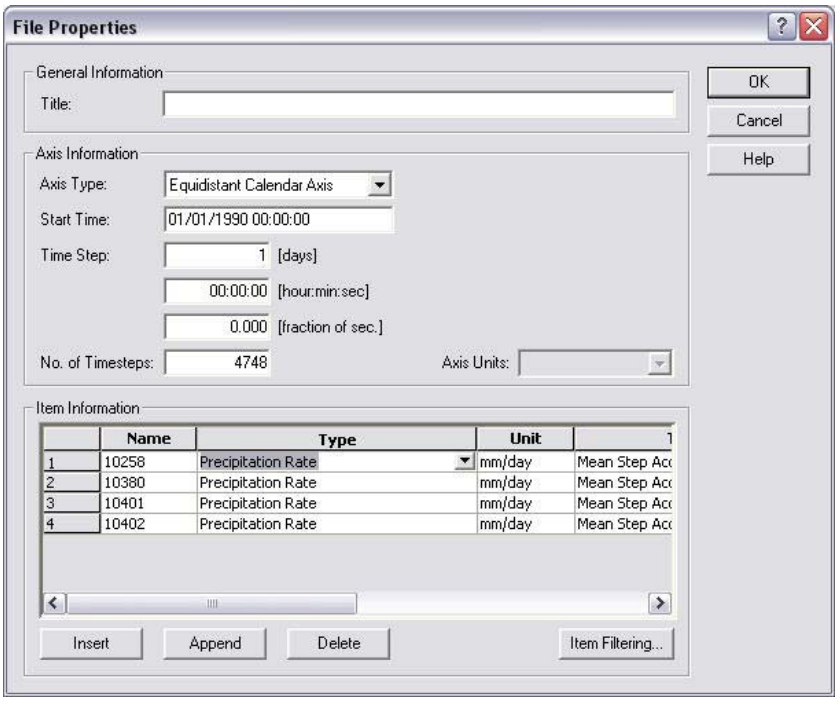
The next two sections outline how to change the EUM Type of an existing file.

7.3.1 Changing the EUM Type of a .dfs0 Parameter

To change the EUM Data Type of a parameter in a .dfs0 file, open the time series in the Time Series Editor and then select the Properties... item from the Edit drop down menu



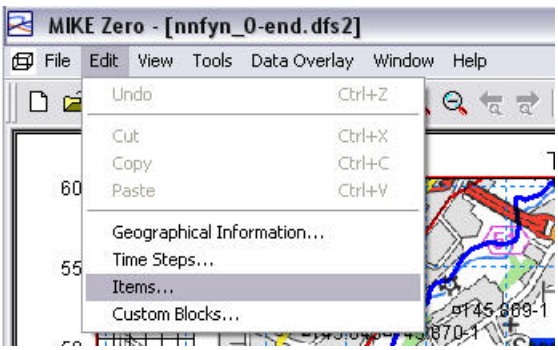
This opens the item properties dialogue



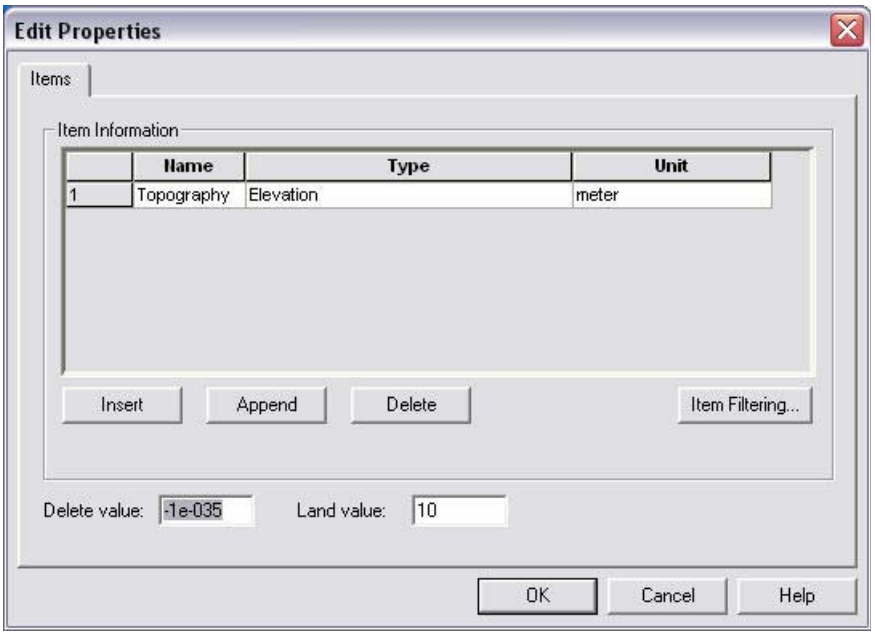
where you can change the EUM Type and the EUM Unit that is assigned for each time series in the file.

7.3.2 Changing the EUM Type of a .dfs2 Parameter

To change the EUM Data Type of a parameter in a .dfs2 file, open the grid file in the Grid Editor and then select the Items... item from the Edit drop down menu



This will open the Edit Properties dialogue for the Grid Editor



where you can change the EUM Type and the associated data EUM Unit of the item.

WORKING WITH DATA

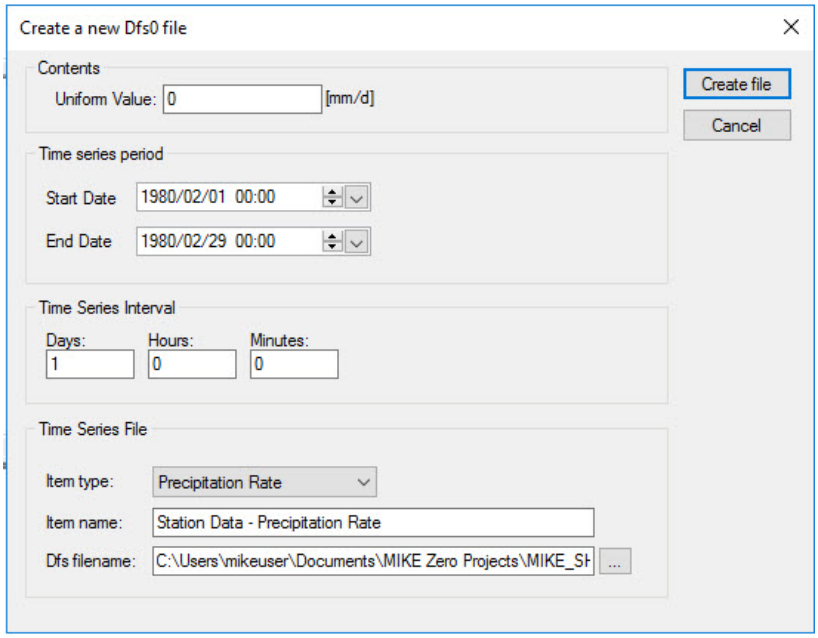
8 Time Series Data

MIKE SHE uses the dfs0 file format for time series data. Various tools are available for converting ASCII and EXEL time series to the dfs0 file format. Time series data is required as input for most transient simulations, for example, daily records of precipitation. Transient simulations can also generate numerous dfs0 output files.

8.1 Creating Time Series in MIKE SHE

In most cases, you will create dfs0 files using the Create buttons in the MIKE SHE Setup dialogues. In this way, you can avoid the confusing task of assigning the Type of time series (e.g. precipitation) and EUM Unit type (e.g. millimetres) and the TS Type (e.g. reverse step accumulated). Each of these items are specified automatically.

If you create time a time series using a Create button, the following dialogue will appear:



Uniform time series

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

Time Series Period

The time series period is the extent of the time series. In a MIKE SHE simulation, all the time series files must cover the Simulation Period *(p. 188)*. The default time series period for a new time series file is the Simulation Period. However, if you change the time series period so that it does not cover the simulation period, you will receive an error message when MIKE SHE tries to run. If you try to add a time series file that does not cover the simulation period, then the OK button will remain greyed out and you will not be able to select the file. The constraints tab in the file selector dialogue gives you the reason that you cannot select the file.

Time Series Interval

The time series interval is the length of the individual time periods. The number of time periods is the length of the time series period divided by the period interval. The last period is shortened if necessary.



Time Series File

Every time series has an **Item Type** which is defined by the valid EUM Data Unit (see EUM Data Units *(p. 143)*) for the particular variable from which the Create dialogue was launched. In most cases, there is only one valid Type. In some cases you may have a choice. For example, in Precipitation, you can chose between Precipitation Rate, which is the average amount of precipitation per time (e.g. mm/hour) in the time interval, and Rainfall.which is the measured amount of precipitation in the time interval (e.g. mm).

The **Name** is simply the name of the data item in the resulting .dfs0 file.

The **file name** has a default value, that you should change if you will be creating several files of the same type, such as multiple rain gauge time series files. Otherwise you may accidentally overwrite the previous file.

8.1.1 Import from ASCII or Excel

The easiest way to import ASCII data into a dfs0 file is via the Windows clipboard. In this case, create a uniform time series file with the correct number of time steps and then highlight all of the data values. Then copy and paste the data from the ASCII file into the table.

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

8.2 Working with Spatial Time Series

In the MIKE SHE Toolbox, there is a Tool in the File Converter section called ***dfs2+dfs0 to dfs2***. In this utility you specify a dfs2 grid file with integer grid codes and a dfs0 file with time series data, where the dfs2 file grid codes are the item numbers in the dfs0 file.

The utility will read the dfs2 file and for each time step in the dfs0 file, it will substitute the grid code with the time series value.

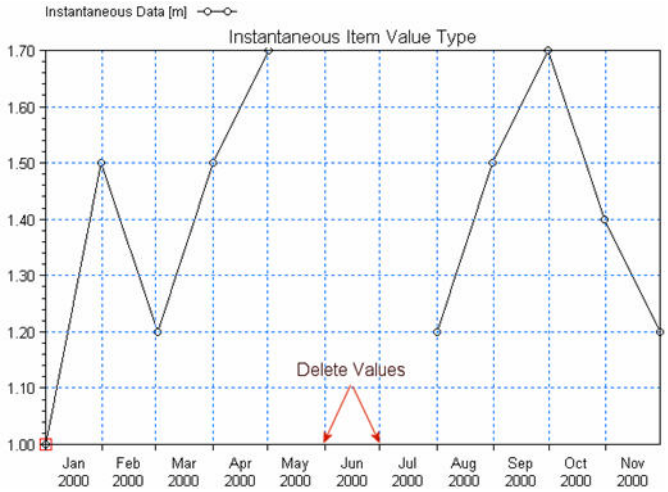
The result is a dfs2 file with one grid for each time step and the grid values are the time series values.

8.3 Time Series Types

Specifies how the time step is being defined and how the measured value is being assigned to the time step. There are five different value types available:

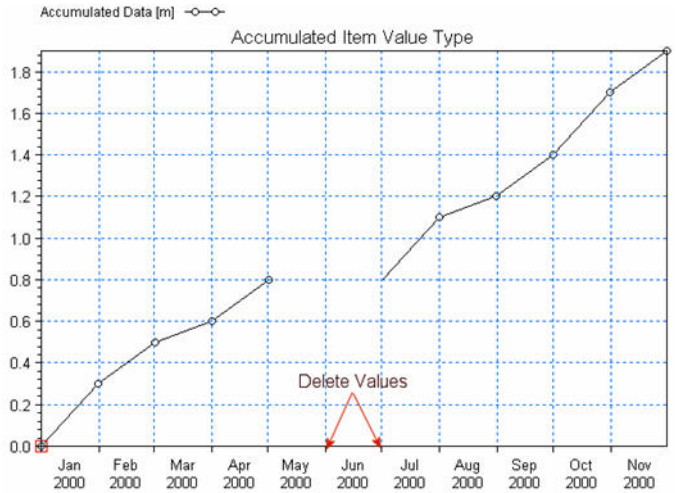
Instantaneous

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


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

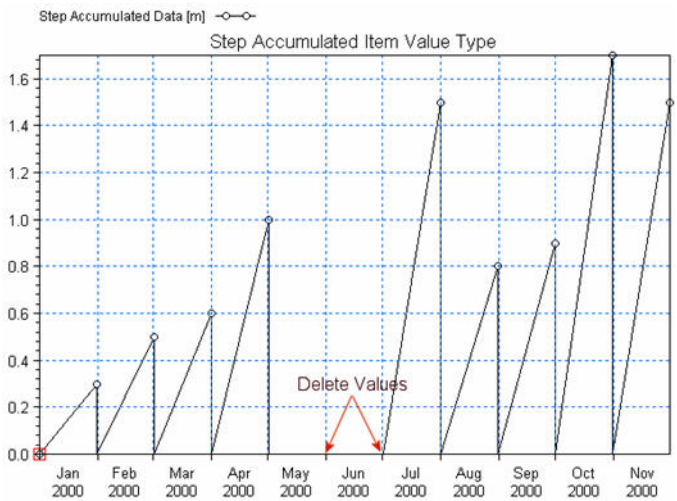
Accumulated

The values are summed over successive intervals of time and always relative to the same starting time. For example, rainfall accumulated over a year with monthly rainfall values.



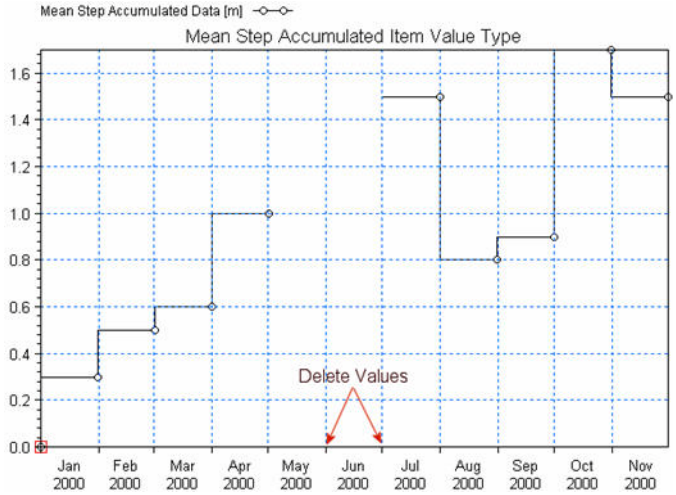
Step Accumulated

The values are accumulated over a time interval, relative to the beginning of the interval. For example, a tipping bucket rain gauge measures step-accumulated rainfall. In this case, the rain gauge accumulates rainfall until the gauge is full, then it empties and starts accumulating again. Thus, the time series consists of the total amount of rainfall accumulated in each time period - say in mm of rainfall.



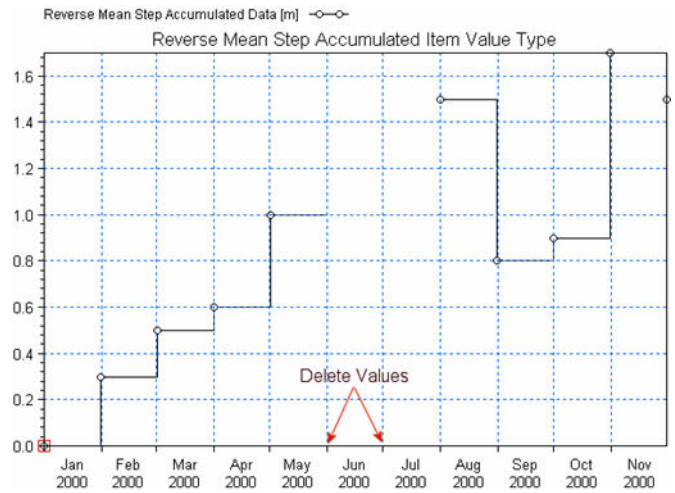
Mean Step Accumulated

The values are accumulated over the time interval as in the Step Accumulated, but the value is divided by the length of the accumulation period. Thus, based on the previous example, the time series consists of the rate of rainfall accumulated in each time period - say in mm of rainfall per hour (mm/hr).



Reverse Mean Step Accumulated

In this case, the values are the same as the Mean Step Accumulated, but the values represent the time interval from now to the start of the next time interval. The Reverse Mean Step Accumulated time series are primarily used for forecasting purposes.



9 Using MIKE SHE with ArcGIS

MIKE SHE has been designed to work smoothly with ArcGIS files. In most cases, distributed data can be linked directly to shape files created by ArcGIS or any other application. The type of shape file depends on the type of data. Distributed data, such as initial water levels can be input as point and line themes, whereas spatial data that is referenced to a time series, such as precipitation, can be added as a polygon theme. In this case, each polygon can be assigned a time series of values.

In the reverse direction, all gridded data in the MIKE SHE Setup Editor can be easily saved as a point theme shape file from the pop-menu when you right click on a colour shaded map. This includes both interpolated data in the Setup tab and pre-processed data in the Pre-processed tab.

9.1 Converting ArcGIS ASCII Grids to dfs

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

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

9.2 Converting dfs files to ASCII Grids and shape files

The MIKE Zero Tool box also contains tools for converting dfs2 files to ArcGIS shape files (Mike2Shp) and Grid files (Mike2Grd). These tools can be useful if you have manipulated your grid files in the MIKE Zero Grid Editor, since it does not directly support shape file export.

Alternatively, you can right click on any map view in the MIKE SHE Setup Editor and then use the right mouse function to export to a shape file.

If you want to convert a dfs3 file to a shape file or a grid file then you will need to extract a dfs2 file from the dfs3 first using the 2D Grid from 3D file tool that is found under the Extraction item in the MIKE Zero Toolbox.

10 Spatial Data

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

10.1 The Grid Editor

The Grid Editor is a generic MIKE Zero grid tool for all MIKE by DHI software. It is the primary means to edit and manipulate gridded data in MIKE SHE.

The Grid Editor was originally developed for the Marine programs MIKE 21 and MIKE 3. However, this often leads to confusion in the node and layer numbering because MIKE 21 and MIKE 3 use a different nodal system because they are based on a node-centred finite difference scheme. Whereas, MIKE SHE is based on a block-centred finite difference scheme.

Node numbering in the Grid Editor

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

Layer numbering in the Grid Editor

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

10.2 Gridded Data Types

There are two basic types of spatial data in MIKE SHE - Real and Integer. Real data is generally used to define model parameters, such as hydraulic conductivity. Integer data is generally used to define parameter zones. Thus, model cells with the same integer value can be associated with a time series or other characteristic.

Furthermore, real spatial parameters can be distinguished by whether or not they vary in time. At the moment Integer zones cannot vary with time.

Thus, spatial parameters can be divided into the following:

- Stationary Real Parameters
- Time Varying Real Parameters, and
- Integer Grid Codes

Stationary Real Parameters

Stationary Real Parameters can vary spatially but do not usually vary during the simulation, such as hydraulic conductivity. If such parameters do vary in time, then you must divide the simulation into time periods and run the each

time period as a separate simulation, starting each simulation from the end of the previous simulation. This is most easily accomplished using the Hot Start facility, which is found in the Simulation Period dialogue.

The spatial distribution of stationary real parameters are entered using the Stationary Real Data dialogue

Time Varying Real Parameters

Many spatial parameters are time dependent, such as precipitation rate. In this case, both a spatial distribution , as well as a time series for each cell in the model, must be defined. Spatially distributed parameters that also vary in time are entered using the Time-varying Real Data dialogues

10.3 Integer Grid Codes

Integer Grid Codes are required when Real data varies in time or when model functions, such as soil profiles and paved areas, are assigned to particular zones. Integer Grid Codes are always integer values and do not vary with time.

For information on entering Integer Codes see the Integer Grid Codes section.

The following is an outline of the parameters that require Integer Grid Codes.

Model Domain

Integer Grid Codes are used to define the inactive areas both inside and outside the model domain. Inactive areas outside of the model and the edge of the model are defined in the Model Domain and Grid section, while inactive, subsurface areas inside of the model are defined as Internal boundary conditions.

Component Calculations

Integer Grid Codes are used to delineate such things as paved areas. In this case, the integer code acts like a flag and the calculations that are done are different depending on how the flag is set.

Model Properties

Integer Grid Codes are used to delineate areas with similar properties. In this case, the integer value defines the zone to which the cell belongs. Thus, it defines which set of model properties is to be assigned to the particular cell.

For example, a model may be divided into a five zones each with a different soil profile for the unsaturated zone. In this case, the data tree will expand under the model property to include five separate sub-branches, where the soil profile can be defined.

Time Series

Integer Grid Codes are used to define zones for which Real data varies in time. Thus, a time series for a parameter, such as precipitation rate, can be assigned to a model zone. Similarly to the Model Properties above, the model tree will expand under the parameter to include a separate sub-branch for each zone, where the time series file can be defined.

Time Varying Integers

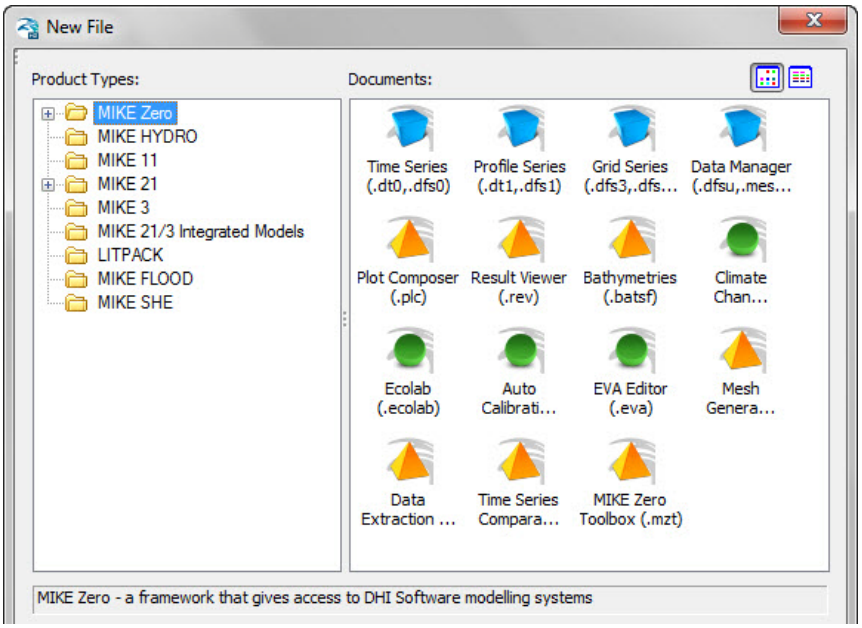
Grid Codes and Integer values do not normally vary with time. If such parameters do vary in time, then you must divide the simulation into time periods and run each time period as a separate simulation, starting each simulation from the end of the previous simulation using the Hot Start options (see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

10.4 Gridded (.dfs2) Data

If the parameter is defined using gridded data, then the data must be in DHI's .dfs2 file format.

The easiest way to create the .dfs2 file is to use the Create... button, which creates a new grid with the proper default values and attribute type. You can then edit this grid in the MIKE Zero Grid Editor, which can be accessed using the Edit... button.

Alternatively, a .dfs2 file can be created using the Grid Series editor, which can be accessed by clicking on File|New in the pull-down menu, or using the New File icon,  , in the toolbar, and then selecting Grid Series.



If you create the file from these tools you must be careful to ensure that the EUM Data Type matches the parameter that you are creating the file for. For more information on the EUM data types, see EUM Data Units.

The grid for the .dfs2 file does not have to be the same as the numerical model grid. However, if the grids are not subsets of one another then the grids will be interpolated using the bilinear interpolation during the pre-processing stage.

The parameter grid and the model grid are aligned with one another if the parameter grid or the model grid contain an even multiple of the other grid's cells. For example, if the parameter grid was two times finer, then every model grid cell must contain exactly four parameter grid cells.

If the grids are aligned then the parameter grid will be averaged to the model grid during the pre-processing stage. However, in some cases it does not make sense to average parameter values. For example, Van Genuchten soil parameters cannot really be averaged, since they are a characteristic of the soil. In such cases, you should ensure that the model grid and the parameter grid file are identical.

10.4.1 Stationary Real Data

Spatially distributed Real parameters, such as conductivity or topography, can be defined in three ways, namely they can be defined as a uniform (global) value or they may be distributed and defined using either gridded data (.dfs2 file), GIS points and polygons (ArcView .shp file), or irregularly distributed point data (x, y, value coordinate file).

It does not make sense to interpolate some parameters to the model grid. In such cases, the use of line and point data should be avoided.

Uniform

A uniform, global value means that all the grid cells in the model will have the same value.

GIS point and line data or Distributed point data

If the parameter is defined using irregularly distributed point data or an ArcView shape (.shp) file, then the data will be interpolated to the model grid during the pre-processing stage, using the interpolation method selected.

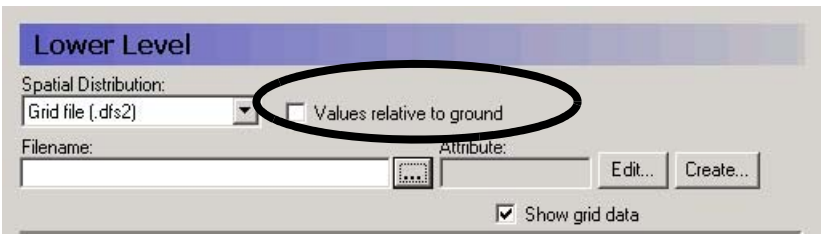
The following interpolation methods are included:

- Bilinear Interpolation *(p. 169)*, or
- Triangular Interpolation *(p. 172)*
- Inverse Distance Interpolation *(p. 173)*.

It does not make sense to interpolate some parameters to the model grid. In such cases, the use of line and point data should be avoided.

Elevation Data

Elevation data, such as Layer elevations, is handled exactly the same as all other Stationary Real Parameters, except that the value may be optionally specified as a depth below the ground surface rather than absolute elevation above the datum.



Note: The value must be negative if it is below the ground level.

Tip: The current tools do not allow you to specify a polygon shape file with real values. However, this would be desirable in some cases, such as when implementing Manning's M values based on vegetation distributions. A trick to get around this limitation is the following:

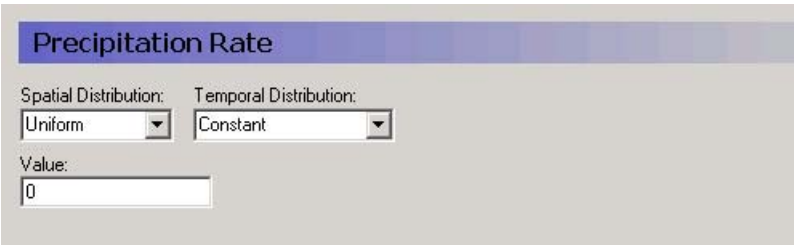
1. Temporarily assign an integer grid code to each of the polygons.
2. Specify this file as an input file for one of the data items that needs integer grid codes, such as drain codes.
3. Right click on the map that will be displayed and save the map view to a dfs2 file
4. Open this dfs2 file in the grid editor and use the grid editor tools to replace the integer values with real values
5. In the Grid Editor, change the EUM unit to the appropriate value
6. Save the file and then load it into the Data item for which you wanted it.

10.4.2 Time-varying Real Data

If the time-varying Real parameter does not vary spatially then the parameter must be defined as Global with either a Fixed or Time-varying value (see Uniform + Constant and Uniform + Time Varying).

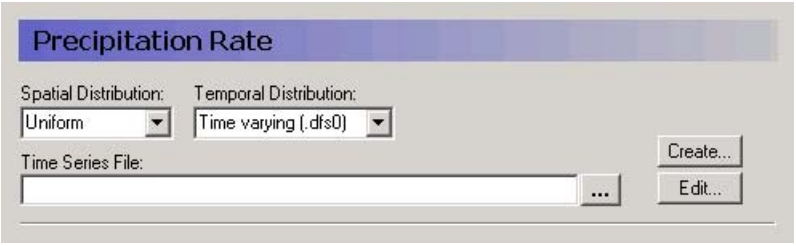
Often, time-varying data, such as precipitation rate, are spatially distributed using measurement stations, which in the model are translated into model zones using, for example, Thiessen polygons. In this case, each station is associated with a .dfs0 time series file that contains the time series of precipitation rate. Sub area-based zones are defined using Integer Grid Codes in either a .dfs2 file as Grid Codes, or in a Shape (.shp) file as polygons with an Integer Code (see Sub area-based + Grid Codes or Polygons).

Uniform + Constant



The parameter Value will be assigned to every cell in the model or layer as appropriate and will remain constant throughout the simulation.

Uniform + Time Varying



The time series in the .dfs0 file will be assigned to every cell in the model or layer as appropriate.

Sub area-based + Grid Codes or Polygons

Sub area-based time varying data means that the model domain is divided into zones that are defined by an Integer Grid Code.

If a .dfs2 file is used, then the Integer Grid Codes are defined on a regular grid, which is interpreted to the model grid during the Pre-processing stage.

If the Integer Grid Codes are defined using polygons then you must supply an ArcView .shp file containing polygons each with an Integer Grid Code. The item **Fill Gaps with**: allows you to define the Integer Grid Code to use in the event that a cell is not included within one of the polygons.

Once the file containing Integer Grid Codes has been defined, a new level in the data tree will appear below the current level, containing one entry for every unique Integer Grid Code in the file.

On this level, you must then supply a time series values for every Integer Grid Code. However, the time series can also be fixed, in the sense that a constant value over time is used. This makes it easy to use detailed time series for some zones and constant values for zones where little information exists.

The time series dialogue itself includes two graphical views. The upper graphic displays the time series that is being applied and the lower graphic shows where the time series will be applied.

10.4.3 Integer Grid Codes

The dialogues for Integer Codes function essentially same as those for Stationary Real Data, except that interpolation does not make sense for integer grid codes.

If Integer Grid Codes are being used to assign Model Properties, such as soil profiles or time series, then new sub-branches will appear in the data tree corresponding to the number of unique Integer Grid Codes in the .dfs2 file.

Uniform Value

A Uniform, global value means that all the grid cells in the model will have the same value. Thus, all cells would belong to the same zone.

Grid File (.dfs2)

If the Integer Code is defined using a grid file, then the Integer Code is defined on a grid. This grid may be different than the numerical model grid. However, the grids must be subsets of one another. That is, the Integer Code grid and the model grid must be aligned with one another and the Integer Code grid or the model grid must contain an even multiple of the other grid's cells. For example, if the Integer Code grid was two times finer, then every model grid cell must contain exactly four Integer Codes.

Normally, the Integer Code will be assigned to the model grid based on the most prevalent Integer Code in the cell. However, this can lead to problems when the a particular code is both infrequent and widely dispersed. For example, if a model area contained many small wetland areas that were much smaller than a grid cell.

For this reason, a bookkeeping count is kept of the assignments to reduce any bias in the assignment of Integer Codes and ensure that less frequently occurring Integer Codes will be represented in the resulting model grid. For example, if their were two different Integer Codes, A and B, used in the model and A always occurred more frequently in each model cell, the bookkeeping count would ensure that B would actually be assigned to some of the model cells. The final frequency of occurrence of the Integer Codes in the model cells would reflect the underlying frequency of occurrence of the Integer Codes. That is, if A occurred twice as often as B, the model grid would also contain twice as many A's as B's.

Thus, in our widely dispersed wetland example, if every model grid cell contained 9 Integer Codes for Land Use, and 1/9 of the Land Use grid codes were for wetlands, then every ninth Model Cell would be assigned a Land Use grid code for wetlands.

Polygons

In the current version, only some of the parameters are set up to accept .shp file polygons. Currently, .shp file polygons are only allowed in:

- Model Domain and Grid *(p. 228)*
- Precipitation Rate *(p. 234)*
- Vegetation *(p. 249)*,
- Reference Evapotranspiration *(p. 238)*
- UZ Soil Profile Definitions *(p. 293)*,
- SZ Internal boundary conditions *(p. 327)*, and
- Horizontal Extent *(p. 356)* of SZ Lenses.

Note: The Horizontal Extent *(p. 356)* of SZ Lenses accepts polygons, but the dialogue is still set up for point/line .shp files and an error is given in the Data Verification window.

Model grid codes are assigned based in which polygon the centre of the cell is located in.

10.5 Interpolation Methods

The gap filling is based on the concept that we have to calculate the depth in the point (x_c, y_c). We define this as the function $z_c = f(x_c, y_c)$. If we place our self in this point, we can divide the world up into four quadrants Q1 - Q4. From here it's a matter of finding some points from the raw data set relatively close to this point. The search radius for all possible techniques can be entered - in grid cell distance. Points outside this distance will never be taken into account.

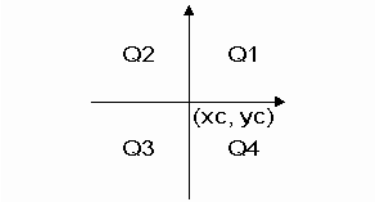


Figure 10.1 Definition of quadrants

10.5.1 Bilinear Interpolation

This technique finds four points from the raw data set - one in each quadrant. The search is done in the following way. A mask of relative indices is created. The cells in this mask are sorted according to the distance. For the quadrant Q1 the cells are sorted in the following way, the grid point it self being excluded.

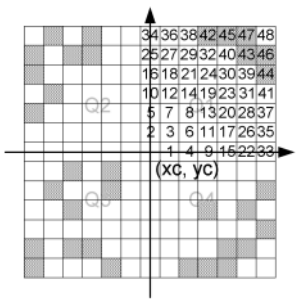


Figure 10.2 Illustration of the neighbouring grid cells being sorted

Note that the grid cells with a crosshatch pattern contain raw data points. When the closest raw data point in each quadrant is found, we have four points that form a quadrangle. This quadrangle contains the centre point, where we want to calculate the z-value. This is illustrated on Figure 10.3.

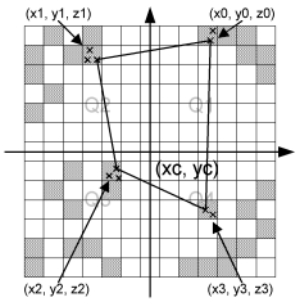


Figure 10.3 Illustration of the closest raw data points in each quadrant.

Note that each grid cell might contain more raw data points. If this is the case, the closest of these is chosen. We now have an irregular quadrangle, where the elevation is defined in each vertex. We need to compute the elevation in (x_c, y_c). If we transform our quadrangle into a square, we can perform bilinear interpolation. This is illustrated on Figure 10.4.

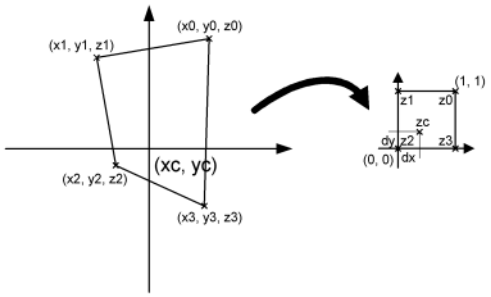


Figure 10.4 Illustration of bilinear interpolation.

First the interpolation requires the transformation from quadrangle to a normalized square. This is done in the by computing 8 coefficients in the following way:

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

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

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

where the coefficients are

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

Solving equation (10.2) gives us *dx*.

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

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

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

or

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

Choosing between (10.5) and (10.6) is done in such a way, that division by zero is avoided. (x_c, y_c) has been mapped to (dx, dy). The task was to compute the elevation in the point (x_c, y_c) and this is done in the following way using regular bilinear interpolation:

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

If less than four points are found (if one or more quadrants are empty), the double linear interpolation is replaced with reverse distance interpolation (RDI). This is done according to the following scheme:

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

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

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

The method works fairly efficiently, but it has one drawback. The quadrant search is heavily dependent on the orientation of the bathymetry. If the bathymetry is rotated 45 degrees 4 completely different points might be used for the interpolation. For this reason there is also a Triangular interpolation method, which can be used, and this method should be direction independent.

10.5.2 Triangular Interpolation

As mentioned previously the ‘Bilinear Interpolation’ is dependent on the orientation of the bathymetry. The ‘Triangular Interpolation’ is made as an answer to this problem. First the closest point to (x_c, y_c) is found. The following figure shows this:

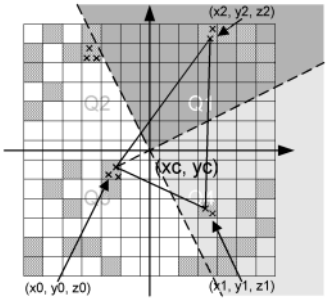


Figure 10.5 Illustration of triangular interpolation

In this example the point (x_0, y_0, z_0) is the closest point. When this point is identified, two quadrants are identified – indicated by the light grey and the dark grey areas. The closest point in these two quadrants are then found. They can be seen on the figure as (x_1, y_1, z_1) and (x_2, y_2, z_2). The interpolation is then done in two steps. First the coefficients describing the plane defined by the 3 found points are computed:

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

And secondly the actual interpolation is done:

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

If less than 3 points are found, reverse distance interpolation (RDI) is used. The triangular interpolation is more time consuming due to the more complex direction independent search, but better end results should be achieved with this method.

10.5.3 Inverse Distance Interpolation

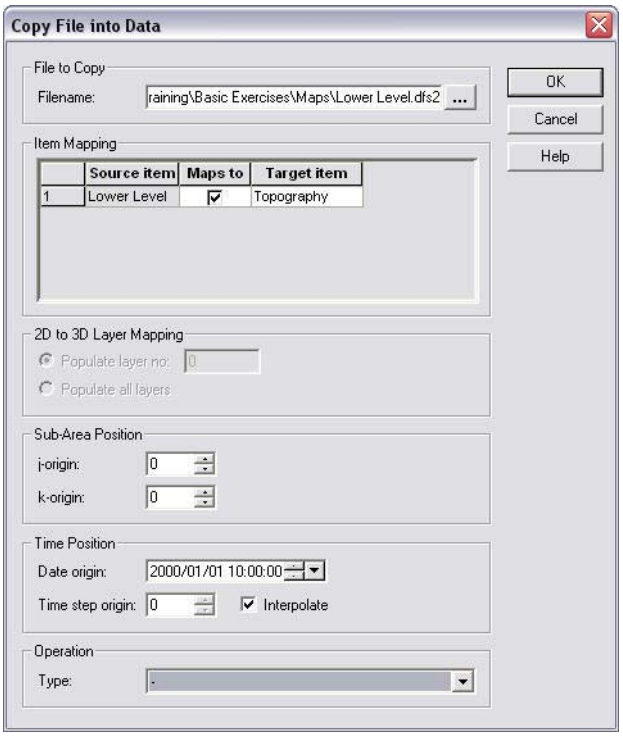
The two inverse distance interpolation methods both use the nearest point in each quandrant to calculate the interpolated value, weighted by the distance or the distance squared respectively.

10.6 Performing simple math on multiple grids

In the upper menu of the Grid Editor, under tools, there is an item called **Copy File into Data**.



If you select this item then a dialogue appears where you can insert an existing dfs2 or dfs3 file into the current dfs2 or dfs3 file that you are editing in the Grid editor.



Alternatively, you can define an operation that you want to do with the file. For example, if you were editing a topography file, you could subtract all of the values in a lower elevation file, to obtain a thickness distribution for a layer.

The principle advantage of this tool, is that time varying dfs2 and dfs3 files can be manipulated. However, if the operations are complex, but not time varying then

Target file

The target file is the current file you are editing in the Grid editor. The operations that you do are performed on the target file. So, if you don't want to edit the target file, copy it to a new name first and edit the copy.

File to Copy

The top section of the dialogue is the name of the source file that you want to insert into, subtract from, add to, etc. the target file.

Item mapping

If the target file or the source file has more than one item in it, then all of the items will be listed here and you will be able to choose whether or not to map the various items to one another.

2D to 3D Layer Mapping

If you are mapping a 2D dfs2 file into a 3D dfs3 file, then you can choose to map all of the layers or only a single layer.

Sub-area position

You select to map the source file onto the target file starting at a different location than the origin. In this case, you must specify the coordinates in the target grid where the origin of the source grid should be positioned. For example, if you have a 20x20 grid and we wish to copy data into the 4x4 rectangle given by the four nodes (10,14), (13,14), (13,17) and (10,17), then you should select a 4x4 grid file and specify j-origin=10 and k-origin=17. **Note: the Grid editor starts its nodal numbering at 0,0**.

Time Position

The source grid and target grid do not have to have equal time steps or the same time origin. In this section of the dialogue, you can specify the time at which the source grid should be added to the target grid. In this way, you can add additional time steps to the end of a time varying dfs2 file, or insert hourly information into a monthly time series, for example.

Operation

Finally, you can specify how the source grid file should interact with the target file.

Copy - all values are copied such that they replace the existing data in the data set

Copy if target differs from delete value - values in the source file will be copied into the target file, only if the target value is a delete value

Copy if source differs from delete value - values in the source file will be copied into the target file, only if both the source value and the target values are not delete values

Copy if source AND target differs from delete value - values in the source file will be copied into the target file, only if the source value is not a delete value

+ - the source values will be added to the target values

- - the source values will be subtracted from the target values

***** - the source values will be multiplied by the target values

/ - the source values will be divided by the target values

10.7 Performing Complex Operations on Multiple Grids

In the Toolbox, under MIKE SHE/Util, there is a Grid calculator tool, which allows you to perform complex operations on .dfs2 grid files. The only real limitation is that the grid files must have the same grid dimensions. Thus, this tool is much more flexible than the grid operations available in the Grid Editor. With this tool, you can make complex chains of operations, save the setup, and create batch files. You can also run it from a command line, which can

save you a lot of time if you are doing the same operation many times.or after each simulation

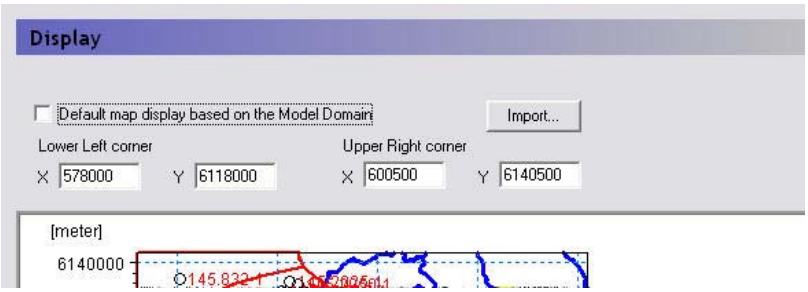
The Grid Calculator works like a wizard, with Next and Back buttons to move between dialogues.

11 Setup Data Tab

- This chapter is organized around the Setup Data Tree. For each branch in the data tree there is a corresponding subsection. The main sections include:
- **Display** *(p. 180)* - display of map overlays
- **Simulation Specification** *(p. 186)* - control and selection of water movement engines
- **Water Quality Simulation Specification** *(p. 207)* - control and selection of water quality engines
- **Species** *(p. 223)* - specification of species for water quality simulations
- **Water Quality Sorption and Decay** *(p. 225)*
- **Model Domain and Grid** *(p. 228)* - definition of model extent and grid
- **Sub-catchments** *(p. 230)* - definition of catchment boundaries for lumped parameter water movement engines
- **Topography** *(p. 232)* - specification of land surface elevation
- **Climate** *(p. 233)* - specification and extent of climate measurements, such as precipitation and evapotranspiration
- **Land Use** *(p. 248)* - specification of vegetation and irrigation
- **Rivers, Lakes and Sewers** *(p. 268)* - link to a MIKE 1D channel flow or sewer model
- **Overland Flow** *(p. 271)* - specification of 2D overland sheet flow parameters for both water movement and water quality
- **Unsaturated Zone** *(p. 288)* - specification of 1D unsaturated zone columns
- **Groundwater table for lower UZ boundary** *(p. 304)* - specification of static lower boundary condition for unsaturated flow, if saturated zone not included
- **Saturated Zone** *(p. 304)* - specification of 3D saturated zone parameters for both water movement and water quality
- **Sources** *(p. 339)* - location and extent of solute sources for water quality simulation
- **Storing of Results** *(p. 341)* - output selection for calibration time series and gridded data

- **Extra Parameters** *(p. 353)* - extra input data for model options not yet available in the data tree

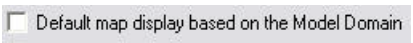
11.1 Display



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

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

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



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

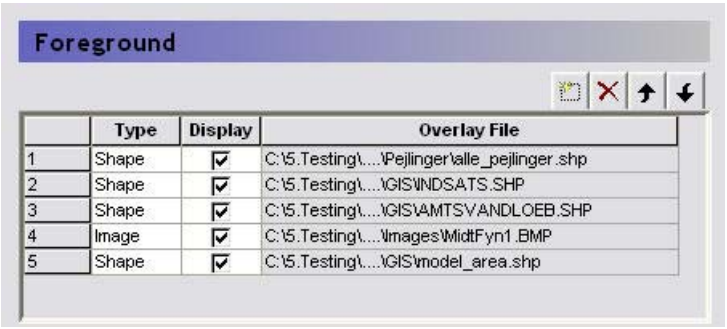
Import - The Import button allows you to read the coordinate extents from a map file, such as a .dfs2 file or a .shp file.

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

Related Items:

- Model Domain and Grid *(p. 228)*

11.1.1 Foreground/Background



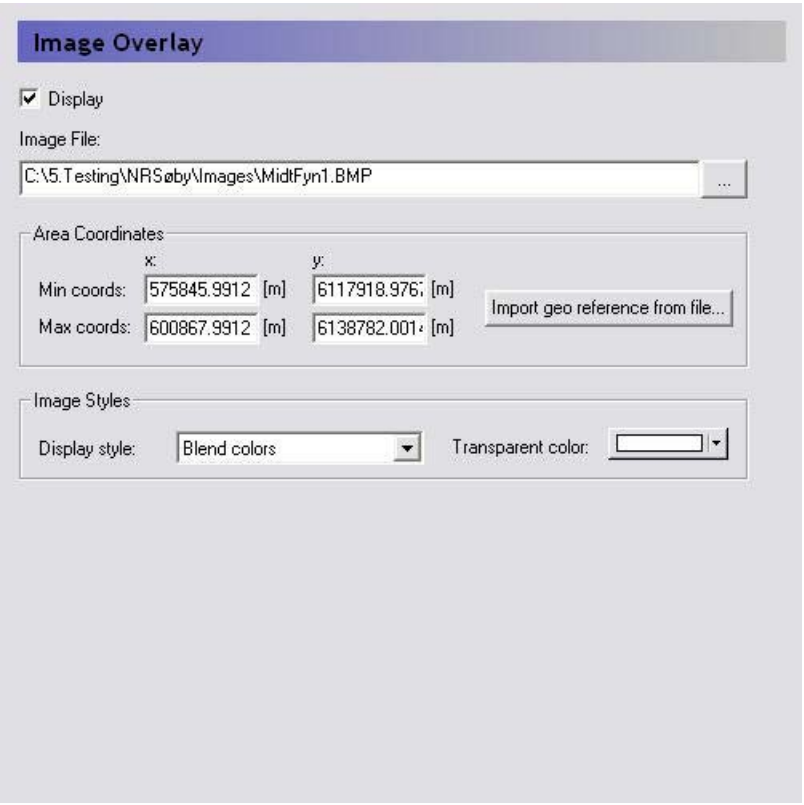
The Foreground and Background items are used to add map overlays to the map view. The table gives you an overview of the defined overlays and allows you to add, delete and hide overlays.

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

The available map types includes,

- ESRI shape (.shp) files,
- Grid (.dfs2) files,
- Image (.bmp, .gif and .jpg) files,
- MIKE Hydro River (.mhydro), and
- MIKE SHE well database (.wel) files.

11.1.2 Image Overlays



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

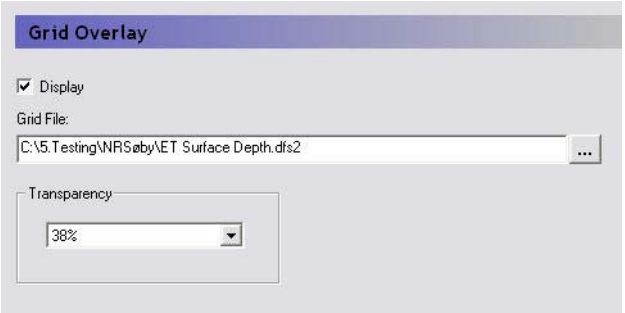
Area Coordinates - Since image files do not contain geographic information, you must specify the spatial location of the image. This is done by specifying the map coordinates of the lower left corner of the image (Minimum X and Y) and the upper right corner of the image (Maximum X and Y).

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

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

Blend Colours selection., in which case the Transparent colour option is not used.

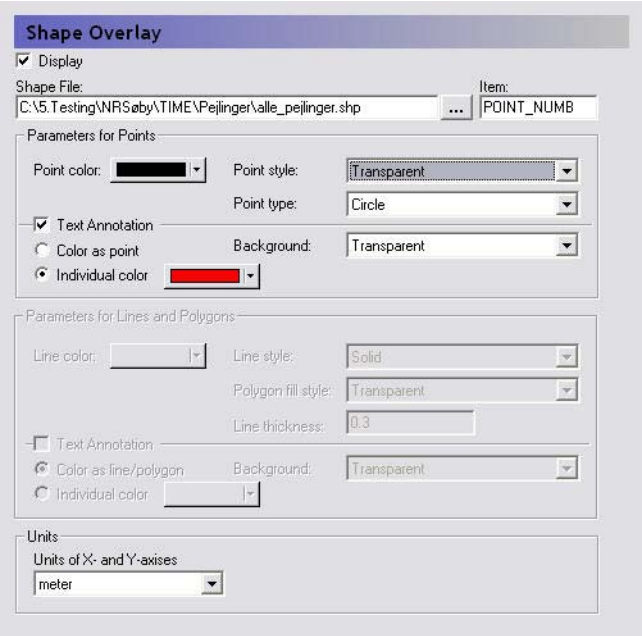
11.1.3 Grid Overlays



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

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

11.1.4 Shape Overlays



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

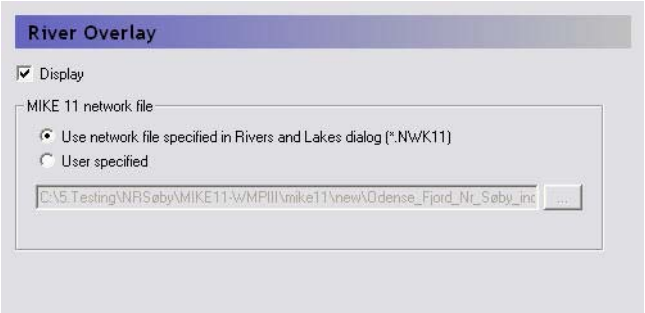
Shape File and Item - In the file browser dialogue you can select from the available items in the .shp file.

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

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

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

11.1.5 River Overlays

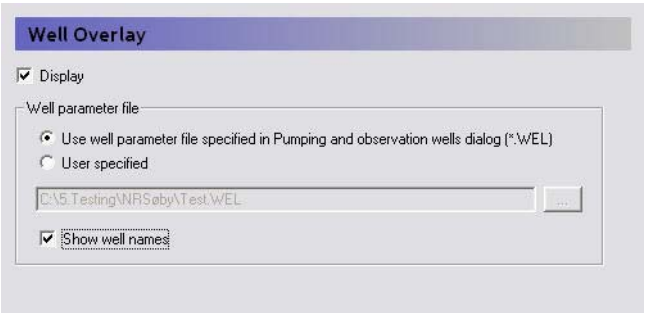


If you want to display a MIKE Hydro River network in your map views, then you can add a River Overlay. By default, the river network defined in the Rivers, Lakes and Sewers dialogue is displayed. If you would rather display a different river network, for example, an overview network with fewer branches,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

Related Items:

- Rivers, Lakes and Sewers *(p. 268)*

11.1.6 MIKE SHE Well Database Overlays



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

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

Related Items:

- Pumping Wells *(p. 338)*

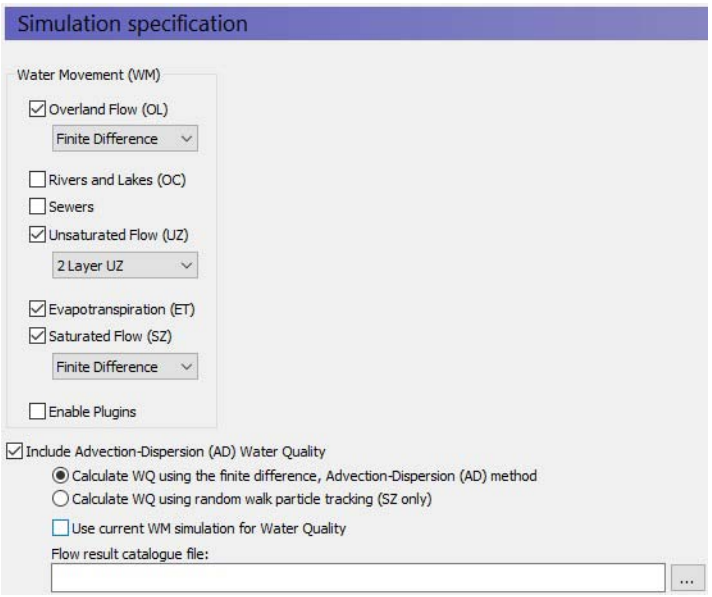
11.1.7 Current Layer



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

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

11.2 Simulation Specification



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

- Overland Flow - see Overland Flow - Technical Reference *(p. 515)*)
  - Finite Difference Method *(p. 515)*
  - Simplified Overland Flow Routing *(p. 533)*
- Rivers and Lakes - see Channel Flow - Technical Reference *(p. 471)*

- Unsaturated Flow - see Unsaturated Zone - Technical Reference *(p. 575)*
  - 1D Richards Equation *(p. 576)*
  - simplified 1D Gravity Flow *(p. 587)*
  - Two-Layer Water Balance *(p. 589)* for shallow water tables
- Evapotranspiration - see Evapotranspiration - Technical Reference *(p. 437)*)
- Saturated Flow - see Saturated Flow - Technical Reference *(p. 623)*)
  - 3D Finite Difference Method *(p. 623)*
  - Linear Reservoir Method *(p. 638)*

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

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

Water Quality options

Include Advection Dispersion (AD) Water Quality

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

PT versus AD

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

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

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

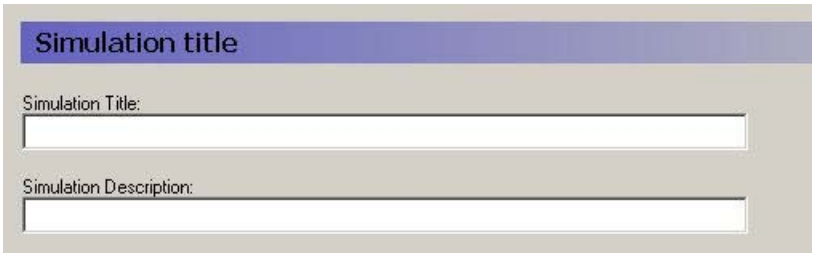
Use Current WM simulation for Water Quality

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

Related Items:

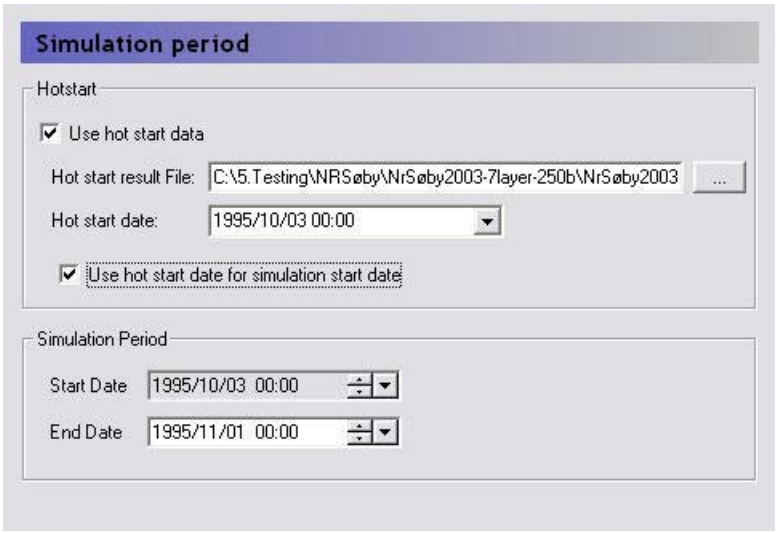
- Working with Solute Transport - User Guide *(p. 715)*
- Particle Tracking-Reference *(p. 737)*
- Advection Dispersion - Reference *(p. 673)*

11.2.1 Simulation Title



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

11.2.2 Simulation Period



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

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

However, the simulation can be started from a hot start file. A hot start file is useful for simulations requiring a long warm up period or for generating initial conditions for scenario analysis. To start a model from a previous model run, you must first save the hot start data, in the Storing of Results *(p. 341)* dialogue.

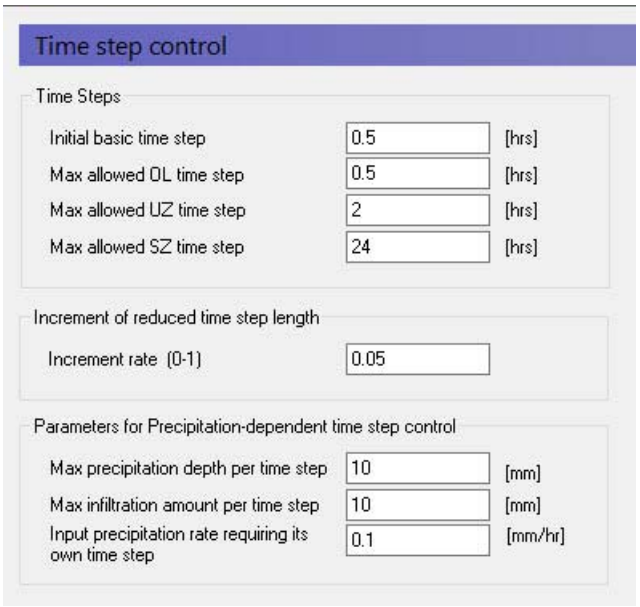
Hot start date - The hot start information is saved at specified intervals and the list of hot start dates is automatically filled in from the hot start file.

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

Related Items:

- Storing of Results *(p. 341)*

11.2.3 Time Step Control



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

Table 11.1

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

Time Steps

Initial basic time step This is the initial time step for the Basic time step,

unless the component's maximum allowed time step is less than the initial basic time step.

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

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

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

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

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

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

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

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

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

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

Increment of reduced time step length

Increment rate This is a factor for both decreasing the time step length and increasing the time step length back up to the maximum time step, after the time step has been reduced. See Parameters for Precipitation-dependent time step control *(p. 192)* for more details on when and how the time step is changed. A typical increment rate is about 0.05.

Parameters for Precipitation-dependent time step control

Periods of heavy rainfall can lead to numerical instabilities if the time step is too long. To reduce the numerical instabilities, the a time step control has been introduced on the precipitation and infiltration components. You will notice the effect of these factor during the simulation by suddenly seeing very small time steps during storm events. If your model does not include the unsaturated zone, or if you are using the 2-Layer water balance method, then you can set these conditions up by a factor of 10 or more. However, if you are using the Richards equation method, then you may have to reduce these factors to achieve a stable solution.

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

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

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

melting temperature and the wet snow storage is greater than 95% of the maximum wet snow storage fraction, then the potential snow release equals the total snow storage.

Input precipitation rate requiring its own time step If the precipitation rate [mm/hr] in the precipitation time series is greater than this amount, then the simulation will break at the precipitation time series measurment times. This option is added so that measured short term rainfall events are captured in the model. For example, assume you have hourly rainfall data and 6-hour time steps. If an intense rainfall event lasting for only one hour was observed 3 hours after the start of the time step, then MIKE SHE would automatically break its time stepping into hourly time steps during this event. Thus, instead of a 6-hour time step, your time steps during this period would be: 3 hours, 1 hour, and 2 hours. This can also have an impact on your time stepping, if you have intense rainfall and your precipitation measurements do not coincide with your storing time steps. In this case, you may see occasional small time steps when MIKE SHE catches up with the storing timestep.

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

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

Actual time step for the different components

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

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

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

until the maximum allowed time step is reached.

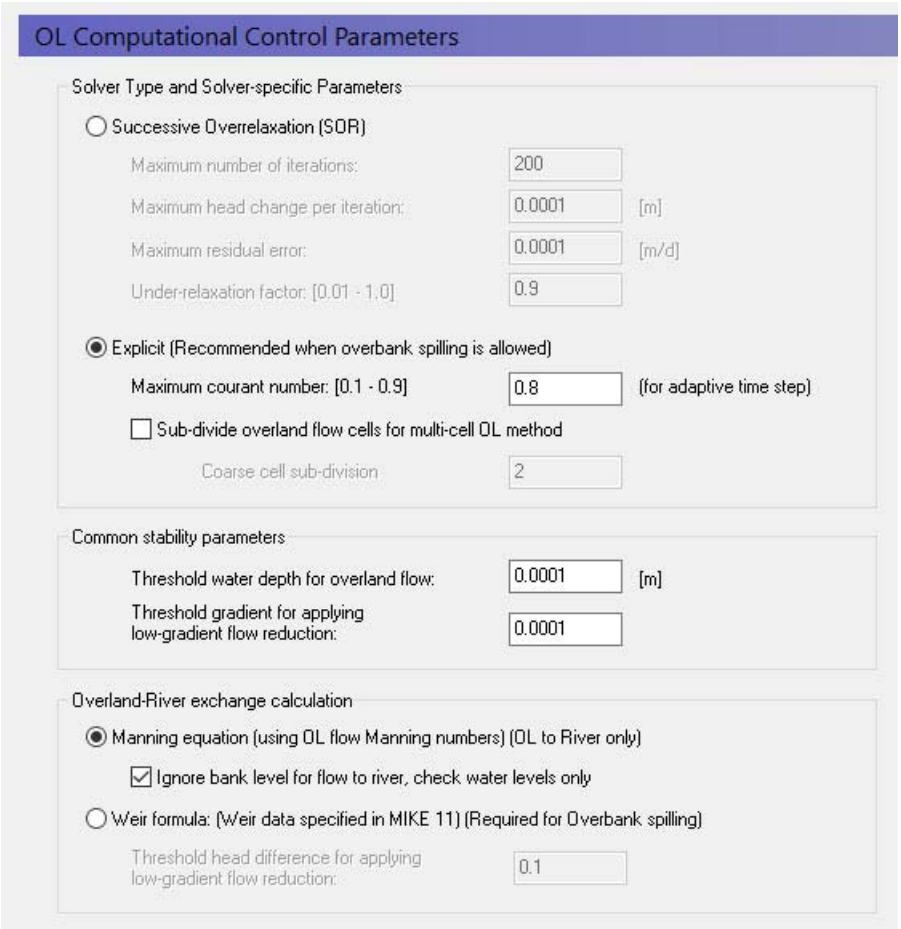
Relationship to Storing Time Steps

The Storing Time Step specified in the Detailed WM time series output *(p. 344)* dialogue, must also match up with maximum time steps. Thus,

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

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

11.2.4 OL Computational Control Parameters



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

Table 11.2

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

Table 11.2

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

Solver Type for Overland Flow

Overland flow can be solved using either using the implicit Successive Over-Relaxation (SOR) Numerical Solution *(p. 519)* or an Explict Numerical Solution *(p. 520)*).

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

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

SOR parameters

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

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

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

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

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

Explicit parameters

Maximum courant number The courant number represents the ratio of the speed of wave propagation to the grid spacing. In other words, a courant number greater than one would imply that a wave would pass through a grid cell in less than one time step. This would lead to severe numerical instabilities in an explicit solution. The courant number must be greater than 0.1 and less then 1.0. For a detailed discussion of the courant criteria see the Courant criteria *(p. 521)*.

Subdivide overland flow cells for multi-cell flow method The explicit overland flow solver includes an additional option for solving the water levels on a finer grid scale than the flow velocities. This method also modifies the intercell conductance to account for average flow area across the cell. See Multi-cell Overland Flow Method *(p. 529)* for more information.

Common stability parameters

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

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

The threshold depth for overland flow should not be confused with the Detention Storage *(p. 274)*. The detention storage is related to the

amount of water stored in local depression on the ground surface, which must be filled before water can flow laterally to an adjacent cell.

Threshold gradient for applying low gradient flow reduction In flat areas with ponded water, the head gradient between grid cells will be zero or nearly zero and numerical instabilities will be likely. To dampen these numerical instabilities in areas with low lateral gradients a damping function has been implemented. The damping function essentially increases the resistance to flow between cells. This makes the solution more stable and allows for larger time steps. However, the resulting gradients will be artificially high in the affected cells and the solution will begin to diverge from the Mannings solution. At very low gradients this is normally insignificant, but as the gradient increases the differences can become significant.

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

For a detailed discussion of the damping function see the Low gradient damping function *(p. 523)* or Threshold gradient for overland flow *(p. 546)*.

Overland River Exchange Calculation

Generally, Overland flow will discharge into the River Link if the water elevation in the cell is higher than the bank elevation of the River Link.

Mannings equation

The rate of discharge to the river is dictated by the Mannings calculation for overland flow. However, this flow is only one way, that is ponded water will flow to the river.

There is a tick box to defined the level used to control if flow occurs or not.

- If the tick box is ON, then ponded water will flow to the river if the water level in the cell is above the water level in the River Link (default).
- If the tick box is OFF, then ponded water will flow to the river only if the water level in the cell is above the BANK elevation in the River Link. This was the only option prior to Release 2017. So, to be compatible with older simulations, the tick box must be off.

Weir formula

If you want to include overbank spilling from the river to the overland grid cells, then you must use the weir formula, which provides a mechanism for

water to flow back and forth across the river bank. In this case, the bank is treated like a broad crested wier.

Note Whether or not to allow overbank spilling from the river to overland flow is set in the river model for each coupling reach. If you do not allow overbank spilling in MIKE+, then the overland river exchange is only one way, but uses the weir formula instead of the Manning's formula for calculating the amount of exchange flow.

If you do not use the overbank spilling option, then you can still use the flood inundation option to "flood" a flood plain. In this case, though, the flooding is not calculated as part of the overland flow, but remains part of the water balance of the river model. For more information on the flood inundation method see the section on Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes *(p. 492)*.

Threshold head difference for applying low gradient flow reduction

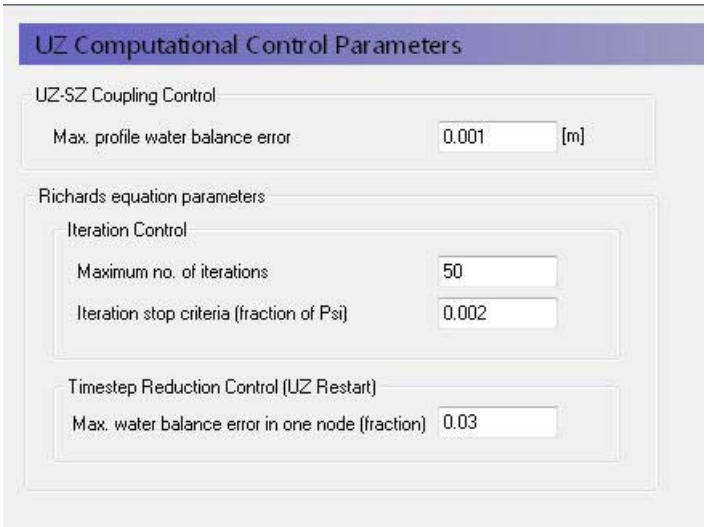
If the difference in water level between the river and the overland flow cell is less than this threshold, then the flow over the weir is reduced to dampen numerical instabilities. In this case, the same damping function is used as in low gradient areas. The damping function essentially increases the resistance to flow between the cell and the river link. This makes the solution more stable and allows for larger time steps. However, the resulting gradients will be artificially high in the affected cells and the solution will begin to diverge from the Mannings solution. At very low gradients this is normally insignificant, but as the gradient increases the differences can become significant.

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

Related Items:

- Low gradient damping function *(p. 523)*
- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*

11.2.5 UZ Computational Control Parameters



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

Table 11.3

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

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

UZ-SZ Coupling Control

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

Maximum profile water balance error The tolerance criteria in the UZ-SZ coupling procedure is the maximum allowed accumulated water bal-

ance error in one UZ column. If this value is exceeded the location of the groundwater table will be adjusted and additional computations in the UZ component will be done until this criteria is met. The recommended value is 1e-5 m or less. Details on this coupling procedure can be found in Coupling the Unsaturated Zone to the Saturated Zone *(p. 601)*.

Additional Richards Equation Control Parameters

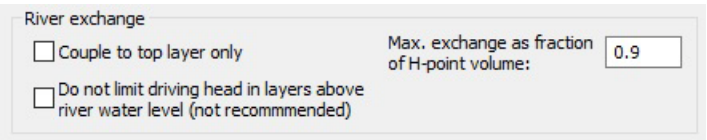
When the Richards Equation method is used, you can specify:

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

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

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

11.2.6 SZ Computational Control Parameters



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

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

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

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

Steady-state vs Transient Simulations

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

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

Iteration Control

Table 11.4

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

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

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

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

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

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

Sink de-activation in drying cells

Table 11.5

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

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

River exchange

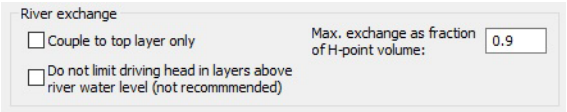


Table 11.6

Variable	Units
Maximum exchange as fraction of H-point volume	-

Max. exchange as fraction of H-point volume - This represents the maximum amount of water that can be removed from the river during one SZ time step. Removing larger amounts of water could effectively dry out the river. If this occurs, then the SZ solver will issue a warning and only this fraction of water will be removed, which prevents rapid drying out of the river during a single time step.

Couple to top Layer only - Calculate groundwater river exchange for the top layer only

Do not limit driving head in layers above river water level - If the river water level drops below the bottom of the layer, the driving head physically does not depend on the river water level. Instead it is the difference between the head in the layer and the layer bottom. Mainly for backwards compatibility with older models' results this can be switched on so that the full head difference between groundwater and river will be used.

Pre-conditioned Conjugate Gradient

Advanced Settings

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

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

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

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

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

Successive Over-relaxation



Table 11.7

Variable	Dimensions
Relaxation Factor	-

Over-relaxation

Relaxation factor - The speed of convergence also depends on the relaxation coefficient. Before you set up your model for a long simulation, you

should test the iteration procedure by running a few short simulations with different relaxation coefficients. This coefficient must be between 1.0 and 2.0, with a typical value between 1.3 and 1.6.

11.3 Python Plugins

The MShePy module, which enables the use of Python scripting with MIKE SHE. This includes:

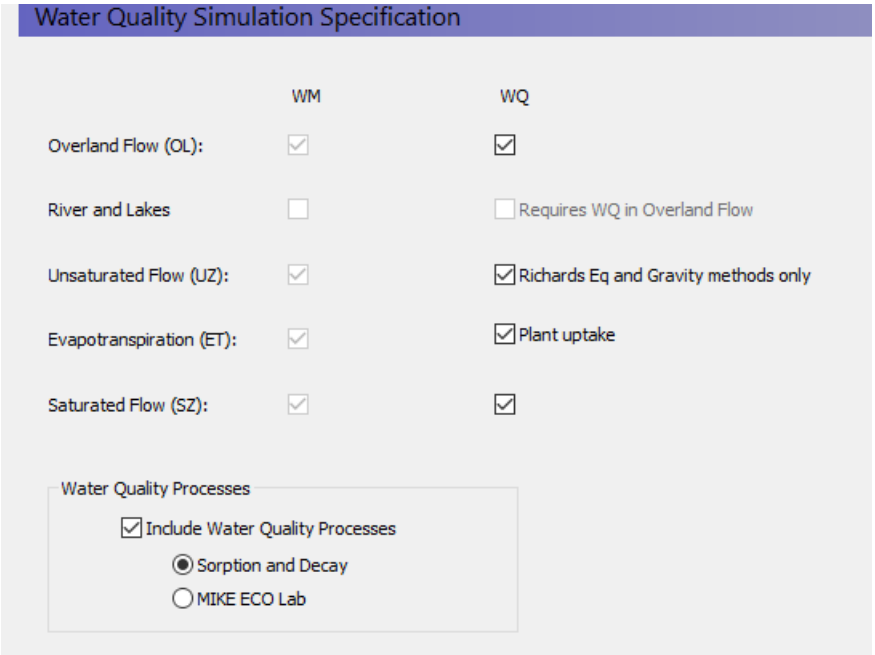
1. running Python plugins during a MIKE SHE simulation;
2. running MIKE SHE simulations by Python scripts.

In the first case, the user can add plugins to a MIKE SHE model. A Plugin consists of a collection of code functions in Python performing custom functionalities. During preprocessing, the Python plugins are attached to the model and once the simulation is started, functions (or "slots") in the plugins will be connected to signals (e.g. before a time step). Signals will be emitted by MIKE SHE and trigger execution of one or more custom functions defined in the plugin. Once the function has finished to perform arbitrary tasks (e.g. exchange data with MIKE SHE), MIKE SHE will continue the normal execution.

Related Items:

- Python Scripting in MIKE SHE *(p. 383)*

11.4 Water Quality Simulation Specification



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

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

Solute transport in surface water bodies is calculated by MIKE 1D. If solute exchange to the river is to be simulated, then the water quality must also be simulated in overland flow.

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

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

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

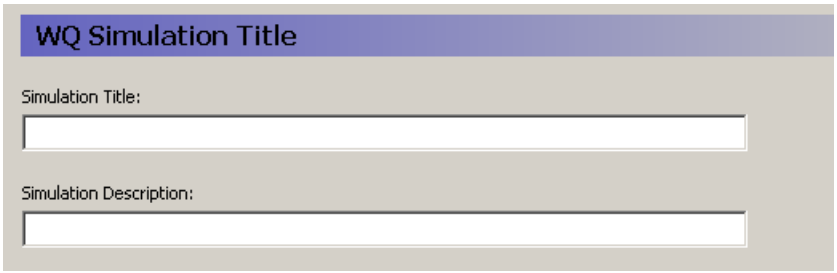
Sorption and Decay -

MIKE ECO Lab - MIKE ECO Lab options (see Working with MIKE ECO Lab in MIKE SHE - User Guide *(p. 725)*)

Related Items:

- Working with Solute Transport - User Guide *(p. 715)*
- Advection Dispersion - Reference *(p. 673)*
- Reactive Transport - Reference *(p. 699)*

11.4.1 WQ Simulation Title



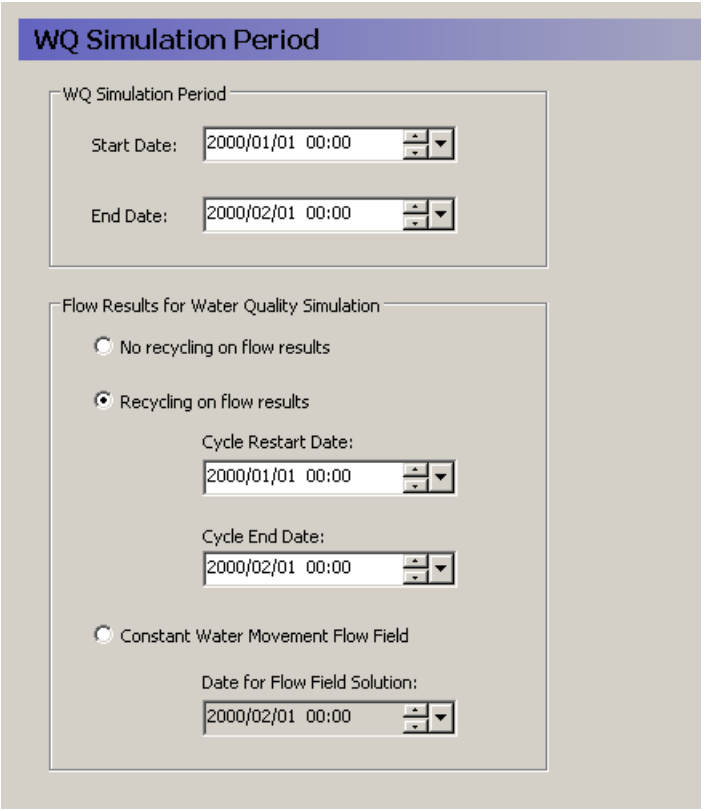
WQ Simulation Title	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

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

Related Items:

- Working with Solute Transport - User Guide *(p. 715)*
- Advection Dispersion - Reference *(p. 673)*

11.4.2 WQ Simulation Period



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

WQ Simulation Period

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

Flow Results for Water Quality Simulation

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

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

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

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

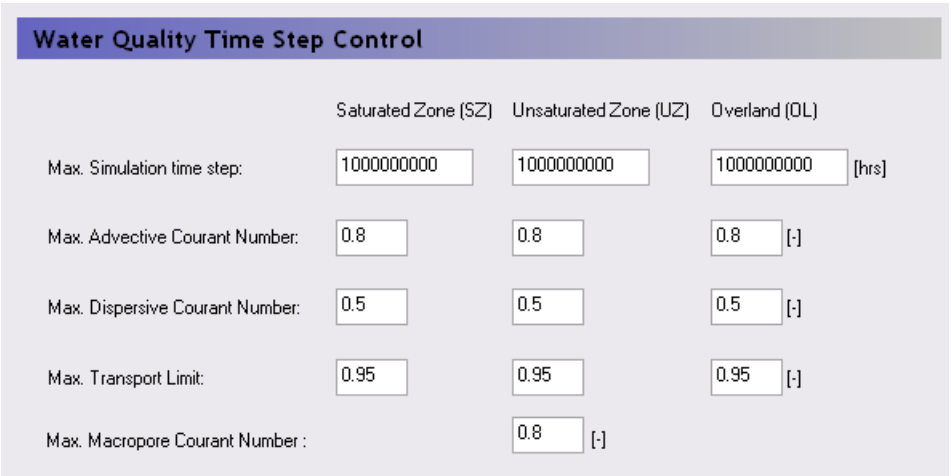
For example, you may have a two-year water movement simulation but you may want to simulate water quality for 10 years. To do this, you would specify the start and stop dates of the part of the water movement simulation that you want repeated. If you want to repeat the whole water movement simulation,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

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

Related Items:

- Working with Solute Transport - User Guide *(p. 715)*
- Advection Dispersion - Reference *(p. 673)*

11.4.3 Water Quality Time Step Control



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

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

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

Stability Criteria

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

Max. Advective Courant Number - The advective courant number represents the ratio of cell size to the time it would take a particle to move across a cell. This criteria is likely to be controlling if your flow velocities are high, or your dispersivity values are very low or zero. The

default value is 0.8. If your actual time step is being controlled by this criteria, then you could increase it to make the simulation run faster. However, you will need to check to make sure the simulation has converged properly and that the mass balance is reasonable.

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

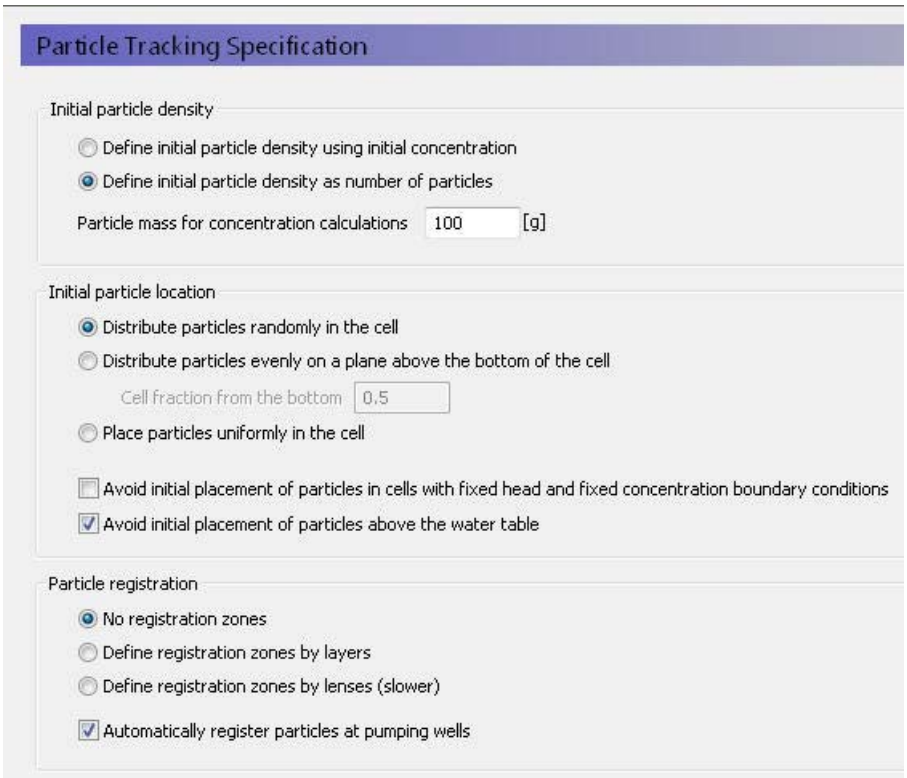
Max. Transport Limit - The transport limit restricts the fraction of the total amount of mass that can leave the cell in one time step. The default value 0.95, which usually does not limit the time step.

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

Related Items:

- Working with Solute Transport - User Guide *(p. 715)*
- Advection Dispersion - Reference *(p. 673)*

11.5 Particle Tracking Specification



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

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

Initial particle density

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

Define initial particle density as initial concentration - The initial concentration is defined for each SZ layer. See Initial (secondary) concentration *(p. 329)*. The number of particles in each cell is calculated by

dividing the mass of solute in the cell by the particle mass specified in the text box below.

Define initial particle density as number of particles - If this option is selected, then an Initial number of particles per cell *(p. 330)* item is added to the data tree for each SZ layer.

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

Initial particle location

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

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

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

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

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

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

Particle registration

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

zone. If the particle passes through the registration zone, then this will be recorded.

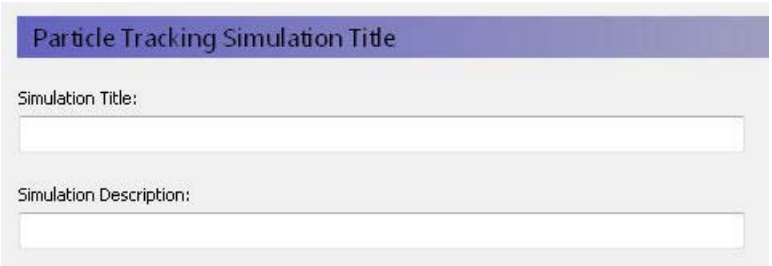
You can define the registration zones by numerical layer, or you can define them be lenses similar to the way you define geologic lenses. In both cases additional items are added to the SZ data tree. Defining the registration codes by lense is slower, because the PT engine has to continually check to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

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

Related Items:

- PT Registration Codes/Lenses *(p. 330)*
- Particle Tracking-Reference *(p. 737)*

11.5.1 Particle Tracking Simulation Title



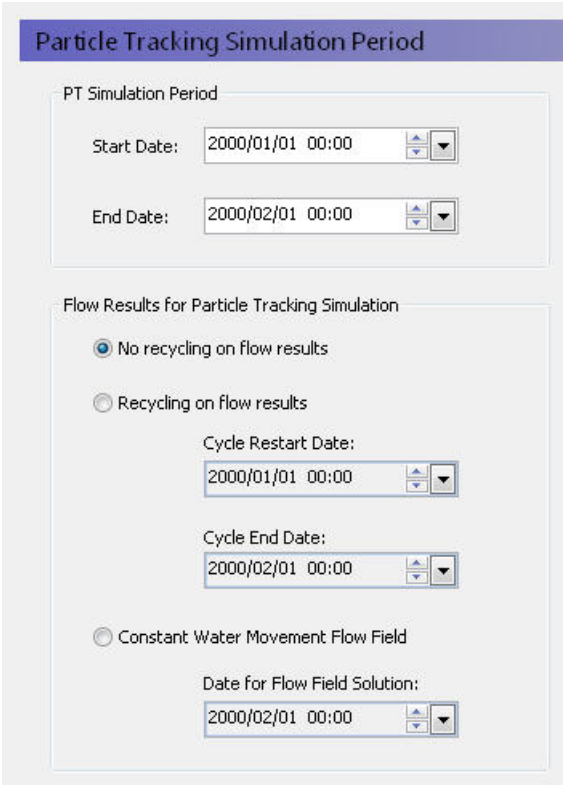
Particle Tracking Simulation Title	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue and the Random Walk Particle Tracking suboption is selected.

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

Related Items:

- Working with Solute Transport - User Guide *(p. 715)*
- Particle Tracking-Reference *(p. 737)*

11.5.2 Particle Tracking Simulation Period



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

PT Simulation Period

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

Flow Results for Particle Tracking Simulation

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

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

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

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

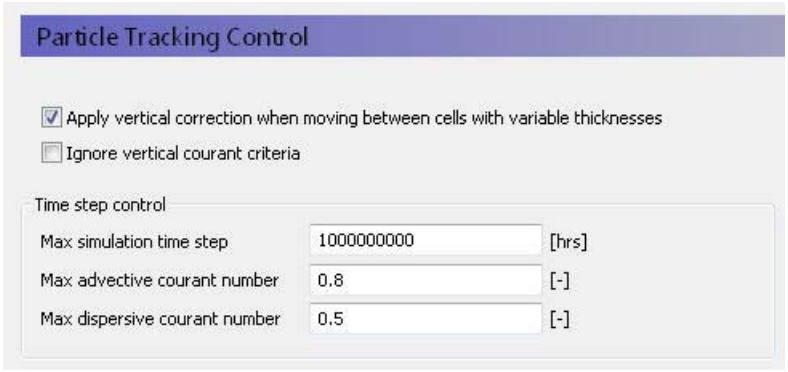
For example, you may have a two-year water movement simulation but you may want to simulate water quality for 10 years. To do this, you would specify the start and stop dates of the part of the water movement simulation that you want repeated. If you want to repeat the whole water movement simulation,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

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

Related Items:

- Working with Solute Transport - User Guide *(p. 715)*
- Particle Tracking-Reference *(p. 737)*

11.5.3 Particle Tracking Control



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

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

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

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

Time step control

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

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

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

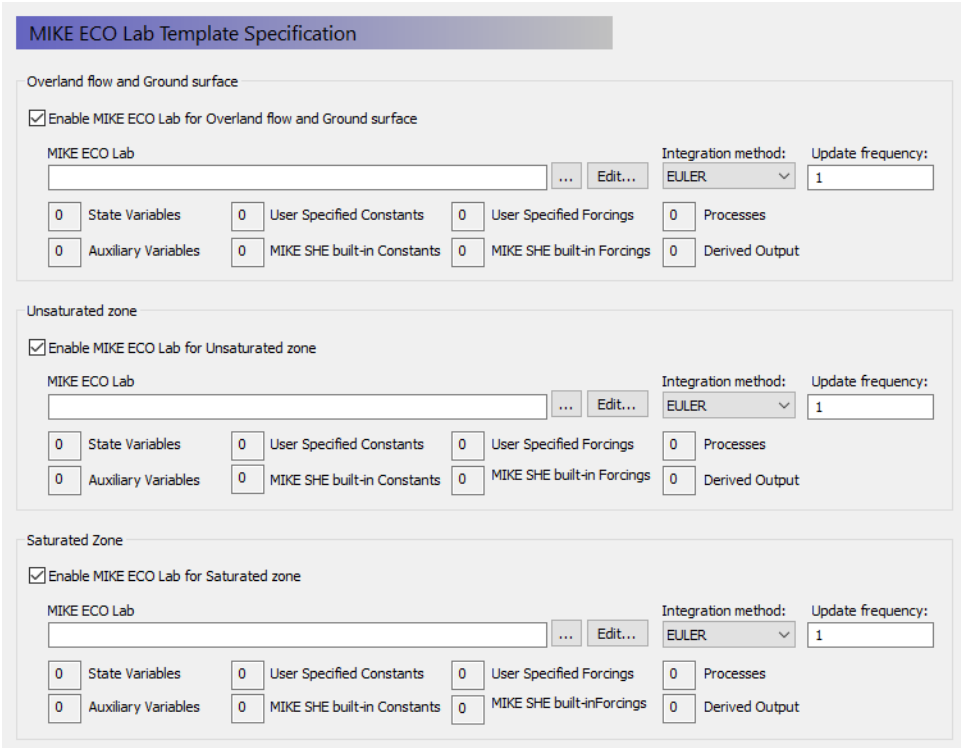
Max. Dispersive Courant Number - The dispersive courant number represents the ratio of the cell size to time it would take a particle to move across a cell due to dispersion. This criteria is likely to be controlling when the velocities are very slow and the dispersivity is non-zero. The default value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

Related Items:

- Working with Solute Transport - User Guide *(p. 715)*
- Particle Tracking-Reference *(p. 737)*

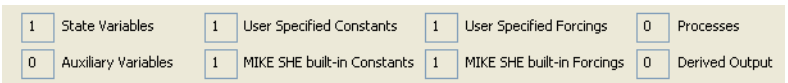
11.6 MIKE ECO Lab Template Specification

.



In the MIKE ECO Lab Template Specification dialog, there is a checkbox for each of the processes. These checkboxes are active if the Water Quality is activated for the process in the WQ Simulation Specification dialog. When you enable MIKE ECO Lab for the specific process, you will be able to browse to the required MIKE ECO Lab template. Specified templates can be directly modified by clicking on the Edit button.

When you browse to a template, the template file is read by the Setup Editor and the number of components (i.e. State Variables, Forcings, Processes, etc) that have been specified in the template are displayed in the Template summary.



Interpolation Method

When calculating the concentrations (State Variables) for the next time step, an explicit time-integration of the transport equations is made. Depending on the desired accuracy of this numerical integration, you can chose one of three different integration methods. The methods are in increasing level of accu-

racy (and numerical effort), starting with the Euler method and finishing with the Runge Kutta 5th Order method).

Update Frequency

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

Important Note: Units

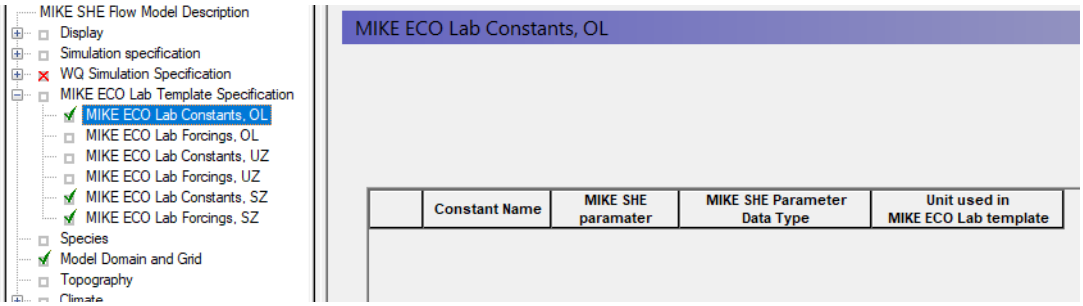
All concentrations passed from MIKE SHE to MIKE ECO Lab are in units of [g/m^3], which is equivalent to [mg/L].

Thus all parameters and equations defined in the MIKE ECO Lab template must reflect these units - either directly or via an appropriate scaling factor. For example, the correct units for a decay rate constant might be [g/m^3/day] or [mg/L/day].

Related Items:

- Working with MIKE ECO Lab in MIKE SHE - User Guide *(p. 725)*
- MIKE ECO Lab Overview Tables *(p. 221)*
- User Specified MIKE ECO Lab Constants and Forcings *(p. 222)*
- Advection Dispersion - Reference *(p. 673)*

11.6.1 MIKE ECO Lab Overview Tables



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

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

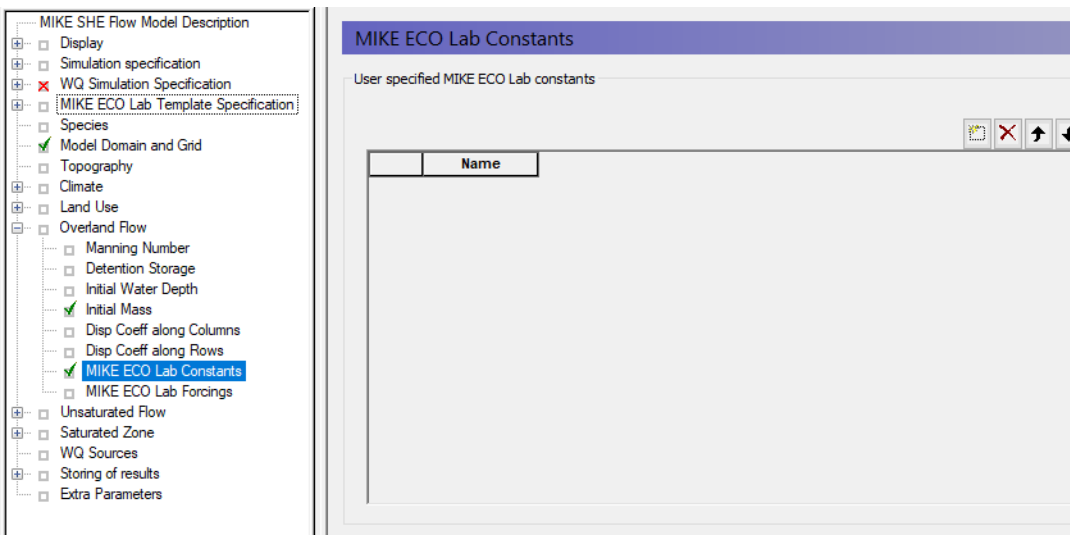
If the Forcing or Constant is not user defined (User Defined = “NO”), then the Forcing or Constant is an internal value in MIKE SHE and will be passed automatically to MIKE ECO Lab. The list of available parameters is quite short - primarily geometry related (e.g. cell volume), plus a few domain specific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

If the Forcing or Constant is user defined (User Defined = “YES”), then the Forcing or Constant must be specified explicitly in MIKE SHE. In this case, there is nothing to specify on the main list of Forcings and Constants, but a Forcing or Constant item is added to the data tree down under the appropriate branch in the data tree. In this branch you will find a table of user specified Forcings and Constants, plus individual sections for each item. Each item can be spacially defined similarly to other constants or time varying values in MIKE SHE.

Related Items:

- Working with MIKE ECO Lab in MIKE SHE - User Guide *(p. 725)*
- MIKE ECO Lab Template Specification *(p. 220)*
- User Specified MIKE ECO Lab Constants and Forcings *(p. 222)*
- Advection Dispersion - Reference *(p. 673)*

11.7 User Specified MIKE ECO Lab Constants and Forcings



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

In this case, a Forcing or Constant item is added to the data tree under the appropriate branch in the data tree. In this branch, you will find a table of user

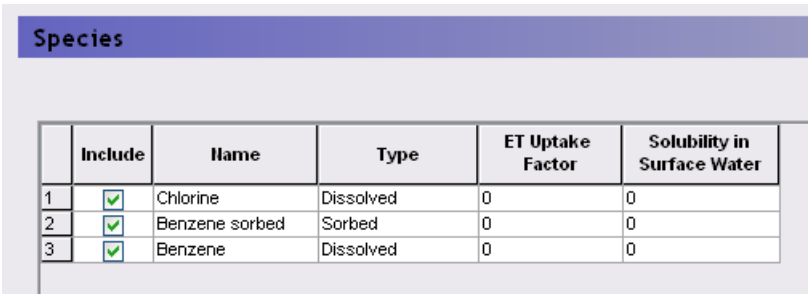
specified Forcings and Constants, plus individual sections for each item. Each item can be spacially defined similarly to other constants or time varying values in MIKE SHE.

Note: The Forcings and Constants are defined by Water Quality layer in the Saturated Zone. Thus, you have to define at least one Water Quality Layer in the Saturated Zone.

Related Items:

- Working with MIKE ECO Lab in MIKE SHE - User Guide *(p. 725)*
- MIKE ECO Lab Template Specification *(p. 220)*
- MIKE ECO Lab Overview Tables *(p. 221)*
- Advection Dispersion - Reference *(p. 673)*

11.8 Species



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

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

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

- **Include** - Turning off the include checkbox allows you to exclude a species from the simulation without having to remove it and all of its accompanying sources, etc.

- **Name** - This is the displayed identifier in all subsequent dialogues and in the data tree.

  MIKE ECO Lab note: The Name must be exactly the same as the State Variable Name used in the MIKE ECO Lab Template (except in the case of dual domain mass transfer, which uses the reserved suffix "_2").

- **Type** - There are four species types. Species can be either:
  - **Dissolved** - Dissolved species are mobile in the water. They are active in the subsurface and surface water. Disolved species have a default concentration of [μg/m^3].
  - **Sorbed** - Sorbed species are only available in the subsurface. They are fixed to the soil matrix and do not move with the water. Sorbed species have a default concentration of [g/g].
  - **Suspended** - Suspended species are only available in ponded water. They do not infiltrate to the UZ or SZ, and they cannot become Sorbed species. If the ponded water infiltrates, the species is left behind. Suspended species have a default concentration of [μg/m^3].
  - **Fixed** - A fixed species is neither disolved or nor sorbed. It is used as an immobile state variable by MIKE ECO Lab. This allows MIKE ECO Lab to read and write arbitrary values to MIKE SHE during the simulation. Fixed species have an undefined unit.

- **ET uptake transpiration factor** - This is the factor that determines the rate at which plants will remove the mobile solute from the water.

- **Solubility in Surface Water** - Surface water (overland flow) sources are specified as a mass. Thus, the source will be active until all of the solute has been dissolved. This is important because the life of the source depends on the amount of surface water flow.

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

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

Related Items:

- Working with Solute Transport - User Guide *(p. 715)*

- Working with MIKE ECO Lab in MIKE SHE - User Guide *(p. 725)*
- Advection Dispersion - Reference *(p. 673)*

11.9 Water Quality Sorption and Decay

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

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

Decay Processes

Decay Processes

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

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

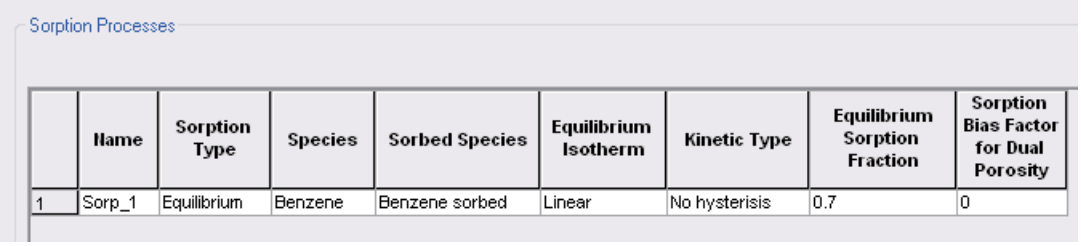
Reference Temperature - This is the reference temperture for the decay function in Equation (37.15).

Temp Decay Exponent - This is the decay exponent, α, in Equation (37.15).

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

Water Content Decay Exponent - This is the decay exponent, β, in Equation (37.14).

Sorption Processes



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

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

Species - This is the name of the species that will be sorbed. The combo box includes only of the **dissolved** species listed in the main Species *(p. 223)* dialog.

Sorbed Species - This is the name of the sorbed version of the dissolved species. This combo box includes only the **sorbed** species from the Species *(p. 223)* dialog.

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

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

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

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

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

For more details on the sorption isotherms, see Equilibrium Sorption Isotherms *(p. 700)*.

Kinetic Type - The three equilibrium sorption isotherms described above can be extended to include kinetically controlled sorption. In the MIKE SHE AD module, a two-domain approach is used, where the sorption is assumed to be instantaneous for a fraction of the sorbed solute and rate-controlled for the remainder. (see below). If Equilibrium-Kinetic is selected in this combo box, then an additional data tree sub-item will be added in the UZ and SZ WQ sections for the kinetic rate constant specified in Equation (37.9).

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

Sorption Bias Factor for Dual Porosity - Sorption depends on the porosity and the bulk density of the soil. In dual porosity systems this is rather complicated. The distribution of sorption between the matrix and the fractures should be calculated based on the bulk density and different porosities. However, this is not always practically possible, so MIKE SHE includes a sorption bias factor that allows you to explicitly control the sorption distribution between the fractures and the matrix. The sorption bias factor function is shown in Figure 37.3.

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

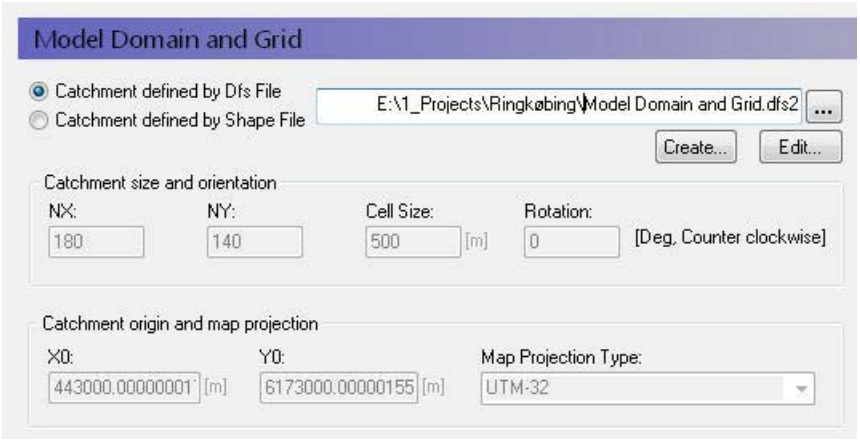
Sorption bias factor = 0 The distribution of sorption sites between macro pores and matrix is assumed to be proportional to the distribution of porosities.

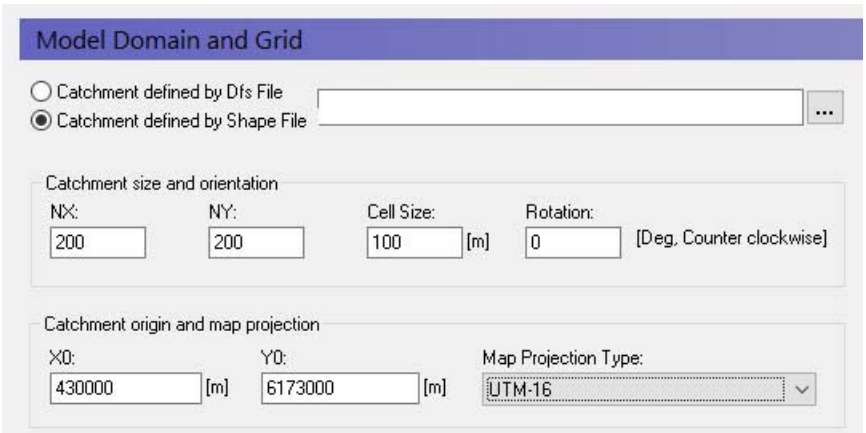
Sorption bias factor = 1 Sorption is only in the macro pores.

Related Items:

- Working with Solute Transport - User Guide *(p. 715)*
- Advection Dispersion - Reference *(p. 673)*
- Reactive Transport - Reference *(p. 699)*

11.10 Model Domain and Grid





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

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

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

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

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

Using a dfs2 file

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

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

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

The catchment definition is displayed in the greyed out text boxes but is not editable, since the catchment definition is part of the .dfs2 file format. If you want to change the cell size, origin, number of cells etc., you must change the .dfs2 file itself. For more information on editing and setting up the Model Domain and Grid see Conceptual geologic model for the finite difference approach *(p. 63)*.

Using a shp file

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

Map projections

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

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

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

Map projections in MIKE SHE

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

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



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

11.11 Sub-catchments



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

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

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

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

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

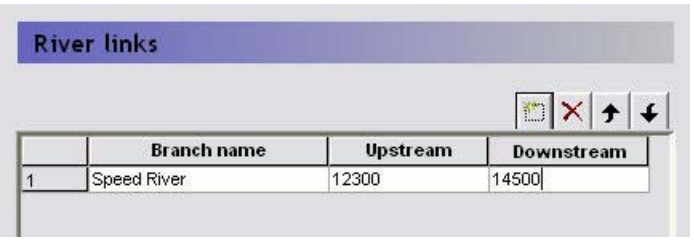
If you uncheck this checkbox, a sub-item will appear where you can specify the river branch and chainage to link the subcatchment to.

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

Related Items

- Simplified Overland Flow Routing *(p. 533)*
- Overland Flow Zones *(p. 286)*
- Linear Reservoir Method *(p. 638)*
- Interflow Reservoirs *(p. 308)*

11.11.1 River Links



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

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

Related Items

- Coupling of MIKE SHE and a MIKE 1D River Model *(p. 477)*

11.12 Topography

Topography	
dialogue Type:	Stationary Real Data
EUM Data Units	Elevation

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

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

- Lower Level *(p. 355)*,
- Upper Level *(p. 355)*,
- Initial Potential Head *(p. 324)*, and
- Drain Level *(p. 334)*.

Depth parameters, such as ET Surface Depth *(p. 302)*, are also measured from the topography.

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

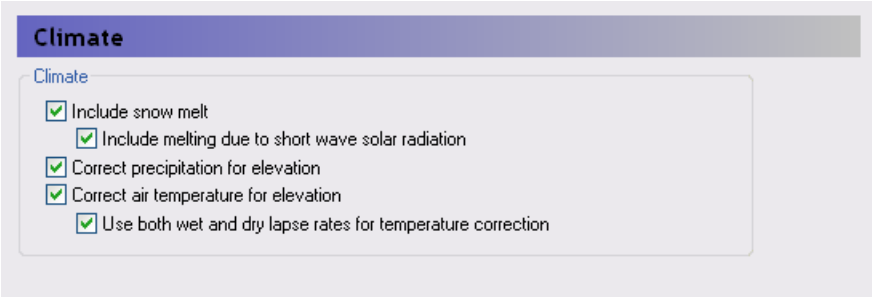
Non-dfs2 files or dfs2 files that have a different grid definition than the model grid are all interpolated to the grid defined in the Model Domain and Grid.

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

Related Items

- Interpolation Methods *(p. 169)*

11.13 Climate



Precipitation is always included in the data tree.

Include snow melt

If snowmelt is included, then Evapotranspiration must also be specified in the Simulation Specification *(p. 186)* dialog. When snow melt is included, then a new Snow Melt *(p. 243)* section is added to the data tree where the various snow melt parameters can be specified.

Include melting due to short wave radiation - When the Snow melt option is selected, then the check box for radiation melting will become editable. This option will add a new section to the climate data for incoming short wave radiation. Also added will be an item in the Snow Melt *(p. 243)* section for the Melting Coefficient for Solar Radiation *(p. 245)*, which defines the rate at which the snow melts per unit of incoming radiation.

Elevation corrections

Correct precipitation for elevation - In mountainous areas, precipitation can vary significantly with elevation - especially on the windward side of the mountains. However, there are rarely enough rain gauging stations to measure this spatial variability of rainfall. Choosing to correct the precipitation for elevation allows you to define a spatially variable correction factor, if you are using sub area-based rainfall data. See Precipitation Lapse Rate *(p. 236)* for the correction equations.

Correct temperature for elevation - In mountainous areas, temperature varies significantly with elevation. Choosing to correct the temperature for elevation allows you to define a spatially variable correction factor, if you are using sub area-based temperature data. See Temperature Lapse Rate *(p. 241)* for the correction equations.

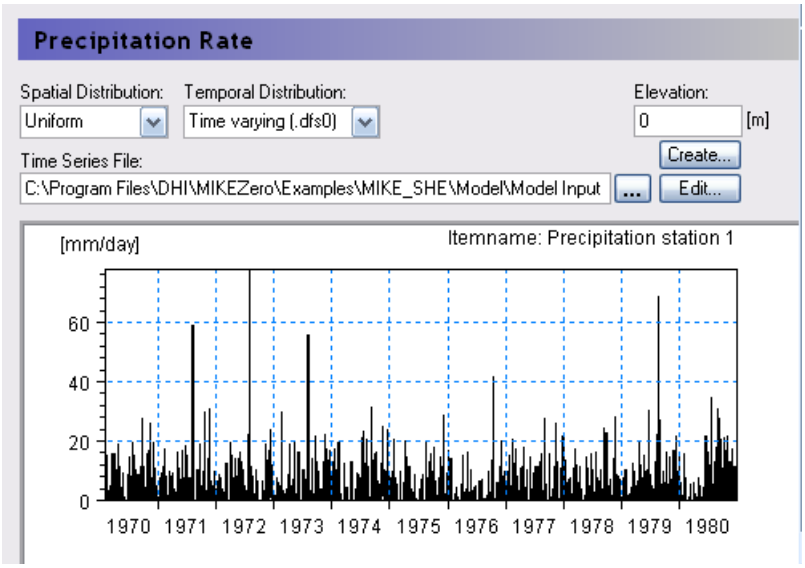
Use both wet and dry lapse rates for temperature correction - The moisture content of the air is a significant factor for the elevation correction of temperture. If this option is chosen, then you can specify a different correction factor for the time steps when it is raining.

Related Items

- Time Series Data *(p. 153)*

- Air Temperature *(p. 240)*
- Precipitation Rate *(p. 234)*
- Reference Evapotranspiration *(p. 238)*
- Short Wave Solar Radiation *(p. 239)*
- Snow Melt *(p. 243)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.1 Precipitation Rate



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

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

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

If you use a rate, then the EUM Data Units must be Precipitation and the time series must be Mean Step Accumulated *(p. 158)*.

If you use an amount, then the EUM Data Units must be Rainfall and the time series must be Step Accumulated *(p. 157)*.

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

If Include snow melt is included and the Air Temperature *(p. 240)* is below the Threshold Melting Temperature *(p. 244)*, The threshold melting temperature is the temperature at which the snow starts to melt - usually 0 C. If the air temperature is above this threshold, then the snow will melt at the rate specified in the Degree-day Melting or Freezing Coefficient (p. 245) item. If the air temperature falls belowthen the precipitation will accumulate as snow.

Elevation - This is the station elevation. It is used as the reference elevation if the cell-by-cell precipitation is to be corrected for elevation differences (See Elevation corrections *(p. 233)*). The actual precipitation in each cell will then be calculated based on the elevation of the cell relative to the station elevation and the Precipitation Lapse Rate *(p. 236)*. Elevation correction is not available for fully-distributed precipitation data.

Related Items

- Precipitation Lapse Rate *(p. 236)*
- Creating Time Series in MIKE SHE *(p. 153)*
- Working with Spatial Time Series *(p. 155)*
- Time Series Types *(p. 155)*

11.13.2 Precipitation Lapse Rate

Precipitation Lapse Rate	
Conditions	if Correct precipitation for elevation checkbox is on in the Climate *(p. 233)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Correction of precipitation (e.g. % per 100m)

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

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

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

Related Items

- Precipitation Rate *(p. 234)*

11.13.3 Net Rainfall Fraction

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

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

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

$$R_{net} = Prec \cdot Rainfall_{net} \cdot Infil_{frac} \quad (11.3)$$

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

Related Items

- Precipitation Rate *(p. 234)*
- Infiltration Fraction *(p. 237)*

11.13.4 Infiltration Fraction

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

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

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

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

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

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

$$R_{net} = Prec \cdot Rainfall_{net} \cdot Infil_{frac} \quad (11.4)$$

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

Related Items

- Precipitation Rate *(p. 234)*
- Net Rainfall Fraction *(p. 236)*

11.13.5 Reference Evapotranspiration

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

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

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

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

Related Items

- Creating Time Series in MIKE SHE *(p. 153)*
- Working with Spatial Time Series *(p. 155)*
- Time Series Types *(p. 155)*
- Vegetation Properties Editor *(p. 407)*
- Evapotranspiration - Technical Reference *(p. 437)*
- Working with Evapotranspiration - User Guide *(p. 457)*

11.13.6 Short Wave Solar Radiation

Short Wave Solar Radiation	
Conditions	if melting due to short wave radiation is checked on in Climate *(p. 233)* dialog
dialogue Type	Time-varying Real Data
EUM Data Units	Grid Code
Time Series EUM Data Units	Radiation intensity; Instantaneous

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

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

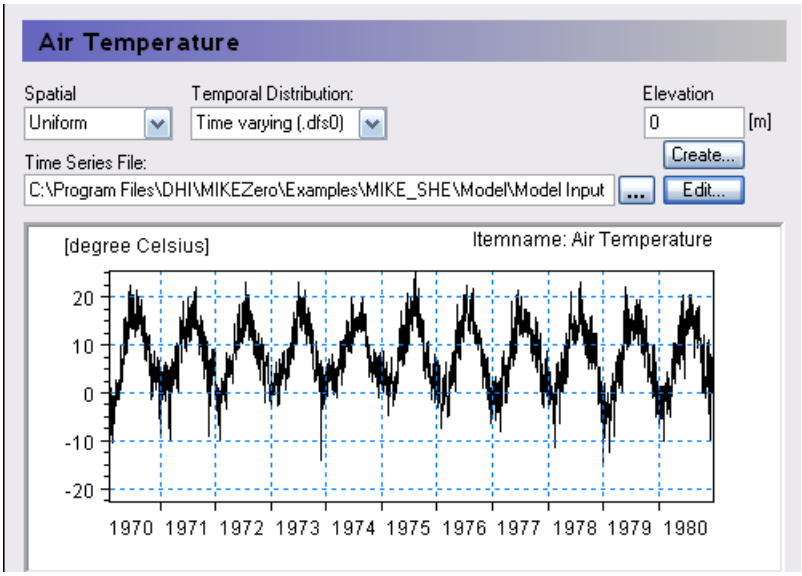
Daily average values will typically underestimate the amount of snow melt. So model time steps and input data that account for the diurnal fluctations in radiation will give better results.

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

Related Items

- Creating Time Series in MIKE SHE *(p. 153)*
- Working with Spatial Time Series *(p. 155)*
- Time Series Types *(p. 155)*
- Climate *(p. 233)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.7 Air Temperature



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

Elevation - This is the station elevation. It is used as the reference elevation if the cell-by-cell temperatures are to be corrected for elevation differences (See Elevation corrections *(p. 233)*). The elevation correction is specified by the Temperature Lapse Rate *(p. 241)*, or optionally by the Wet Lapse Rate *(p. 242)* if it is raining. Elevation correction is not available for fully distributed temperature data.

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

- If the Air temperature is below the Threshold Melting Temperature *(p. 244)* value, then the precipitation accumulates as snow; above it is treated as rain.

- If the Air temperature is above the Threshold Melting Temperature *(p. 244)* value, then dry snow is converted to wet snow using the Degree-day Melting or Freezing Coefficient *(p. 245)*.
- If the Air temperature is below the Threshold Melting Temperature *(p. 244)* value, then wet snow is converted to dry snow using the Degree-day Melting or Freezing Coefficient *(p. 245)*.

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

Related Items

- Creating Time Series in MIKE SHE *(p. 153)*
- Working with Spatial Time Series *(p. 155)*
- Time Series Types *(p. 155)*
- Snow Melt *(p. 243)*
- Threshold Melting Temperature *(p. 244)*
- Degree-day Melting or Freezing Coefficient *(p. 245)*
- Decay *(p. 704)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.8 Temperature Lapse Rate

Temperature Lapse Rate	
Conditions	if Correct temperature for elevation checkbox is on in the Climate *(p. 233)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Temperature Lapse Rate (e.g. C per 100m)

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

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

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

However, the moisture content of the air is a significant factor for the elevation correction of temperture. Thus, you have the option in the Climate *(p. 233)* dialog to specify a different correction factor for the time steps when it is raining.

Related Items

- Climate *(p. 233)*
- Air Temperature *(p. 240)*
- Wet Lapse Rate *(p. 242)*

11.13.9 Wet Lapse Rate

Wet Lapse Rate	
Conditions	if Correct temperature for elevation checkbox is on in the Climate *(p. 233)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Temperature Lapse Rate (e.g. C per 100m)

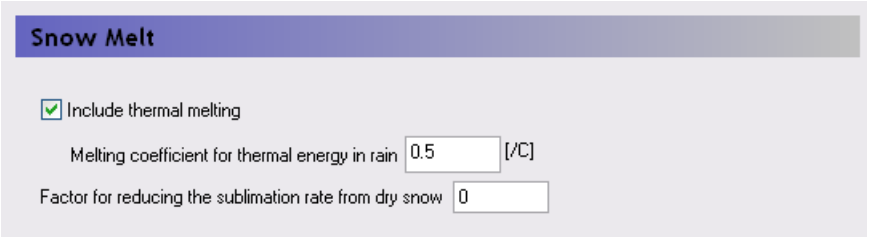
The moisture content of the air is a significant factor for the elevation correction of temperture. Thus, you have the option in the Climate *(p. 233)* dialog to specify a different correction factor for the time steps when it is raining.

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

Related Items

- Climate *(p. 233)*
- Air Temperature *(p. 240)*
- Temperature Lapse Rate *(p. 241)*

11.13.10Snow Melt



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

Snow melts by converting from dry snow to wet snow. Wet snow is converted to overland flow when the Maximum Wet Snow Fraction in Snow Storage *(p. 246)* is exceeded.

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

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

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

where M_{rain} is the melting rate, C_{rain} is the melting coefficient due to the energy content of the rain, P is the precipitation rate, T_{air} is the current air temperature and T_0 is the Threshold Melting Temperature *(p. 244)*.

Factor for reducing the sublimation rate from dry snow -

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

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

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

*Maximum ET = (Reference ET) * (Crop Coefficient) * (Sublimation Factor)*

Related Items

- Climate *(p. 233)*
- Air Temperature *(p. 240)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.11 Threshold Melting Temperature

Threshold Melting Temperature	
Conditions	if Snow melt included in n the Climate *(p. 233)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Temperature

The threshold melting temperature is the temperature at which the snow starts to melt - usually 0 C. If the air temperature is above this threshold, then the snow will melt at the rate specified in the Degree-day Melting or Freezing Coefficient *(p. 245)* item. If the air temperature falls below

Related Items

- Degree-day Melting or Freezing Coefficient *(p. 245)*
- Air Temperature *(p. 240)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.12Degree-day Melting or Freezing Coefficient

Degree-day Melting or Freezing Coefficient	
Conditions	if Snow melt included in the Climate *(p. 233)* dialog
dialogue Type	Time-varying Real Data
EUM Data Units	Grid code
Time Series EUM Data Units	Melting coefficient [e.g. mm snow/day/degree C]

The degree day factor is the amount of snow that melts per day for every degree the Air Temperature *(p. 240)* is above the Threshold Melting Temperature *(p. 244)*.

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

Related Items

- Threshold Melting Temperature *(p. 244)*
- Air Temperature *(p. 240)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.13Melting Coefficient for Solar Radiation

Melting Coefficient for Solar Radiation	
Conditions	if radiation melting is included in the Climate *(p. 233)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Radiation melting coefficient [e.g. mm/(KJ/$m^{2)}$]

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

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

Related Items

- Short Wave Solar Radiation *(p. 239)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.14Minimum Snow Storage for Full Area Coverage

Minimum Snow Storage for Full Area Coverage	
Conditions	if Snow melt included in the Climate *(p. 233)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Storage depth [e.g. mm]

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

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

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

Related Items

- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.15Maximum Wet Snow Fraction in Snow Storage

Maximum Wet Snow Fraction in Snow Storage	
Conditions	if Snow melt included in the Climate *(p. 233)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Fraction [-]

Melting snow does not instantly lead to runoff. When snow melts, the liquid water content of the snow first increases until a critical water content is

reached. When this water content is reached additional melted snow will form runoff. If the air temperature drops below freezing the liquid water stored in the snow will re-freeze. This repeated thawing and freezing leads to snow compaction and changes in the snow structure over the winter. This in turn justifies the time dependancy of the Degree-day Melting or Freezing Coefficient *(p. 245)*.

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

Related Items

- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.16Initial Total Snow Storage

Initial Total Snow Storage	
Conditions	if Snow melt included in the Climate *(p. 233)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Storage depth [e.g. mm]

This is the initial total storage depth of snow in the cell. This value is used to also determine the initial snow coverage on the cell. Thus, if this value is less than the Minimum Snow Storage for Full Area Coverage *(p. 246)*, then the area coverage fraction will be less than one.

Related Items

- Minimum Snow Storage for Full Area Coverage *(p. 246)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.13.17 Initial Wet Snow Fraction

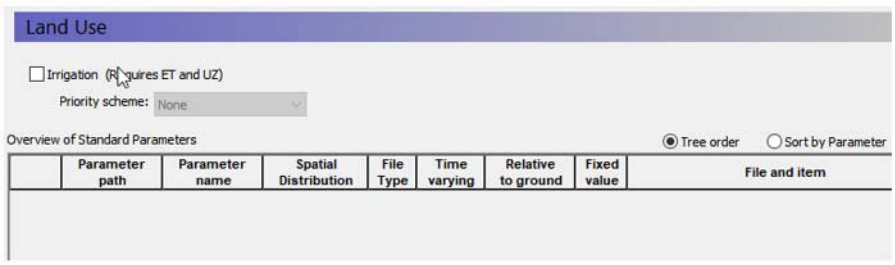
Initial Wet Snow Fraction	
Conditions	if Snow melt included in the Climate *(p. 233)* dialog
dialogue Type	Stationary Real Data
EUM Data Units	Fraction [-]

The Initial Wet Snow Fraction determines the initial liquid water content of the snow storage. If this value is greater than the Maximum Wet Snow Fraction in Snow Storage *(p. 246)*, then runoff will be generated in the first time step. If this value is zero, then the initial snow is dry.

Related Items

- Maximum Wet Snow Fraction in Snow Storage *(p. 246)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*

11.14 Land Use



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

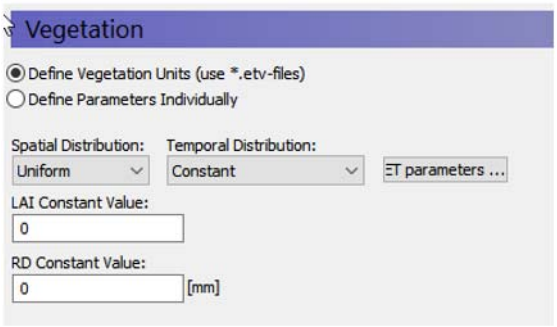
- Vegetation distribution, and
- Irrigation.

In the main menu you will also find an overview table of the main vegetation parameters used in the model. The table is blank to start with and is populated as the vegetation options and parameters are added in the sub-menus. The table of standard parameters is only active when the vegetation option from the Vegetation menu: Define Parameters Individually is selected (see section 11.14.1).

Irrigation - The Irrigation option allows you to specify a demand driven irrigation scheme with priorities. Activating the Irrigation option creates several sub-items in the data tree for the irrigation parameters. The Irrigation option requires that both Evapotranspiration and Unsaturated Flow be simulated. For more information see Irrigation Command Areas *(p. 258)*

Priority Scheme - The priority scheme is used by the Irrigation module to rank the model areas in terms of priority for irrigation. Two options are allowed: Equal Volume or Equal Shortage. If the water is to be distributed based on equal volume, then all cells with the same priority number will receive an equal amount of water, regardless of their actual demand. If the water is to be distributed based on equal shortage, then all cells with the same priority number will receive an amount of water that satisfies an equal percentage of their actual demand. For more information see Irrigation Priorities *(p. 267)*

11.14.1 Vegetation



The main Vegetation dialogue is used to define the distribution of vegetation across your model area. It works the same way as the other dialogues with options for specifying constant or time varying uniform values, grid codes with associated time series or fully distributed time varying values. There are two options for specifying vegetation parameters:

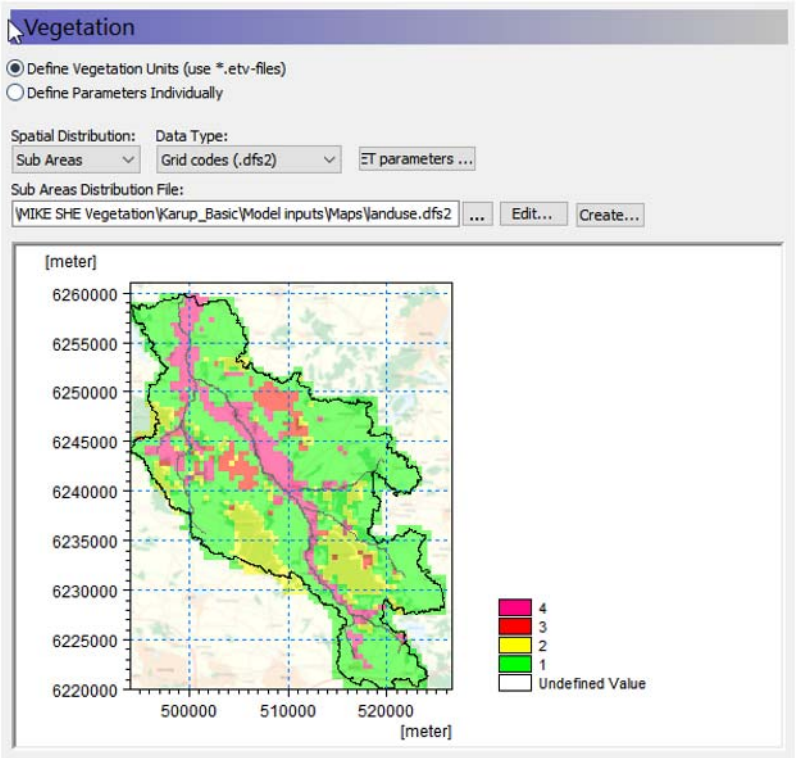
- Define Vegetation Units using grids and associated time series or from a Vegetation Properties file (*.etv)
- Define Parameters Individually

Vegetation Units option

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

If the Vegetation Properties file option is used for the Leaf Area Index and Root Depth parameters, it is possible to specify crop rotations (p. 257).

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



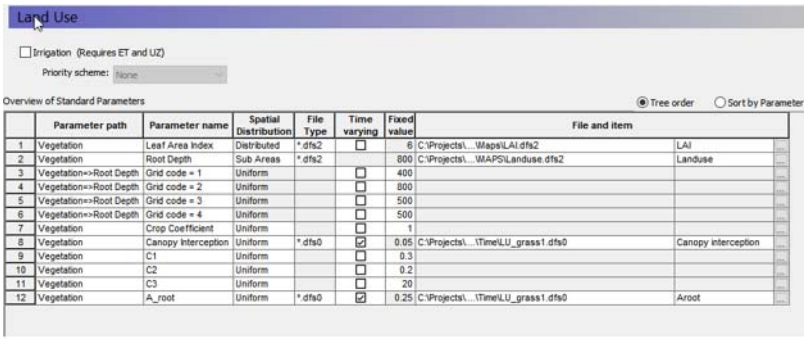
Vegetation	
Conditions:	If Evapotranspiration selected in the Simulation Specification dialogue
EUM Data Units	Grid Code
Time Series EUM Data Units	Leaf Area Index and Root Depth, or Vegetation Property File (see Vegetation Properties Editor *(p. 407)*)
dialogue Type:	Special version of Time-varying Real Data

Individual Parameters option

As an alternative to the Vegetation Units option, it is possible to specify the vegetation parameters individually. This option allows you to specify the main vegetation parameters: Leaf Area Index, Root Depth and Crop coefficient individually with all available options for distributions in space and time.

Evapotranspiration parameters are also specified individually, similarly to how the vegetation parameters are specified.

An overview table of the main parameters is included in the main Land Use dialogue. From here you can also modify the input values and time series links.

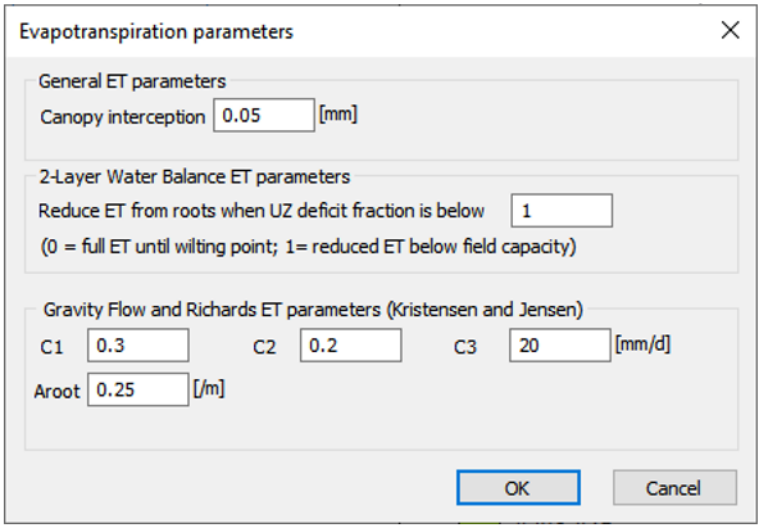


	Parameter path	Parameter name	Spatial Distribution	File Type	Time varying	Fixed value	File and item	
1	Vegetation	Leaf Area Index	Distributed	*.dfs2	☐	6	C:\Projects\...\Maps\LAI.dfs2	LAI
2	Vegetation	Root Depth	Sub Areas	*.dfs2		800	C:\Projects\...\MAPS\Landuse.dfs2	Landuse
3	Vegetation=>Root Depth	Grid code = 1	Uniform		☐	400		
4	Vegetation=>Root Depth	Grid code = 2	Uniform		☐	800		
5	Vegetation=>Root Depth	Grid code = 3	Uniform		☐	500		
6	Vegetation=>Root Depth	Grid code = 4	Uniform		☐	500		
7	Vegetation	Crop Coefficient	Uniform		☐	1		
8	Vegetation	Canopy Interception	Uniform	*.dfs0	☑	0.05	C:\Projects\...\Time\LU_grass1.dfs0	Canopy interception
9	Vegetation	C1	Uniform		☐	0.3		
10	Vegetation	C2	Uniform		☐	0.2		
11	Vegetation	C3	Uniform		☐	20		
12	Vegetation	A_root	Uniform	*.dfs0	☑	0.25	C:\Projects\...\Time\LU_grass1.dfs0	Aroot

Evapotranspiration Parameters

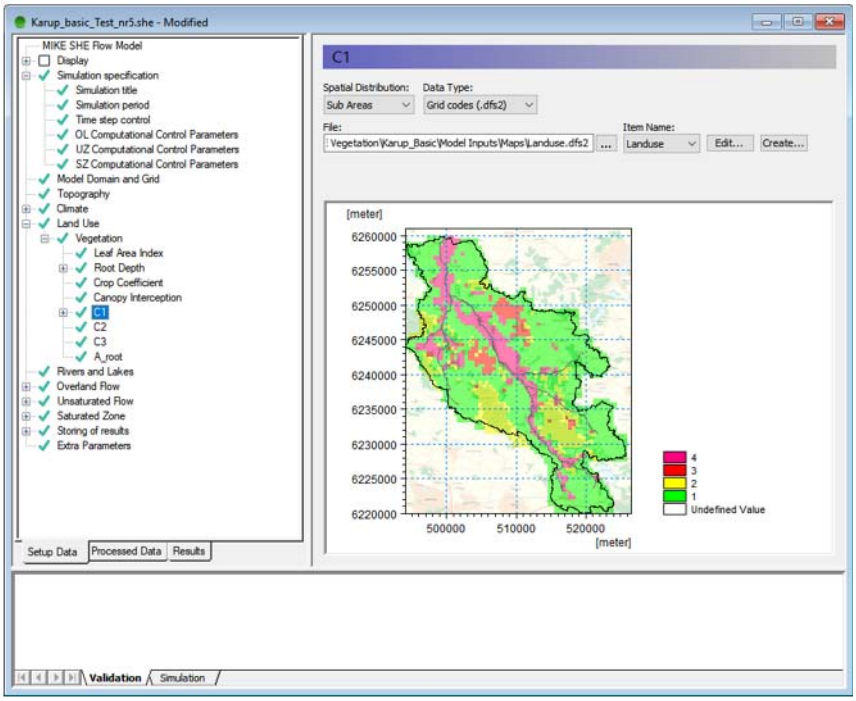
Evapotranspiration parameters are either specified through clicking on a button labelled ET Parameters... in case the Vegetation Units option is chosen, or the values are specified individually exactly like the vegetation parameters.

Clicking on the ET Parameters menu pops up the dialogue for specifying the default ET parameters.



The Evapotranspiration parameters in this dialogue do not vary in time and are global for the model. However, if the Vegetation Properties file option is used for the Leaf Area Index and Root Depth parameters, these values can be overridden by crop specific evapotranspiration values specified in the vegetation properties file.

If the Individual Parameter option is chosen it is possible to specify fully distributed values in time and space for each of the evapotranspiration parameters.



The following sections are condensed from the Evapotranspiration - Technical Reference *(p. 437)* chapter, which should be consulted for more detailed information.

Canopy Interception

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

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

Deficit Fraction for reduced ET (2-Layer ET only)

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

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

The deficit fraction is the fraction of this maximum deficit. When this deficit fraction is reached, MIKE SHE will start to reduce the rate of ET. In other words, transpiration will be removed at the full rate until this deficit fraction is reached. Then the transpiration rate will decrease linearly until θmin is reached, at which point transpiration will stop.

- A typical value for this deficit fraction is 0.5. This means that the ET will be removed from the UZ at the full rate until the deficit reaches half the maximum deficit.
- A deficit fraction of 0 means that ET will be start being reduced as soon as the water content falls below θmax.
- A deficit fraction of 1 means that ET will not be reduced, and ET will be removed at the maximum rate until the wilting point is reached.

For more information on this variable, see the Section The 2-Layer Water Balance Method *(p. 447)* in the Technical documentation.

C_1, C_2 and C_3

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

C1 - C_1 is plant dependent. For agricultural crops and grass, C_1 has been estimated to be about 0.3. C_1 influences the ratio soil evaporation to transpiration. This is illustrated in Figure 21.7. For smaller C_1 values the soil evaporation becomes larger relative to transpiration. For higher C_1 values, the ratio approaches the basic ratio determined by C_2 and the input value of LAI.

C2 - For agricultural crops and grass, grown on clayey-loamy soils, C_2 has been estimated to be about 0.2. Similar to C_1, C_2 influences the distribution between soil evaporation and transpiration, as shown in Figure 21.8. 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 21.8, where 23 per cent and 61 per cent of the total extraction takes place in the top node for C_2 values of 0 and 0.5 respectively.

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

Therefore, the total actual evapotranspiration will become sensitive to the ability of the soil to draw water upwards via capillary action.

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

Root Mass Distribution Parameter, AROOT

In the Kristensen and Jensen model, water extraction by the roots for transpiration varies over the growing season. In nature, the exact root development is a complex process, which depends on the climatic conditions and the moisture conditions in the soil. Thus, MIKE SHE allows for a root distribution determined by the root depth (time varying) and a general, vertical root-density distribution, defined by *AROOT*, see Figure 21.3. In the above dialogue, AROOT is not time varying, but can be specified as a time series using the Vegetation Properties Editor *(p. 407)*.

How the water extraction is distributed with depth depends on the *AROOT* parameter. Figure 21.4 shows the distribution of transpiration for different values of *AROOT*, assuming that the transpiration is at the reference rate with no interception loss ($C_{int}=0$) and no soil evaporation loss ($C_2=0$). *The figure shows* that the root distribution, and the subsequent transpiration, becomes more uniformly distributed as *AROOT* approaches *0*. During simulations, the total actual transpiration tends to become smaller for higher values of *AROOT* because most of the water is drawn from the upper layer, which subsequently dries out faster. The actual transpiration, therefore, becomes more dependent on the ability of the soil to conduct water upwards (capillary rise) to the layers with high root density.

Figure 21.5 shows the effect of the root depth, given the same value of *AROOT*. A shallower root depth will lead to more transpiration from the upper unsaturated zone layers because a larger proportion of the roots will be located in the upper part of the profile. However, again, this may lead to smaller actual transpiration, if the ability of the soil to conduct water upwards is limited.

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

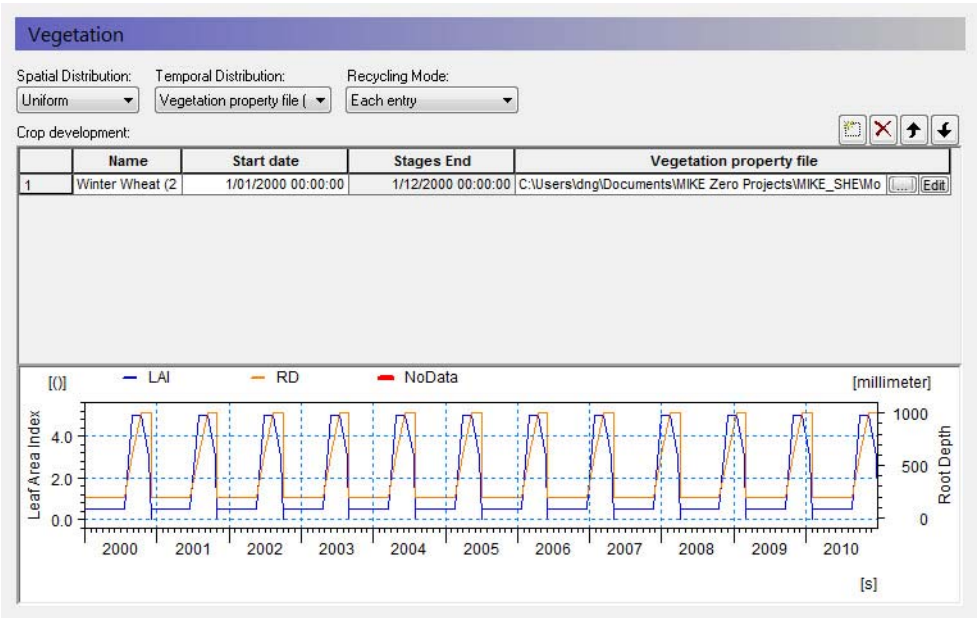
Thus, *AROOT* is an important parameter for estimating how much water can be drawn from the soil profile under dry conditions.

Related Items

- Kristensen and Jensen method *(p. 439)*

- The 2-Layer Water Balance Method *(p. 447)*
- Vegetation Properties *(p. 256)*
- Vegetation Properties Editor *(p. 407)*

11.14.2 Vegetation Properties



Vegetation	
Conditions:	If Evapotranspiration selected in the Simulation Specification dialogue
EUM Data Units	Grid Code
Time Series EUM Data Units	Leaf Area Index and Root Depth, or Vegetation Property File (see Vegetation Properties Editor *(p. 407)*)
dialogue Type:	Special version of Time-varying Real Data

There are three relevant time series parameters: the Leaf Area Index, the Root Depth and the Crop Coefficient, Kc. These parameters can be defined as constants, via .dfs0 files, or they can be defined from a Vegetation Properties file.

Using a Vegetation Properties file

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

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

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



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

Recycling Mode

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

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

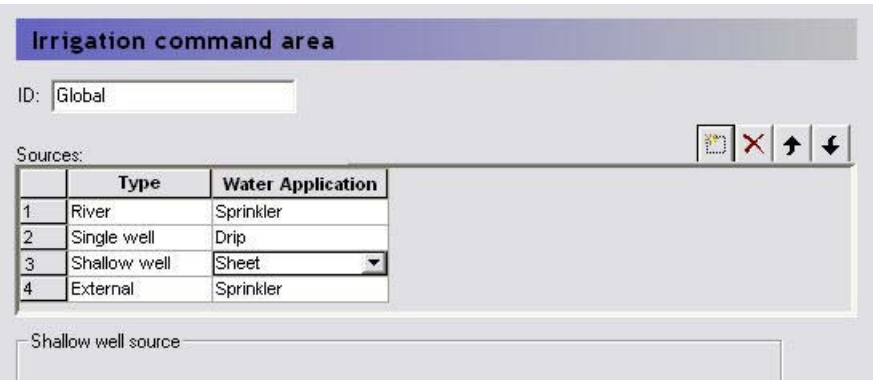
The Vegetation dialogue also includes a button for the Evapotranspiration Parameters, which are global stationary parameters.

Related Items

- Kristensen and Jensen method *(p. 439)*

- The 2-Layer Water Balance Method *(p. 447)*
- Vegetation Properties *(p. 256)*
- Vegetation Properties Editor *(p. 407)*

11.14.3 Irrigation Command Areas



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

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

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

Each source can also be limited by a licensed maximum amount of water in any period - License Limited Irrigation *(p. 263)*.

Calculation sequence and shortage handling for SZ Linear Reservoirs

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

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

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

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

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

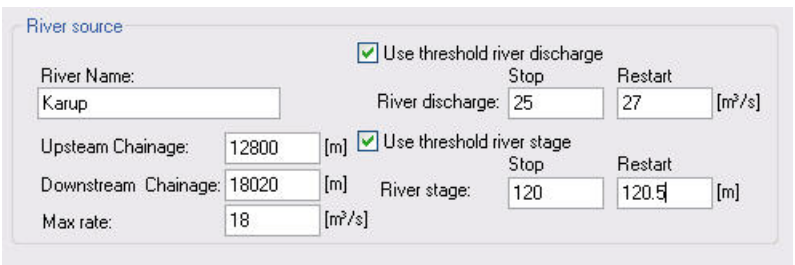
$$V_2 = (D_r - D_w) \cdot S_y \cdot A_T \cdot (1 - F_1) \tag{11.8}$$

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

Water Source Types

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

River Sources



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

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

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

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

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

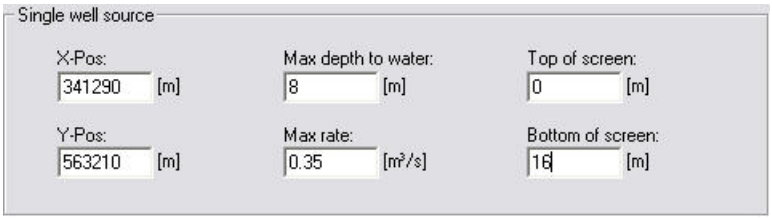
Use threshold river stage - If the water level in the river falls below the **Stop** value, then water will no longer be extracted from the River. However, if the water level in the river increases again and reaches the **Restart** value, the river source will be reactivated. The river stage threshold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

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



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

Single Well Sources



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

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

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

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

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

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

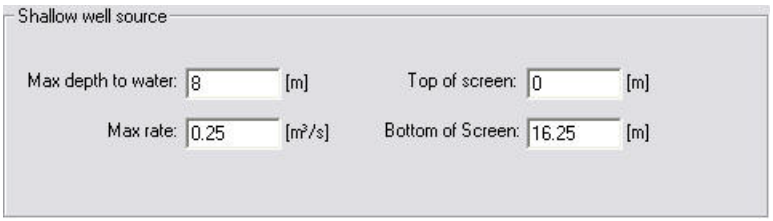
In the linear reservoir groundwater method, multiple single wells are allowed in each baseflow reservoir. No warnings are given.

When the linear reservoir method is used, the screen interval is ignored and the water is pumped from the two baseflow reservoirs. The distribution between the two reservoirs is determined by the faction give in the Baseflow Reservoirs *(p. 310)* dialogue. If the demand from one of the reservoirs exceeds the available water, the pumping will be reduced. The pumping rate at the other reservoir will not be increased to compensate.

Also in the Linear Reservoir method, the specified “max depth to water” for the actual command area and source is used. In other words, it is not using the “threshold depth for pumping” in the baseflow reservoir menu. Pumping is

allowed when the depth to the water table is less than the specified threshold value at the start of the time step.

Shallow Well Sources



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

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

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

Max rate - This is the maximum extraction rate for the shallow well **in each cell**. If more water is required for irrigation, then the next source will be activated.

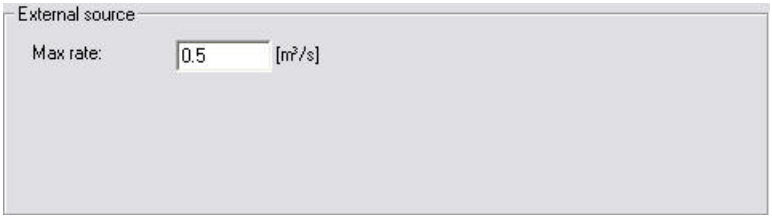
Top/Bottom of Screen - The depth of the top and bottom of the screen is used to define from which numerical layers water can be extracted. Pumping will stop if the water table falls below the bottom of the layer that contains the filter bottom.

Shallow well sources are removed from baseflow Reservoir 1 if the Linear Reservoir groundwater method is used. The screen interval is ignored.



Note: Shallow wells can be located in cells containing single sources. The preprocessor will give a warning for such violations. Multiple shallow wells are not allowed in the same command area.

External Sources



In some case, the irrigation water can be from outside of the watershed being modelled. In this case, the only constraint is the maximum amount of water than can be extracted from the source.

Max rate - This is the maximum extraction rate for the source. If more water is required for irrigation, then the next source will be activated.

Water Application methods

There are three ways to apply the irrigation water in the model.

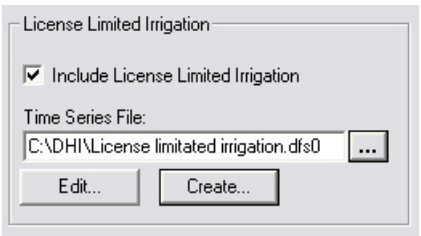
Sprinkler - If the water is applied as sprinkler irrigation, it is added to the precipitation component.

Drip - If the water is applied as Drip irrigation, it is added directly to the ground surface as ponded water.

Sheet - If the water is applied as Sheet irrigation, then an additional data tree item is required to define where the water is to be added within the command area. The idea behind this option is that water is flooded onto one or more cells of the command area and then distributed to the adjoining cells as overland flow. The sheet irrigation is applied directly to the cells as ponded water.

All three methods are allowed in the Simple, sub-catchment based overland flow method. However, the sheet method does not really make sense if the subcatchment overland flow method is used.

License Limited Irrigation



Sometimes, the total amount of irrigation water that a user can apply is limited by a license over a certain period (e.g. 10000 m^3 / year). The license limited option, allows you to specify a dfs0 time series file with a time series of maxi-

mum amounts. If the maximum amount is reached within the license period, then the irrigation will be stopped until the next license period, when it will be started again. The license period length is defined by the time steps in the specified dfs0 file.

During the simulation the license data is included in the calculation of the available water volume of each source. The module keeps track of the “actual available license volume”. Whenever this is reached or exceeded, the source will be closed until a new license period starts (or the end of the simulation). When a source is closed for this reason, a message is printed in the wm_print.log file.

Notes

- The dfs0 file EUM Data Units *(p. 143)* must be Water volume (m^3 or other volume unit) and the time series-type must be Step Accumulated *(p. 157)*.
- The specified volumes cover the period from the previous value (or start of simulation) until the date of the actual value.
- The files may contain delete values. These are simply ignored. This makes it possible to include licenses for several sources in one file, even when the dates of the different source licenses differ.
- An irrigation log file is included in the results output: *projectname*_IrrigationLicenseLog.dfs0.
- This log file contains the “actual available license volume” of each source with license included, stored as instantaneous values at the end of every time step. This makes it easy to identify the periods where sources have been closed due to “license shortage”.



Note: Unused license volumes are NOT carried over to the next license period (use it or loose it !).

11.14.4 Sheet Application Area

Sheet Application Area	
Conditions:	Irrigation selected in Land Use and UZ and ET simulated and Sheet selected as an application method
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

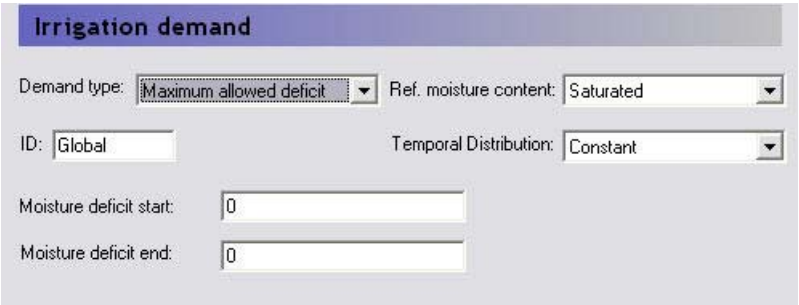
The sheet application area is used to define where the irrigation will be applied. The program does not make any distinction between sheet applica-

tion areas. The sheet irrigation will be distributed on every cell with a non-delete value or non-zero integer code within the command area.

Example If the calculated demand in the command area is 100 mm/day and the area of the command area is 5000 m^2, then the total amount of irrigation water required will be 100 mm x 5000 m^2 per day.

The total amount of irrigation water will be divided equally among the cells in the sheet application area. If each cell is 100 m^2 and the irrigation is applied to 10 cells, then each of the 10 cells will receive 500 mm/day of irrigation water as ponded water.

11.14.5 Irrigation Demand



Irrigation Demand	
Conditions:	Irrigation selected in Land Use and UZ and ET simulated
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The Irrigation Demand is used to describe when the water will be applied in the model.

The Irrigation Demand data item is divided into two dialogues. The first is the distribution dialogue for the Demand Areas and the second contains the information on when the water will be applied in each Demand area.

Demand Type

User Specified - For the User Specified demand type, the demand is not calculated. Rather it is simply specified as a constant value or as a time series.

Crop stress factor - The crop stress factor is the minimum allowed fraction of the crop specific reference ET that the actual ET is allowed to drop to before irrigation starts . That is, the minimum allowed *(Actual ET)/(Reference ET x K_c)* relationship. This should be a value between 1 and 0. If the actual Crop Stress factor falls below the given value, irrigation will be added.

Ponding depth - When using this option, the demand will be equal to the difference between the actual ponding depth and specified ponding depth. The option is typically used for modelling irrigation of paddy rice. If the ponding depth falls below the specified value then more irrigation water is added.

Max allowed deficit - The available water for crop transpiration (AW) is the difference between the actual water content and the water content at the wilting point for the root zone. The maximum available water for crop transpiration (MAW) is then the value of AW for the reference moisture content, where either the saturated water content or the field capacity can be specified for the reference moisture content. The deficit can then be defined as the fraction of the MAW that is missing and is a value between 0 and 1, where 0 is the deficit when the actual moisture content is equal to the reference moisture content, and 1 is the deficit when the available water for crop transpiration is zero, which is when the water content drops below the water content at the wilting point. When using the Max Allowed Deficit method, irrigation is started when the deficit exceeds the moisture deficit start value and stops at the moisture deficit end value.

If, for example, the reference moisture content is the field capacity and irrigation should start when 60 % of the maximum available water in the root zone is used and cease when field capacity is reached again, the value for the Start should be 0.6, and the value for the Stop value should be 0

Reference Moisture Content

If the Maximum allowed deficit is used, then the reference moisture content can be either for calculating the deficit is either the saturated water content or the field capacity water content.

Temporal Distribution

In each of the demand types the demand factor can be specified as a constant value or as a time series. However, an additional option in this combobox is to use the Vegetation Properties file. To use this option, you must include a vegetation properties file when you specify the vegetation. Further, you must specify the irrigation properties in the vegetation properties file itself. In this case, the Vegetation properties file, will contain all of the values needed by each of the different methods and the demand values cannot be input in this dialogue.

11.14.6 Irrigation Priorities

Irrigation Priorities	
Conditions:	Irrigation selected in Land Use and UZ and ET simulated and Priorities option selected in Land Use dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

If there is insufficient water available to satisfy all of the irrigation demand, then the irrigation areas can be prioritised. In this case, each area of the model can be assigned a priority number (1 = highest priority). All the areas with the highest priority will be irrigated first. If there is sufficient water after the first areas have been irrigated, then the areas with the next highest priority will be irrigated.

However, if there is insufficient water to completely satisfy the demand of a particular priority region (all of the cells with priority value = 1, for example), then the water will be distributed to each of the cells based on either Equal Volume or Equal Shortage. The choice of the two priority schemes is assigned in the main **Land Use** *(p. 248)* dialogue.

Equal Volume - If the water is to be distributed based on equal volume, then all cells with the same priority number will receive an equal amount of water, regardless of their actual demand. For example, if there is a demand for 100 m^3 of water in 10 cells, but only 50 m^3 is available, then each cell will receive 5 m^3 of water, regardless of the actual demand.

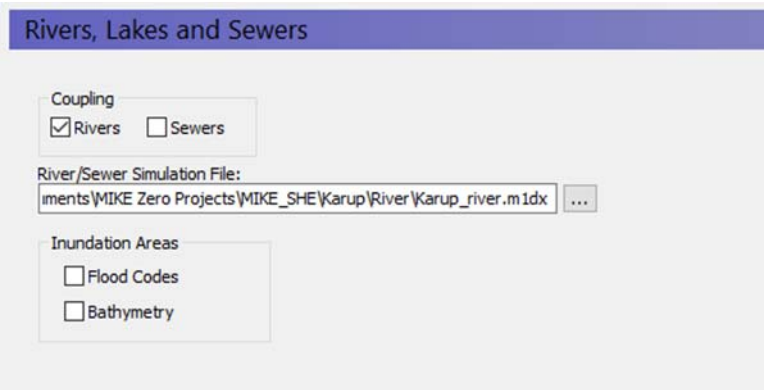
Equal Shortage - If the water is to be distributed based on equal shortage, then all cells with the same priority number will receive an amount of water that satisfies an equal percentage of their actual demand. For example, if there is a demand for 100 m^3 of water in 10 cells, but only 50 m^3 is available, and 5 of the cells have a demand of 15 m^3, while 5 have a demand of only 5 m^3, then the first cells will receive 7.5 m^3, while the latter will receive only 2.5 m^3.

11.14.7 Irrigation Log Areas

Irrigation Log Areas	
Conditions:	Irrigation selected in Land Use and UZ and ET simulated and Priorities option selected in Land Use dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

For each of the specified areas one time series file (.dfs0) will be created summarising the simulated irrigation. Note that there is a technical limit to the number of dfs files that can be written.

11.15 Rivers, Lakes and Sewers



In this dialogue, you can link MIKE SHE to a MIKE 1D model that simulates rivers, a collection system, or both. The text box will accept a MIKE 1D setup (.m1dx) file exported from MIKE+. In addition, as of release 2025, the dialog will still accept extensions for models built in MIKE Hydro River (.hydro) file and MIKE 11 (.sim11). MIKE HYDRO River and MIKE 11 were discontinued after the 2023 release of MIKE software.

If the MIKE 1D setup includes a model of the collection system, then a .couple file is also required. It needs to be in the same directory as the .m1dx file and have the same name. Both the .m1dx and the .couple file need to be exported from the same MIKE+ setup. The couple file doesn't need to be specified in MIKE SHE.

Specification of sewer pipes and manholes to include in a coupled model is done in the MIKE+ GUI. Please have a look at the MIKE+ documentation for more information.

Considerations when linking MIKE SHE to a river model

In principle, there are three basic steps for developing an integrated MIKE SHE/river model:

1. Establish a hydraulic river model as a stand-alone model, make a performance test and, if possible, a rough calibration using prescribed inflow and stage boundaries.
2. Establish a MIKE SHE model that includes the overland flow component and (optionally) the saturated zone and unsaturated zone components.
3. Couple MIKE SHE and the river model by defining branches (reaches) where the river should interact with MIKE SHE.

The chapter Coupling of MIKE SHE and a MIKE 1D River Model *(p. 477)* describes in more detail the three steps above.

Flooding and Inundation

Flow on the flood plain can be simulated by overbank spilling. However, this can be numerically intensive.

In many cases, you do not need to calculate the actual flow but only the water levels on the ground surface, for example in wetland areas on the flood plain. For such cases, you can use Flood Codes to define the inundated areas. See Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes *(p. 492)*.

Flood codes - With this option, you can specify the Flood Code map that shows which cells flood during a storm event, or are otherwise covered with water (e.g. permanent wetlands).

Bathymetry - The bathymetry is used to more accurately simulate the topography of the flood code cells, when the MIKE SHE topography is specified on a larger grid.

Water Quality with Rivers

If you have selected Water Quality calculation for Rivers and Lakes in the Water Quality Simulation Specification *(p. 207)* dialog, then an extra river model setup specification will be displayed. In this text box, you must specify the file that contains the water quality model setup for the river. This may or may not be the same as the one specified for the water movement calculation.

The extra river model setup specification is required because the water quality model is run independently of the water movement model. In other words, the water movement model is run first, which calculates the exchange between the groundwater and surface flow, with the river model. Then the

water quality model uses the water movement results to calculate the solute fluxes between the different water components. Thus, the water quality river model setup file could be different than the water movement setup file if you are running several solute transport scenarios. However, the river model setup files must be consistent. This consistency will be checked at run time.

The river model water movement results cannot be recycled in the same way that the rest of the MIKE SHE results can be. Your river simulation must cover the entire period of you water quality simulation.

Related Items:

- Water Quality Simulation Specification *(p. 207)*
- Channel Flow - Technical Reference *(p. 471)*
- Coupling of MIKE SHE and a MIKE 1D River Model *(p. 477)*

11.15.1 Flood codes

Flood Codes	
Conditions:	If Rivers and Lakes selected in the Simulation Specification dialogue and Flood Codes selected in the River and Lakes dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

Flood codes are required when using the Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes *(p. 492)*. This requires a .dfs2 file with Integer Grid Codes which are then used for making the flood mapping for the coupling reaches.

Related Items:

- Coupling of MIKE SHE and a MIKE 1D River Model *(p. 477)*
- Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes *(p. 492)*

11.15.2 Bathymetry

Bathymetry	
Conditions	If Rivers and Lakes selected in the Simulation Specification dialogue and Bathymetry selected in the Rivers and Lakes dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

The bathymetry option allows you to specify a detailed flood plain and river bottom topography, which can then be used for more accurate definition of the topography in the flood code cells.

Note: This file is only used if the Bed Topography Option is set in the river model.

Related Items:

- Coupling of MIKE SHE and a MIKE 1D River Model *(p. 477)*
- Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes *(p. 492)*

11.16 Overland Flow

The main dialogue for overland flow includes several options when the Finite difference method is selected. It does not contain any options when the Sub-catchment based method is selected.



If the Finite Difference method is selected in the Simulation Specification dialogue, the basic items required for the calculation of Overland Flow are

- the Manning number, which is equivalent to the Stickler roughness coefficient,
- the Detention Storage, and
- the Initial Water Depth on the ground surface (ponded water).

There are three main options available in the Overland Flow dialogue.

Separated overland flow areas

The first allows you to divide the model area into overland flow zones, which are conceptually areas separated by dikes or embankments. With this option, overland flow will not be allowed to flow between zones. If this is checked, then an additional item, Separated Flow Areas *(p. 279)*, will be added to the data tree.

Surface-subsurface exchange

The surface-subsurface exchange option allows you to specify an exchange coefficient to reduce the exchange of water between the overland flow and the saturated zone when the water table is at or above the ground surface.

This coefficient is also taken into account when calculating vertical infiltration from ponded water to the unsaturated zone.

If the reduced vertical exchange option is chosen then a new item, the Surface-Subsurface Leakage Coefficient *(p. 276)*, must be specified.

In specified areas, reduce exchange only for ponded areas - An additional sub-item is also available that allows you to apply the exchange coefficient in sub-areas where it is only used when water is ponded. During rainfall, this will not have an effect, as the rainfall is always added to ponded storage first. However, after the rainfall, this will allow water to flow downhill along a drainage pathway with a reduced infiltration rate.



Note: This option is only available if you are using Multi-cell Overland Flow *(p. 547)*

Reduced vertical exchange by paved area fraction - Paving is common in urban areas and has a significant impact on infiltration and runoff. This option allows you to additionally specify a paved area fraction that further reduces the infiltration in the cell by the fraction of paving.

When a Paved Area Fraction is specified, it is used as a linear scaling fraction for the Surface-Subsurface Leakage Coefficient. That is, the effective leakage coefficient is reduced by the Paved Area Fraction.

$$EffLeakCoef = (1-PAreaFrac) \times SurfSubSurfLeakCoef$$

Overland Drainage

In natural systems, runoff does not travel as 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 Overland Drainage (OL Drainage) allows you to specify conceptual drainage networks. Conceptually, the OL Drainage is similar to the SZ Drainage in that a drainage network is calculated based on a downhill flow path from each node until it reaches a stream, a boundary, or a local depression.

Selecting this option creates additional sub-items in the data tree for the drainage options.

Related Items:

- Separated Flow Areas *(p. 279)*
- Surface-Subsurface Leakage Coefficient *(p. 276)*
- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Overland Drainage *(p. 558)*
- Coupling of MIKE SHE and a MIKE 1D River Model *(p. 477)*

11.16.1 Manning number

Manning number	
Condition	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	ManningsM

The Manning M is equivalent to the Stickler roughness coefficient, the use of which is described in Overland Flow - Technical Reference *(p. 515)*.

The Manning M is the inverse of the more conventional Mannings n. The value of n is typically in the range of 0.01 (smooth channels) to 0.10 (thickly vegetated channels). This corresponds to values of M between 100 and 10, respectively. Generally, lower values of Mannings M are used for overland flow compared to channel flow.

If you don’t want to simulate overland flow in an area, a Mannings M of 0 will disable overland flow. However, be aware that this will also prevent overland flow from entering into the cell.

Note: A time-varying value can be specified by using a time-varying dfs2 file and checking ON the time-varying option.

Related Items:

- Overland Flow - Technical Reference *(p. 515)*
- Mean-Step-Accumulated time varying parameters *(p. 777)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*

11.16.2 Detention Storage

Detention Storage	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Storage Depth

Detention Storage is used to limit the amount of water that can flow over the ground surface. The depth of ponded water must exceed the detention storage before water will flow as sheet flow to the adjacent cell. For example, if the detention storage is set equal to 2mm, then the depth of water on the surface must exceed 2mm before it will be able to flow as overland flow. This is equivalent to the trapping of surface water in small ponds or depressions within a grid cell.

The detention storage also limits the inflow into the river. Thus, overland flow into the river will only occur if the ponded depth is above the detention storage. If you are using Flood codes, then the river model will only control the water level in the cell if the water level in the river link is above the topography + detention storage.

Water trapped in detention storage continues to be available for infiltration to the unsaturated zone and to evapotranspiration. Using detention storage, you can simulate water that is trapped in depressions that are smaller than a grid cell.

Detention storage can be used effectively to improve the Overland simulation performance. In areas of permanently ponded water, you can set the Detention Storage equal to depth of ponding. This will remove these areas from the Overland Flow calculation and prevent the time step from be reduced while MIKE SHE tries to calculate lateral circulation in the ponded areas.

Related Items:

- Overland Flow - Technical Reference *(p. 515)*

- Overland Flow Performance *(p. 544)*
- OL Drainage Options *(p. 760)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*

11.16.3 Initial Water Depth

Initial Water Depth	
Condition	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Water Depth

This is the initial condition for the overland flow calculations, that is the initial depth of water on the ground surface. The initial water depth is usually zero.

The initial water depth for overland flow is also the boundary condition for overland flow. In other words, if you specify an initial depth of 2mm, then the boundary will always have 2mm of water and there will be an inflow of water to the model whenever the internal cell has less than 2mm of ponded water.



Note: You can specify a time varying OL boundary condition using an Extra Parameter option.

Related Items:

- Overland Flow - Technical Reference *(p. 515)*
- Time-varying Overland Flow Boundary Conditions *(p. 756)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*

11.16.4 Surface-Subsurface Leakage Coefficient

Surface-Subsurface Leakage Coefficient	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue AND the Reduced vertical exchange in specified areas option selected in the main Overland Flow dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Leakage Coeff./Drain Time Const.

The Surface-subsurface leakage coefficient reduces the infiltration rate at the ground surface. It works in both directions. That is, it reduces both the infiltration rate and the seepage outflow rate across the ground surface.

Conceptually, the leakage coefficient is used to account for soil compaction and fine sediment deposits on flood plains in areas that otherwise have similar soil profiles.

If the groundwater level is at the ground surface, then the exchange of water between the surface water and ground water is based on the specified leakage coefficient and the hydraulic head between surface water and ground water. In other words the UZ model is automatically replaced by a simple Darcy flow description when the profile becomes completely saturated.

If the groundwater level is below the ground surface, then the vertical infiltration is determined by the minimum of the moisture dependent hydraulic conductivity (from the soils database) and the leakage coefficient.

This option is often useful under lakes or on flood plains, which may be permanently or temporarily flooded, and where fine sediment may have accumulated creating a low permeable layer (lining) with considerable flow resistance. It is also used to simulate soil compaction and paved areas.

The value of the leakage coefficient may be found by model calibration, but a rough estimate can be made based on the saturated hydraulic conductivities of the unsaturated zone or in the low permeable sediment layer, if such data is available.

The leakage coefficient is also applied to the vertical leakage from ponded water to the UZ model. In this case, the vertical leakage is a series calculation based on the harmonic mean of the UZ hydraulic conductivity and the leakage coefficient.

The specified leakage coefficient is used wherever it is specified. In areas where a delete value is specified, it is ignored.

In the processed data, the item, Surface-Subsurface Exchange Grid Code, is added, where areas with full contact are defined with a 0, and areas with reduced contact are defined with a 1.



Note: A time-varying value can be specified by using a time-varying dfs2 file and checking ON the time-varying option to support reduced infiltration due to frozen soil in the winter.

Related Items:

- Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes *(p. 492)*
- Mean-Step-Accumulated time varying parameters *(p. 777)*
- Groundwater exchange with Rivers *(p. 485)*
- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*

11.16.5 Sub-areas for reduced ponded exchange

Sub-areas for reduced ponded exchange	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue AND Surface-subsurface leakage coefficient AND In specified areas, reduce exchange only for ponded areas are both selected in the main Overland Flow dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

An additional sub-item is also available that allows you to apply the exchange coefficient in sub-areas where it is only used when water is ponded. During rainfall, this will not have an effect, as the rainfall is always added to ponded storage first. However, after the rainfall, this will allow water to flow downhill along a drainage pathway with a reduced infiltration rate.

Related Items:

- Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes *(p. 492)*
- Groundwater exchange with Rivers *(p. 485)*
- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*

11.16.6 Paved Area Fraction

Paved Area Fraction	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue and Reduced vertical exchange by paved areas is selected in the main Overland Flow dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Fraction

Paving is common in urban areas and has a significant impact on infiltration and runoff. This option allows you to additionally specify a paved area fraction that further reduces the infiltration in the cell by the fraction of paving.

When a Paved Area Fraction is specified, it is used as a linear scaling fraction for the Surface-Subsurface Leakage Coefficient. That is, the effective leakage coefficient is reduced by the Paved Area Fraction.

$$EffLeakCoef = (1\text{-}PAreaFrac) \times SurfSubSurfLeakCoef$$



Note: A time-varying value can be specified by using a time-varying dfs2 file and checking ON the time-varying option.

Related Items:

- Overland Flow - Technical Reference *(p. 515)*
- Unsaturated Zone *(p. 763)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*

11.16.7 Separated Flow Areas

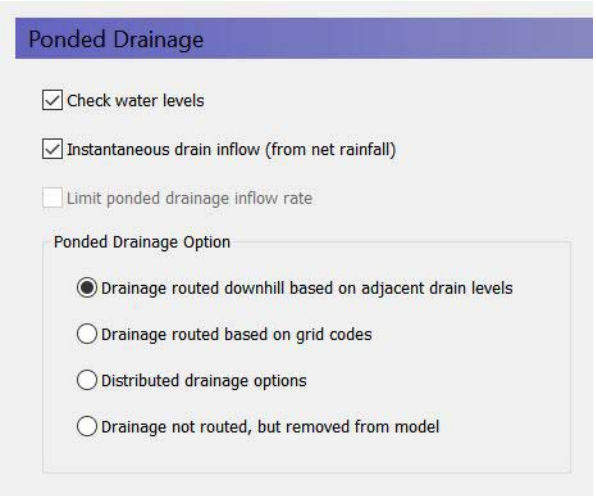
Separated Flow Areas	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue and Separated Flow Areas selected in the main Overland Flow dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

By specifying separated overland flow areas you can simulate areas that are separated by dikes or embankments. Separated overland flow areas are defined by specifying a .dfs2 Integer Grid Code file, containing a unique code value for each flow area. The model will then disable overland flow between grids with different flow codes. Thus, embankments can be simulated by defining different flow codes on each side of the embankment. Legal code values are 1 and higher. Delete values are not allowed inside the model area.

Related Items:

- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*

11.16.8 Overland Drainage



The OL Drainage function requires a reference system for linking the drainage to a recipient node or cell. The recipient can be a river node, another OL grid cell, or a model boundary.

The Overland Drainage module is complex. The section Overland Drainage *(p. 558)* includes a lot of helpful information on using this module

Check water levels - This means that the destination water level must be lower than the ponded water level in the cell.

Instantaneous drain inflow (from net rainfall) - If this option is ON (default), then the OL Drainage is instantaneous, and it will only be taken from net precipitation, disregarding any ponded water. If this option is OFF, then you must specify an Inflow Time Constant (new data tree item), which controls the rate of inflow to the drain. In other words, some ponding will be retained on the surface at the end of the time step - even if the time constant is very high.

Limit OL drainage inflow rate - You can optionally also specify a maximum inflow rate. If the Limited ponded drainage inflow rate option is ON, then a new data tree item for the maximum inflow rate to the drain will be added. This supports culverts, etc. that restrict drain inflow.

There are four different options for setting up the drainage source-recipient reference system

- **Routed downhill based on drain levels** - routed to the nearest downhill river, boundary or internal depression
- **Routed based on grid codes** - routed to the nearest river or boundary with the same grid code.

- **Distributed drainage options** - routing to specified River Links or sewer manholes
- **Drainage not routed, but removed**

Related Items

- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Overland Drainage *(p. 558)*
- Unsaturated Zone *(p. 763)*

Runoff Coefficient

Runoff Coefficient	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Overland Drainage selected in Overland Flow dialogue
dialogue Type:	Stationary Real Data
EUM Data Units	Fraction

The Runoff Coefficient defines the fraction of calculated drain flow that will effectively be routed to storm sewers and other surface drainage features in paved areas.

The Runoff Coefficient acts in two ways:

1. it tells MIKE SHE where it should be applied, and
2. the value specifies how much should be 'drained away' before the remaining water is subject to other processes like infiltration, runoff, etc.

Thus, in non-drainage cells, you should use a value of zero or an Undefined Value (-1e-35 by default). Whereas, in drainage cells, you must specify a value greater than 0 and less than or equal to 1.

If the option "instantaneous drain inflow (from net rainfall)" is not selected, the coefficient value is rather a scaling factor than a true runoff coefficient. In this case the flow into the drainage system will also be controlled by the inflow time constant, so the coefficient could be seen as the area percentage contributing to drainage.

If the "instantaneous drain inflow (from net rainfall)" option is selected, the parameter comes close to a true runoff coefficient: Any ponded water will be left on the cell, and only the specified fraction of net rainfall will be routed to the drain without any delay. Note that interception storage and when using

the Richards- and gravity UZ-solvers also evapotranspiration have already been calculated at this point, so the specified fraction will be taken from the resulting net precipitation.



Note: A time-varying value can be specified by using a time-varying dfs2 file and checking ON the time-varying option.

Related Items

- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Overland Drainage *(p. 558)*
- Unsaturated Zone *(p. 763)*

Maximum Storage Change Rate

Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Overland Drainage selected in Overland Flow dialogue
dialogue Type:	Stationary Real Data
EUM Data Units	Storage change rate

If the **Specify max rate of change in storage depth** option is checked, then MIKE SHE will restrict the rate at which ponded water is drained to the paved drainage function. This allows you to retain ponded water on a cell and drain it away at a specified rate to the river.



Note: A time-varying value can be specified by using a dfs2 file and checking ON the time-varying option.

Related Items

- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Overland Drainage *(p. 558)*
- Unsaturated Zone *(p. 763)*

Inflow Time Constant

Inflow Time Constant	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Overland Drainage selected in Overland Flow dialogue and Drain Inflow Retention is ON
dialogue Type:	Stationary Real Data
EUM Data Units	Leakage coefficient/Drain time constant

This is the time constant that controls the inflow rate to the Overland drainage storage.



Note: A time-varying value can be specified by using a time-varying dfs2 file and checking ON the time-varying option.

Related Items

- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Overland Drainage *(p. 558)*
- Unsaturated Zone *(p. 763)*

Outflow Time Constant

Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Overland Drainage selected in Overland Flow dialogue
dialogue Type:	Stationary Real Data
EUM Data Units	Leakage coefficient/Drain time constant

This is the time constant that controls the outflow rate from the Overland Drainage storage to the recipient.



Note: A time-varying value can be specified as an Extra Parameter to support changing paving conditions over time.

Related Items

- Overland Flow - Technical Reference *(p. 515)*

- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Overland Drainage *(p. 558)*
- Unsaturated Zone *(p. 763)*

Drain Level

Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Overland Drainage selected in Overland Flow dialogue, and the Drain Level option selected
dialogue Type:	Stationary Real Data
EUM Data Units	Height above ground

This is the level that controls the source - destination reference system, the sill for flow into the drain (if the drain level is above ground) as well as the bottom of the drain storage for determining the head difference between the drain storage and the recipient (if the option "Check water levels" is ON).

Note: A time-varying value can be specified by using a dfs2 file and checking ON the time-varying option. However, this will exclude using any of the drainage distribution methods that rely on the Drain Level.

Related Items

- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Overland Drainage *(p. 558)*
- Unsaturated Zone *(p. 763)*

Drain Codes

Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Overland Drainage selected in Overland Flow dialogue, and the Drain code option selected
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

The drainage reference system will be created only within a common drain code.

Related Items

- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Overland Drainage *(p. 558)*
- Unsaturated Zone *(p. 763)*

Drainage Option Distribution

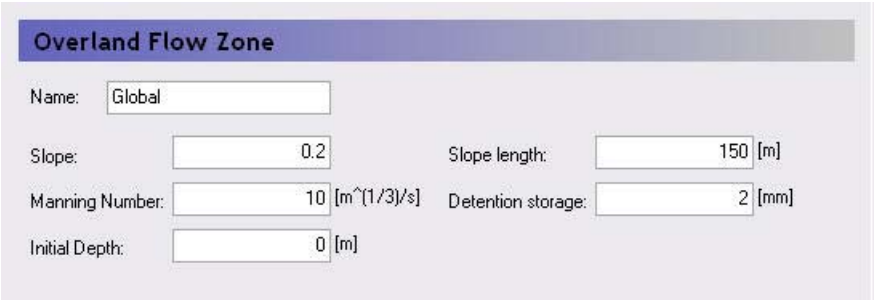
Maximum storage change rate	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Overland Drainage selected in Overland Flow dialogue, and the Distributed option selected
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

This allows you to defined drainage to specified River Links or sewer manholes, via an Extra Parameter option.

Related Items

- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Overland Drainage *(p. 558)*
- OL Drainage to Specified River H-points *(p. 761)*
- Unsaturated Zone *(p. 763)*

11.16.9 Overland Flow Zones



Overland Flow Zones	
Conditions:	If Overland Flow + the Subcatchment-based method is chosen in the Simulation Specification dialogue
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The Overland flow zones are used to defined the topographic zones for the simple, catchment-based overland flow solution. The overland flow zones are typically defined as areas with similar topography. For example, the river flood plain would be a typical topographic zone, although it might be included in many subcatchments.

Each unique grid code in the Overland Flow Zones map will generate a sub-item in the data tree where the following parameters must be specified.

Slope - The Slope variable is a representative slope in the topographic area. It can be thought of as the average slope, but it is really a calibration parameter as it can't really be calibrated to a true average.

Slope Length - Like the Slope itself, the Slope Length is a representative distance that water must travel as overland flow before reaching a ditch or stream. It can be thought of as the average travel distance, but, like the Slope, it is really a calibration parameter as it can't really be calibrated to a true average.

Manning Number - The Manning M is equivalent to the Stickler roughness coefficient. The Manning M is the inverse of the more conventional Mannings n. The value of n is typically in the range of 0.01 (smooth channels) to 0.10 (thickly vegetated channels). This corresponds to values of M between 100 and 10, respectively. Generally, lower values of Mannings M are used for overland flow compared to channel flow.

Detention Storage - Detention Storage is used to limit the amount of water that can flow over the ground surface. For example, if the detention storage is set equal to 2mm, then the depth of water on the surface must exceed 2mm before it will be able to flow as overland flow. Water trapped in detention storage continues to be available for infiltration to the unsaturated zone and to evapotranspiration. Using detention storage, you can simulate water that is trapped in depressions that are smaller in area than a grid cell.

Initial Depth - This is the initial condition for the overland flow calculations, that is the initial depth of water on the ground surface.

Related Items:

- Simplified Overland Flow Routing *(p. 533)*
- Overland Flow - Technical Reference *(p. 515)*
- Working with Overland Flow and Ponding- User Guide *(p. 541)*

11.16.10Initial Mass (per Species)

Initial Mass	
Condition	when water quality for Overland Flow is selected in the Water Quality Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Mass per unit area [e.g. g/m^2]

The initial mass for overland transport is given as a surface concentration [e.g kg/m^2]. This makes it easier to control the mass of solute introduced because you do not have to consider surface water depth.

Related Items:

- Solute Transport in Overland Flow *(p. 693)*
- Initial Conditions *(p. 696)* in Overland Transport

11.16.11 Dispersion coefficient along columns/rows

Dispersion coefficient along columns/rows	
Condition	when water quality for Overland Flow is selected in the Water Quality Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Dispersion coefficient

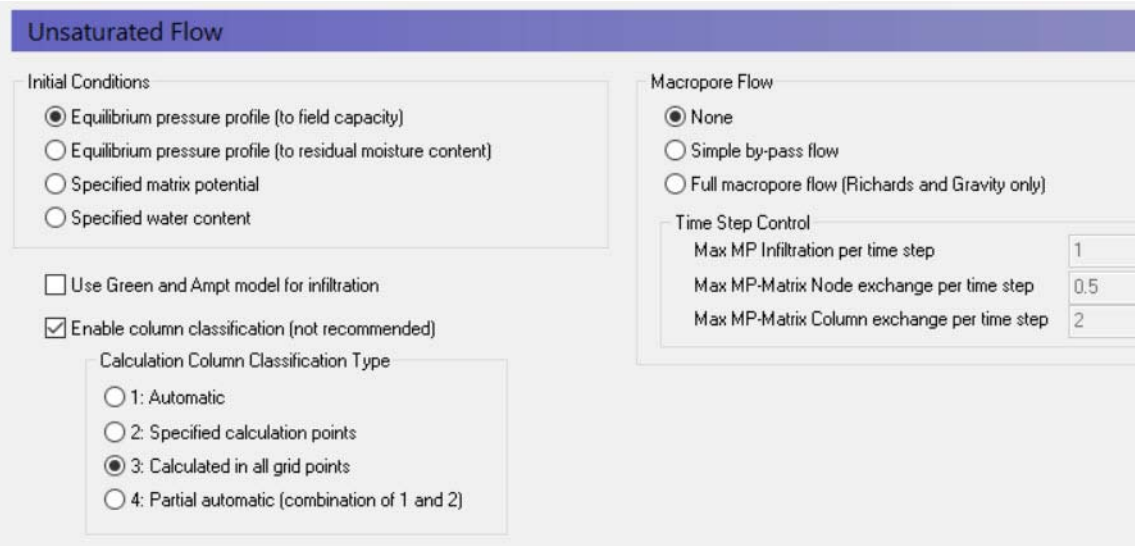
For the overland flow transport, two dispersion coefficients are required - one along the rows and the other along the columns.

Note Unlike for the unsaturated and saturated flow, the overland transport module requires the actual dispersion coefficient [m^2/s] - not the dispersivity [m].

Related Items:

- Solute Transport in Overland Flow *(p. 693)*

11.17 Unsaturated Zone



There are three methods in MIKE SHE to calculate Unsaturated Flow:

- Richards Equation *(p. 576)*,
- the Gravity Flow *(p. 587)*, and

- the Two-Layer Water Balance *(p. 589)*.

The available options in this dialog and the items in the data tree depend on which UZ calculation method chosen in the Simulation Specification *(p. 186)* dialog.

Initial Conditions

The initial conditions section defines the initial moisture content for the soil profile.

- **Equilibrium pressure profile (to field capacity)** This is the default option. In this case, the initial soil moisture distribution follows the water content-pressure relationship from the soil database with a minimum water content equal to the calculated field capacity. In other words, the profile will be saturated at the water table and decrease above the water table until field capacity is reached. All cells above this will have a water content equal to field capacity. This is a reasonable initial condition for most temperate climates, where the water content is regularly recharged by rainfall. Generally, this will lead to an initial drainage from the UZ as the water content equilibrates with the rainfall rate.
- **Equilibrium pressure profile (to residual moisture content)** This option is useful in arid and semi-arid conditions where the natural soil moisture below the root zone is low. When the rate of rainfall is low, then the natural soil moisture distribution will approximate the soil-moisture pressure curve. This option is useful in dry conditions because drainage to dry soil moisture with depth may require a long simulation to reach equilibrium.
- **Specified matrix potential** and **Specified water content** These two options for specifying the water content or matrix potential with depth are typically only used in special conditions where the models are very detailed and accurate soil moisture or pressures have been measured. For example, in column experiments. Specifying these values requires you to define either a uniform value, or values and specified depths.

Green and Ampt Infiltration

The Green and Ampt infiltration 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.

Enable column classification

The column classification should be avoided today because the models have become more complex, MIKE SHE has become more efficient and computers have become faster.

If the either Richards Equation or the Gravity Flow Module are chosen for calculating the unsaturated zone flow then the top-level Unsaturated Zone dia-

logue has an option to turn open a section for the column classification. However, if the Two Layer Method is chosen then the Column Classification section is unavailable.

Calculating unsaturated flow in all grid squares for large-scale applications can be time consuming. To reduce the computational burden MIKE SHE allows you to compute the UZ flow in a reduced subset of grid squares. The subset classification is done automatically by the pre-processing program according to soil and, vegetation distribution, climatic zones, and depth to the groundwater table.

Column classification can decrease the computational burden considerably. However, the conditions when it can be used are limited. Column classification is either not recommended or not allowed when

- the water table is very dynamic and spatially variable because the classification is not dynamic,
- if the 2 layer UZ method is used because the method is fast and the benefit would be limited,
- if irrigation is used in the model because irrigation zones are not a classification parameter, and
- if flooding and flood codes are used, since the depth of ponded water is not a classification parameter

If the classification method is used, then there are three options for the classification:

- **Automatic classification**

  The automatic classification requires a distribution of groundwater elevations (see Groundwater Depths used for UZ Classification). This can be either the initial depth to the groundwater based on the initial heads, or you can supply a .dfs2 map of the groundwater elevations. In both cases, you must supply a table of intervals upon which the classification will be based. The number of computational columns depends on how narrow the intervals are specified. If, for example, two depths are specified, say 1 m and 2 m, then the classification with respect to the depth to groundwater will be based on three intervals: Groundwater between 0 m and 1 m, between 1 m and 2 m, and deeper than 2 m.

  If the Linear Reservoir method is used for the groundwater, then the Interflow reservoirs are also used in the classification. However, since feedback to the UZ only occurs in the lowest Interflow reservoir of each subcatchment, the Interflow reservoirs are added to the Automatic Classification in two zones - those that receive feedback and those that don't.

- **Specified classification**

  Alternatively a data file specifying Integer Grid Codes, where UZ computations are carried out can be specified, with grid codes range from 2 up

to the number of UZ columns (see Specified classification). The location of the computational column is specified by a negative code and the simulation results are then transferred to all grids with the an equivalent positive code.

- **Calculated in all Grid points (default)**

  For smaller scale studies, or studies where the classification system becomes intractable, you can specify that computations are to be carried out in all soil columns.

- **Partial Automatic**

  Finally a combination of the Automatic classification and the Specified classification is available. If this option is chosen an Integer Grid Code file must be provide (see Partial automatic classification) with the following grid codes: In grid points where automatic classification should be used the grid code 1 must be given. In grid points where computation should be performed for all cells the grid code 2 must be given.

Macropore Flow

Flow through macropores in unsaturated soil is important for many soil types.

Simple bypass flow - A simple empirical function is used to describe simple bypass flow in macropores. The infiltration water is divided into one part that flows through the soil matrix and another part, which is routed directly to the groundwater table, as bypass flow.

The bypass flow is calculated as a fraction of the net rainfall for each UZ time step. The actual bypass fraction is a function of a user-specified maximum fraction and the actual water content of the unsaturated zone, assuming that macropore flow occurs primarily in wet conditions.

Typically, macropore flow is highest in wet conditions when water is flowing freely in the soil (e.g. moisture content above the field capacity, θ_{FC}) and zero when the soil is very dry (e.g. moisture content at the wilting point, θ_{WP})

Simple bypass flow is described in the Reference section under Simplified Macropore Flow (bypass flow) *(p. 590)*.

Full Macropore Flow - Macropores are defined as a secondary, additional continuous pore domain in the unsaturated zone, besides the matrix pore domain representing the microporous bulk soil. Macropore flow is initiated when the capillary head in the micropore domain is higher than a threshold matrix pressure head, corresponding to the minimum pore size that is considered as belonging to the macropore domain. Water flow in the macropores is assumed to be laminar and not influenced by capillarity, thus corresponding to gravitational flow.

Full Macropore flow is described in the Reference section under Full Macropore Flow *(p. 591)*.

Max macropore infiltration per time step - This is a stability criteria that prevents too much water from entering the macropores in one time step.

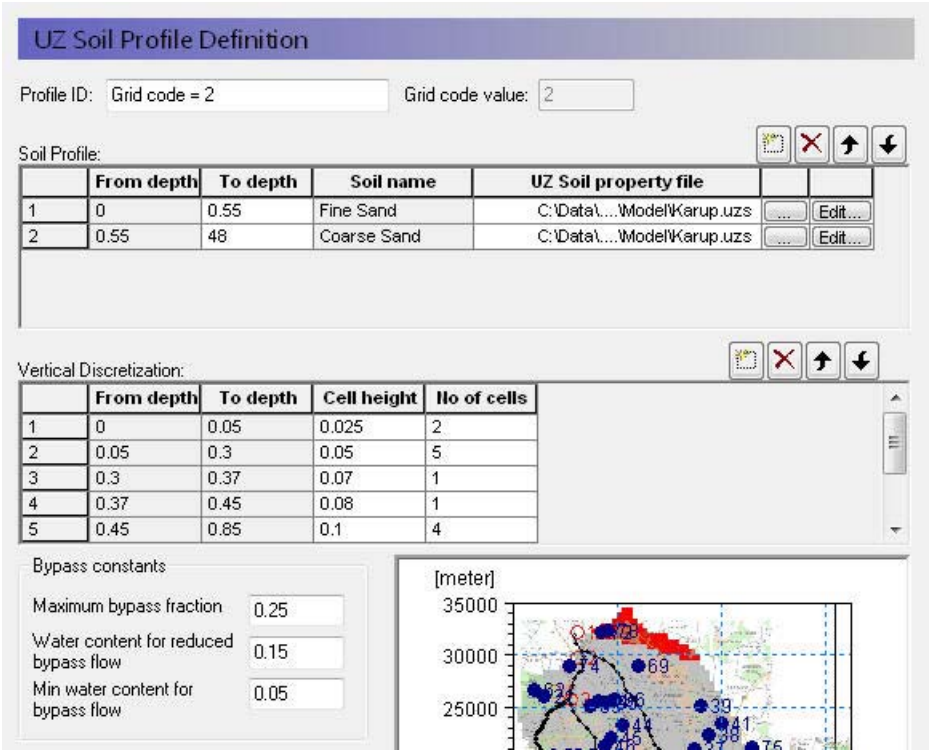
Max macropore matrix exchange per time step - This is a stability criteria that prevents too much water from exchanging between the macropores and the bulk matrix in one node in one time step.

Max macropore-matrix column exchange per time step - This is a stability criteria that prevents too much water from exchanging between the macropores and the bulk matrix in the entire column in one time step.

Related Items:

- Unsaturated Zone - Technical Reference *(p. 575)*
- Richards Equation *(p. 576)*
- Gravity Flow *(p. 587)*
- Two-Layer Water Balance *(p. 589)*
- Green and Ampt Infiltration *(p. 596)*
- Simplified Macropore Flow (bypass flow) *(p. 590)*
- Full Macropore Flow *(p. 591)*
- Coupling the Unsaturated Zone to the Saturated Zone *(p. 601)*

11.17.1 Soil Profile Definitions



Soil Profile Definitions	
Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue and Richards Equation or the Gravity Flow module selected for the numeric engine
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The first part of the soil profile definition is to define the areas with the same soil profiles. This can be done using either a uniform value or distributed using a dfs2 integer grid code file or a polygon .shp file. For each unique grid code or polygon in the distribution file, a separate sub-item will appear in the data tree, in which you can define the soil profile data.

In the sub-item dialogues, the soil type and distribution of the soil layers (i.e. depth and thickness of each soil type) in the individual profiles is specified, as well as the vertical discretisation of the soil profile.

Definition of soil properties

Profile ID - This is the editable name displayed in the data tree for this profile.

Grid code value - This is the current integer grid code or polygon value read from the soil distribution.

Soil Profile - The soil profile section allows you to define the vertical soil profile. Soil layers can be added, deleted and moved up and down using the icons.

From and To Depths refers to the distances to the top and bottom of the soil layer, below the ground surface. Only the To Depth item is editable, as the From Depth item is equal to the bottom of the previous layer.

Soil name is the name of the soil selected in the UZ Soil Property file. It is not directly editable, but must be chosen from the list of available soil names when you assign the UZ Soil property file using the file browser.

UZ Soil property file is the file name of the soil database, in which the soil definition is available. The Edit button opens the specified Soil property database file, whereas the Browse button [...] opens the file browser to select a file.

Vertical discretisation - In this section you specify the vertical discretisation of the soil profile, which typically contains small cells near the ground surface and increasing cell thickness with depth. However, the soil properties are averaged if the cell boundaries and the soil boundaries do not align.

From and To Depths refers to the distances to the top and bottom of the soil layer, below the ground surface. Neither are directly editable since they are calculated from the number of cells and their thicknesses.

Cell Height is the thickness of the numerical cells in the soil profile.

No. of Cells is the number of cells with the specified cell height. Together these two values define the total thickness of the current section.

Dispersivity is the longitutinal dispersivity in the column. This column only appears if WQ is simulated in the unsaturated zone.

The discretisation should be tailored to the profile description and the required accuracy of the simulation. If the full Richards equation is used the vertical discretisation may vary from 1-5 cm in the uppermost grid points to 10-50 cm in the bottom of the profile. For the Gravity Flow module, a coarser discretisation may be used. For example, 10-25 cm

in the upper part of the soil profile and up to 50-100 cm in the lower part of the profile.

Note that at the boundary between two blocks with different cell heights, the two adjacent boundary cells are adjusted during the pre-processing to give a smoother change in cell heights.

Bypass Constants The bypass constants are editable when Simple Bypass flow is checked on in the main Unsaturated Zone *(p. 288)* dialogue. The availabe bypass parameters include:

Maximum bypass fraction - This is the maximum fraction of net rainfall that will infiltrate via simple macropore flow. Valid values are between 0 and 1.0.

Water content for reduced bypass flow - This is the threshold water content below which the bypass fraction is reduced. If the water content 10cm or 50cm below the ground surface is less than this water content, then the soil is dry and the bypass flow will be reduced.

Limit on water content for bypass flow - This is the minimum water content for bypass flow. If the water content 10cm or 50cm below the ground surface is less than this limit, then the soil is very dry and the bypass flow will be zero.

The actual relationship between the bypass constants and the calculation of the bypass flow is described in Simplified Macropore Flow (bypass flow) *(p. 590)*.

Related Items:

- Unsaturated Zone - Technical Reference *(p. 575)*
- Richards Equation *(p. 576)*
- Gravity Flow *(p. 587)*

11.17.2 Mass Transfer Coefficient to Unsaturated Macropores

Mass transfer coefficient to unsaturated macropores	
Condition	when water quality for Unsaturated Flow is selected in the Water Quality Simulation Specification dialogue and Full Macropore Flow is selected in the main Unsatured Flow dialog
dialogue Type	Stationary Real Data
EUM Data Units	1st-order rate WQ model [e.g. 1/day]

Macropores are defined as a secondary, additional continuous pore domain in the unsaturated zone, besides the matrix pore domain representing the microporous bulk soil.

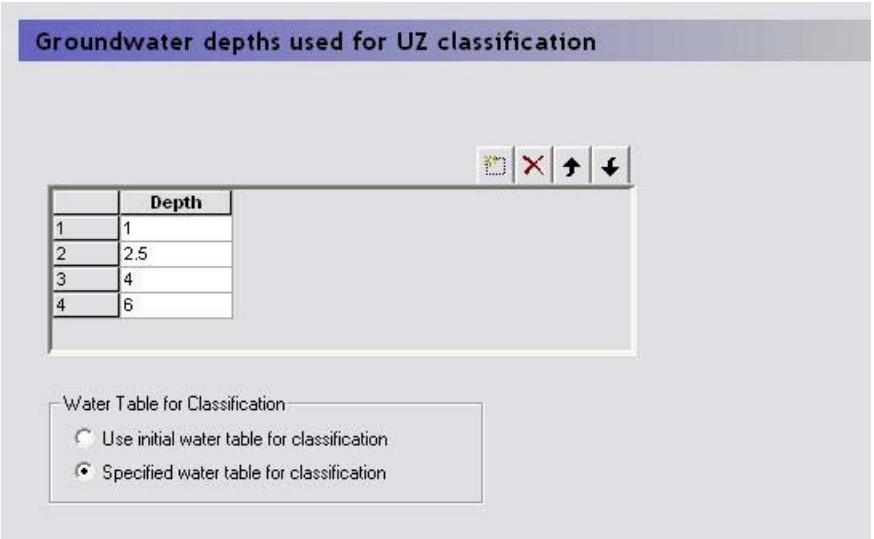
In other words, tthe macropores represent a continous secondary porous media that is “parallel” to the soil matrix porosity. Mass transfer to macropores occurs mostly by means of diffusion from the unsaturated zone soil matrix.

This coefficient represents the rate of transfer from the soil matrix to the macropores. It is defined per species.

Related Items:

Full Macropore Flow *(p. 591)*

11.17.3 Groundwater Depths used for UZ Classification



Groundwater Depths used for UZ Classification	
Conditions:	if either the Automatic or Partially automatic column classifications selected in the main Unsaturated Zone dialogue

The automatic or partially automatic classification requires a distribution of groundwater elevations (see the main Unsaturated Zone dialogue). This can be either the initial depth to the groundwater based on the initial heads (default option), or you can supply a .dfs2 map of the groundwater elevations (second option). If you chose the second option, then a new item, Ground Water Table for UZ classification, will appear in the data tree.

In both cases, you must supply a table of intervals upon which the classification will be based. The number of computational columns depends on how narrow the intervals are specified. If, for example, two depths are specified, say 1 m and 2 m, then the classification with respect to the depth to groundwater will be based on three intervals: Groundwater between 0 m and 1 m, between 1 m and 2 m, and deeper than 2 m.

Related Items:

- Lumped UZ Calculations (Column Classification) *(p. 620)*

11.17.4 Ground Water Table for UZ classification

Groundwater table for UZ classification	
Conditions:	if Specified water table for classification selected in the Groundwater Depths used for UZ Classification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

If the Specified water table for classification is selected in the Groundwater Depths used for UZ Classification dialogue, then this is the ground water table used for the classification.

Related Items:

- Lumped UZ Calculations (Column Classification) *(p. 620)*

11.17.5 Partial automatic classification

Partial Automatic Classification	
Conditions:	if the Partially automatic column classifications selected in the main Unsaturated Zone dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code
Valid Values	1 or 2

A combination of the Automatic classification and the Specified classification is available. If this option is chosen an Integer Grid Code file must be provide with the following grid codes:

- In grid points where automatic classification should be used the grid code must be 1.
- In grid points where computation should be performed for all cells the grid code must be 2.

Related Items:

- Lumped UZ Calculations (Column Classification) *(p. 620)*

11.17.6 Specified classification

Specified Classification	
Conditions:	if Specified classification selected in the main Unsaturated Zone dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code
Valid values	+/- 2 to the number of SZ cells (-1, 0, 1 not valid)

This is a data file specifying Integer Grid Codes, where UZ computations are to be carried out. Grid codes range from 2, up to the number of UZ columns. The location of the computational column is specified by a negative code and the simulation results are then transferred to all grids with the an equivalent positive code.

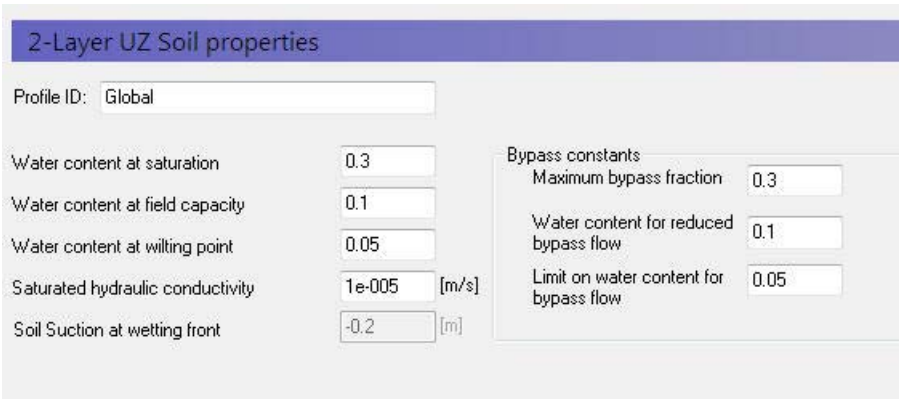
For example, if a grid code holds the value -2, a UZ computation will be carried out for the profile located in that grid. Simulation results will subsequently be transferred to all grid codes with code value +2.

An easy way to generate a .dfs2 file to be used for specification of UZ computational columns is to let the MIKE SHE setup program generate an automatic classification first, and subsequently extract the UZ classification grid codes. The extracted .dfs2 file can be edited in the 2D editor as desired and used to specify UZ computational grids.

Related Items:

- Lumped UZ Calculations (Column Classification) *(p. 620)*

11.17.7 2-Layer UZ soil properties



2-Layer UZ Soil Properties	
Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue and the Two-Layer Water Balance method selected for the numeric engine
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The first part of the soil definition is to define the areas with the same soil profiles. This can be done using either a uniform value or distributed using a dfs2 integer grid code file or a polygon .shp file. For each unique grid code or polygon in the distribution file, a separate sub-item will appear in the data tree, in which you can define the soil profile data.

Definition of soil properties

Profile ID - This is the editable name displayed in the data tree for this profile.

Grid code value - This is the current integer grid code or polygon value read from the soil distribution.

Water content at saturation - This is the maximum water content of the soil, which is usually approximately equal to the porosity,

Water content at field capacity - This is the water content at which vertical flow becomes negligible. It practice, this is the water content that is reached when the soil can freely drain. Although, it is usually higher than the residual saturation, which is usually defined as the minimum saturation that can be achieved in a laboratory test.

Water content at wilting point - This is the lowest water content that plants can extract water from the soil.

Saturated hydraulic conductivity - The saturated hydraulic conductivity of the soil is equal to the maximum infiltration rate of the soil.

Soil suction at wetting front - The soil suction at the wetting front is used in the Green and Ampt infiltration model to increase the infiltration rate beyond the saturated hydraulic conductivity when the soil is dry. This is a soil specific value that accounts for the capillarity of the soil.

Bypass Constants The bypass constants are editable when Simple Bypass flow is checked on in the main Unsaturated Zone *(p. 288)* dialogue. The availabe bypass parameters include:

Maximum bypass fraction - This is the maximum fraction of net rainfall that will infiltrate via simple macropore flow. Valid values are between 0 and 1.0.

Water content for reduced bypass flow - This is the threshold water content below which the bypass fraction is reduced. If the water content 10cm or 50cm below the ground surface is less than this water content, then the soil is dry and the bypass flow will be reduced.

Limit on water content for bypass flow - This is the minimum water content for bypass flow. If the water content 10cm or 50cm below the ground surface is less than this limit, then the soil is very dry and the bypass flow will be zero.

The actual relationship between the bypass constants and the calculation of the bypass flow is described in Simplified Macropore Flow (bypass flow) *(p. 590)*.

Related Items:

- Unsaturated Zone - Technical Reference *(p. 575)*
- Two-Layer Water Balance *(p. 589)*
- Simplified Macropore Flow (bypass flow) *(p. 590)*

11.17.8 ET Surface Depth

ET Surface Depth	
Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue and the Two-Layer Water Balance method selected for the numeric engine
dialogue Type	Stationary Real Data
EUM Data Units	Depth below ground

In unsaturated soils, capillary action can lead to saturated conditions existing some distance above the water table. If the water table is close to the ground surface, ET will continue to occur at the maximum rate, so long as this capillary zone reaches the ground surface. That is, evapotranspiration will not decrease the saturation, but draw water directly from the water table due to capillary action. Similarly, when the water table is deeper, plant roots will draw water directly from the saturated zone as long as the roots reach the capillary zone.

In MIKE SHE, the ET surface depth is the thickness of the capillary fringe. This thickness is added to the root depth to define the thickness of Layer 1 in the Two-Layer Water Balance ET/UZ method.

In the absence of vegetation, if the water table is above this depth then soil evaporation will draw water from the water table (SZ). However, the rate that it can remove water from the water table decreases linearly with depth. That is, if the water table is at the ground surface it will remove water at the full rate and if the water table is at the ET Surface Depth then the rate will go to zero.

This is exactly the same function as MODFLOW, except MODFLOW does not describe it like this. In MODFLOW, because there is no vegetation, MODFLOW users typically use the linear function to also “simulate” root zone ET.

In MIKE SHE, if vegetation is present then the exact same function is used, but instead of being relative to the soil surface, it is relative to the bottom of the root zone. If the water table is at or above the bottom of the root zone then ET will be remove from SZ at the full rate. As the water table falls below the root zone, ET from SZ will be reduced linearly until the water table is at distance below the root zone equal to the ET Surface depth – at which point the ET from SZ becomes zero.

Note, that Layer 1 in the 2-Layer Water Balance UZ/ET method is the thickness of the root zone plus the ET Surface depth. If the water table falls below the bottom of the Layer 1, then ET from the SZ will stop. This does not mean that ET stops. ET will continue to remove water from the available water in

Layer 1. However, Layer 1 is greater than the root thickness, which means that the available storage is increased by the thickness of the ET Surface Depth. This is probably not significant when the roots are thick relative to the thickness of the capillary fringe. However, if the capillary fringe is thick, say more than 1m in a silty-clay, then the amount of water available in Layer 1 may be artificially high.

Richards equation, there is no ET Surface depth, because the capillary thickness is defined by the Sat-Pressure curve. In the Gravity method, the capillarity is ignored.

In coarse to medium sands, the ET surface depth is typically less than 10cm. In fine sands and silts, the ET surface depth could be a half a metre or more.

Note The ET surface depth must be greater than or equal to zero.

Related Items:

- Unsaturated Zone - Technical Reference *(p. 575)*
- Two-Layer Water Balance *(p. 589)*

11.17.9 Initial Conditions (UZ)

Initial conditions for the unsaturated zone	
Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue AND either The Water Quality is turned on for the unsaturated zone, OR An initial pressure or saturation has been selected for the Richards/Gravity flow solution method.
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Concentration or Temperature

There are two different initial conditions for the unsaturated zone.

Water movement - For water flow in the unsaturated zone (Richards/Gravity methods only), the initial conditions can be defined to be in equilibrium, which means the the initial pressure head and water content are derived from the soil pressure-moisture content relationship, or you can specify a water content or water pressure directly. The choice of initial condition is defined in the main Unsaturated Zone *(p. 288)* dialogue.

Water quality - For the water quality module, an initial concentration is required for each mobile and sorbed species. Furthermore, if temperature

dependent decay is specified, then an initial soil temperature must also be specified.

The initial conditions for the unsatured zone can be defined as a constant value, or by layer, or from a fully 3D dfs3 file.

Related Items:

- Unsaturated Zone - Technical Reference *(p. 575)*

11.18 Groundwater table for lower UZ boundary

Groundwater table for lower UZ boundary	
Conditions	If Unsaturated Flow selected in the Simulation Specification dialogue without selecting Saturated Flow, OR If the Unsaturated Flow is selected and the Linear Reservoir Method for groundwater is selected.
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The groundwater table must be explicitly defined if unsaturated flow is simulated without explicitly simulating groundwater flow. The specified groundwater table is used as the lower boundary condition for the unsaturated model.

If the Linear Reservoir Method is used for the groundwater simulation, the water table is not calculated, thus requiring the water table to be explicitly defined.

However, the specified groundwater table is a static variable. If you need to relate your unsaturated zone model to a dynamic water table, you must include the saturated zone in your model based on the 3D Finite Difference Method.

Related Items:

- Saturated Flow - Technical Reference *(p. 623)*
- Linear Reservoir Method *(p. 638)*

11.19 Saturated Zone

In MIKE SHE, the saturated zone is only one component of an integrated groundwater/surface water model. The saturated zone interacts with all of the other components - overland flow, unsaturated flow, channel flow, and evapotranspiration.

By comparison, MODFLOW only simulates the saturated flow. All of the other components are either ignored (e.g. overland flow) or are simple boundary conditions for the saturated zone (e.g. evapotranspiration).

On the other hand, there are very few difference between the MIKE SHE Finite Difference method and MODFLOW. The differences that are present are limited to differences in the discretisation and to some differences in the way some of the boundary conditions are defined.

Main saturated zone dialogue

The top dialog for the saturated zone section is slightly different depending on whether or not the Linear Reservoir Method *(p. 305)* or the 3D Finite Difference Method *(p. 306)* is chosen.

11.19.1 Linear Reservoir Method



Setting up a saturated groundwater model using the Linear Reservoir Method involves defining the Interflow and Baseflow Reservoirs, as well as their respective properties.

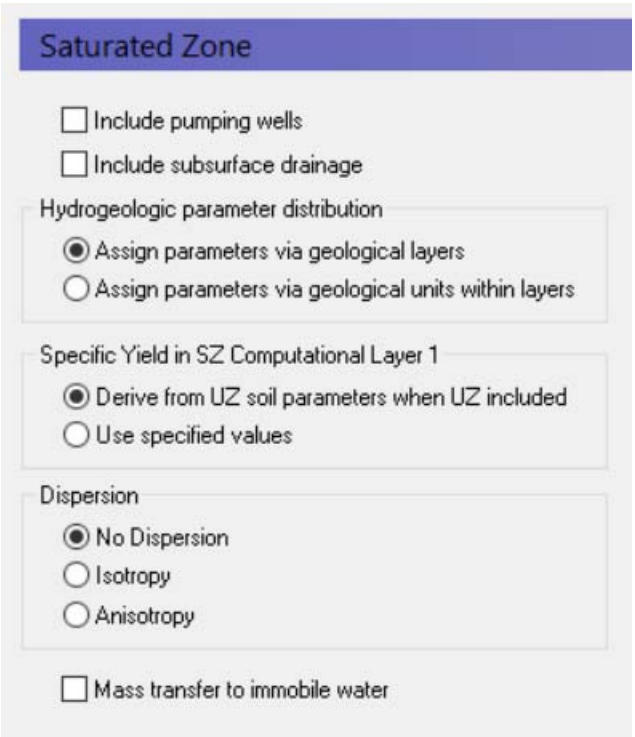
Pumping wells

By default, wells are not included, but in most applications pumping wells play a major role in the hydrology of the area. If wells are included in the model, then this must be checked and a new item in the data tree appears where the well database can be defined. Pumping wells extract water only from the baseflow reservoirs.

Related Items:

- Saturated Flow - Technical Reference *(p. 623)*
- Linear Reservoir Method *(p. 638)*

11.19.2 3D Finite Difference Method



Setting up a saturated zone hydraulic model based on the 3D Finite Difference Method involves defining the:

- the geological model,
- the vertical numerical discretisation,
- the initial conditions, and
- the boundary conditions.

In the MIKE SHE GUI, the geological model and the vertical discretisation are essentially independent, while the initial conditions are defined as a property of the numerical layer. Similarly, subsurface boundary conditions are defined based on the numerical layers, while surface boundary conditions such as wells, drains and rivers are defined independently of the subsurface numerical layers.

The use of grid independent geology and boundary conditions provides a great deal of flexibility in the development of the saturated zone model, thus the same geological model and many of the boundary conditions can be re-used for different model discretisation and different model areas.

Include Pumping wells
By default, wells are not included, but in most applications pumping wells play a major role in the hydrology of the area. If wells are included in the model, then this must be checked and a new item in the data tree appears where the well database can be defined.

Include Subsurface drainage
Subsurface drainage is used to limit the amount of groundwater that reaches the ground surface and to route near surface groundwater to local streams and rivers. There are a number of drainage options for specifying surface drains in MIKE SHE, which are described in more detail in the section Drainage.

Hydrogeologic parameter distribution

The first option allows you to specify the hydrologic parameters of the geologic layers and lenses directly by means of .dfs2 grid files, point/line theme .shp files, or irregular xyz point values. The second option allows you to assign the hydrologic parameters to the geologic layers by means of zones with uniform properties, whereby the zones are defined by integer grid codes.

Specific Yield in SZ Computational Layer 1

By default, the specific yield in the top SZ layer is calculated from the specified UZ field capacity and saturated water content in the UZ layer that contains the initial water table. This is done to ensure the consistency between the SZ specific yield and the UZ properties. If these are inconsistent, then you may have a situation where the change in water table elevation in the UZ and the SZ models will be different.

You must specify a Specific Yield value for every SZ layer including the top SZ layer. However, when a UZ model is included in MIKE SHE, by default this value is not used. The Use specified values option allows you to over-ride the default. This may be a useful option to invoke if, for example, you are using Autocalibration. In this case, you can define and optimize the Specific Yield directly.

Dispersion

If your simulation includes water quality modelling in the saturated zone, then you must also define the type of dispersion you want to simulate. Dispersion is the physical process that causes solutes to spread longitudinally, vertically and horizontally as they move through the soil. The dispersion essentially represents the natural, microscopic variations in pore geometry that cause small scale variations in flow velocity.

No dispersion - Dispersion is ignored and no dispersivities need to be specified.

Isotropic - The transverse horizontal and transverse vertical dispersivities are assumed to be the same. Only two dispersivities need to be specified - the longitudinal and the transverse dispersivities.

Anisotropic - The horizontal and vertical transverse dispersivities are different, which requires the specification of five different dispersivities.

Mass Transfer to Immobile Water

If many cases, immobile water trapped in the bulk soil matrix is both a source and sink of solutes. In other words, as solutes move in the bulk water media, some of the solutes will diffuse into water trapped in the pores of the soil grains.

Furthermore, solutes in fractured media will be transported by diffusion in and out of the soil matrix. The porosity of the soil matrix is often at least ten times less than the fractures. The diffusion of solutes into the soil matrix will retard the initial breakthrough of the solute and the matrix will act as a long term sink as the solute slowly diffuses out of the matrix long after the main solute plume has passed by.

If you turn on this option, then four additional data items may be required, namely

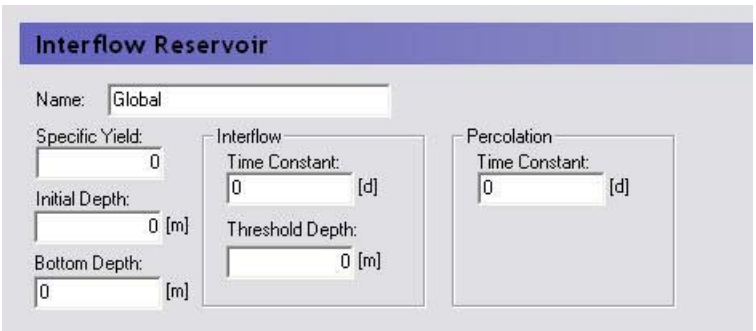
- the Secondary Porosity *(p. 318)* will be added as a geologic property,
- a Dual Porosity transfer coefficient *(p. 322)* per species will be required for each Water Quality layer,
- a Macropore/Secondary Half Life *(p. 357)* coefficient will be required for each species to calculate the solute decay in the matrix, and
- an initial concentration in the matrix for each species.

Note: there is no separate sorption process for the immobile water.

Related Items:

- Saturated Flow - Technical Reference *(p. 623)*
- Solute Transport in the Saturated Zone *(p. 677)*

11.19.3 Interflow Reservoirs



The interflow reservoirs are used to route near-surface groundwater to local streams. In the Linear Reservoir Method, each reservoir is assumed to be like a bathtub, with an inflow from infiltration and the upstream reservoir, as well as an outflow flowing into the next downstream reservoir and down into the baseflow reservoir beneath. Each linear reservoir flows only into the next downstream interflow reservoir, or into a stream if it is the lowest reservoir.

Note Polygon shape files are currently not allowed for defining interflow reservoirs. The flow reference between interflow reservoirs depends precisely on the integer code numbers assigned to the reservoirs. Within a subcatchment, the interflow reservoir with the higher number always flows into the reservoir with the next lowest number.

Each Interflow reservoir requires a value for:

Specific Yield - to account for the fact that the reservoir contains a porous media, and is not an actual bathtub.

Initial depth - the initial depth of water in the reservoir, measured from the ground surface.

Bottom depth - the depth below the ground surface of the bottom of the reservoir. If the water level drops to the bottom of the reservoir, percolation stops.

Interflow time constant - a calibration parameter that represents the time it takes for water to flow through the reservoir to the next reservoir.

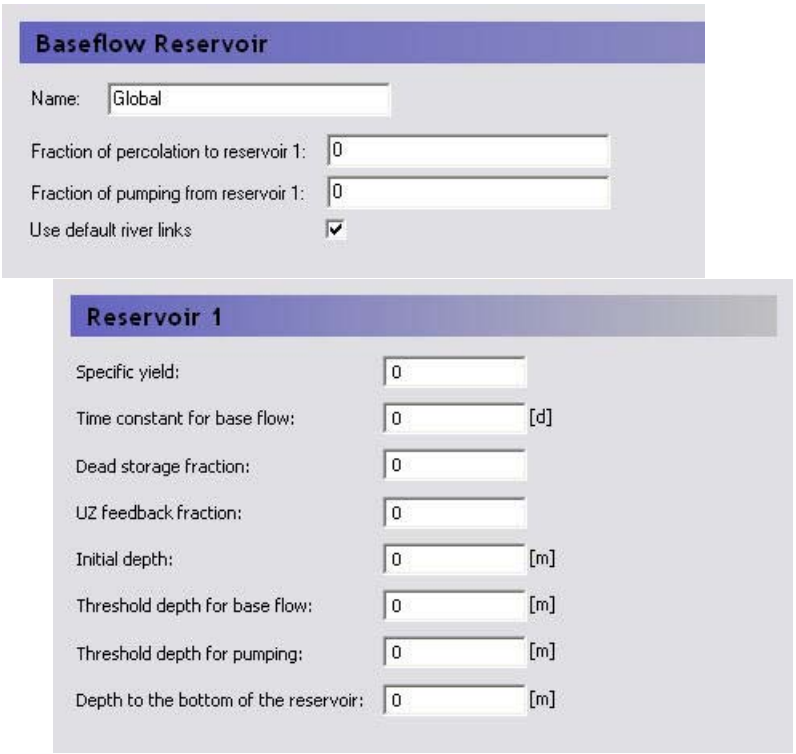
Percolation time constant - a calibration parameter that represents the time it takes for water to seep down into the baseflow reservoir

Interflow threshold depth - the depth below the ground surface when interflow stops. If interflow stops, percolation will continue until the reservoir is empty (i.e the water level reaches the bottom depth). The threshold depth must be less than or equal to the depth to the bottom of the reservoir.

Related Items:

- Saturated Flow - Technical Reference *(p. 623)*
- Linear Reservoir Method *(p. 638)*
- Calculation of Interflow *(p. 644)*

11.19.4 Baseflow Reservoirs



In the Linear Reservoir Method in MIKE SHE, each baseflow reservoir is divided into two parallel baseflow reservoirs. The two parallel baseflow reservoirs each receive a fraction of the percolation water from the interflow reservoirs as their only source of inflow. Each baseflow reservoir can discharge to pumping wells, to the unsaturated zone adjacent to streams and rivers (i.e. the zone beneath the lowest interflow reservoir), as well as directly to the river network.

In the primary baseflow reservoir map view, you can define the number of baseflow reservoirs in your system. You can define any number of baseflow reservoirs, but typically, there are only one or two.

For each baseflow reservoir pair, there are three items to define:

Fraction of percolation to reservoir 1 - this is used to divide the percolation between each of the two parallel baseflow reservoirs.

Fraction of pumping from reservoir 1 - this is used to divide the pumping (if it exists) between each of the two parallel baseflow reservoirs.

Use default river links - in most cases you will link the simplified overland flow and the groundwater interflow to all of the river links found in the

lowest interflow reservoir in each subcatchment. However, in some cases you may want to link the flow to particular river links. For example, if your river network does not extend into the subcatchment, you can specify that the interflow discharges to a particular node or set of nodes in a nearby river network.

If you uncheck this checkbox, a River Links *(p. 231)* sub-item will appear where you can specify the river branch and chainage to link the subcatchment to.

In the sub-dialogue for each of the parallel baseflow reservoirs, you must define the following:

Specific Yield - to account for the fact that the reservoir contains a porous media and is not an actual bathtub.

Time constant for base flow - a calibration parameter that represents the time it takes for water to flow through the reservoir

Dead storage fraction - the fraction of the received percolation that is not added to the reservoir volume but is removed from the available storage in the reservoir.

UZ feedback fraction - the fraction of base flow to the river that is available to replenish the water deficit in the unsaturated zone adjacent to the river (i.e. the lowest interflow reservoir in the subcatchment).

Initial depth - the initial depth to the water in the reservoir measured from the ground surface

Threshold depth for base flow - the depth below the ground surface when base flow stops. The threshold depth must be less than or equal to the depth to the bottom of the reservoir.

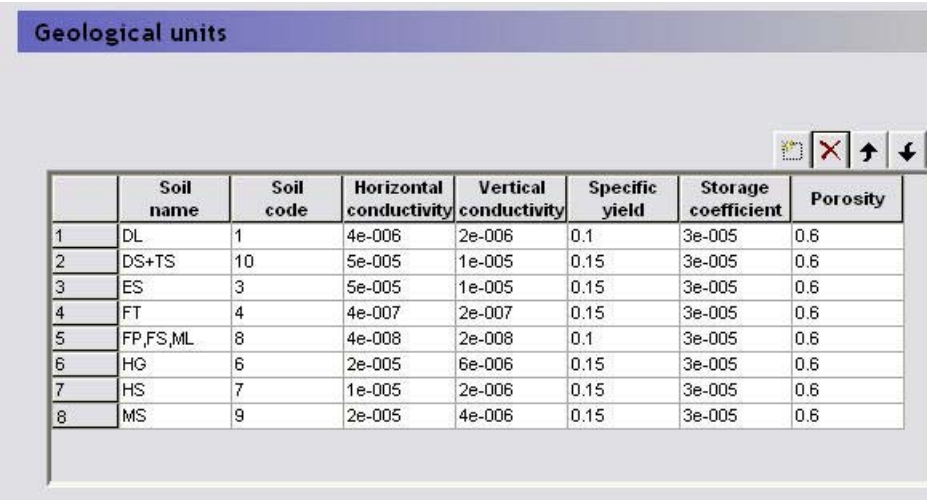
Threshold depth for pumping -the depth below the ground surface when pumping is shut off. The threshold depth must be less than or equal to the depth to the bottom of the reservoir.

Depth of the bottom of the reservoir - the depth below the ground surface of the bottom of the reservoir.

Related Items:

- Saturated Flow - Technical Reference *(p. 623)*
- Linear Reservoir Method *(p. 638)*
- Calculation of Baseflow *(p. 647)*

11.19.5 Geological Units



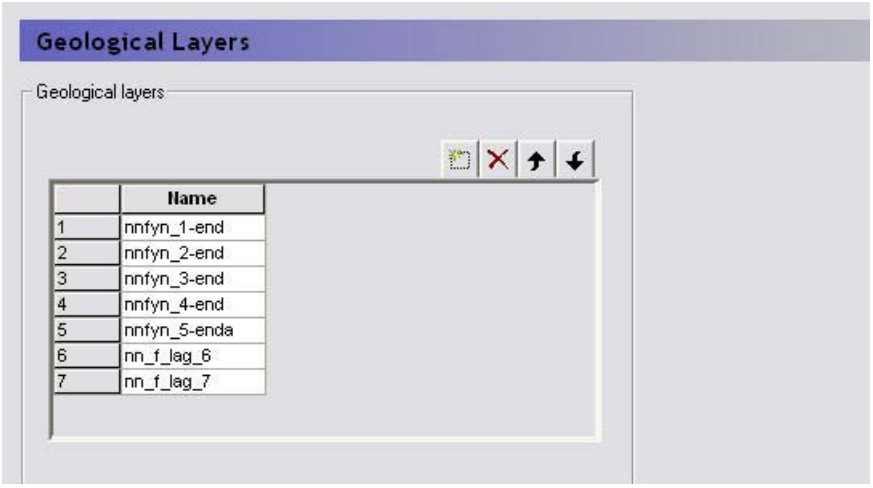
	Soil name	Soil code	Horizontal conductivity	Vertical conductivity	Specific yield	Storage coefficient	Porosity
1	DL	1	4e-006	2e-006	0.1	3e-005	0.6
2	DS+TS	10	5e-005	1e-005	0.15	3e-005	0.6
3	ES	3	5e-005	1e-005	0.15	3e-005	0.6
4	FT	4	4e-007	2e-007	0.15	3e-005	0.6
5	FP,FS,ML	8	4e-008	2e-008	0.1	3e-005	0.6
6	HG	6	2e-005	6e-006	0.15	3e-005	0.6
7	HS	7	1e-005	2e-006	0.15	3e-005	0.6
8	MS	9	2e-005	4e-006	0.15	3e-005	0.6

If you specify your geologic conceptual model via geological units, you can add each of your geologic units and its associated hydrogeologic properties to the table. Then, instead of specifying the hydrogeologic properties for each geological layer, you only need to specify the distribution of the units within the geologic layer or lense.

Related Items:

- Horizontal Hydraulic Conductivity *(p. 315)*
- Vertical Hydraulic Conductivity *(p. 316)*
- Specific Yield *(p. 316)*
- Specific Storage *(p. 317)*
- Porosity *(p. 318)*
- Dispersion Coefficients LHH, THH, TVH, LVV, THV *(p. 320)*

11.19.6 Geological Layers



For each geologic layer, you must specify the hydrogeologic parameters of the layer including

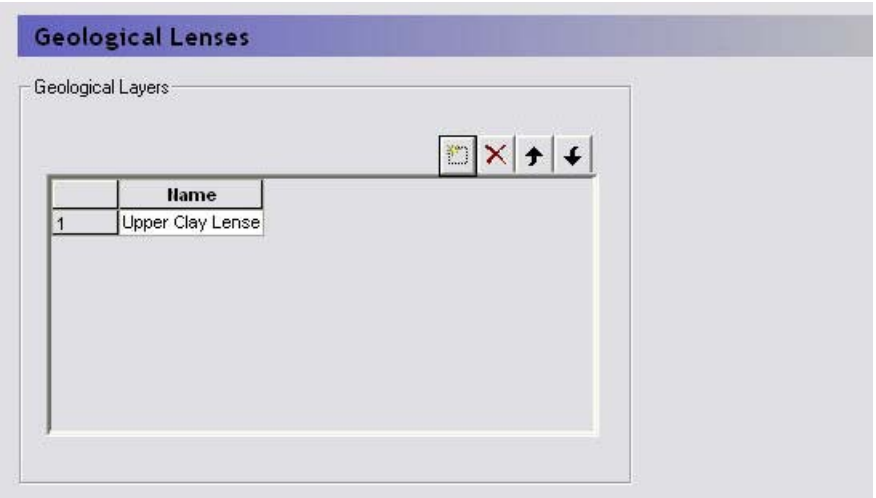
- Lower Level *(p. 355)*,
- Horizontal Hydraulic Conductivity *(p. 315)*,
- Vertical Hydraulic Conductivity *(p. 316)*,
- Specific Yield *(p. 316)*
- Specific Storage *(p. 317)*,

If you define your hydrogeology by , then most of the physical properties will be defined as properties of the Geological Unit and there will be an additional item, the Geological Unit Distribution, in the data tree.

Related Items:

- Saturated Flow - Technical Reference *(p. 623)*

11.19.7 Geological Lenses



For each geologic layer, you must specify the hydrogeologic parameters of the layer including

- Lower Level *(p. 355)*,
- Upper Level *(p. 355)*,
- Horizontal Extent *(p. 356)*,
- Horizontal Hydraulic Conductivity *(p. 315)*,
- Vertical Hydraulic Conductivity *(p. 316)*,
- Specific Yield *(p. 316)*,
- Specific Storage *(p. 317)*,

If you define your hydrogeology by , then most of the physical properties will be defined as properties of the Geological Unit and there will be an additional item, the Geological Unit Distribution, in the data tree.

Related Items:

- Lenses *(p. 64)*

11.19.8 Geological Unit Distribution

Geological Unit Distribution	
Conditions:	If geology defined by Geologic Units
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code
Valid Values	each value must be in the Geological Units table

The Geological Unit Distribution references the geological units defined in the table. Each Integer Code must refer to one of the geological units in the table.

Related Items:

- Geological Units *(p. 312)*
- Lenses *(p. 64)*

11.19.9 Horizontal Hydraulic Conductivity

Horizontal Hydraulic Conductivity	
dialogue Type	Stationary Real Data
EUM Data Units	HydrConductivity

The hydraulic conductivity is a function of the soil texture and is related to the ease with which water can flow through the soil. Loose, coarse uniform soils have a higher conductivity than compacted soils with a range of particle sizes. Thus, a loose, uniform coarse sand can have a horizontal hydraulic conductivity as high as 0.001 m/s. Whereas, a tight, compacted clay can have a have a horizontal hydraulic conductivity as low as $1x10^{-8}$ m/s - which is 5 orders of magnitude.

The horizontal hydraulic conductivity is typically 5 to 10 times higher than the vertical hydraulic conductivity.

MIKE SHE assumes that the horizontal conductivity is isotropic in the x and y directions

Related Items:

- Geological Units *(p. 312)*
- Vertical Hydraulic Conductivity *(p. 316)*

- Lenses *(p. 64)*

11.19.10Vertical Hydraulic Conductivity

Vertical Hydraulic Conductivity	
dialogue Type	Stationary Real Data
EUM Data Units	HydrConductivity

The hydraulic conductivity is a function of the soil texture and is related to the ease with which water can flow through the soil. Loose, coarse uniform soils have a higher conductivity than compacted soils with a range of particle sizes. Thus, a loose, uniform coarse sand can have a horizontal hydraulic conductivity as high as 0.001 m/s. Whereas, a tight, compacted clay can have a have a horizontal hydraulic conductivity as low as $1x10^{-8}$ m/s - which is 5 orders of magnitude.

The vertical hydraulic conductivity is typically 5 to 10 times lower than the horizontal hydraulic conductivity.

Related Items:

- Geological Units *(p. 312)*
- Horizontal Hydraulic Conductivity *(p. 315)*
- Lenses *(p. 64)*

11.19.11Specific Yield

Specific Yield	
dialogue Type	Stationary Real Data
EUM Data Units	Specific Yield

In an unconfined aquifer, the Specific Yield is defined as the volume of water released per unit surface area of aquifer per unit decline in head. The specific yield is much higher than the Specific Storage because the water that is released is primarily from the dewatering of the pores at the water table. This results in a unit of $L^3/L^2/L$, which is dimensionless.

The Specific Yield is only used in transient simulations, but must always be input. Furthermore, the specific yield is only used in the cells that contain the water table. In the cells below the water table, the Specific Storage is used.

Specific Yield of upper SZ layer

The specified value for specific yield is not used for the specific yield of the upper most SZ layer if UZ is included in the simulation.

By definition, the specific yield is the amount of water release from storage when the water table falls. The field capacity of a soil is the remaining water content after a period of free drainage. Thus, specific yield is equal to the saturated water content minus the field capacity.

To avoid water balance errors at the interface between the SZ and UZ models, the specific yield of the top SZ layer is set equal to the he saturated water content minus the field capacity. The value is determined once at the beginning of the simulation. The water content parameters are taken from the UZ layer in which the initial SZ water table is located.

Related Items:

- Specific Yield of the upper SZ numerical layer *(p. 654)*
- Geological Units *(p. 312)*
- Lenses *(p. 64)*

11.19.12Specific Storage

Specific Storage	
dialogue Type	Stationary Real Data
EUM Data Units	Elastic Storage

In a confined aquifer, the specific storage is defined as the volume of water released per volume of aquifer per unit decline in head. This is slightly different than the specific yield because the water released from storage comes primarily from the expansion of the water and aquifer compression due to the reduction in water pressure (increase in effective stress). Thus, the water released from storage is released from the entire column of water in the aquifer, not just at the phreatic surface. This results in a unit of $L^3/L^3/L$, or 1/L.

The Specific Storage Coefficient is only used in transient simulations, but must always be input. Furthermore, the specific storage coefficient is only used in the cells below the water table. In the cells containing the water table, the Specific Yield *(p. 316)* is used.

11.19.13Porosity

Porosity	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue
dialogue Type:	Stationary Real Data
EUM Data Unitse:	Porosity

In a porous media, most of the volume is taken up by soil particles and the actual area available for flow is much less than the nominal area. This distinction is important when calculating flow velocities for solute transport. The porosity is the cross-sectional area available for flow divided by the nominal cross-sectional area. This is often referred to as the effective porosity, since it discounts the dead end pore spaces that are not available for flow. In the absences of dead end pores, the porosity is equal to the specific yield.

The Porosity must be greater than 0 and less than 1. For unconsolidated porous media, the porosity is usually from 0.15 to 0.3 depending of the grain size distribution (the more uniform the higher the effective porosity). For fractured media the porosity is usually much lower, in the from 0.01 to 0.05.

.Related Items:

- Secondary Porosity *(p. 318)*
- Solute Transport in the Saturated Zone *(p. 677)*

11.19.14Secondary Porosity

Secondary Porosity	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialog AND if the Mass Transfer to Immobile Water *(p. 308)* option selected in the Saturated Zone dialog
Dialog Type	Stationary Real Data
EUM Data Units	Porosity

Solutes travelling in a fractured media will be transported by diffusion into and out of the soil matrix of the surrounding media. The velocity in fractured

media due to the lower effective porosity will cause very fast break-through down gradient. However, the solute mass that diffuses into the surrounding media will act like a long term solute source after the main plume has passed, as the solute diffuses back into the fractures.

The matrix source/sink process can be included in MIKE SHE by activating the Mass Transfer to Immobile Water *(p. 308)* option in the Saturated Zone dialog.

Matrix porosity must be between 0 and 1. Matrix porosities are generally very difficult to measure and are usually calibrated against breakthrough curves to estimate a realistic value. Furthermore, the value needs to be the "effective" matrix porosity - that is, the matrix porosity that is "actively" involved in solute diffusion. This can be significantly lower than the matrix porosity measured by core analysis. For a limestone aquifer the matrix porosity has been calibrated to be as small as 4 per cent (core samples indicated 20 to 35%) whereas for a clay till sample it was calibrated to be 20 to 30% - a little less than the total matrix porosity.

Related Items:

- Dual Porosity transfer coefficient *(p. 322)*
- Solute Transport in the Saturated Zone *(p. 677)*
- Transport in Fractured Media *(p. 687)*

11.19.15Bulk Density

Bulk Density	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue AND the SZ solute transport option is turned on AND at least one sorption process is active
dialogue Type:	Stationary Real Data
EUM Data Unitse:	Density

The bulk density is the average density of the in-place soil material, including void space. In other words, if you were to remove one cubic meter of soil, how much would it weigh? A typical bulk density is about 1700 kg/m^3, but representative local values are usually available.

The bulk density is used in the calculation of sorption of solutes, so you need to specify a bulk density whenever any sorption processes are active in SZ.

Related Items:

- Solute Transport in the Saturated Zone *(p. 677)*

11.19.16Dispersion Coefficients LHH, THH, TVH, LVV, THV

Dispersion Coefficients LHH, THH, TVH, LVV, THV	
dialogue Type	Stationary Real Data
EUM Data Units	Dispersion Coefficient

If dispersion is included, then the two different dispersion options differ in the number of dispersion parameters required. If you assume isotropic conditions you need to specify the longitudinal dispersivity, α_L, and the transversal dispersivity, α_T . If you assume anisotropic conditions you need to specify five dispersivities.

The magnitude of the dispersivity coefficient depends on the degree of heterogeneity in your geology and the degree to which these heterogeneities have been described in the model. The more heterogeneous your geology is, the larger the dispersivities should be. On the other hand, the more detailed you have described the heterogeneities with your model geometry, the smaller dispersivities should be.

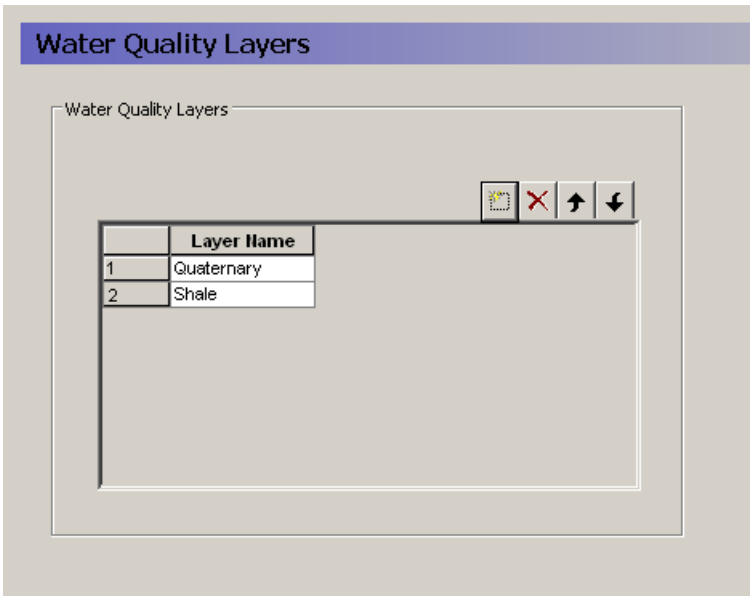
Further, the magnitude of the dispersivities depends on the size of the model and on the model grid size. The larger the model, the larger the dispersivities should be. Whereas, the larger the grid size, the smaller the dispersivities should be due to numerical dispersion.

Thus, it is difficult to give a rule of thumb for the values of dispersivity. Recent field experiments on solute transport, though, indicate that the longitudinal dispersivity should be in the range of 1% or less of the travel distance, the transverse-horizontal dispersivity should be at least 50 times less than this and the transverse-vertical dispersivity should be 2 or more times less than the transverse-horizontal.

Related Items:

- Solute Transport in the Saturated Zone *(p. 677)*

11.19.17Water Quality Layers



Water Quality Layers	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue AND the SZ solute transport option is turned on AND at least one sorption or decay process is active

The water quality layers present a conceptual layer system like the geologic layers that applies to chemistry parameters. In some cases, the water quality layers will mimic the geologic layers, but in other case they will not. For example, redox potential will not generally follow the geologic layers and water quality parameters related reaction processes will be quite different in oxidizing and reducing envrironments. Likewise, you may have several geologic layers in the upper quaternary deposits that affect the flow hydraulics, but the water chemistry can be divided into a quaternary sand and silt zone overlain on an iron rich shale layer.

Unfortunately, the different water quality parameters may not all require identical water quality layers. However, in the current version, only one set of water quality layers is available. If two or more parameters require different distributions, then you will have to divide the overlapping layers into seperate units.

When you add a Water Quality Layer to the table, an item is added to the data tree containing all of the relevant data for the Water Quality Decay Processes *(p. 356)* and Water Quality Sorption processes *(p. 358)* that are active. The location of the WQ Layer is specified by a Lower Level *(p. 355)*.

Related Items:

- Solute Transport in the Saturated Zone *(p. 677)*
- Water Quality Decay Processes *(p. 356)*
- Water Quality Sorption processes *(p. 358)*
- Lower Level *(p. 355)*

11.19.18Dual Porosity transfer coefficient

Mass transfer coefficient to immobile water	
Dialog Type	Stationary Real Data
EUM Data Units	

Solutes travelling in a fractured media will be transported by diffusion into and out of the soil matrix of the surrounding media. The velocity in fractured media due to the lower effective porosity will cause very fast break-through down gradient. However, the solute mass that diffuses into the surrounding media will act like a long term solute source after the main plume has passed, as the solute diffuses back into the fractures.

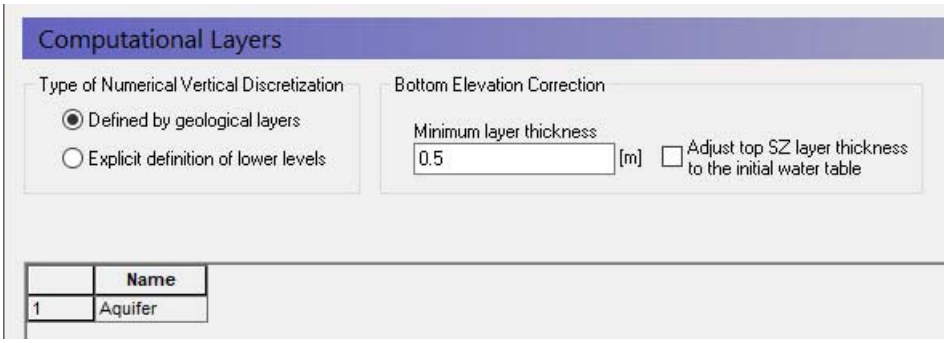
The matrix source/sink process can be included in MIKE SHE by activating the Mass Transfer to Immobile Water *(p. 308)* option in the Saturated Zone dialog.

The mass transfer coefficient represents the rate at which mass is transfered between the fractures and the matrix. It depends on the geology and chemistry. It is defined per species.

Related Items:

- Secondary Porosity *(p. 318)*
- Solute Transport in the Saturated Zone *(p. 677)*
- Transport in Fractured Media *(p. 687)*

11.19.19Computational Layers



The vertical discretisation in the saturated zone can be defined in two ways:

- by the geological layers, in which case there will be one calculation node in each geological layer,
- by explicitly defining the lower level of each calculation layer.

Vertical discretisation

Defined by the geological layers - Groundwater flow in a multi-layer aquifer can be described by a model in which the computational layers follow the interpreted geological layers. Each layer is characterised only by its base level specified either by a constant level or by a distributed file. The number of numerical layers will be identical with the number of geologic layers.

Explicit definition of lower levels - If you define the vertical discretisation explicitly each computational layer is defined by its lower elevation.

Bottom elevation correction

There are two corrections that may be made to the layer elevations.

Minimum layer thickness - The minimum thickness of the calculation layers must be specified to adjust the geological model or the specified levels to prevent layers with zero thickness or very thin layers. Otherwise, very thin layers may cause numerical difficulties. The default value is 0.5, which is usually sufficient. This means that if the two geological layers or specified layer elevations approach one another then the bottom layer will be pushed down to maintain the minimum layer thickness.

Adjust top SZ layer thickness to the initial water table - In principle, the UZ calculates vertical infiltration in the unsaturated zone and the SZ calculates 3D saturated flow below the water table. However, the UZ and SZ layer elevations are defined independently of one another. In MIKE SHE, the UZ flow is only calculated to the water table, or to the bottom of top SZ layer. Any UZ cells below this are ignored and outflow

from the UZ model is added to the top layer of the SZ model. This option allows you to more easily keep the UZ and SZ models consistent based on the depth of the initial SZ water table.

Related Items:

- Coupling the Unsaturated Zone to the Saturated Zone *(p. 601)*
- Definition of vertical UZ numerical grid *(p. 613)*
- Saturated Flow - Technical Reference *(p. 623)*

11.19.20 Initial Potential Head

Initial Potential Head	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

The Initial Potential Head is the starting head for transient simulations and the initial guess for steady-state simulations. The choice of initial head for steady state simulations may affect the rate of numerical convergence depending on the solver used.

Related Items:

- Saturated Flow - Technical Reference *(p. 623)*

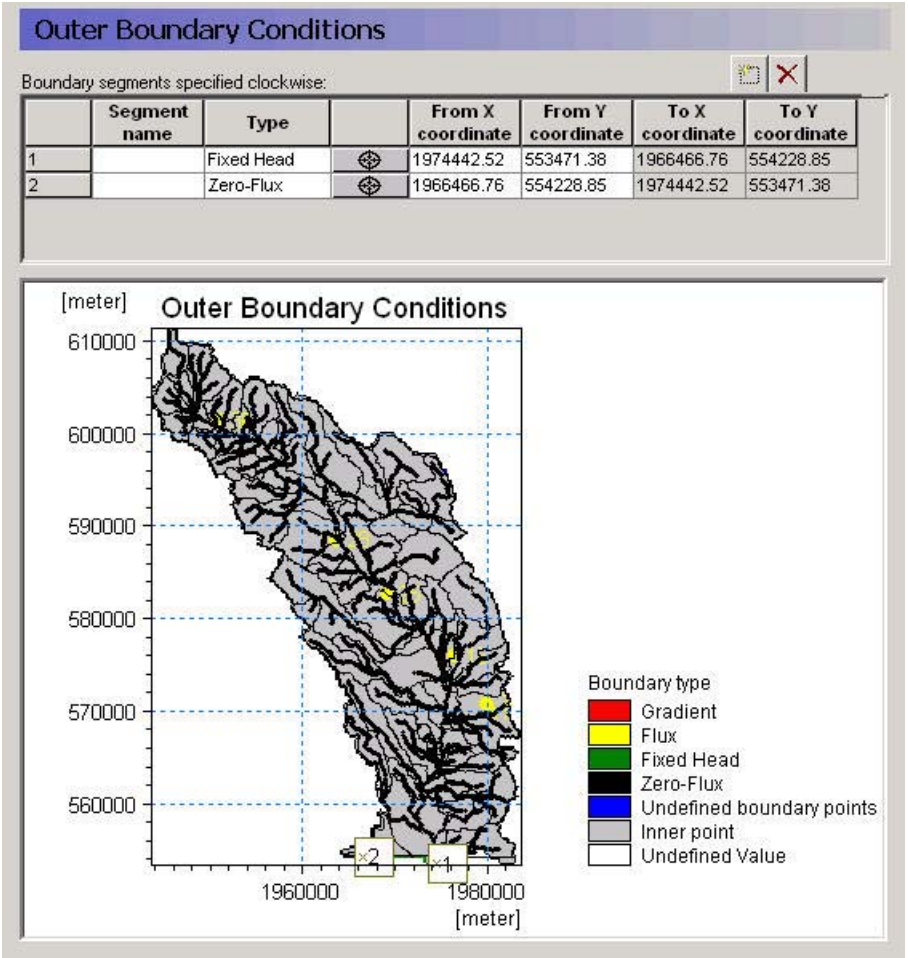
11.19.21 Initial Soil Temperature

The initial soil temperature is required if temperature dependent decay is specified. The current soil temperature is calculated at every time step based on an empirical formula using the air temperature.

Related Items:

- Decay *(p. 704)*

11.19.22Outer boundary conditions



The outer boundary conditions are defined as line segments between two boundary points. The boundary points are, in principle, independent of the model domain because they do not need to lie on the model boundary. Rather they are projected onto the nearest model boundary cell. Thus, the model boundary can be modified slightly without having to modify the boundary locations. However, if the model boundary is moved significantly or if the boundary is convoluted then the calculated ‘nearest’ node might not be the one expected.

Specifying a boundary condition

To specify an outer boundary segment,

1. add a new line to the outer boundary points table
2. click on the target icon,

3. click on one end of your boundary segment,
4. add a second line to the outer boundary points table
5. click on the new target icon,
6. click on the other end of your boundary segment
7. Change the name of the top line of the points table,
8. Select the appropriate boundary condition for the boundary segment.

Available boundary conditions

Fixed Head - This boundary prescribes a head in the boundary cell. The head can be fixed at a prescribed value, fixed at the initial value from the initial conditions, or assigned to a .dfs0 time series file. If a time series input is used, then the actual value used in the model at the current time step is linearly interpolated from the available values.

The last option is a time varying dfs2 file, which is typically extracted from a regional results file. This can be done using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. MIKE SHE then interpolates in both time and space from the .dfs2 file to the local head boundary at each local time step.

Zero flux - This is a no-flow boundary, which is the default.

Flux - This boundary prescribes a flux across the outer boundary of the model. When the spatial distribution is set to "global", the given value will designate the total inflow along the entire boundary stretch, which may contain several cells. A time varying flux can be specified as a mean step-accumulated discharge (e.g. m^3/s) or as a step-accumulated volume (e.g. m^3). A positive value implies an inflow to the model cells.

When the spatial distribution is set to "distributed" the boundary inflow can be specified for each cell in the boundary stretch separately. Also the inflow along x- and y-axis can be specified individually. As before, a positive value implies an inflow to the model cells.

Gradient - This boundary describes a constant or time varying gradient between the node on the outer boundary and the first internal node. A time varying gradient can be specified as an instantaneous dimensionless or percent value. A positive gradient implies a flux into the model.

Notes

1. The head is calculated in a No Flow outer boundary cells, whereas the head is specified in the Fixed Head outer boundary cells, but in both cases all properties must be assigned to all outer boundary cells.
2. An internal model cell in contact with multiple boundary cells will not receive multiple quantities of water.

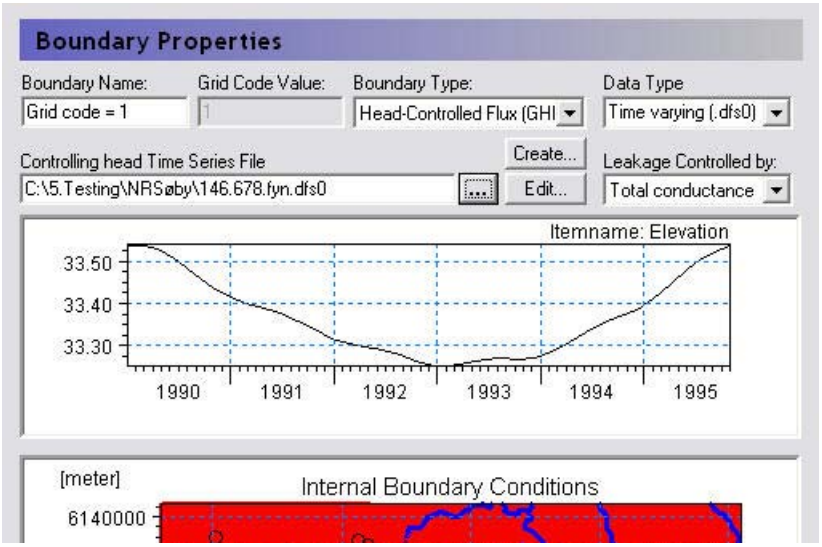
3. Additional detailed information can be found under Boundary Conditions *(p. 632)* in the Saturated Flow - Technical Reference *(p. 623)*.
4. An error will be generated if the flux/gradient input cell
- has zero thickness,
- has a horizontal hydraulic conductivity of zero,
- is an inactive internal boundary cell, or
- is a fixed head internal boundary cell.

5. A warning will be issued if the flux/gradient boundary
- is a head controlled flux (GHB) internal boundary cell, or
- is a fixed head drain internal boundary cell.

Related Items:

- Saturated Flow - Technical Reference *(p. 623)*
- Boundary Conditions *(p. 632)*

11.19.23Internal boundary conditions



Internal Boundary Conditions	
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The map dialogue for the internal boundary conditions allows you to specify the locations of the various boundary conditions. Any Integer Code value is

permissible and a separate item will be added to the data tree below this level with one item for each unique integer code in the domain.

Only one integer code is allowed per cell, which means that no cell can have more than one boundary condition.

If you use a polygon .shp file, each unique polygon will have a separate entry in the data tree.

Available boundary conditions

The following boundary conditions can be defined for each integer code:

Fixed Head For a fixed head boundary, you specify the head in the cell. The model will not calculate the head in the cell. Care should be taken when specifying fixed head boundary conditions, as the cell becomes an infinite source or sink of water. The fixed head can be a prescribed value, fixed at the initial value from the initial conditions, assigned to a .dfs0 time series file, or assigned to a time varying dfs2 file. The last option is typically from a results file. It could be from a regional results file, which can be extracted using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. Or, it could be from a previous run of the same model.

Fixed Head Drain - For a fixed head drain boundary, you specify a reference head. If the cell water level is above the reference level, then the boundary acts like a normal fixed head boundary conditions. If the head in the cell falls below the reference level, then the boundary condition is turned off. That is, if the simulated head drops below the head reference level the flux is set to zero. Thus, the fixed head drain allows only water extraction.

The reference head can be a prescribed value, fixed at the initial value from the initial conditions, assigned to a .dfs0 time series file, or assigned to a time varying dfs2 file. The last option is typically from a results file. It could be from a regional results file, which can be extracted using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. Or, it could be from a previous run of the same model.

Note: This boundary condition was previously called the Head controlled abstraction boundary condition in early versions of MIKE SHE.

Head Controlled Flux (GHB) - The head controlled flux, or General Internal Head Boundary is similar to the fixed head. However, a flow resistance is incorporated via a user specified leakage coefficient.

The head can be a prescribed value, assigned to a .dfs0 time series file, or assigned to a time varying dfs2 file. The last option is typically from a results file. It could be from a regional results file, which can be extracted using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. Or, it could be from a previous run of the same model.

If the GHB is selected, an extra item is added to the data tree below the boundary condition for the leakage coefficient. The leakage coefficient can be specified as either a simple leakage coefficient [1/time] or as a total conductance [length^2/time].

Note: This boundary condition was previously called the General Internal Head Boundary condition in early versions of MIKE SHE

Inactive Cells - This boundary condition is used to make interior cells of the model inactive. It works by assigning a hydraulic conductivity of zero to the cells if the simulation is transient, or a value of 10^{-15} if the simulation is steady-state. Note, though, that this method means that in the pre-processed data, you will see the inactive cells show up in the maps of hydraulic conductivity rather than in the maps of boundary conditions. Also note that since the inactive cells are actually active cells with zero conductivity, the results will also include head values in these points.

Related Items:

- Saturated Flow - Technical Reference *(p. 623)*
- Boundary Conditions *(p. 632)*

11.19.24 Initial (secondary) concentration

Initial (secondary) concentration	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialog
Dialog Type	Stationary Real Data
EUM Data Units	Concentration

There is an initial concentration item for each computational layer. Under each initial concentration item, there is one sub-item for each active species.

The initial concentration is used by the MIKE SHE Water AD engine as the starting concentration for the Water quality simulation.

If you have specified Mass Transfer to Immobile Water *(p. 308)* option selected in the Saturated Zonedialog, then you will also have to specify an initial concentration in the immobile water located in the secondary (matrix) porosity.

Related Items:

- Solute Transport in the Saturated Zone *(p. 677)*

- Initial Conditions *(p. 685)*

11.19.25Initial number of particles per cell

Initial number of particles per cell	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialog and the Random walk particle tracking sub-option is specified and the initial particle density is specified as a number of particles
Dialog Type	Stationary Real Data
EUM Data Units	Integer

There is an initial number of particles item for each computational layer. The initial number of particles is used by the MIKE SHE Random Walk particle tracking engine as the starting concentration.

Related Items:

- Particle Tracking-Reference *(p. 737)*

11.19.26PT Registration Codes/Lenses

PT Registration Codes/Lenses	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialog and the Random walk particle tracking sub-option is specified and PT Registration codes are defined by layer or lense
Dialog Type	Stationary Real Data
EUM Data Units	Integer

When you run the random walk particle tracking option, registration areas can be defined. All particles passing through a registration area will be tagged. This allows you to create a point shape file that contains the starting location of all particles that passed through the registration area.

Registration codes by layer

If you specify the registration codes by layer, then you can specify a dfs2 integer grid code array for each numerical layer. Each unique integer codes will be used as a registration zone.

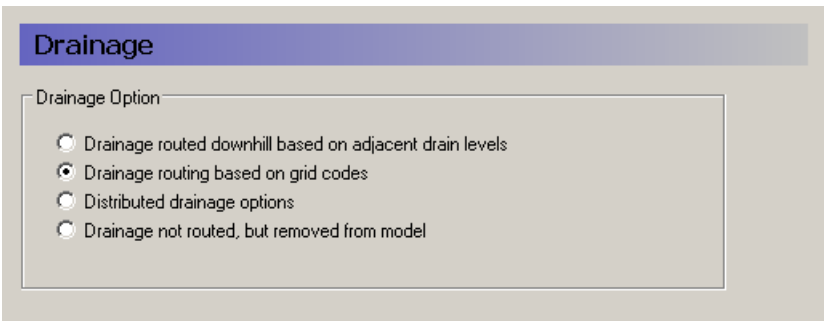
Registration codes by lenses

If you specify the registration zones by lenses, then you have to specify an Horizontal Extent *(p. 356)*, as well as an Upper Level *(p. 355)* and a Lower Level *(p. 355)* for the lense. This is a more flexible way to define the registration zone, but is significantly slower because every particle has to be checked against the extent and the top and bottom elevations.

Related Items:

- PT Registration Extraction *(p. 377)*
- Particle Tracking-Reference *(p. 737)*

11.19.27Drainage



Saturated zone drainage is a special boundary condition in MIKE SHE used to define natural and artificial drainage systems that cannot be defined in MIKE 1D. It can also be used to simulate simple overland flow in a lumped conceptual approach. Saturated zone drainage is applied to the layer of the Saturated Zone model containing the drain level. Water that is removed from the saturated zone by drains is routed to local surface water bodies.

Drain flow is simulated using an empirical formula. Each cell requires a drain level and a time constant (leakage factor). Both drain levels and time constants can be spatially defined. A typical drainage level is 1m below the ground surface and a typical time constant is between 1e-6 and 1e-7 1/s.

MIKE SHE also requires a reference system for linking the drainage to a recipient node or cell. The recipient can be a river node, another SZ grid cell, or a model boundary. Whenever drain flow is produced during a simulation, the computed drain flow is routed to the recipient point using a linear reservoir routing technique.

Drainage routed downhill based on adjacent drain levels
This option was originally the only option in MIKE SHE. The reference system is created automatically by the pre-processor using the slope of the drains calculated from the drainage levels in each cell.

Thus, the pre-processer calculates the drainage source-recipient reference system by

1. looking at each cell in turn and then
2. look for the neighbouring cell with the lowest drain level.
3. If this cell is an outer boundary cell or contains a river link, the search stops.

If the cell does not contain a boundary or river link, then the next search is repeated until either a local minimum is found or a boundary cell or river link is located.

The result of the above search for each cell is used to build the source-recipient reference system.

If local depressions in the drainage levels exist, the SZ nodes in these depressions may become the recipients for a number of drain flow producing nodes. This often results in the creation of a small lake at such local depressions. If overland flow is simulated, then the drainage water will become part of the local overland flow system.

Note: Be aware that the drainage is routed to a destination. It does not phyisically flow downhill. The drain levels are only used to build the drainage source-recipient reference system. In other words, any drainage that is generated at a node, is immediately moved to the recipient node. The assumption here, is that the time step length is longer than the time it takes for the drainage to reach its destination.

Drainage routing based on grid codes

This method is often used when the topography is very flat, which can result in artificial depressions, or when the drainage system is very well defined, such as in agricultural applications.

In this method, the drainage levels and the time constants are defined as in the previous method and the amount of drainage is calculated based on the drain levels and the time constant.

If the drainage routing is specified by Drain Codes, a grid code map is required that is used to restrict the search area for the source-recipient reference system. In this case, the pre-processer calculates the reference system within each grid code zone, such that all drainage generated within one zone is routed to recipient nodes with the same drain code value.

When building the reference system, the pre-processor looks at each cell and then

1. looks for the nearest cell with a river link with the same grid code value,
2. if there is no cells with river links, then it looks for the nearest outer boundary cell with the same grid code,
3. if there are no cells with outer boundary conditions, then it looks for the cell with the same grid code value that has the lowest drain level.

The result of the above search for each cell is used to build the source-recipient reference system.

The above search algorithm is valid for all **positive** Drain Code values. However, all cells where

Drain Code = 0 - will not produce any drain flow and will not receive any drain flow, and

Drain Code < 0 (negative) - will not drain to river links, but will start at Step 2 above and only drain to either a outer boundary or the lowest drain level.

Distributed drainage options

Choosing this method, adds the Option Distribution item to the data tree. With the Option Distribution, you can specify an integer grid code distribution that can be used to specify different drainage options in different areas of your model.

Code = 1 - In grid cells with a value of 1, the drainage reference system is calculated based on the Drain Levels.

Code =2 - In grid cells with a value of 2, the drainage reference system is calculated based the Drain Codes.

Code = 3 - Drainage in grid cells with a value of 3 is routed to a specified Branch and chainage. At the moment, this option requires the use of Extra Parameters *(p. 353)* and is described in SZ Drainage to Specified River H-points *(p. 770)*.

Code = 4 - Drainage in grid cells with a value of 4 is routed to a specified sewer man hole. This option is described in the section Working with Sewers - User Guide *(p. 509)*.

Drain flow not routed, by removed from model

The fourth option is simply a head dependent boundary that removes the drainage water from the model. This method does not involve routing and is exactly the same as the MODFLOW Drain boundary.

Related Items:

- Groundwater drainage *(p. 65)*
- Saturated Zone Drainage *(p. 635)*
- SZ Drainage to Specified River H-points *(p. 770)*

- Coupling of MIKE SHE and a MIKE 1D Sewer Model *(p. 653)*

11.19.28Drain Level

Drain Level	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

If surface drainage is routed by drain levels, the drainage routing reference system is created automatically using the slope of the drains calculated from the drainage levels in each cell.

The drain levels are only used for two things:

- to calculate the drain cell-drain recipient relations, and
- to calculate the amount of drain flow produced in each node when the water table is above the drain level.

The drain level also determines from which SZ layer the drain water will be extracted.

Important note: If the drain level is set equal to the topography in a cell, then the drainage will be turned off in the cell. Drain levels above the topography are not allowed. In this case, the drain level will be automatically adjusted to just below the topography.

Related Items:

- Drainage *(p. 331)*
- Groundwater drainage *(p. 65)*
- Saturated Zone Drainage *(p. 635)*
- SZ Drainage to Specified River H-points *(p. 770)*
- Coupling of MIKE SHE and a MIKE 1D Sewer Model *(p. 653)*

11.19.29Drain Time Constant

Drain Time Constant	
dialogue Type	Stationary Real Data
EUM Data Units	Leakage Coefficient./Drain Time Constant

Drainage flow is simulated using an empirical formula that requires a drainage level and a time constant (leakage factor) for each cell. Mathematically, the time constant is exactly the same as a leakage coefficient - it is simply a factor that is used to regulate how quickly the water can drain. A typical time constant is between 1e-6 and 1e-7 1/s.

Related Items:

- Drainage *(p. 331)*
- Groundwater drainage *(p. 65)*
- Saturated Zone Drainage *(p. 635)*
- SZ Drainage to Specified River H-points *(p. 770)*
- Coupling of MIKE SHE and a MIKE 1D Sewer Model *(p. 653)*

11.19.30Drain Codes

Drain Codes	
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

If the drainage routing is specified by Drain Codes, a grid code map is required that is used to link the drain flow producing cells to recipient grid cells. The drain levels are still used to calculate the amount of drain flow produced in each node, but the routing is based only on the code values in the drain code file.

The Drain Code can be any integer value, but the different values have the following special meanings:

Code = 0 - Grid cells with an Drain Code value of zero will not produce any drain flow and will not receive any drain flow.

Code > 0 - Grid cells with **positive** Drain Code values will drain to the nearest river, boundary or local depression in the drain level - in that priority - located next to a cell with the same Drain Code value. Thus, if a grid cell produces drainage,

1. If there are one or more cells with the same drain code next to a river link, then the drain flow will be routed to the nearest of these cells.

2. If there are no cells with the same Drain Code located next to a river link, then the drain flow will be routed to the nearest boundary cell with the same Drain Code value.

3. If there are no boundary cells with the same Drain Code value, the drain flow will be routed to the cell with the lowest drain level that has the same Drain Code value (which may create a lake).

Code < 0 - Grid cells with **negative** Drain Code values will drain to either a boundary or a local depression, in that order. Thus, if a grid cell produces drainage,

1. If there are no cells with the same Drain Code located next to a river link, then the drain flow will be routed to the nearest boundary cell 2ith the same Drain Code value.

2. If there are no boundary cells with the same Drain Code value, the drain flow will be routed to the cell with the lowest drain level that has the same Drain Code value (which may create a lake).

Related Items:

- Drainage *(p. 331)*
- Groundwater drainage *(p. 65)*
- SZ Drainage to Specified River H-points *(p. 770)*
- Coupling of MIKE SHE and a MIKE 1D Sewer Model *(p. 653)*

11.19.31 Option Distribution

Option Distribution	
Conditions:	always when Surface Drainage active AND when the Distributed drainage option is used.
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code
Valid Values:	1, 2, 3 and 4 only

The drain type distribution is used to distinguish areas of the model where different drainage options are used.

Code = 1 - Drainage in grid cells with a value of 1 is routed downhill based on the value of the drain level specified in Drain Level data item.

Code =2 - Drainage in grid cells with a value of 2 is routed via Drain Codes as specified in the Drain Codes data item.

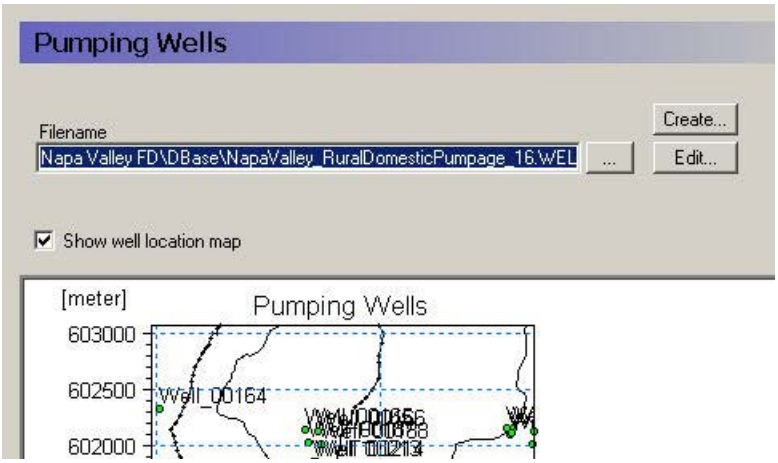
Code = 3 - Drainage in grid cells with a value of 3 is routed to a specified branch and chainage. At the moment, this options requires the use of Extra Parameters.

Code = 4 - Drainage in grid cells with a value of 4 is routed to a specified sewer man hole. At the moment, this options requires the use of Extra Parameters.

Related Items:

- Saturated Zone Drainage *(p. 635)*
- Extra Parameters *(p. 353)*
- SZ Drainage to Specified River H-points *(p. 770)*
- Coupling of MIKE SHE and a MIKE 1D Sewer Model *(p. 653)*

11.19.32Pumping Wells



If pumping wells are active in the model domain, then you must specify the name of the well database to include in the model setup.

Edit The Edit button will open the current Well Database with the current maps and overlays open in the well editor.

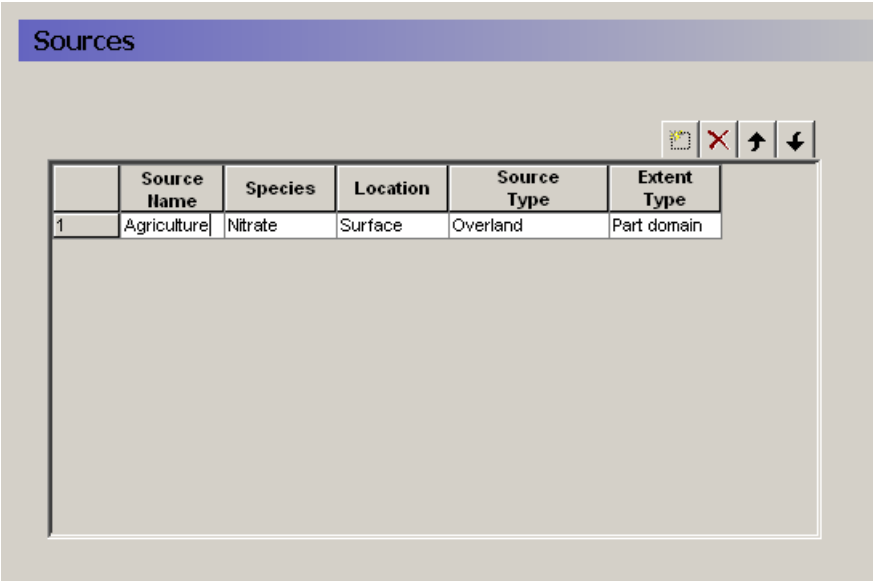
You may get an error in when you open the Well Database if the model has not yet been pre-processed or if the model data has changed without re-preprocessing the model. This error happens because MIKE SHE is trying to reconcile and plot both the geologic layers (input data) and the numerical layers (preprocessed data) for each cell containing a pumping well.

Create The Create button will create a new Well Database file

Related Items:

- Well editor *(p. 401)*

11.20 Sources



The specification of water quality sources is very flexible. The Sources dialogue allows you to add and delete sources, as well as define the type and location of the source. The table provides an overview of all of your sources in your model.

An important feature of the source location definition is the partial extent distribution function. This allows you to define, for example, a distributed global source file - say of the field scale agricultural inputs in your catchment - and run individual water quality scenarios for each sub-catchment (modelled as an partial extent) to assess the subcatchment contributions to the global stream impact.

Source Name - The name appears in the data tree for reference.

Species - You can only choose from the list of available species that you have defined in the Species *(p. 223)* dialogue.

Location - The location is defines whether the source is located on the ground surface (Surface) or in the saturated or unsaturated zone (Subsurface). The available source types depend on where the source is located.

Source Type - If the source is located on the ground surface, then it can be either a Precipitation source (concentration in precipitation water) or an Overland source (mass on the surface). In both cases, the solute can infiltrate or runoff as lateral overland flow.

Extent Type - The source can cover the entire domain (Full domain) or only part of the domain (Part domain). In both cases, the actual source strength can very spatially and temporally. The Extent is used simply to restrict the source data to a zone smaller than the model domain.

Related Items:

- Solute Transport in the Saturated Zone *(p. 677)*
- Solute Transport in the Unsaturated Zone *(p. 688)*
- Solute Transport in Overland Flow *(p. 693)*

11.20.1 Strength

Strength	
dialogue Type:	Time-varying Real Data
EUM Data Units	Grid Code
Time Series EUM Data Units	Concentration or Mass per time

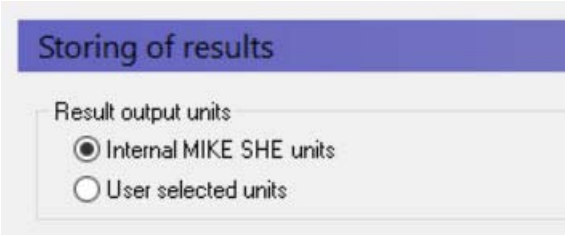
Since the source is a common data item for overland, unsaturated and saturated flow, the units of the source strength depend on the type of source being simulated.

The source strength comprises both a distribution and a value. The distribution can be either uniform, sub area-based or fully distributed. If the data is sub area-based then for each station a sub-item will appear where you can enter the time series of values for the station. If the data is fully distributed, then you can enter a time varying dfs2 file.

Related Items:

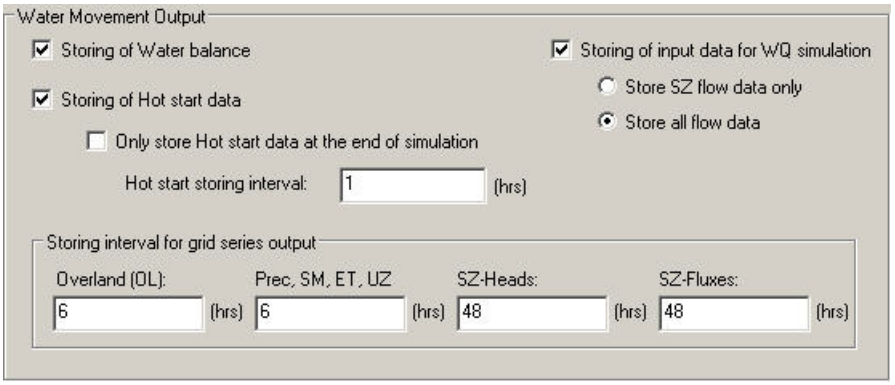
- Solute Transport in the Saturated Zone *(p. 677)*
- Source/Sinks, Boundary Conditions and other Exchanges *(p. 685)* in the saturated zone
- Solute Transport in the Unsaturated Zone *(p. 688)*
- Source/Sinks, Boundary Conditions and other Exchanges *(p. 692)* in the unsaturated zone
- Solute Transport in Overland Flow *(p. 693)*
- Source/Sinks, Boundary Conditions and other Exchanges *(p. 696)* in overland flow

11.21 Storing of Results



Result Output units - By default, the output files are created using the internal MIKE SHE SI units. To support Imperial (American) units, the output files can be optionally output in the units as defined under the menu: File/Options/Edit Unit Base Groups...

Water Movement Output



Storing of water balance data - When this option is selected, MIKE SHE will store all of the relevant output data for the analysis of the water balance. This will automatically select the required items in the Grid Series Output section.

Storing of Hot start data - The option allows you to save a simulation that can be used as the start a new simulation. See Time Step Control for more information on using Hot Start data as the initial data for a simulation.

Only store hot start data at the end of the simulation - Typically, the follow on simulation starts at the end of the previous simulation. However, if you want to test the sensitivity of the results to the starting condition, for example, you may want to save hot start data more frequently. However, the hot start file can become very large if the hot start data is saved frequently.

Store AD input data during water movement simulation - A MIKE SHE water quality simulation is calculated based on the cell-by-cell water fluxes calculated by the water movement module. If water quality is included in the model setup, then the necessary data is automatically saved and this item is hidden. However, if water quality is not included in the simulation, you can optionally tell MIKE SHE to save the necessary data for the water quality model by checking this box on and selecting the save option. The first option only saves the saturated zone data, which is suitable if you are only going to calculate the water quality in the saturated zone. For example, the random walk particle tracking is only available in the saturated zone and there is, therefore, no need to save overland flow data for a particle tracking simulation. The second option saves all of the data necessary for a water quality simulation in the complete integrated model.

Storing interval for grid series output

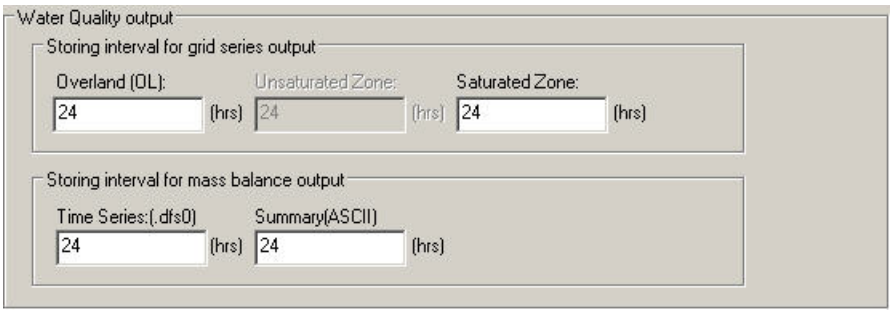
Gridded output can create very large output files if the data is stored at every time step. Gridded data is not usually compared to frequent measurements, such as daily groundwater levels, so the output frequency can be much less than the time step length. In fact, the output frequency of gridded output is often determined by visualization needs, such as to make smooth animations.

The gridded output for the different processes can be saved at different frequencies - the overland frequency is separate from unsaturated zone frequency is separated from the saturated zone frequency. However, you cannot save individual items at different frequencies. Thus, since precipitation, evapotranspiration and unsaturated flow output items are related, they are all saved at the same frequency. The saturated zone heads and fluxes are separated into different frequencies, however, because the gridded output files for a detailed 3D model can get very large.

Note: The Storing Time Step for SZ must be an integer multiple of the Maximum Allowed SZ Time Step that is specified in the Time Step Control dialogue. In other words, if the Maximum allowed SZ time step is 24 hrs, then the Storing Time Step for SZ can only be a multiple of 24 hours (e.g. 24, 48, 72 hours, etc.).

Similarly, the storing frequency for SZ Fluxes must be an integer multiple of the SZ heads frequency, and the storing frequency for overland flow and unsaturated flow items, must be an integer multiple of their respective maximum allowed time step length.

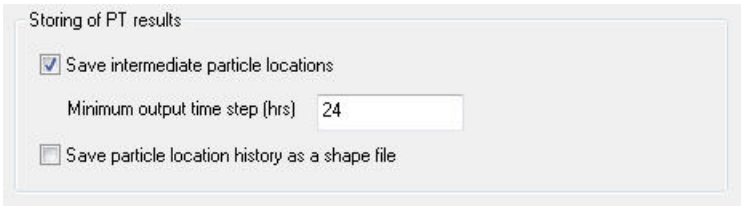
Water Quality Output



Storing interval for grid series output - similar to the storing interval for water movement data, the gridded output data files for a detailed transport simulation can get very large. These three frequencies allow you to save only the data you need.

Storing interval for mass balance output - similar to the storing interval for grid series output, these separate storing frequencies allow you to save only the data you need.

Particle Tracking Output



Save intermediate particle locations - If you want to plot pathlines, then you need to turn this option on. You should be careful, when using this option, as the file size can easily become very large.

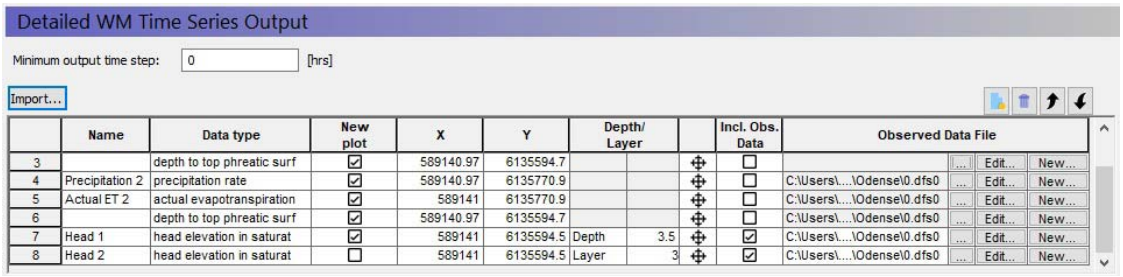
Minimum output time step - This value determines how often the particle locations will be saved. The more frequent the saving, the larger the output file.

Save particle location history as a shape file - The intermediate points can be saved directly into a shape file. However, since there can be hundreds or even thousands of particles in a PT simulation, this can generate very large shape files.

Related Items:

- Time Step Control *(p. 190)*
- Detailed WM time series output *(p. 344)*
- Grid series output *(p. 352)*

11.21.1 Detailed WM time series output



The Detailed time series output dialogue allows you to specify the location at which you want detailed time series output and the item that you want output. For each specified point, the output variable is stored in a .dfs0 file with one value for every simulation time step. Finally, for each item in the detailed time series table, an HTML plot is created in the Result Tab.

Note All of the detailed time series items are stored in one .dfs0 file. This can lead to file size and disk space errors, if you have a long detailed simulation or more than ~200 detailed time series items. Also, the display in the Results Tab will become very slow if you have a lot of items, since it has to read the entire .dfs0 file and generate all of the graphs every time you access the Detailed Time Series page in the Results Tab.

Minimum Output time step - In some simulations, the output time step can become very small due to the automatic time step adjustment. If the output is being saved every simulation time step, then the output files can become very large. This value allows you to set a lower limit on the output time step. Note, however, that the sub-time step values are averaged over the Minimum output time step.

Name - This is a text field that can be used to specify a reference name for the location, for example, a borehole name. This is also the name that will be used for the time series item in the Dfs0 file created during the simulation.

Data Type - This is the list of available output items is dynamic in the sense that the list changes in response to the processes included in the Simulation Specification dialogue. Further, additional items are available that are related to simulation variables, such as the number of iterations during each Saturated Zone time step. A list of available Data Types can be found in Output Items *(p. 92)*.

New plot - If this is checked, then the a new detailed time series HTML-plot will be created on the Results Tab. If this is unchecked, then the output will be added to the previous plot. You can use the Up and Down arrows to arrange the output points so that relevant points are plotted together

X, Y - Often, detailed time series are associated with measurement stations. That is, locations at which a time series of measurements are available, for example, water levels in a well or water depths on a flood plain. This is the (X, Y) map coordinates of the point in the same EUM units (ft, m, etc.) as specified in the EUM Database for Item geometry 2-dimensional. (see EUM Data Units)

Depth/Layer -This is the depth of the observation point below land surface for subsurface observation points. . It can be specified either as a 1-based layer index (top down), or as a depth in the same EUM units (ft, m, etc.) as specified in the EUM Database for Depth Below Ground (see EUM Data Units).

Target Icon - You can use the target icon to locate the output point exactly. Alternatively, you can type the exact coordinates or import the items from an ASCII file.

Include observation data - If this is checked, then a .dfs0 file can be specified that includes observation points. The observation points are automatically plotted along with the results in the plot on the Results tab. The .dfs0 item is selected in the file browser dialogue. The **Edit** button opens the specified .dfs0 file and the **New** button can be used to create a new .dfs0 file with the correct item type etc. and at the same time import data from an Excel spreadsheet.

Importing data

Detailed MIKE SHE Time Series data can be imported directly into the Detailed MIKE SHE Time Series dialogue using the Import button. The data file must be a tab- delimited ASCII file without a header line. The file must contain the following fields and be in the format specified below.

Name>DataTypeCode>NewPlot>X>Y>UseLayer>Depth/Layer>UseObsdata>dfs0file name>dfs0ItemNumber

where the > symbol denotes the Tab character and

Name - is the user specified name of the observation point. This is the name that will be used for the time series item in the Dfs0 file created during the simulation.

DataTypeCode - This is a numeric code used to identify the output data type. A list of available Data Type Codes can be found in Output Items *(p. 92)*.

NewPlot - This is a flag to specify whether a new detailed time series HTML-plot will be created on the Results Tab:

0 = the output will be added to the previous plot.
1 = Create a new plot

X, Y - This is the (X, Y) map coordinates of the point in the same EUM units (ft, m, etc.) as specified in the EUM Database for Item geometry 2-dimensional. (see EUM Data Units)

UseLayer - select whether the next field contains a depth (value 0) or a layer index (value 1)

Depth/Layer -This is the depth of the observation point below land surface for sub-surface observation points. It can be specified either as a 1-based layer index (top down), or as a depth in the same EUM units (ft, m, etc.) as specified in the EUM Database for Depth Below Ground (see EUM Data Units). The previous field defines whether this is a depth or a layer index.

UseObsData - This is a flag to specify whether or not an observation file needs to be input: 0 = No; 1 = Yes

dfs0file name - This is the file name of the dfs0 time series file with observation data. The path to the dfs0 file must be relative to the directory containing the MIKE SHE *.she document. The dfs0 extension is optional. If it is not included, it will be added to the file name automatically. For example,

.\Time\Calibration\GroundwaterObs

refers to the file *GroundwaterObs.dfs0* located in the subdirectory *Time\Calibration,* which is found in the same directory as the .she model document.

dfs0ItemNumber - This is the Item **number** of the observation data in the specified DFS0 file.

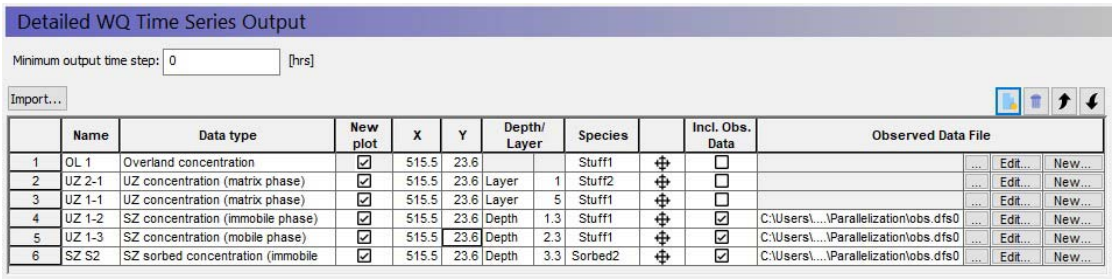
The following is a simple example with three MIKE SHE observation points, where the file name is obsdata.dfs0.

OL depth	61	GrandRiver	34500	18300			0		
CZ head 1	101	GrandRiver	34500	18300	0	5.5	1	.\time\obsdata.dfs0	1
CZ Head 2	101	GrandRiver	34500	18300	1	4	1	.\time\obsdata.dfs0	2

Related Items:

- Time Step Control *(p. 190)*
- Detailed River Time Series Output *(p. 350)*
- Grid series output *(p. 352)*
- MIKE SHE Detailed Time Series *(p. 370)*

11.21.2 Detailed WQ time series output


Detailed WQ Time Series Output

Minimum output time step: 0 [hrs]

Import...

	Name	Data type	New plot	X	Y	Depth/ Layer		Species		Incl. Obs. Data	Observed Data File			
1	OL 1	Overland concentration	☑	515.5	23.6			Stuff1		☐		...	Edit...	New...
2	UZ 2-1	UZ concentration (matrix phase)	☑	515.5	23.6	Layer	1	Stuff2		☐		...	Edit...	New...
3	UZ 1-1	UZ concentration (matrix phase)	☑	515.5	23.6	Layer	5	Stuff1		☐		...	Edit...	New...
4	UZ 1-2	SZ concentration (immobile phase)	☑	515.5	23.6	Depth	1.3	Stuff1		☑	C:\Users\....\Parallelization\obs.dfs0	...	Edit...	New...
5	UZ 1-3	SZ concentration (mobile phase)	☑	515.5	23.6	Depth	2.3	Stuff1		☑	C:\Users\....\Parallelization\obs.dfs0	...	Edit...	New...
6	SZ S2	SZ sorbed concentration (immobile	☑	515.5	23.6	Depth	3.3	Sorbed2		☑	C:\Users\....\Parallelization\obs.dfs0	...	Edit...	New...

The Detailed WQ Time Series output dialogue allows you to specify the location at which you want detailed time series output of concentrations. For each specified point, the output variable is stored in a .dfs0 file with one value for every simulation time step. Finally, for each item in the detailed time series table, an HTML plot is created in the Result Tab.

Note All of the detailed time series items are stored in one .dfs0 file. This can lead to file size and disk space errors, if you have a long detailed simulation or more than ~200 detailed time series items. Also, the display output in the Results Tab will become very slow if you have a lot of items, since it has to read the entire .dfs0 file and generate all of the graphs every time you access the Detailed Time Series page in the Results Tab.

Minimum Output time step - In some simulations, the output time step can become very small due to the automatic time step adjustment. If the output is being saved every simulation time step, then the output files can become very large. This value allows you to set a lower limit on the output time step. Note, however, that the sub-time step values are averaged over the Minimum output time step.

Name - This is a text field that can be used to specify a reference name for the location, for example, a borehole name. This is also the name that will be used for the time series item in the Dfs0 file created during the simulation.

Data Type - This is the list of available output items. It is dynamic in the sense that the list changes in response to the processes included in the Water Quality Simulation Specification dialogue. A list of available Data Types can be found in MIKE SHE Output Items *(p. 787)*.

New plot - If this is checked, then the a new detailed time series HTML-plot will be created on the Results Tab. If this is unchecked, then the output will be added to the previous plot. You can use the Up and Down arrows to arrange the output points so that relevant points are plotted together

X, Y - Often, detailed time series are associated with measurement stations. That is, locations at which a time series of measurements are available, for example, water levels in a well or water depths on a flood plain. This is the (X, Y) map coordinates of the point in the same EUM units (ft, m, etc.) as specified in the EUM Database for Item geometry 2-dimensional. (see EUM Data Units)

Depth/Layer -This is the depth of the observation point below land surface for subsurface observation points. It can be specified either as a 1-based layer index (top down), or as a depth in the same EUM units (ft, m, etc.) as specified in the EUM Database for Depth Below Ground (see EUM Data Units).

Species -This is the list of Species in the WQ simulation.

Target Icon - You can use the target icon to locate the output point exactly. Alternatively, you can type the exact coordinates or import the items from an ASCII file.

Include observation data - If this is checked, then a .dfs0 file can be specified that includes observation points. The observation points are automatically plotted along with the results in the plot on the Results tab. The .dfs0 item is selected in the file browser dialogue. The **Edit** button opens the specified .dfs0 file and the **New** button can be used to create a new .dfs0 file with the correct item type etc. and at the same time import data from an Excel spreadsheet.

Importing data

Detailed MIKE SHE Time Series data can be imported directly into the Detailed MIKE SHE Time Series dialogue using the Import button. The data file must be a tab- delimited ASCII file without a header line. The file must contain the following fields and be in the format specified below.

Name>DataTypeCode>NewPlot>X>Y>UseLayer>Depth/Layer>Use-Obsdata>dfs0file name>dfs0ItemNumber

where the > symbol denotes the Tab character and

Name - is the user specified name of the observation point. This is the name that will be used for the time series item in the Dfs0 file created during the simulation.

DataTypeCode - This is a numeric code used to identify the output data type. A list of available Data Type Codes can be found in MIKE SHE Output Items *(p. 787)*.

NewPlot - This is a flag to specify whether a new detailed time series HTML-plot will be created on the Results Tab:

0 = the output will be added to the previous plot.
1 = Create a new plot

X, Y - This is the (X, Y) map coordinates of the point in the same EUM units (ft, m, etc.) as specified in the EUM Database for Item geometry 2-dimensional. (see EUM Data Units)

UseLayer - select whether the next field contains a depth (value 0) or a layer index (value 1)

Depth/Layer -This is the depth of the observation point below land surface for subsurface observation points. It can be specified either as a 1-based layer index (top down), or as a depth in the same EUM units (ft, m, etc.) as specified in the EUM Database for Depth Below Ground (see EUM Data Units). A depth value must always be included.

Species -This is the name of the species to be used.

UseObsData - This is a flag to specify whether or not an observation file needs to be input: 0 = No; 1 = Yes

dfs0file name - This is the file name of the dfs0 time series file with observation data. The path to the dfs0 file must be relative to the directory containing the MIKE SHE *.she document. The .dfs0 extension is optional. If it is not included, it will be added to the file name automatically. For example,

.*Time**Calibration**GroundwaterObs*

refers to the file *GroundwaterObs.dfs0* located in the subdirectory *Time**Calibration,* which is found in the same directory as the .she model document.

dfs0ItemNumber - This is the Item **number** of the observation data in the specified DFS0 file.

The following is a simple example with three MIKE SHE observation points, where the file name is obsdata.dfs0.

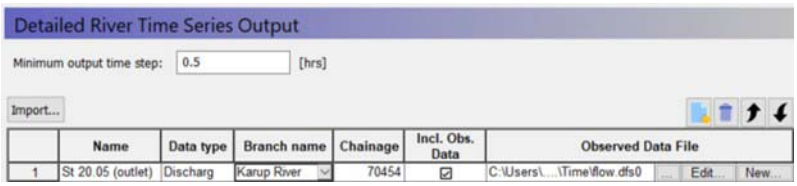
OL conc	216	GrandRiver	34500	18300		Species 1	0		
CZ conc 1	101	GrandRiver	34500	18300	0	Species 2	1	.\time\obsdata.dfs0	1
CZ conc 2	101	GrandRiver	34500	18300	1	Species 3	1	.\time\obsdata.dfs0	2

Related Items:

- Time Step Control *(p. 190)*

- Detailed River Time Series Output *(p. 350)*
- Grid series output *(p. 352)*
- MIKE SHE Detailed Time Series *(p. 370)*

11.21.3 Detailed River Time Series Output



The Detailed time series output for the river allows you to specify the river chainage location at which you want detailed time series output and the item that you want output. For each specified point, the output variable is stored in a .dfs0 file with one value for every simulation time step. Finally, for each item in the detailed time series table, an HTML plot is created in the Result Tab - with or without observation data.

The principle advantage of this option is that you can now easily create calibration plots of calculated versus observed water levels without opening and having to create specific plots in MIKE View.

Minimum Output time step - In some simulations, the output time step can become very small due to the automatic time step adjustment. If the output is being saved every time step, then the output files can become very large. This value allows you to set a lower limit on the output time step. Note, however, that the sub-time step values are averaged over the Minimum output time step.

Name - This is a text field that can be used to specify a reference name for the location, for example, a gage name.

Data Type - This is the list of available output items, which for MIKE 1D only contains two items - water level and flow rate.

Branch name - the Branch name must be a valid branch name in the river model. However, this is not checked until run time, at which point an error message will be generated if it is not valid and the simulation will be stopped.

Chainage - like the branch name, the chainage must be a valid river chainage.

Include observation data - If this is checked, then a .dfs0 file can be specified that includes observation points. The observation points are automatically plotted along with the results in the HTML plot on the Results tab. The .dfs0 item is selected in the file browser dialogue. The **Edit** button opens the specified .dfs0 file and the **New** button can be used to create a new .dfs0 file with the correct item type etc. and at the same time import data from an Excel spreadsheet.

Importing data

Detailed River Time Series data can be imported directly into the dialogue using the Import button. The data file must be a tab- delimited ASCII file without a header line. The file must contain the following fields and be in the format specified below.

Name>DataTypeCode>BranchName>Chainage>UseObsdata>dfs0file name>dfs0ItemNumber

where the > symbol denotes the Tab character and

Name - is the user specified name of the observation point. This is the name that will be used for the time series item in the Dfs0 file created during the simulation.

DataTypeCode - This is a numeric code used to identify the output data type: 0 for Water Level and 1 for Discharge.

Branch name - the Branch name must be a valid branch name in the river model. However, this is not checked until run time, at which point an error message will be generated if it is not valid and the simulation will be stopped.

Chainage - like the branch name, the chainage must be a valid river chainage.

UseObsData - This is a flag to specify whether or not an observation file needs to be input: 0 = No; 1 = Yes

dfs0file name - This is the file name of the dfs0 time series file with observation data. The path to the dfs0 file must be relative to the directory containing the MIKE SHE *.she document. The .dfs0 extension is added to the file name automatically and must be not be included in the file name. For example,

.*Time**Calibration**RiverstageObs*

refers to the file *RiverStageObs.dfs0* located in the subdirectory *Time**Calibration,* which is found in the same directory as the .she model document. The file name must not be empty even if UseObsData is set to 0.

dfs0ItemNumber - This is the Item **number** of the observation data in the specified DFS0 file.

The following is a simple example with three river observation points, where the file name is obsdata.dfs0:

Obs_1 > ? > GrandRiver > 34500. >0 >.\time\obsdata >1

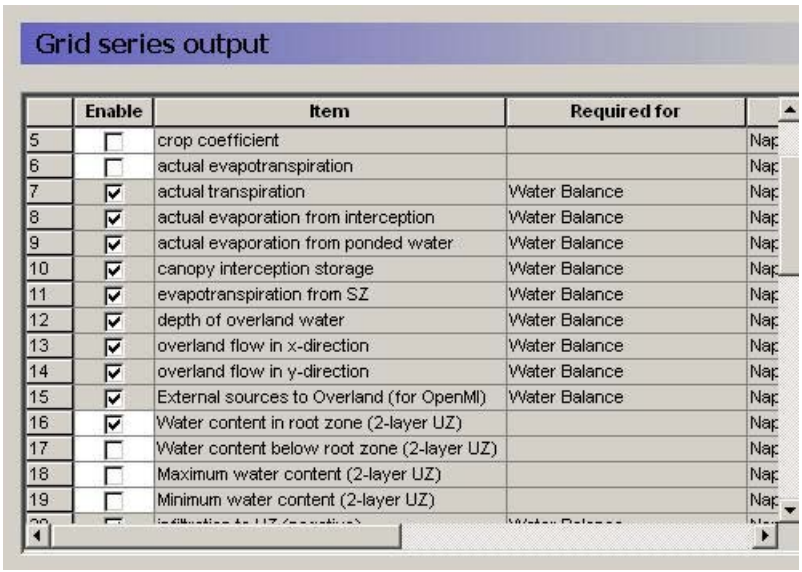
Obs_2 > ? > GrandRiver > 22500. >1 >.\time\obsdata >2

Obs_3 > ? > GrandRiver > 1500. >1 >.\time\obsdata >3

Related Items:

- Time Step Control *(p. 190)*
- Detailed WM time series output *(p. 344)*
- Grid series output *(p. 352)*
- Detailed River Time Series *(p. 373)*

11.21.4 Grid series output



The Grid series output dialogue allows you to specify the frequency at which you want detailed output of gridded data and the items that you want output. A list of available Data Types can be found in Output Items *(p. 92)*. The list is dynamic in the sense that the list changes in response to the processes included in the Simulation Specification dialogue.

In some cases, such as when the Water Balance output has been specified (see Storing of Results), some of the items will be automatically selected and cannot be unselected. This will be noted in the Required for column of the dialogue.

Related Items:

- Time Step Control *(p. 190)*
- Detailed WM time series output *(p. 344)*
- Detailed River Time Series Output *(p. 350)*
- Gridded Data Results Viewer *(p. 371)*

11.22 Extra Parameters

The Extra Parameters Section is available to support new features in MIKE SHE that are not yet supported in the dialogues and data tree. Detailed descriptions of the features that use Extra Parameters are found in Extra Parameters *(p. 749)*.

If you need to activate a feature that is only supported in the Extra Parameters section, you must first add the necessary number of lines to the Extra Parameters table. Then fill in the data that is required for the module.

Name - this is the name of the parameter that is required by the unsupported feature. It must be spelled exactly as specified in the documentation. In may be the actual name of the feature or the name of a parameter.

Type - The type is the type of parameter. The following types are available:

Float - Real, floating point number

Integer - Integer number

Boolean - an On/Off checkbox - typically used to turn a feature on or off

Text - a character string

file name - this is typically the file where more detailed input data is recorded

Value - this is the value associated with the Type above

11.23 Data description

11.23.1 Layer description

In some cases an input item may be defined as either a constant value or more detailed using a Layer based approach. In the latter case you need to create a layer for each input. Then for each specified layer a sub-item will appear where you can specify the input data.

11.23.2 Item description

For some items an input item comprises both a spatial distribution and a value that may be time varying.

The spatial distribution of the item can be defined in three ways:

- Uniform
- Station based
- Fully distributed

Uniform

The Uniform spatial distribution defines the same value in the entire area. The value can be defined by a constant or by a time-varying dfs0 file.

Station based

If the data is sub area-based the area is divided into a number of sub areas, one for each station. You define the spatial area related to each station by grid codes in a dfs2 file or by polygons in a shape file (.shp). Then for each station a sub-item will appear where you can enter the time series of values for the station.

Fully distributed

If the data is fully distributed you need to specify a dfs2 file with the values to be used.

11.24 Geometry

Many items in MIKE SHE are defined by a surface elevation. For example, numerical and geologic layers both require bottom surfaces. Some items, such as geologic lenses also require an upper surface. Finally, some items are only defined in parts of the domain, which requires a definition of the areal extent of the parameter.

11.24.1 Upper Level

Upper Level	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The Upper Level is the upper elevation of the lense or water quality source. It is used by the interpolation algorithm to assign geologic and water quality properties to the model cells.

Related Items:

- Geological Lenses *(p. 314)*
- Lenses *(p. 64)*
- Water Quality Sources *(p. 339)*

11.24.2 Lower Level

Lower Level	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The Lower Level value is used to define the bottom of geologic layers and lenses, numerical layers, and water quality layers and sources.

The bottom of a layer always equal to the top of the layer underneath. However, in the case of lenses and sources, the lower level is used to interpolate geologic and source properties to the model cells. In the case of numerical layers, the lower level is only used when you are using the explict definition option.

Related Items:

- Initial Conditions (UZ) *(p. 303)*
- Geological Layers *(p. 313)*
- Geological Lenses *(p. 314)*
- Lenses *(p. 64)*
- Water Quality Sources *(p. 339)*

11.24.3 Horizontal Extent

Horizontal Extent	
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

Lenses

The horizontal extent is used to define the lateral extents of geologic lenses. The horizontal extents is usually a .shp file polygon, or a dfs2 grid file. In either case, the polygon name or the .dfs2 codes are ignored. Any cell within a polygon or with a grid code different than 0 is treated as part of the lense.

Water Quality Sources

If the source extent is Part Domain, then you can define a local extent for which the larger source file is applied. This allows you to define, for example, a distributed global source file - say of the field scale agricultural inputs in your catchment - and run individual water quality scenarios for each sub-catchment (modelled as an partial extent) to assess the subcatchment contributions to the global stream impact.

Related Items:

- Geological Lenses *(p. 314)*
- Lenses *(p. 64)*
- Water Quality Sources *(p. 339)*

11.25 Water Quality Decay Processes

If WQ in unsaturated flow is to be calculated, then each decay process specified in the Water Quality Sorption and Decay *(p. 225)* dialog will add one item to this data tree branch, with the name equal to that specified in the dialog. Each branch will include the first-order decay half life for the species specified in the decay process.

11.25.1 Half Life

Half Life	
Condition	when Water quality processes are selected in the Water Quality Simulation Specification dialogue AND at least one decay process is active
dialogue Type	Stationary Real Data
EUM Data Units	Half-life

The half life equals the length of time for half of the current solute to disappear. For rapidly decaying solutes a typical half life could be a few days to a few months. For slowly decaying species this could be a few years to many centuries. But, for long-lived radioactive elements the half life can be 10000 years or more.

Related Items:

- Water Quality Sorption and Decay *(p. 225)*
- Solute Transport in Overland Flow *(p. 693)*
- Solute Transport in the Unsaturated Zone *(p. 688)*
- Solute Transport in the Saturated Zone *(p. 677)*

11.25.2 Macropore/Secondary Half Life

Macropore/Secondary Half Life	
Condition	when Water Quality processes are selected in the Water Quality Simulation Specification dialogue AND at least one decay process is active AND Full Macropore Flow selected in the main Unsaturated dialog OR Transfer to immobile water selected for SZ
dialogue Type	Stationary Real Data
EUM Data Units	Half-life

The half life equals the length of time for half of the current solute to disappear. For rapidly decaying solutes a typical half life could be a few days to a few months. For slowly decaying species this could be a few years to many

centuries. But, for long-lived radioactive elements the half life can be 10000 years or more.

The decay half life in macropores may or may not be the same as in the bulk media. For example, water in the macropores may be more oxygenated.

Related Items:

- Water Quality Sorption and Decay *(p. 225)*
- Solute Transport in Overland Flow *(p. 693)*
- Solute Transport in the Unsaturated Zone *(p. 688)*
- Solute Transport in the Saturated Zone *(p. 677)*

11.26 Water Quality Sorption processes

If WQ in unsaturated flow is to be calculated, then each sorption process specified in the Water Quality Sorption and Decay *(p. 225)* dialog will add one item to this data tree branch, with the name equal to that specified in the dialog. Each branch will include the appropriate sorption coefficients for the selected sorption isotherm for the species specified in the sorption process.

11.26.1 Linear Kd

Linear Kd	
Condition	when water quality for Unsaturated or Saturated flow is selected in the Water Quality Simulation Specification dialogue and the Linear Isotherm is selected for the sorption process
dialogue Type	Stationary Real Data
EUM Data Units	Kd

The linear sorption isotherm is mathematically the simplest isotherm and can be described as a linear relationship between the amount of solute sorbed onto the soil material and the aqueous concentration of the solute:

$$c^* = K_d c \tag{11.9}$$

where K_d is known as the distribution coefficient.

The distribution coefficient is often related to the organic matter content of the soil by an experimentally determined parameter (K_{oc}) which can be used to calculate the K_d values.

$$K_d = f_{oc} \cdot K_{oc} \tag{11.10}$$

where f_{oc} is the fraction of organic carbon.

Commonly referred to is the retardation factor (R), which is the ratio between the average water flow velocity (v) and the average velocity of the solute plume ($_vc$). The retardation factor is calculated as

$$R = \frac{v}{v_c} = 1 + \frac{\rho_b}{\theta} \cdot K_d \tag{11.11}$$

Related Items:

- Water Quality Sorption and Decay *(p. 225)*
- Solute Transport in the Unsaturated Zone *(p. 688)*
- Solute Transport in the Saturated Zone *(p. 677)*
- Equilibrium Sorption Isotherms *(p. 700)*

11.26.2 Freundlich Kf and N

Freundlich Kf and N	
Condition	when water quality for Unsaturated or Saturated flow is selected in the Water Quality Simulation Specification dialogue and the Freundlich Isotherm is selected for the sorption process
dialogue Type	Stationary Real Data
EUM Data Units	Kd and Dimensionless exponent

The Freundlich sorption isotherm is a more general equilibrium isotherm, which can describe a non-linear relationship between the amount of solute sorbed onto the soil material and the aqueous concentration of the solute

$$c^* = K_f \cdot c^N \tag{11.12}$$

where K_f and N are constants. The relationship between K and N is shown in Figure 37.1.

Related Items:

- Water Quality Sorption and Decay *(p. 225)*
- Solute Transport in the Unsaturated Zone *(p. 688)*
- Solute Transport in the Saturated Zone *(p. 677)*
- Equilibrium Sorption Isotherms *(p. 700)*

11.26.3 Langmuir alpha and beta

Langmuir alpha and beta	
Condition	when water quality for Unsaturated or Saturated flow is selected in the Water Quality Simulation Specification dialogue and the Langmuir Isotherm is selected for the sorption process
dialogue Type	Stationary Real Data
EUM Data Units	Kd and Concentration_4

Both the linear and the Freundlich isotherm suffer from the same fundamental problem. That is, there is no upper limit to the amount of solute that can be sorbed. In natural systems, there is a finite number of sorption sites on the soil material and, consequently, an upper limit on the amount of solute that can be sorbed. The Langmuir sorption isotherm takes this into account. When all sorption sites are filled, sorption ceases. The Langmuir isotherm is often given as

$$\frac{c}{c^*} = \frac{1}{\alpha\beta} + \frac{c}{\beta} \tag{11.13}$$

or

$$c^* = \frac{c\alpha\beta}{1 + \alpha c} \tag{11.14}$$

where α is a sorption constant related to the binding energy and β is the maximum amount of solute that can be absorbed by the soil material. The relationship between α and β is shown in Figure 37.2.

Related Items:

- Water Quality Sorption and Decay *(p. 225)*
- Solute Transport in the Unsaturated Zone *(p. 688)*

- Solute Transport in the Saturated Zone *(p. 677)*
- Equilibrium Sorption Isotherms *(p. 700)*

11.26.4 Kinetic Sorption/Desorption Rate

Kinetic Sorption Rate	
Condition	when water quality for Unsaturated or Saturated flow is selected in the Water Quality Simulation Specification dialogue AND at least one active Sorption process is using the Kinetic/Equilibrium isotherm option
dialogue Type	Stationary Real Data
EUM Data Units	1st order rate WQ model

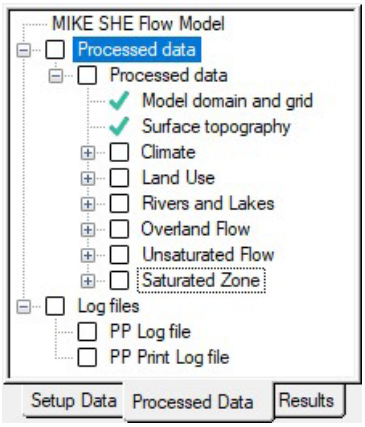
The kinetic sorption rate is the specified rate at which the solute is absorbed onto the soil matrix. The kinetic rate is used whenever a kinetic sorption isotherm is specified.

The kinetic de-sorption rate is the specified rate at which the solutes go back into solution from the soil matrix. The desorption rate is used whenever hysteresis is allowed for the kinetic sorption isotherm is used.

Related Items:

- Water Quality Sorption and Decay *(p. 225)*
- Solute Transport in the Unsaturated Zone *(p. 688)*
- Solute Transport in the Saturated Zone *(p. 677)*
- Kinetic Sorption Isotherms *(p. 702)*

12 Preprocessed Data Tab



In the Setup Tab, you specify the input data required by the model - including the size of the model and the numerical grid. However, most of the setup data is independent of the model extent and grid. When you pre-process your model set up, MIKE SHE's pre-processor program scans through your model set up and interpolates all spatial data to the specified model grid. This interpolated set up data is stored in a .fif file, which is read during the simulation by the MIKE SHE engine. However, the pre-processed data does not include any time information. All time series information must be interpolated dynamically during the run because MIKE SHE dynamically changes the time step during the simulation in response to stresses on the system.

The Preprocessed Data Tab is used to display the pre-processed data.

Before you run your simulation, you should carefully check the preprocessed data for errors. Errors found in the preprocessed data are typically related to incorrectly specified parameters, file names, etc. in the Setup Tab.

The Preprocessed Tab includes a data tree with two items:

- Processed Data *(p. 363)*
- Log Files *(p. 367)*

12.1 Processed Data

On the main pre-processed dialog, there is an uneditable textbox containing the file and location of the pre-processed data. This is a .pfs ASCII file containing the file references for all of the data. The actual data is stored in a .fif file, as well as a number of dfs2 and dfs3 files.

After you have successfully preprocessed your model, the pre-processed data will be automatically loaded when you expand the data tree. The data

tree reflects all the spatial data defined in the model set up tab. In other words, if the overland flow is not included in the Simulation Specification *(p. 186)* dialogue, then the Overland item will not be included in the pre-processed data tree.

Note If you change your model setup data, the pre-processed data will not reflect the changes until you pre-process your model again.

12.1.1 Viewing the pre-processed data

In all map and time series views, there is an edit button. This edit button will open the dfs0, dfs2 or dfs3 file that was generated by the pre-processor in either the Grid Editor or the Time Series Editor.

In all map views, there is also a view button. This view button will open the dfs2 or dfs3 file that was generated by the pre-processor in the results viewer.

Each of these files usually contains a large number of data items. The Grid- or the Time Series Editor opens at the first item, so you must use the scrolling function in the editor to find the data item that you want. In the results viewer the selected item will be displayed right away.

12.1.2 Editing the pre-processed data

MIKE SHE reads only the .fif file during the simulation. The .dfs2 and dfs3 files are created to make it easier to view and plot the preprocessed data. If you edit the dfs2 or dfs3 files, the changes will not be used in the simulation.

If you want to change the pre-processed data and use the changed data in the simulation, you have a couple of options.

Option 1

1. Right click on the map view and save the data to a new dfs2 file,
2. open the new dfs2 file in the Grid Editor, and
3. make the changes in the new dfs2 file and save the file.

Option 2

1. Use the edit button to open the dfs2 or dfs3 pre-processed file in the Grid Editor,
2. make your changes in the file, and
3. save the file with a new name.

In both options above, you then use the new dfs2 or dfs3 file as input in the Setup tab.

12.1.3 Rivers

The coupling between the river model and MIKE SHE is made via river links, which are located on the edges that separate adjacent grid cells. The river link network is created by the pre-processor, based on the river model coupling reaches. The entire river system is always included in the hydraulic model, but MIKE SHE will only exchange water with the coupling reaches.

The location of each of MIKE SHE river link is determined from the co-ordinates of the river points, where the river points include both digitised points and H-points on the specified coupling reaches. Since the MIKE SHE river links are located on the edges between grid cells, the details of the river geometry can be only partly included in MIKE SHE, depending on the MIKE SHE grid size. The more refined the MIKE SHE grid, the more accurately the river network can be reproduced. This also leads to the restriction that each MIKE SHE grid cell can only couple to one coupling reach per river link. Thus, if, for example, the distance between coupling reaches is smaller than half a grid cell, you will probably receive an error, as MIKE SHE tries to couple both coupling reaches to the same river link.

The river links are shown on Rivers and Lakes data tree pages, as well as the SZ Drainage to River page.

Related Items:

- Rivers, Lakes and Sewers *(p. 268)*
- Coupling of MIKE SHE and a MIKE 1D River Model *(p. 477)*
- Channel Flow - Technical Reference *(p. 471)*

12.1.4 Land Use

The vegetation distribution is displayed on a map, but if you use the vegetation database for specifying the crop rotation, this information will not be displayed in the pre-processor.

12.1.5 Unsaturated Flow

The Unsaturated Flow data tree in the pre-processed data contains a two noteworthy data items.

Soil profiles

Under the unsaturated zone, you will find a map with the grid codes for each of the soil profiles used. Accompanying this map is a text page containing the details of all the soil profiles. At the top of this page is the path and file name of the generated txt file, which you can open in any text editor.

Note If you are using one of the finite difference methods, the pre-processor modifies the vertical discretisation wherever the vertical cell size changes.

Thus, if you have 10 cells of 20cm thickness, followed by 10 cells of 40cm thickness, the location of the transition will be moved such that the two cells on either side will be have an equal thickness. In this case, cells 10 and 11 will both be 15cm.

UZ Classification Codes

If certain conditions are met, then the flow results for a 1D unsaturated zone column can be applied to columns with similar properties. If you chose to use this option, then a map will be generated that shows the calculation cells and the corresponding cells to which the results will be copied.

The cell with a calculation is given an integer grid code with a negative value. The flows calculated during the simulation in the cells with the negative code, will be transferred to all the cells with the same positive grid code value. For example, if an UZ recharge to SZ of 0.5 m^3/day is calculated for UZ grid code -51, then all the SZ cells below the UZ cells with a grid code of +51 will also be given the same recharge.

Tip This map can be difficult to interpret without using the Grid Editor.

Related Items:

- Soil Profile Definitions *(p. 293)*
- Unsaturated Zone *(p. 288)*
- Enable column classification *(p. 289)*
- Partial automatic classification *(p. 298)*
- Specified classification *(p. 299)*

12.1.6 Saturated Flow

The saturated zone data is generally written to a dfs3 file. In the map view, there is a combo box where you can specify the layer that you want to view.

Saturated Zone Drainage

The rate of saturated zone drainage is controlled by the drain elevation and the drain time constant. However, the destination of the drainage water is controlled by the drain levels and the drain codes, which determine if the water flows to a river, a boundary, or a local depression. The algorithm for determining the drainage source-recipient reference system is described in Groundwater drainage *(p. 65)*.

During the preprocessing, each active drain cell is mapped to a recipient cell. Then, whenever drainage is generated in a cell, the drain water will always be moved to the same recipient cell. The drainage source-recipient reference system is displayed in the following two grids

Drainage to local depressions and boundary - This grid displays all the cells that drain to local depressions or to the outer boundaries. All drainage

from cells with the same negative value are drained to the cell with the corresponding positive code. If there is no corresponding positive code, then that cell drains to the outer boundary, and the water is simply removed from the model. Cells with a value of zero either do not generate drainage, or they drain to a river link.

Drainage to river - This grid displays all of the cells that drain to river links. All drainage from cells with the same negative value are drained to the cell with the corresponding positive code. Cells with a value of zero either do not generate drainage, or they drain to a the outer boundary or a local depression.

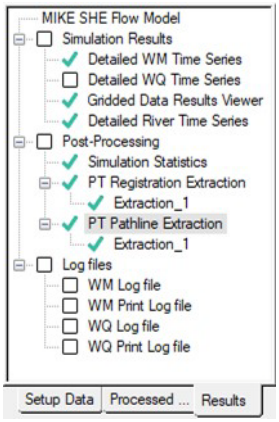
Related Items:

- Groundwater drainage *(p. 65)*
- Drainage *(p. 331)*
- Drain Level *(p. 334)*
- Drain Time Constant *(p. 335)*
- Drain Codes *(p. 335)*
- Option Distribution *(p. 337)*

12.2 Log Files

The log files section contains a view of two log files from the preprocessor. The "PP Log File" contains messages, warnings and errors. This is the file created in the same folder as the .she file. It is also displayed in the simulation tab when running the preprocessor. The "PP print log file" contains additional details. It is created in the results directory.

13 Results Tab



All the simulation results are collected in the Results tab.

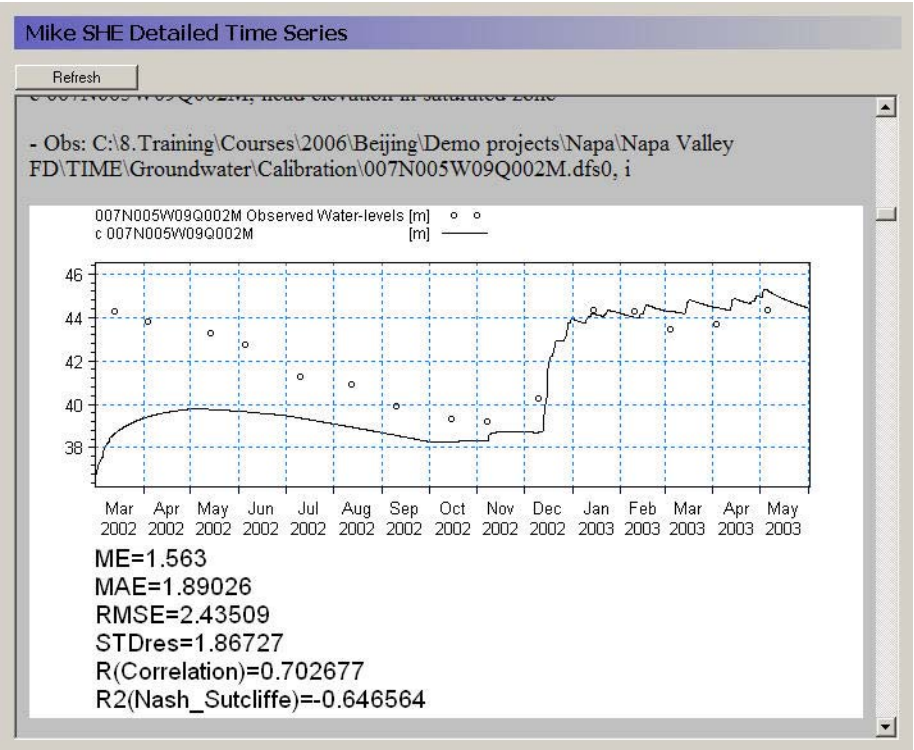
The Simulation results are grouped under the Simulation Results section. This includes Detailed time series output for both MIKE SHE and the river, as well as Grid series output for MIKE SHE.

Under the Results Post-processing, you can find tools for extracting and manipulating the simulation results. This is where you will find the tools for extracting the shape file outputs for the Random walk particle tracking simulation. A Simulation Statistics tool is also available for helping you assimilate the calibration statistics for each of the detailed time series plots.

13.1 Results of Simulation

The Results section includes the direct visualization and analysis of the simulation results

13.1.1 MIKE SHE Detailed Time Series



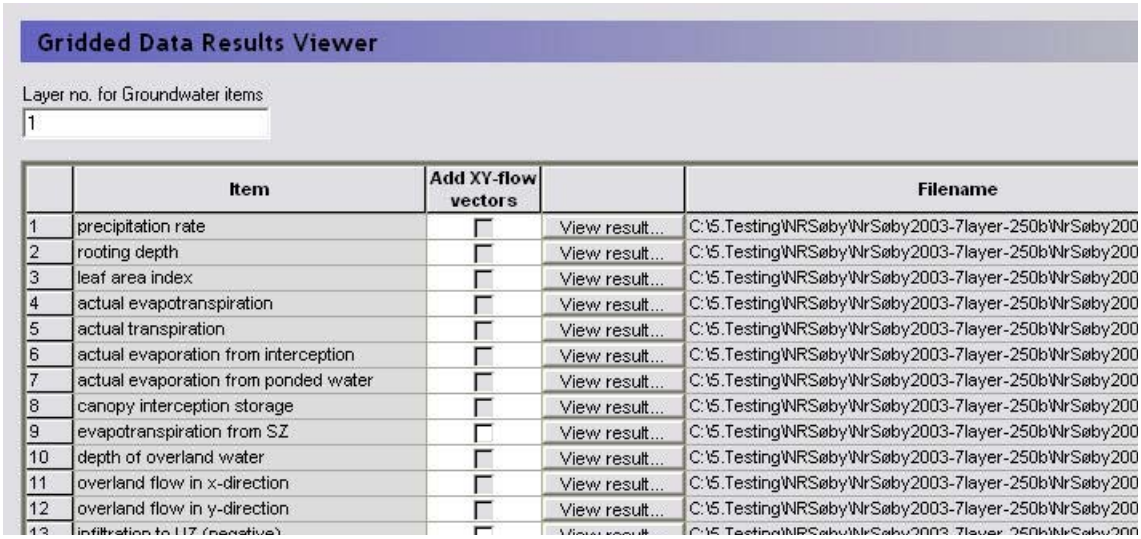
The MIKE SHE Detailed time series tab includes an HTML plot of each point selected in the Detailed WM time series output *(p. 344)* dialogue. The HTML plots are updated during the simulation whenever you enter the plot, or click on the Refresh button.

When you have more than five detailed time series items, the plots are divided into seperate pages. In this case the main page is a list of the links to the individual plots and each html page includes five plots.

Related Items:

- Detailed WM time series output *(p. 344)*
- Statistic Calculations *(p. 375)*

13.1.2 Gridded Data Results Viewer



This table is a list of all gridded data saved during a MIKE SHE simulation. The items in the list originate from the list of items selected in the Grid series output *(p. 352)* dialogue from the Setup tab.

View Result... - Clicking on the View result button will open the Results Viewer to the current item. All overlays from MIKE SHE (e.g. shape files, images, and grid files) will be transferred as overlays to the result view. However, the river network is not transferred as an overlay.

Layer number for Groundwater Items - For 3D SZ data files, the layer number can be specified at the top of the table. However, the layer number can also be changed from within the Results Viewer. By default the top layer is displayed.

Add XY-flow Vectors - Vectors can be added to the SZ plots of results, by checking the *Add X-Y flow vectors* checkbox. These vectors are calculated based on the *Groundwater flow in X-direction* and *Groundwater flow in Y-direction* data types if they were saved during the simulation. In the current version, velocity vectors cannot be added for overland flow output.

file name - The file name column shows the name of the result file from which the gridded data will be extracted.

"The Result Viewer setup file already exists" warning

When the Result Viewer opens one of the items in the table, it creates a setup file for the particular view with the extension .rev. The name of the current setup file is displayed in the title bar of the dialogue. Initially, the .rev file includes only the default view settings and the overlay information from MIKE

SHE. However, if you make changes to the view, such as changes the way contours are displayed, then when you close the view, you will be asked if you want to save your changes.

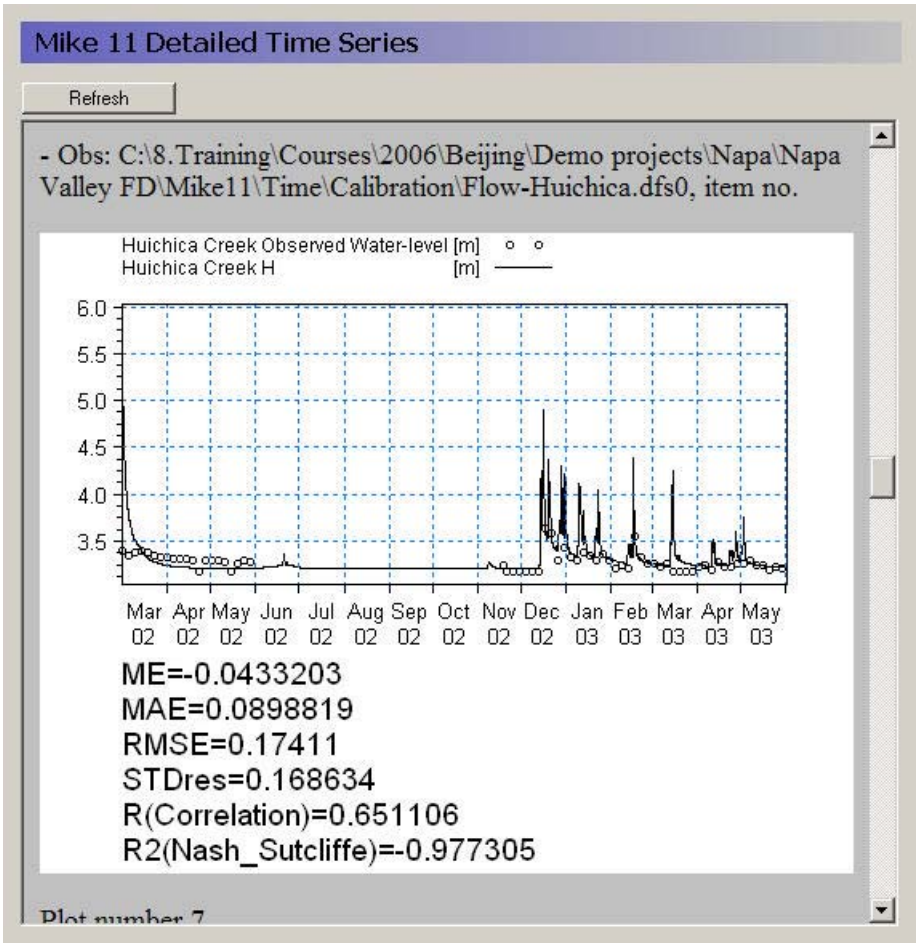
The next time you open the item in the table, you will be asked if you want to overwrite the existing .rev file. If you click on “Yes”, then a new .rev file will be created with the default values. If you click on “No”, then your previous settings will be re-loaded. This is a convenient way to set up the contouring, legend, etc., the way you want and then re-use the settings.

The .rev file can also be loaded directly in the Results Viewer by double clicking on the .rev file or loading the file into the MIKE Zero project explorer.

Related Items:

- Grid series output *(p. 352)*
- The Results Viewer *(p. 97)*

13.1.3 Detailed River Time Series



The detailed river time series page includes an HTML plot of each point selected in the Detailed River Time Series Output *(p. 350)* dialogue. The HTML plots are updated during the simulation whenever you enter the view. Alternatively, you can click on the Refresh button to refresh the plot.

Note that the HTML plot is regenerated every time you enter the view. So, if you have a lot of plots and a long simulation, then the regeneration can take a long time.

When you have more than five detailed time series items, the plots are divided into seperate pages. In this case the main page is a list of the links to the individual plots and each html page includes five plots.

Related Items:

- Detailed River Time Series Output *(p. 350)*

- Statistic Calculations *(p. 375)*

13.2 Results Post-processing

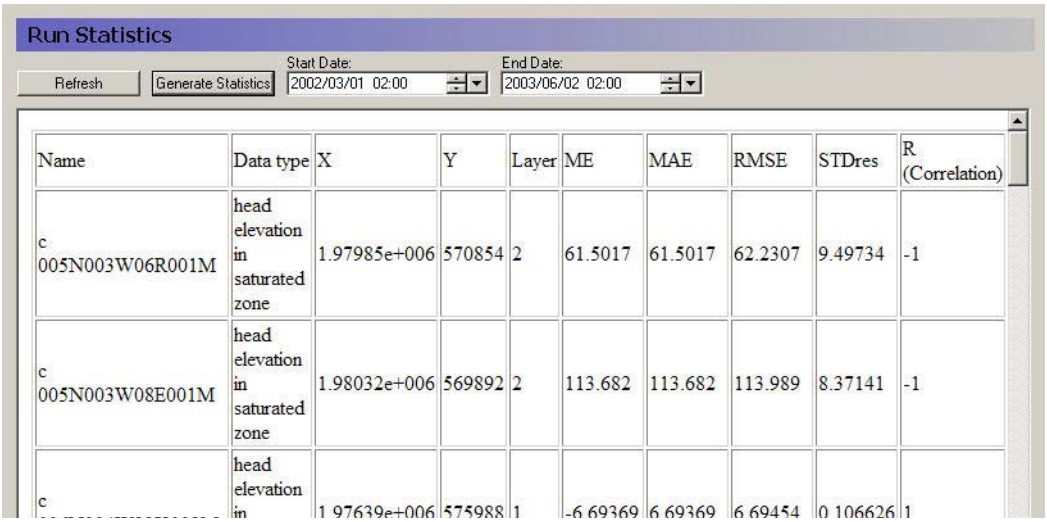
The post-processing section includes options for extracting the results from the various results files.

13.2.1 Run Statistics

Run statistics is a summary of the run statistics for the river model and MIKE SHE detailed time series. The statistics are generated in HTML and shape file format. The calculations include all the detailed time series items that have observation data.

To calculate Run Statistics for a simulation, select the period you want to summarize, then click on the Generate Statistics button. For some simulations with long simulation periods and/or a lot of calibration data it can take several minutes to generate the run statistics.

The Run Statistics are displayed in HTML format on the Run Statistics page (see below).The HTML table can be copied and pasted directly into Microsoft Word or Excel.



Name	Data type	X	Y	Layer	ME	MAE	RMSE	STDres	R (Correlation)
c 005N003W06R001M	head elevation in saturated zone	1.97985e+006	570854	2	61.5017	61.5017	62.2307	9.49734	-1
c 005N003W08E001M	head elevation in saturated zone	1.98032e+006	569892	2	113.682	113.682	113.989	8.37141	-1
c	head elevation in	1.97639e+006	575988	1	-6.69369	6.69369	6.69454	0.106626	1

Similar to the detailed time series output, the Run Statistics can be viewed during a simulation. Press the Refresh button on the Run Statistics page to update the Run Statistics using the most recent model results during a simulation

Shape file output for run statistics

A shape file of statistics is also generated when the HTML document is generated. The shape file contains all of the information contained in the HTML document and can be used to generate maps of model errors that can be used to evaluate spatial bias. The shape file is created in the simulation directory and is named *projectname*_Stat.shp where SimulationName is the name of the *.she file for the simulation. Note: the Run Statistics shape file does not have a projection file associated with it. This file must be created using standard ArcGIS methods.

Statistic Calculations

The statistics contained in the HTML document and the shape file are calculated using the same methods used to calculate statistics for the detailed time series output.

The standard calibration statistics calculated based on the differences between the measured observations and the calculated values at the same location and time. Thus, the error, or residual, for an calculation - observation pair is

$$E_{i,t} = Obs_{i,t} - Calc_{i,t} \tag{13.1}$$

where $E_{i,t}$ is the difference between the observed and calculated values at location i and time t.

Mean (ME)

The mean error at location i where n observations exist is

$$ME_i = \overline{E}_i = \frac{\sum_t (E_{i,t})}{n} \tag{13.2}$$

Mean Absolute Error (MAE)

The mean of the absolute errors at location i where n observations exist is

$$MAE_i = \overline{|E_i|} = \frac{\sum_t |E_{i,t}|}{n} \tag{13.3}$$

Root Mean Square Error (RMSE)

The root mean square error at location i where n observations exist is

$$RMSE_i = \sqrt{\frac{\sum_t (E_{i,t})^2}{n}} \tag{13.4}$$

Standard Deviation of the Residuals (STDres)

The standard deviation of the residuals at location *i* where *n* observations exist is

$$STDres_i = \sqrt{\frac{\sum_t ((E_{i,t}) - \overline{E}_i)^2}{n}} \tag{13.5}$$

The standard deviation is a good measure to evaluate how well the dynamics of a certain observation are simulated.

Correlation Coefficient (R)

The correlation coefficient is a measure of the linear dependency between simulated and measured values. The closer the value is to 1.0, the better the match. The correlation coefficient at location *i* is

$$r_i = \frac{\sum_t (Calc_{i,t} - \overline{Calc}_{i,t}) \cdot (Obs_{i,t} - \overline{Obs}_{i,t})}{\sqrt{\sum_t (Calc_{i,t} - \overline{Calc}_{i,t})^2 \cdot \sum_t (Obs_{i,t} - \overline{Obs}_{i,t})^2}} \tag{13.6}$$

where $\overline{Obs}_i$ and $\overline{Calc}_i$ are the means of the observations and calculations at location *i* respectively.

Nash Sutcliffe Correlation Coefficient (R2)

The Nash-Sutcliffe coefficient at location *i* where *n* observations exist is

$$R2 = 1 - \frac{\sum_t (E_{i,t})^2}{\sum_t (Obs_{i,t} - \overline{Obs}_i)^2} \tag{13.7}$$

where $\overline{Obs}_i$ is the mean of the observations at location *i*. R2 will be 1.0 if there is a perfect match.

13.2.2 Water Balances

It's possible to add Water Balance results to the Results tab. New water balances can be added by clicking on the Insert icon in the Water Balance entry under Post-Processing. The following fields are available for each water balance.

- Enable: If this check box is ticked, then the water balance will be re-computed at the end of each simulate run.

- Name: Optional name field. If blank, the name of the water balance file will be used.
- Water balance configuration file: Path to the water balance file. The Water Balance application can also be opened from here using the Edit button.

After adding a new water balance file, the name is displayed under Water Balance in the Results tab. Individual water balance results are displayed under the water balance name. Individual results are displayed in the same way as in the Water Balance application, and results are updated after each simulation run (if the Enable check box is ticked).

13.2.3 PT Registration Extraction

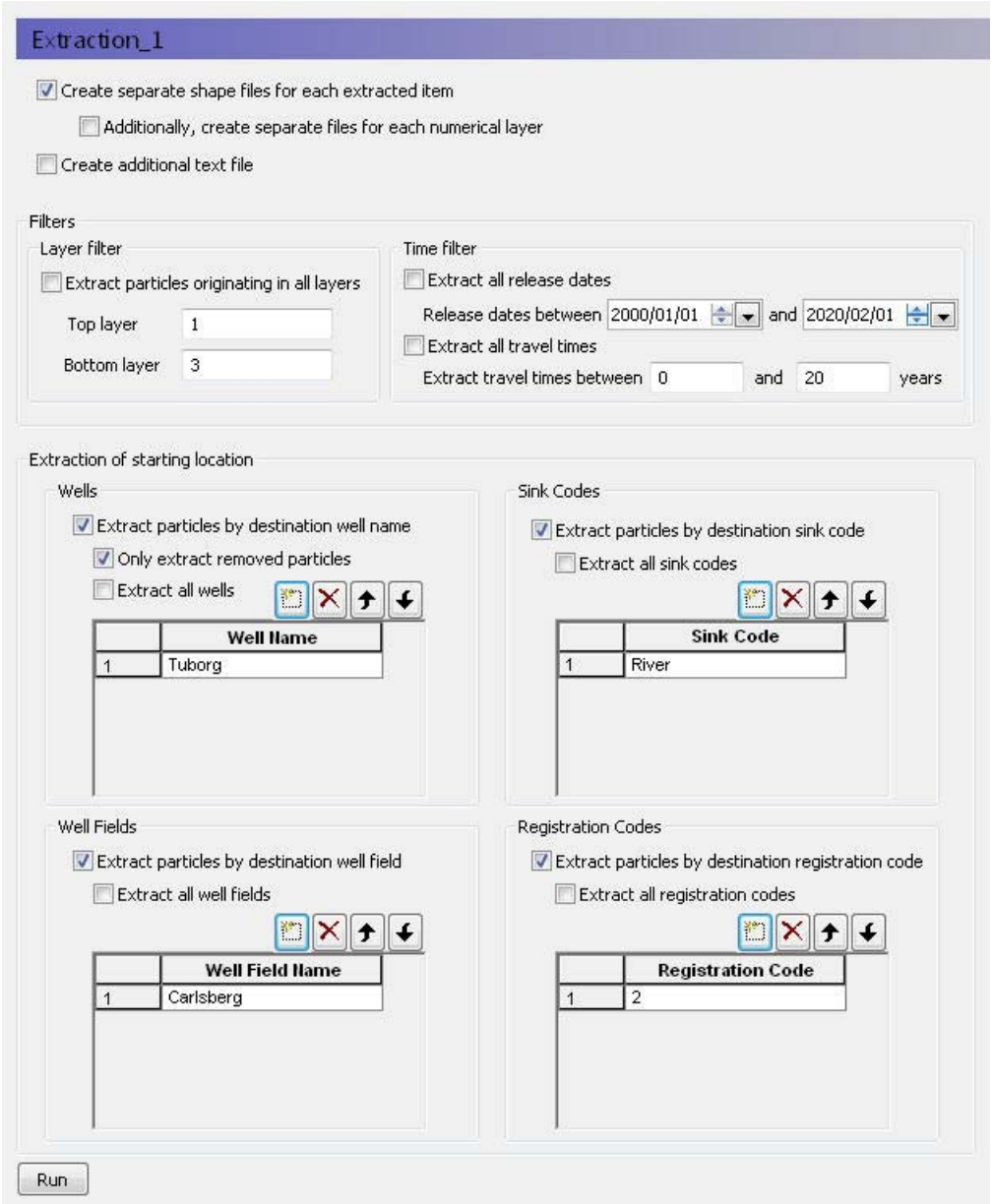
The SZ random walk particle tracking module is very useful for finding out the source of water to an area of interest, for example a well field or a wetland. A property of each particle is its starting location. If the particle is removed from the model or enters a specified area, then the particle is tagged, or 'registered'. At the end of the simulation, you can then sift through all of the particles and find the starting location for all the ones that you want. With this information, you can build up a dyanamic picture of the source of water entering an area or leaving the model through a specific boundary. The extraction itself will compile a point shape file with the starting locations of all the particles satisfying the specified criteria.



In the top level of this data tree item, you can specify any number of different particle extractions. These will be saved from simulation to simulation, allowing you to re-run the extraction for different scenarios.

The buttons at the bottom, allow you to run the selected extraction or all of the specfied extractions.

Extraction properties



The PT module can generate thousands of particles during a simulation. The extraction definition allows you to specify the criteria finding just the particles that you are interested in.

Create separate shape files for each extracted item - If this option is selected, then you will get a seperate shape file for Wells, Well Fields, Sink Codes and Registration codes.

Additionally, create separate files for each extracted layer - If you have selected the option above, then you also have the option to divide the shape files into one for each numerical layer.

Create additional text file - If this option is selected, a space delimited txt file is created with the fields:

ID / XBirth(meter) / YBirth(meter) / ZBirth(meter / IXBirth / IYBirth / LayerBirth / XReg(meter) / YReg(meter) / ZReg(meter) / IXReg / IYReg / LayerReg / BirthTime(year) / TravelTime(year / RegZoneCode / SinkCode / WellName / WellFieldName.

Filters

For each extraction, the search for relevant particles can be restricted to specific layers and time periods.

Extraction of particle starting locations

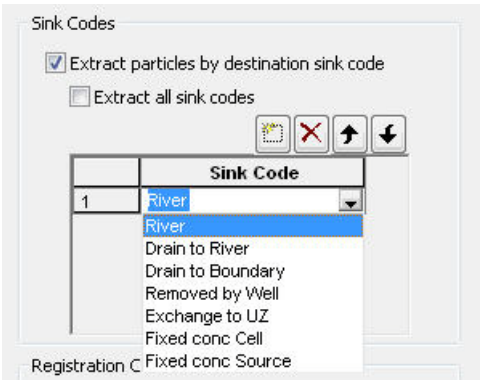
The extraction process finds the starting location of all particles that were registered based on the criteria you specify. By default, all the extraction options are turned off. When you active an extraction criteria, by default all particles that meet the criteria will be extracted. For example, particles for all of the sink codes will be extracted. Each different sink code will have a seperate attribute in the shape file. If you want to refine the search criteria, you can optionally specify the criteria in the table.

Note: an individual particle can be registered multiple times. For example, it might pass through a registration zone, then a well cell without being extracted, and then discharge into a river.

Wells - By default, all particles removed by a well are registered. However, particles that enter a cell containing a well are also registered, even if they are not actually removed by the well. For example, they could flow through the cell or not make it to the well before the simulation ends. There is an option in this section to separate out just the particles that were actually removed by the well. The table allows you to specify individual well names as specified in the Well editor *(p. 401)*.

Well Fields - Each well in the Well Editor can be optionally assigned to a Well Field in the Well Locations *(p. 402)* table. Using the Well Field option, you can group wells together and calculate the influence area of a the entire group of wells.

Sink Codes - Particles can be removed by various sinks. The sinks can be water sinks, such as a river, or they can be concentration sinks.



Drain to River/Boundary - Drain sinks refer to saturated zone drainage and are separated by the discharge point, that is SZ drainage flowing to a river or SZ drainage flowing to a boundary node.

Removed by Well - The particle is removed by a well. Note that, particles that may pass through a cell containing a well.

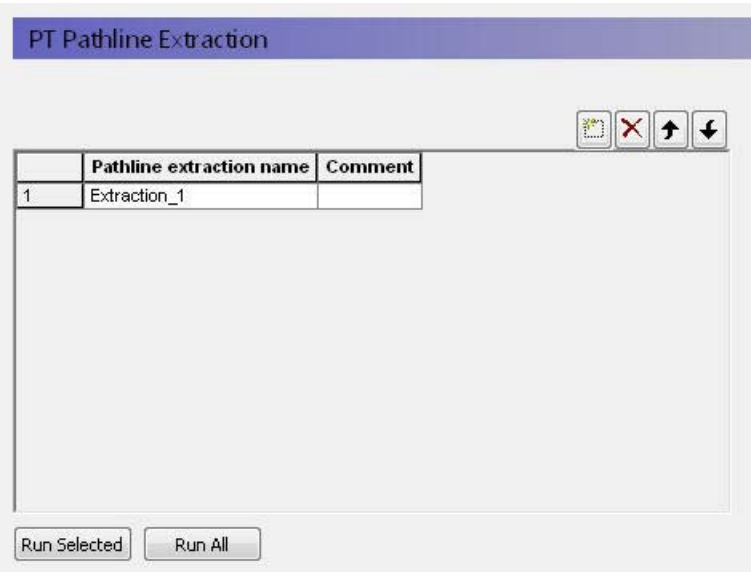
Exchange to UZ - This refers to particles that are removed by capillary flow from SZ into the UZ, or particles that become trapped in the UZ when the water table falls. However, a particle may become trapped and then later reactivated when the water table rises. Each time it is trapped it will be re-registered.

Fixed Conc Cell/Source - Fixed concentration is less than the incoming concentration. Note that the concentration is determined by the particle mass defined in the Particle Tracking Specification *(p. 213)* dialog.

Registration Codes - The registration codes are user defined zones that create a particle registration whenever a particle passes **into** the zone. A particle can be registered multiple times, if it passes in and out of the zone. The registration zones are defined in the PT Registration Codes/Lenses *(p. 330)* dialog. There is no cross-referencing to the list of Registration Codes defined in the simulation, so you have to be careful to specify the code that you want.

13.2.4 PT Pathline Extraction

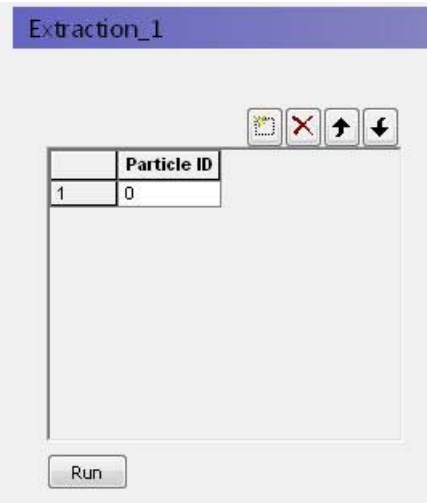
It is too cumbersome to extract and plot the pathlines for all of the thousands of particles that can be generated during a PT simulation. The PT Pathline Extraction utility allows you to extract the pathlines for specific particles if you have saved the intermediate locations in the Storing of Results *(p. 341)* dialog.



In the top level of this data tree item, you can specify any number of different pathline extractions. These will be saved from simulation to simulation, allowing you to re-run the extraction for different scenarios.

The buttons at the bottom, allow you to run the selected extraction or all of the specified extractions.

Extraction Properties



To extract a particle pathline you need a Particle ID. These can only be found after the simulation by evaluating the PT output. For example, you can find the particle ID by extracting the particle start locations that end in a specific

well and then finding the ID numbers of the particles that you want in the shape file that was created.

Keep in mind that the PT Pathline Extraction text output is of 'fixed width' type. This output may then be opened in 3rd party software for further processing and analysis.

13.3 Log Files

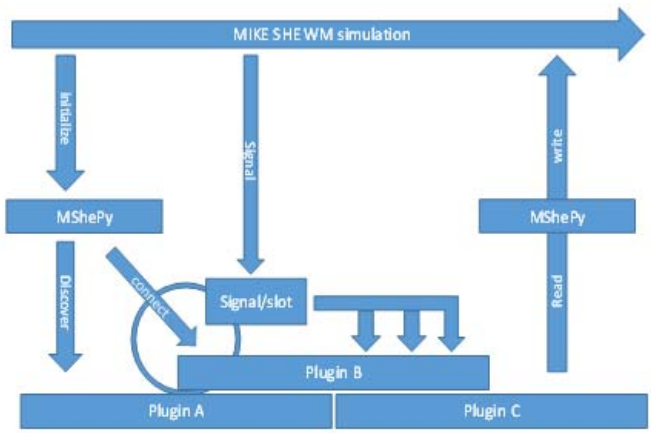
The log files section contains a view of two log files per engine. The "WM Log File" and (if included) the "WQ Log File" contain messages, warnings and errors from the WM and the WQ engines. These are the files created in the same folder as the .she file. They are also displayed in the simulation tab when running the engine. The "WM print log file" and the "WQ print log file" contain additional details. They are created in the results directory.

14 Python Scripting in MIKE SHE

This section describes the MShePy module, which enables the use of Python scripting with MIKE SHE. This includes:

1. running Python plugins during a MIKE SHE simulation;
2. running MIKE SHE simulations by Python scripts.

In the first case, the user can add plugins to a MIKE SHE model. A Plugin consists of a collection of code functions in Python performing custom functionalities. During preprocessing, the Python plugins are attached to the model and once the simulation is started, functions (or "slots") in the plugins will be connected to signals (e.g. before a time step). Signals will be emitted by MIKE SHE and trigger execution of one or more custom functions defined in the plugin. Once the function has finished to perform arbitrary tasks (e.g. exchange data with MIKE SHE), MIKE SHE will continue the normal execution.



In the second case, the user can call and execute the water movement of a MIKE SHE model from a Python shell. This allows controlling the simulation time from the Python shell and exchange items between the MIKE SHE model and the Python environment before, during, and after the simulation.

Requirements

The MShePy module is available from MIKE Zero 2019 version 1. Only water movement (WM) can be accessed through MShePy. Thus, water quality simulations, the preprocessor and other components are currently not supported.

All water movement components can be accessed by MShePy. This includes:

- Overland flow
- River exchange items
- Unsaturated flow
- Evapotranspiration
- Saturated flow

The Python (e.g. from https://www.python.org/) 64-bit versions currently supported are version 3.7, 3.8 and 3.9.

14.1 Python Plugins

14.1.1 How to enable Python plugins in the MIKE SHE user interface

In the Simulation Specification panel (below), tick the Enable Plugins option.

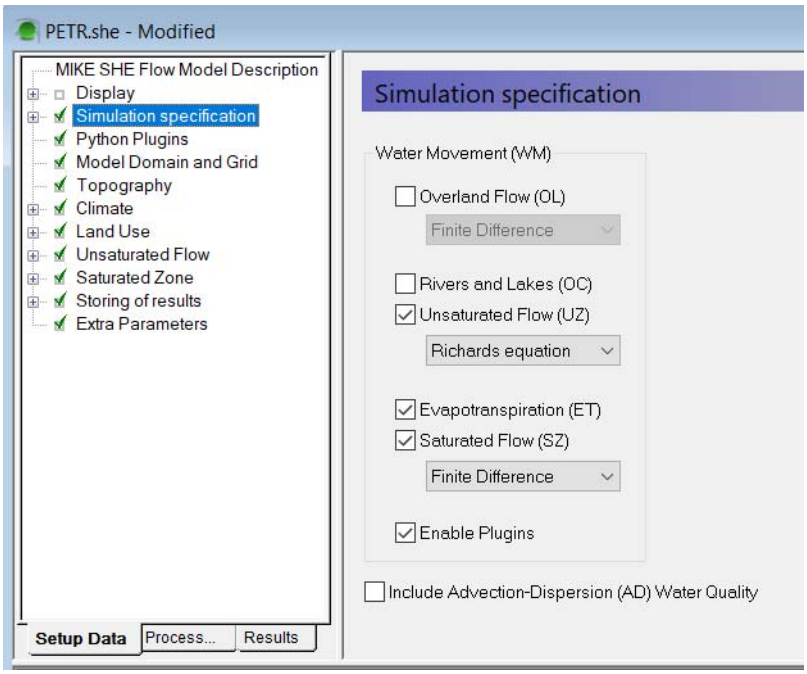


Figure 14.1 How to enable plugins

Then open the Python Plugins panel (Figure 14.2). Under Python Library you can specify how to find the Python dynamic library. "Search default locations" means the operating system´s default search order will be used. On MS Windows this is the application´s directory, several system folders, the current directory and directories in the system PATH variable. The most recent Python version will be preferred. When you tick "Specify" instead you can

provide the location of the Python library file (e.g. python39.dll for version 3.9). You can find it in the installation directory of your Python distribution.

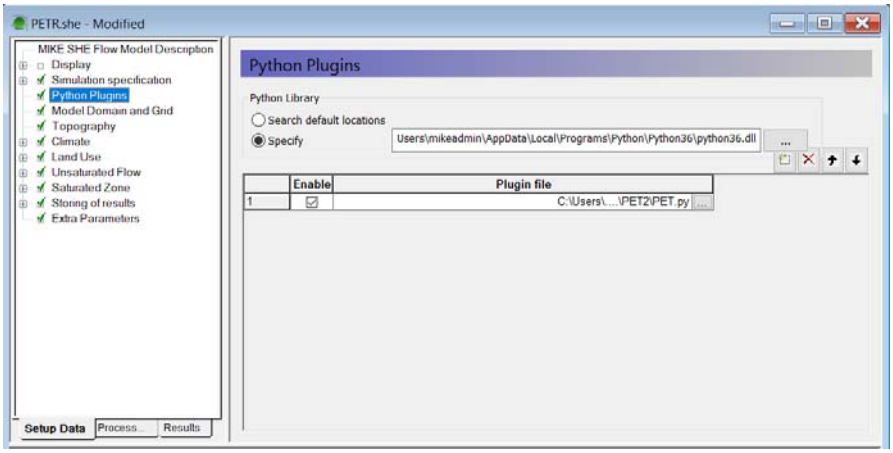


Figure 14.2 Python Plugins panel in editor

Use the insert button to add plugin files to the simulation. Several plugin files can be added to the same MIKE SHE model. Provide the location of the plugin Python file (*.py) and tick the box in the Enable column to activate a plugin file. The order of plugin files in this table will determine the order of execution during the simulation.

14.1.2 Creating a Python plugin

The first step is to import the MShePy module by the command *import MShePy*. The command will import the MShePy version corresponding to the specified Python version (see previous section).

The second step is to define at which point of the simulation custom functions defined in the plugin should be executed. MIKE SHE emits six different signals at predefined moments in the simulation. For each signal that is being emitted one or more plugin functions can be executed. The connection between the signal and the plugin function (or “slot”) is simply made by a naming convention. The function definitions are shown in Table 14.1 below.

Table 14.1 Function definitions

Moment in the simulation	Python function definition
During early initialization	*def enterSimulator():*
At the end of the initialization	*def postEnterSimulator():*
At the beginning of each time step	*def preTimeStep():*

Table 14.1 Function definitions

Moment in the simulation	Python function definition
At the end of each time step	*def postTimeStep():*
After the last time step	*def preLeaveSimulator():*
Right before closing MIKE SHE, after output files have been closed	*def leaveSimulator():*

The third step is to describe actions, which are executed at the defined moments during the simulation, possibly using functions available in the Python environment. An overview of functions included in the MShePy library is provided in the following section. The same plugin file can contain different functions executed at different moments in the simulation. A simple example is shown below for a function printing in the log file the initial timestep count (0) and then the timestep number at each timestep.

```
import MShePy                                  #Import MShePy module
timestep = 0                                   #Define a global variable

def enterSimulator():                          #Define the time slot
  global timeStep
  MShePy.wm.log("The timestep is {0}".format(timeStep))   #Print initial timestep in log file

def postTimeStep():                            #Define the time slot
  global timeStep
  MShePy.wm.log("The timestep is {0}".format(timeStep))   #Print timestep in log file
  timeStep +=1                                 #Calculate next time step
```

14.1.3 Functions available in the MShePy library

A description of the functions available in the MShePy library can be obtained by typing *help(MShePy)* in Python or by typing the name of a specific function (e.g. *help(getValues)*).

14.1.4 How to exchange MIKE SHE items with Python

MShePy provides access to certain MIKESHE exchange items. Some of these are both for in- and output, others may be for either in- or output only. All MIKE SHE items that can be accessed from Python are listed in the summary tables at the end of this chapter. The availability of MIKE SHE items depends on the processes selected in the Simulation specification panel. For example, if the evapotranspiration component is not included in the simula-

tion, then no parameters related to evapotranspiration will be available. The accessible variables are listed in the *_WM.log file, once preprocessing and water movement initialization has been executed.

In order to exchange MIKE SHE items between the Python plugin and the MIKE SHE simulation two functions are available in the MShePy module:

- *MShePy.wm.getValues()* to retrieve a copy of items from MIKE SHE to Python;
- *MShePy.wm.setValues()* to change the value of MIKE SHE items in the simulation.

The input of the *MShePy.wm.getValues()* is a MIKE SHE parameter ID. This can be defined using *MShePy.paramTypes.** followed by the abbreviation corresponding to the parameter (see summary tables at the end of this chapter for the list of abbreviations). The output of the *getValues* function is a tuple containing the starting time as a datetime.datetime object, the current time as a datetime.datetime object, and a MshePy dataset containing the value of the selected MIKE SHE item (see following section for a description of theMShePy dataset).

```
>>> MShePy.wm.getValues(MShePy.paramTypes.P_RATE)
(datetime.datetime(2000, 1, 5, 22, 0),
datetime.datetime(2000, 1, 6, 0, 0),
MShePy36.dataset: "precipitation rate", Precipitation Rate (meter/sec),
in: Global value, out: BaseGrid, shape: [100, 3], size: 296)
```

The input to *MShePy.wm.setValues()* is a MShePy dataset. How to create and handle a MShePy dataset is described in the following section.

14.1.5 How to create and handle MShePy datasets

The function *MShePy.dataset()* can be used to create a dataset. The argument of the function is a MIKE SHE parameter ID (*MShePy.paramTypes.**). The characteristics of the dataset, such as the dimension and the unit, depend on the properties of the selected MIKE SHE parameter and will be automatically set to match the model setup. An overview of these parameters and their properties is given in the summary tables at the end of this chapter.

The string representation of a MShePy dataset object contains some of its properties (see example below). The displayed properties include:

- The parameter description (e.g. “precipitation rate”)
- The current unit (e.g. meter/sec)
- The input and output dimension (e.g. Global value)
- The shape, which indicates the number of elements in each dimension
- The size, or total number of elements

MShePy dataset objects can be indexed using square brackets. The number of inputs in the square brackets depends on the dimension of the object. For instance, 2 inputs can be given for the precipitation rate object in the example below (e.g. *data[1,1]*). Slices can be selected by using the colon (e.g. *data[1:3,1]*, *data[:,1]* or *data[1:3:2,1]*). The **.value()* attribute can also be used to display all values inside a MShePy dataset or to change all values in a dataset by specifying the new value as an argument (e.g. *data.value(10.0)*)

```
>>> (startTime,endTime,data)=MShePy.wm.getValues(MShePy.param-
Types.P_RATE)
>>> data
MShePy36.dataset: "precipitation rate", Precipitation Rate (meter/sec),
in: Global value, out: BaseGrid, shape: [100, 3], size: 296
>>> data[1,1]
0.0
>>> data[1:3,1]
[0.0, 0.0]
>>> data.value(10.0)
>>> data[1:3,1]
[10.0, 10.0]
```

The dimension shown in the summary tables at the end of this chapter refers to the standard dimension the MIKE SHE item has, which is also the dimension the output item has. However, the real dimension of input items may change depending on the selected option in the MIKE SHE interface. For example, the precipitation rate is by standard 2 dimensional (BaseGrid in the previous example), but the input item can also be a global value when the option Uniform is selected in the Precipitation Rate panel in the MIKE SHE user interface (see Figure 14.3 below).

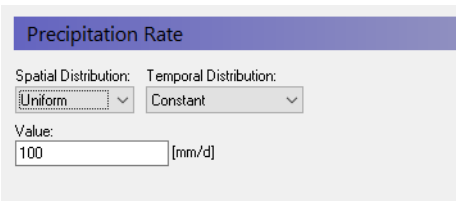


Figure 14.3 Precipitation rate panel

The number of elements in each dimension of a MShePy dataset object can be retrieved by the function **.shape()*. In the example below, data.shape() returns [100,3]. Similarly, **.size()* function returns the total number of elements in a MShePy dataset object (e.g. 196), or the extent in a given dimension when the index of the dimension is given as a parameter, e.g. **.size(1)*.

In order to retrieve the unit of a MShePy dataset object, the function **.unit()* is available. No arguments are required and the function returns the unit of the dataset object. Note that this unit may be different from the unit selected in the MIKE SHE GUI. In the following example, *data.unit()* would return

'*meter/sec*', while the default unit of the precipitation rate in the MIKE SHE GUI is '*mm/d*'.

```
>>> data
MShePy36.dataset: "precipitation rate", Precipitation Rate (meter/sec),
in: Global value, out: BaseGrid, shape: [100, 3], size: 296
>>> data.shape()
[100, 3]
>>> data.size()
296
>>> data.unit()
'meter/sec'
```

In the Python environment, the unit of a MShePy dataset object can be changed by using the function **.convert()*. The argument to the function is the new unit (e.g. *value.convert('mm/year')*). Each MShePy dataset can converted to a unit matching the corresponding item type. The valid units for a given item type can be found in File->Options->Edit Unit Base Groups. The unit conversion in Python will not affect the units of the regular MIKE SHE grid- or detailed time series output items. It is not required to reset the unit before committing the dataset with the setData method. This will be taken care of by the MShePy module.

It is also possible to set a different unit without doing any conversion by calling **.unit()* with an argument. This may be useful when creating a new dataset or for setting the correct unit when making calculations that are supposed to result in a different unit.

```
>>> (startTime, endTime, data) = MShePy.wm.getValues(MShePy.paramTypes.SZ_HEAD)
>>> data[20,22,1]
9.202210426330566
>>> data.unit()                    # get default unit
'meter'
>>> data.convert("inch")           # cannot do this conversion, see error:
Traceback (most recent call last):   File "<stdin>", line 1, in <module>
LookupError: "inch" is a known unit, but it does not match the item type
"Elevation" of the dataset.
>>> data.convert("feet")
MShePy36d.dataset: "head elevation in saturated zone", Elevation
(feet), in: SZ3DGrid, out: SZ3DGrid, shape: [50, 50, 3], size: 5652
>>> data[20,22,1]                  # get the value that now is in feet
30.190979089011048
```

The MShePy library allows arithmetic operations with MShePy datasets: addition, subtraction, multiplication, division, exponentiation, absolute value, negative value, and positive value. The resulting MShePy dataset will have the properties of the first MShePy dataset in the equation (in the example

below, *NEW* is the same dataset type as *HEAD*, while *NEW1* is the same as *ET*).

Arithmetical operations where MShePy datasets are sliced using square brackets(e.g. *ET[:] = HEAD[:,:,0]*), will not replace dataset object, but only the values inside it.

```
>>> (startTime, currentTime, ET) = MShePy.wm.getValues(MShePy.paramTypes.ET_REF_EXC)
>>> (startTime, currentTime, HEAD) = MShePy.wm.getValues(MShePy.paramTypes.SZ_HEAD)

>>> NEW=HEAD+ET
>>> NEW
MShePy36.dataset: "head elevation in saturated zone", Elevation (meter), out: SZ3DGrid, shape: [100, 3, 1], size: 296

>>> NEW1=ET+HEAD
>>> NEW1
MShePy36.dataset: "reference evapotranspiration (OpenMI)", Evapotranspiration Rate (meter/sec), in: Global value, out: BaseGrid, shape: [100, 3], size: 296

>>> ET[:] = HEAD[:,:,0]
>>> ET
MShePy36.dataset: "reference evapotranspiration (OpenMI)", Evapotranspiration Rate (meter/sec), in: Global value, out: BaseGrid, shape: [100, 3], size: 296
```

14.2 Running MIKE SHE simulations from Python scripts

This section describes the steps to execute a MIKE SHE model using a Python script. Functions are included in the MShePy module to initiate and run a MIKE SHE simulation from a Python script.

The MIKE SHE model has to be preprocessed before it can run using the MShePy module. The MIKE SHE model can contain Python plugins.

14.2.1 Initializing a MIKE SHE model

The MShePy module resides in the MIKE installation´s bin/x64 folder. This needs to be in your search path for python modules, e.g. by calling sys.path.append([MIKE installation]/bin/x64). Import the MShePy module by using the command *import MShePy*.

In order to initialize the MIKE SHE model, use the *MShePy.wm.initialize(r"*.she")* function. The function argument is the path to the MIKE SHE model file. While a MIKE SHE model is initialized from a Python script, the model can be opened, but not run in MIKE ZERO.

14.2.2 Running MIKE SHE water movement

Three functions are available in the MShePy module to run a timestep in the MIKE SHE model:

- *MShePy.wm.runAll()* which performs the entire simulation. When using this method you must not initialize or terminate the simulation, differently from all other functions;
- *MShePy.wm.performTimeStep()* which performs one timestep
- *MShePy.wm.runToTime(time)* which performs timesteps until reaching the simulation time specified as an argument (a datetime.datetime object). Be aware that source and sinks assigned from the Python script (including parameters such as a fixed hydraulic head which are converted to source and sinks in the MIKE SHE engine) will remain active for all time steps taken during the execution of this method and be reset only right before the method returns.

All MShePy functions can be called to retrieve and assign variable values in MIKE SHE during the simulation.

14.2.3 Termination and postprocessing

The function *MShePy.wm.terminate()* can be used to terminate the simulation. The user should provide as function input False if an error occurred during the simulation or True if no error occurred.

Postprocessing and the visualization of the results can be done in MIKE ZERO and MIKE VIEW. Results can also be exported using Python and handled in an external software.

14.2.4 Python script example

A Python script describing the steps of initializing, running and terminate a MIKE SHE model is shown below.

```
import datetime                                    #Import daytime module

import MShePy                                      #Import MShePy module

MShePy.wm.initialize(r"C:\Users\PETR.she")         #Initialize the model

currentTime=MShePy.wm.currentTime()                #Find the initial timestep

EndTime=currentTime+datetime.timedelta(days=5)     #Calculate the
end timestep after 5 days

MShePy.wm.runToTime(EndTime)                       #Run the model
until the end timestep
```

```
MShePy.wm.terminate(True)          #Terminate the simulation
```

14.3 Plugins application example

14.3.1 Calculation of the reference evapotranspiration

Several methods are available to calculate the reference crop evapotranspiration, depending on the data available. One of these methods is the Blaney-Criddle equation from Brouwer and Heibloem (1986):

$$ET_0 = p(0.46T_{mean} + 8) \quad (14.1)$$

where ET_0 is the reference evapotranspiration as an average of a period of one month [mm/day], p is the mean daily percentage of annual daytimehours, and T_{mean} is the daily mean air temperature [C].

The following Python plugin calculates the reference evapotranspiration for a 1-year MIKE SHE simulation using the Blaney-Criddle method accordingly to the example provided by Brouwer and Heibloem (1986).

The plugin is composed of three parts:

1. Importing MShePy module
2. Creating global variables (P, T_max, and T_min)
3. Calculating and assigning the evapotranspiration at the beginning of each time step

In order to calculate the evapotranspiration (point 3), the evapotranspiration object is created. Since the temperature and the percentage of daytime hours is given in monthly average, the month of the current time step is used to select the corresponding values for the month. These values are used to calculate the reference evapotranspiration value and assign it to the MIKE SHE simulation. For this example, the evapotranspiration value was chosen to be a global value for the whole model domain.

```
import MShePy #importe the MShePy module

#define global variables
#monthly average of percentage of daytime hours
P = [0.26, 0.26, 0.27, 0.28, 0.29, 0.29, 0.29, 0.28, 0.28, 0.27, 0.26, 0.25]
#monthly average of daily maximum temperature (?)
T_max = [32.1, 35.8, 38, 38.7, 39, 36.6, 32.6, 30.8, 31.8, 34.8, 35, 32]
#monthly average of daily minimum temperature (?)
T_min = [15.5, 18.8, 21.8, 24.5, 26, 25, 22.7, 22, 23, 21.3, 18.7, 16.6]
```

```
def preTimeStep():                          #define the preTimeStep slot
PET = MShePy.dataset(MShePy.paramTypes.ET_REF_EXC)    #Create
PET dataset

time = MShePy.wm.currentTime()          #Find current time step
month = time.month                      #Find current month
T_mean = (T_max[month-1]+T_min[month-1])/2   #Average temperature
ET = P[month-1]*(0.46*T_mean+8)         #Calculate ET (mm/d)
PET.value(ET)                           #Assign value to PET dataset
MShePy.wm.setValues(PET)                #Assign ET value to simulation
```

14.3.2 Reference evapotranspiration adjusted for the altitude

The reference evapotranspiration can be calculated using the modified version of the Penman-Monteith equation presented by Allen et al. (1998):

$$ET_0 = \frac{0.408\Delta(R_n - G) + \gamma\frac{900}{T+273}u_2(e_s - e_a)}{\Delta + \gamma(1 + 0{,}34u_2)} \quad (14.2)$$

where ET_0 is the refence evapotranspiration [mm/day], R_n is the net radiation at the crop surface [MJ/m^2/day], G is the soil heat flux density [MJ/m^2/day], T is the mean daily air temperature at a 2 m height [^{o}C], u_2 is the wind speed at a 2 m height [m/s], e_s is the saturation vapor pressure [kPa], e_a is the actual vapor pressure [kPa], Δ is the slope vapor pressure curve [kPa/^{o}C], and γ is the psychrometric constant [kPa/^{o}C].

This plugin example considers the changes in pressure and, thus, on the psychrometric constant due to the elevation. The psychrometric constant can be expressed as function of the elevation:

$$\gamma = 0{,}00065\left(101{,}3\left(\frac{293 - 0{,}0065Z}{293}\right)^{5{,}26}\right) \quad (14.3)$$

where Z is the ground elevation [m].

The plugin below reproduces example 17 by Allen et al. (1998). It consists of three parts:

- Importing MShePy module
- Creating global variables
- Calculating and assigning the evapotranspiration at the end of the initialization

The calculation of the γ and, thus, of the evapotranspiration depends on the surface elevation. In the plugin example, *Gamma* (γ) is calculated using *Elevation*, which is a MShePy dataset for surface elevation. Thus, *Gamma* is

also a MShePy surface elevation dataset, but it contains value of γ, whose unit is [kPa/^{o}C] and not [m]. This is also true for the *ET* variable, which contains the calculated evapotranspiration. This discrepancy is solved by assigning the value of *ET* to *PET*, which is a MShePy reference evapotranspiration dataset. It would have not been possible to calculate the values of *PET* directly from an equation containing the elevation dataset (Elevation), because then *PET* would have been changed to an elevation dataset.

```
import MShePy #import the MShePy module

#define global variables
Delta = 0.246 #Slope vapour pressure curve (oC/kPa)
Rn = 14.33    #Net radiation at the crop surface (MJ/m2/day)
G = 0.14      #Soil heat flux density (MJ/m2/day)
T = 30.2      #Monthly average mean temperature (oC)
u2 = 2        #Monthly average daily wind speed measured at 2 m (m/s)
Es = 4.42     #Monthly average daily saturation vapour pressure (kPa)
Ea =2.85      #Monthly average daily actual vapour pressure (kPa)

def postEnterSimulator():                    #define the slot

#Create PET dataset
(startTime, currentTime, PET) = MShePy.wm.getValues(MShePy.param-
Types.ET_REF_EXC)
PET.convert('mm/day')          #Convert the unit of the PET dataset

#Get the elevation from the MIKE SHE model
(startTime, currentTime, Elevation) = MShePy.wm.getVal-
ues(MShePy.paramTypes.DEM_Z)

#Calculate the psychrometric constant from the elevation (kPa/oC)
Gamma = 0.00065*101.3*((293-0.0065*Elevation)/293)**5.26

#Calculate reference ET (mm/d)
ET = (0.408*Delta*(Rn-G)+Gamma*(900/(T+273))*u2*(Es-
Ea))/(Delta+Gamma*(1+0.34*u2))
PET[:]=ET[:]                   #Assign calculated value to PET dataset

MShePy.wm.setValues(PET)       #Assign evapotranspiration value
to the simulation
```

14.3.3 Managing irrigation as function of precipitation in the previous months

Periods with dry conditions will typically require more irrigation compared to wet periods. The following plugin describes a simplified method to optimize the irrigation rate based on the precipitation rate of the previous weeks.

At the beginning of each time step, the precipitation rate of the previous month, compared to the current simulation time step, is analyzed. For precipitation rates lower than the annual mean precipitation rate, irrigation is applied

to the model. When the previous month has been a wet one (precipitation rate higher than the annual mean), then no irrigation is applied.

In this plugin, the irrigation rate is applied to the model as part of the precipitation rate. Thus, a new precipitation rate is calculated as the sum of the observed precipitation and calculated irrigation and passed to the MIKE SHE simulation.

```
import MShePy

Precipitation = [1.3, 0.7, 1.3, 1, 1.3, 2, 1.9, 1.9, 2, 1.6, 2, 1.6]   #Monthly average precipitation (mm/d)

P_mean = 1.6                                    #Annual mean precipitation (mm/d)

def preTimeStep():
    PreIrr = MShePy.dataset(MShePy.paramTypes.P_RATE)       #Create Precipitation+irrigation dataset
    time = MShePy.wm.currentTime()      #Current time step
    month = time.month                  #Current month in the simulation

if Precipitation[month-2]<P_mean:       #Condition for irrigation
    Irrigation = P_mean-Precipitation[month-2]      #Calculate the irrigation rate (mm/d)
    PreIrr_rate=Precipitation[month-1]+Irrigation       #Calculate irrigation+recharge (mm/d)
    PreIrr.value(PreIrr_rate)               #Assign irrigation + recharge
else:
    PreIrr.value(PreIrr_rate)               #Assign recharge rate

MShePy.wm.setValues(PreIrr)     #Assign precipitation+irrigation value
```

14.4 Summary tables of exchangeable items between MIKE SHE and Python

Table 14.2 is an overview of MIKE SHE items that can be exchanged as input with Python. The dimension may change for input items depending on the selected option for the item in the MIKE SHE interface. The unit may also change depending on the option selected in File->Options->Edit Unit Base Groups.

Table 14.2 Input items working with Python

Parameter	Abbreviation	Unit	Dimension
Precipitation rate	P_RATE	m/s	2
Air temperature	AIR_TEMP	C	2
Rooting depth	ROOTZONE_WC	m	2
Leaf area index	LAI		2

Table 14.2 Input items working with Python

Parameter	Abbreviation	Unit	Dimension
Crop coefficient	KC		2
reference evapotranspiration	ET_REF_EXC	m/s	2
updated precipitation rate	P_RATE_UPDT	m/s	2
depth of overland water	OL_D	m	2
External sources to Overland (for OpenMI)	OL_SOURCE	m^3/s	2
overland water elevation	OL_H	m	2
Inflow (flux) to OL drain	OLDR_IN_FLX	m/s	2
Inflow (flow) to OL drain	OLDR_IN_FLO	m^3/s	2
water content in unsaturated zone	UZ_WC		3
head elevation in saturated zone	SZ_HEAD	m	3
SZ horizontal conductivity (for DA-OpenMI)	SZ_K_HOR	m/s	3
SZ vertical conductivity (for DA-OpenMI)	SZ_K_VER	m/s	3
Leakage (flux) to SZ	SZ_LEAD_FLX	m/s	2
Leakage (flow) to SZ	SZ_LEAK_FLO	m^3/s	2
Inflow (flux) to SZ drain	SZDR_IN_FLX	m/s	2
Inflow (flow) to SZ drain	SZDR_IN_FLO	m^3/s	2
groundwater extraction	SZ_EXTR	m^3/s	3
External sources to SZ (for OpenMI)	SZ_SOURCE	m^3s	3
pumping from baseflow reservoir	PMP_BASE	m^3/s	

This table is an overview of MIKE SHE items that can be exchanged as output with Python. The dimension may change for input items depending on the selected option for the item in the MIKE SHE interface. The unit may also change depending on the option selected in File->Options->Edit Unit Base Groups.

Table 14.3 Output items working with Python

Parameter	Abbreviation	Unit	Dimension
precipitation rate	P_RATE	m/s	2
air temperature	AIR_TEMP	C	2
average water content in the rootzone	ROOT_DPTH		2
rooting depth	ROOTZONE_WC	m	2
leaf area index	LAI		2
crop coefficient	KC		2
reference evapotranspiration	ET_REF_EXC	m/s	2

Table 14.3 Output items working with Python

Parameter	Abbreviation	Unit	Dimension
precipitation rate of next time step	P_RATE_NEXT	m/s	2
updated precipitation rate	P_RATE_UPDT	m/s	2
ETref x Kc	ET_REF_X_KC	m/s	2
actual evapotranspiration	ET_ACT	m/s	2
actual transpiration	TRANSP_ACT	m/s	2
actual soil evaporation	ESOIL_ACT	m/s	2
actual evaporation from interception	EI_ACT	m/s	2
actual evaporation from ponded water	EOL_ACT	m/s	2
canopy interception storage	SI	mm	2
evapotranspiration from SZ	ESZ	m/s	2
snow evaporation	SNOW_ET	m/s	2
depth of overland water	OL_D	m	2
overland flow in x-direction	OL_FLOW_X	m^3/s	2
overland flow in y-direction	OL_FLOW_Y	m^3/s	2
OL Drain Storage Depth	OLDR_S	m	2
net precipitation rate for AD	P_NET_AD	m/s	2
overland water elevation	OL_H	m	2
infiltration to UZ (negative)	INF	m/s	2
exchange between UZ and SZ (pos.up)	UZ_SZ_EX_POSUP	m/s	2
bypass flow UZ (negative)"	BYP_FLO	m/s	2
total recharge to SZ (pos.down)	RECHARGE_POSDN	m/s	2
total recharge (flow) to SZ (pos.down)	RECH_TOT_FLO	m^3/s	2
groundwater levels used by UZ	GW_LVL	m	2
unsaturated zone flow	UZ_FLO	mm/h	3
water content in unsaturated zone	UZ_WC		3
root water uptake	ROOT_UPTK	m/s	3
macropore water content	MP_WC		3
macropore flow	MP_FLO	mm/h	3
exchange from matrix to macropores	MX_MP_EX	mm/h	3
exchange from macropores to matrix	MP_MP_EX	mm/h	3
depth to phreatic surface (negative)	PHR_DP	m	2
elevation of phreatic surface	PHR_ELV	m	2
head elevation in saturated zone	SZ_HEAD	m	3
SZ horizontal conductivity (for DA-OpenMI)	SZ_K_HOR	m/s	3
SZ vertical conductivity (for DA-OpenMI)	SZ_K_VER	m/s	3
seepage flow SZ -overland	SZ_OL_EX	mm/day	2

Table 14.3 Output items working with Python

Parameter	Abbreviation	Unit	Dimension
seepage flow overland - SZ (negative)	OL_SZ_EX	mm/day	2
3D UZ recharge to SZ (negative)	SZ_RE_NEG	mm/day	3
groundwater flux in x-direction	SZ_X_FLO	m^3/s	3
groundwater flux in y-direction	SZ_Y_FLO	m^3/s	3
groundwater flux in z-direction	SZ_Z_FLO	mm/day	3
groundwater extraction	SZ_EXTR	m^3/s	3
SZ exchange flow with river	SZ_RI_EX	m^3/s	3
SZ drainage flow from point	SZDR_POINT_FLO	m^3/s	3
SZ flow to MOUSE	SZ_MOUSE_FLO	m^3/s	3
pumping from baseflow reservoir	PMP_BASE	m^3/s	
overland to river flow (positive)	OL_RI_POS	m^3/s	
river to overland flow (negative)	RI_OL_NEG	m^3/s	
positive baseflow (SZ to river)	SZ_RI_POS	m^3/s	
negative baseflow (river to SZ)	RI_SZ_NEG	m^3/s	
exchange from river to SZ	RI_SZ_EX	m^3/s	
exchange from river to OL	RI_OL_EX	m^3/s	
river water level	RI_WL	m	2
elevation of ground surface	DEM_Z	m	2



MIKE SHE EDITORS

15 Well editor

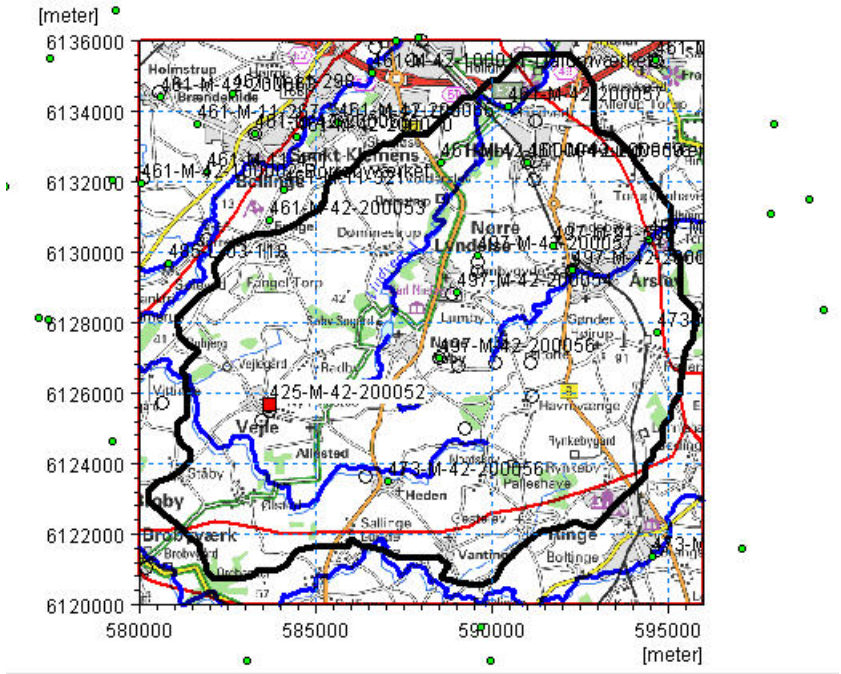
The Well Editor is the MIKE SHE tool for managing pumping wells. The data file for the well editor is independent of the numerical model. That is, the well file often contains all of the wells in the entire model region, including wells outside the model area. The preprocessing takes care of including only the relevant wells in the numerical model.

The dialogue for the Well Editor is divided into

- an Interactive Map *(p. 401)* of well locations,
- a table of Well Locations *(p. 402)*,
- a table of Well Filters *(p. 403)* for the current well, and
- a schematic Layers Display *(p. 404)* showing the relationship between the well screen, the current geologic model and the numerical layers.

The most common way to add data to the Well database is via ASCII import: Importing Data *(p. 404)*.

15.1 Interactive Map



The interactive map displays all of the wells in the well file. Clicking on individual wells will select the corresponding item in the table of well locations. Similarly, selecting an item from the list will change the icon of the well on the map to a red square.

The overlays are automatically carried over from the model Setup Tab. You can't add or modify overlays directly in the Well Editor. This must be down from the Setup Tab

Right clicking on the map, allows you to control the zoom and a number of other functions:

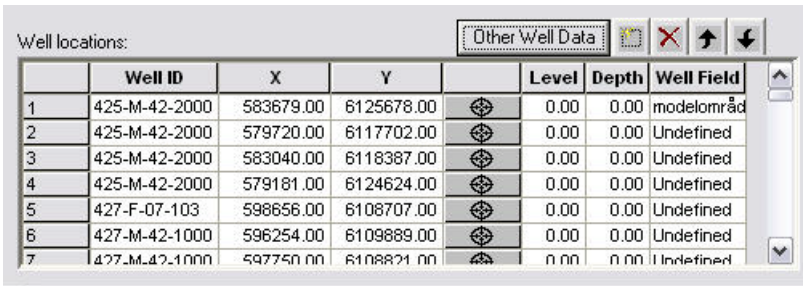
Grid - turns on/off a faint coordinate grid that changes with the zoom factor

Set new area coordinates - allows you to change the displayed area of the map

Text - turns on/off the display of the Well ID labels for the wells

Export Graphic - allows you to save the view to the clipboard, or a .bmp or wmf graphic file for importing into MSWord, for example.

15.2 Well Locations



	Well ID	X	Y		Level	Depth	Well Field
1	425-M-42-2000	583679.00	6125678.00		0.00	0.00	modelområd
2	425-M-42-2000	579720.00	6117702.00		0.00	0.00	Undefined
3	425-M-42-2000	583040.00	6118387.00		0.00	0.00	Undefined
4	425-M-42-2000	579181.00	6124624.00		0.00	0.00	Undefined
5	427-F-07-103	598656.00	6108707.00		0.00	0.00	Undefined
6	427-M-42-1000	596254.00	6109889.00		0.00	0.00	Undefined
7	427-M-42-1000	597750.00	6108821.00		0.00	0.00	Undefined

Well_ID - This is the user specified name of the well. The Well_ID cannot contain any spaces.

X, Y - These are the X and Y map coordinates of the well.

EUM Data Units: Item geometry 2-dimensional

Level - The Level defines the maximum elevation shown on the profile view of the geologic layers, calculation layers, and screened intervals for the well. The topography is shown if the Level is less than the topography.

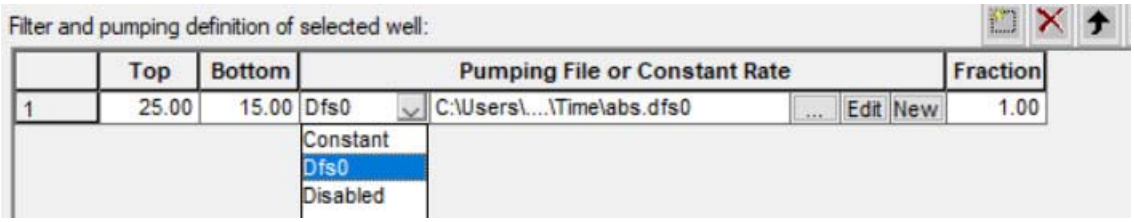
EUM Data Units: Elevation

Depth - The Depth is defined from the Level. It defines the maximum depth shown on the graphical view displaying the profile view of the geologic layers, calculation layers, and screened intervals for the well. The bottom of the geologic layers is shown if the Level minus the Depth is higher than the bottom of the geologic layers.

EUM Data Units: Depth below ground

Well Field - The Well Field item is used for filtering the displayed boreholes. The Mask item in the top menu bar uses the Well Field for it selection criteria. The Well Field item is also used to defined registration zones in the Random Walk Particle Tracking (PT) module. For more information, see Particle Tracking (PT) *(p. 741)*, and Particle Tracking-Reference *(p. 737)*.

15.3 Well Filters



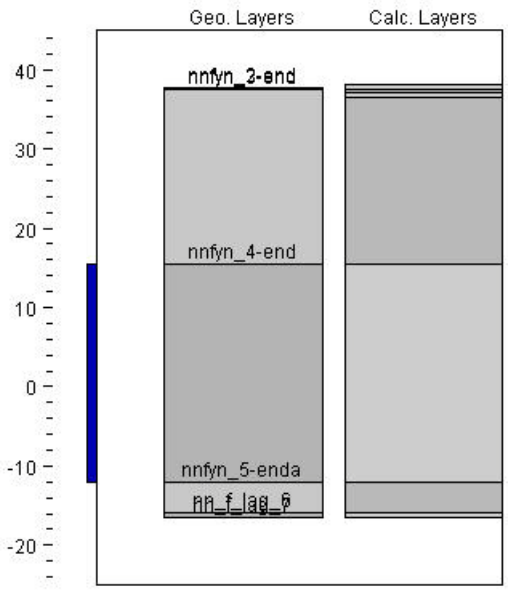
Top - This is the elevation of the top of the screen or open hole interval for the well (in the same units (ft, m, etc.) as specified in the EUM Database for item geometry 2-dimensional).

Bottom - The elevation of the bottom of the screen or open hole interval for the well (in same units (ft, m, etc.) as specified in the EUM Database for item geometry 2-dimensional).

Pumping File or Constant Rate - You have three choices here. Either a constant value for the pumping rate, name of the .dfs0 file with groundwater pumping data for the well (When using the Browse button, [...], to select the file, you will be given the option of specifying the Item number in the .dfs0 file.), or to disable pumping at this well.

Fraction - This is a multiplier for the groundwater pumping rate specified in the DFS0 File.

15.4 Layers Display



The Layer display section displays the location of all well screens assigned to the well. The Geo Layers column displays the geologic layers assigned in the Setup Tab for the well, and the Calc Layers is the numerical layers for the column of model cells in which the well is located.

Both the Geo Layers and the Calc Layers items require that the model has been successfully pre-processed. If you have not pre-processed the model yet, or if during the preprocessing an error occurred, then a warning message dialogue may appear saying that the model must be pre-processed first. If this happens, the Well Editor will function normally, but the Geo and Calc layers may not be shown.

15.5 Importing Data

In the top menu bar there is an Import menu, that allows you to import the following well data.

Zeus data (Adm and Lit) - The Zeus data is a specialized data format from the Geological Survey of Denmark and Greenland.

Tab delimited ASCII files - This is the most common way of imported well data.

15.5.1 Importing TAB delimited text file

The most common file format to import is a TAB delimited ASCII file, typically generated from Excel or a database program.

Several screened intervals can be specified for each well. Each screened interval should be imported on a different line. The line should include well properties, as well as screened properties.

Below is the format that each line in the ASCII file must follow:

```
Well_ID>X>Y>Level>Depth>Well_Field>Top>Bottom>Frac-
tion>dfs0_File>dfs0_item
```

A simple example with three groundwater wells is given below.

```
CW1 7780.00 20331.00 0.00 0.00 CW   0. -60. 1.
.\Time\ClassPumpage.dfs0 1

CW2 8000.00 19000.00 0.00 0.00 CW -10. -50. 0.5
.\Time\ClassPumpage.dfs0 3

CW3 7600.00 21300.00 0.00 0.00 CW  10. -60. 1.
.\Time\ClassPumpage.dfs0 2
```

The dfs0-item-number field is used to import Constant values or to disable the well as follows:

- if the `dfs0_item` value is positive, then the value is read as the dfs0 Item Number
- if the `dfs0_item` value is 0, then the `dfs0_File` value is read as the constant pumping rate
- if the `dfs0_item` is -1, then the pumping well is disabled.

Note - The import function does not allow spaces in the text fields. Also, you cannot import filter definitions as “disabled” or “fixed pumping rate”.

16 Vegetation Properties Editor

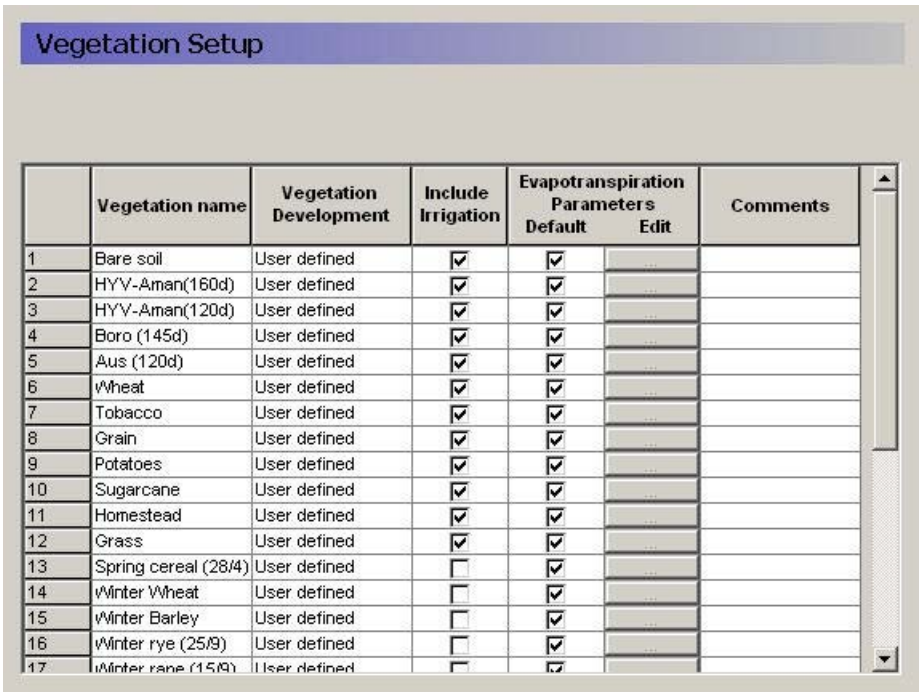
The vegetation editor is used to specify vegetation data for the evapotranspiration and irrigation management modules. The vegetation database contains the time varying vegetation characteristics for each type of vegetation that is specified in the model domain.

The vegetation database is optional and can be used only when the Evapotranspiration (ET) and Unsaturated Zone components are included in the model.

16.1 Vegetation Database Items

The vegetation database is organized around a data tree similar to the setup editor. To create a vegetation type in the database and populate with the corresponding data, simply add a vegetation item in the main dialogue and then fill out the tables in the dialogues that appear in the data tree under the new vegetation item.

16.1.1 Specifying Vegetation Properties in a Database



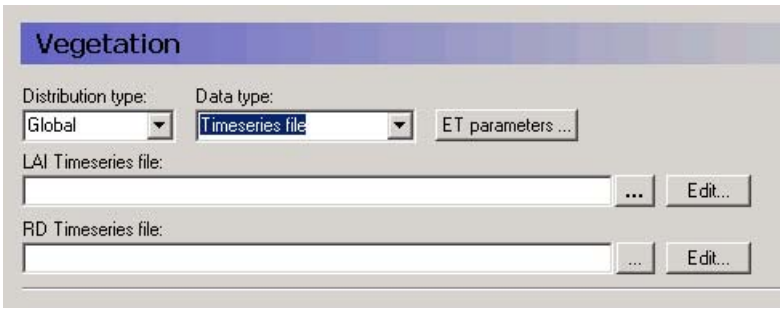
The vegetation database is populated with data in a number of steps. Firstly, all the vegetation types to be included in the database are entered in a table in the Vegetation Setup Menu. The data needed are:

- Vegetation name;

- Vegetation Development;
- Include Irrigation;
- Evaporation parameters.

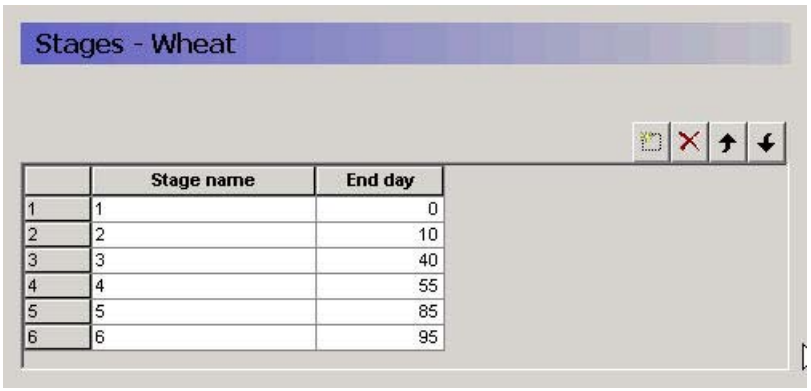
If the Irrigation module is used in the model you, may chose to use the irrigation demand values from the vegetation file. In this case, you should select the 'Include Irrigation' option and then specify the irrigation demand parameters by vegetation type.

You can also specify specific Evaporation Parameters for each vegetation type. If the 'Default' value is checked, then the global values defined in the main vegetation dialogue (shown below) will be used:



If the default value is unchecked, then you can specify the ET parameters by vegetation type.

16.1.2 Vegetation stages



The first thing to do, is to specify the standard vegetation stages for each crop or land use type in the database. These temporal variations in vegetation characteristics can normally be described by a number of characteristic stages of specific length. The changes are defined as a set of linear changes between two consecutive crop stages. Three parameters describe the stage, the leaf area index (LAI), the root depth (RD), and the crop coefficient K_c.

IMPORTANT NOTE: The values of LAI, RD and K_c are defined as Instantaneous parameters. Therefore, it is critical that you start with a Stage at 0 days. Otherwise, your initial value for LAI, RD or K_c will be zero.

The day number indicates the cumulative days from crop establishment (e.g. sowing) to the end of the specific crop stage.

If in the Vegetation *(p. 249)* dialogue, subsequent start dates overlap with the development cycle, a warning will be issued in the log file that says the crop development was not over yet before the new crop was started. MIKE SHE will then start a new crop cycle at the new start date.

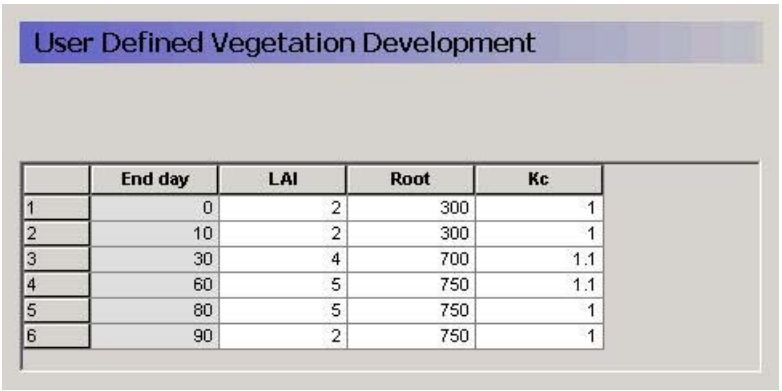
16.1.3 Evapotranspiration Parameters

The parameters used in the evapotranspiration calculations can be divided into three groups, which regulate interception, soil evaporation and plant transpiration, respectively.

The d

Note: The C1, C2, C3 ,and AROOT parameters are only used in the Richards Equation and Gravity Flow methods, and not in the 2-Layer UZ method.

16.1.4 Vegetation Development Table



	End day	LAI	Root	Kc
1	0	2	300	1
2	10	2	300	1
3	30	4	700	1.1
4	60	5	750	1.1
5	80	5	750	1
6	90	2	750	1

For each crop stage, three vegetation parameters need to be specified:

- **LAI** - The Leaf Area Index, which is the (Area of leaves)/(Area of the ground), can vary between 0 and 7 depending of the vegetation type;
- **Root** - The Rooting Depth of the crop. It will normally vary over the season. Consideration about the soil type should be taken because some crops may develop different root distribution depending on the soil characteristics;
- **Kc** - The crop coefficient.

The leaf area index and the root depth should be specified at the end of each crop stage. The development of LAI and root depth between the specified values are then interpolated linearly by the model. In addition to these parameters, it is often necessary to supply the crop coefficient (**Kc**), which is used to adjust the reference evapotranspiration relative to the actual evapotranspiration of the specific crop.

By the FAO definition, the reference evapotranspiration represents the potential evapotranspiration for a 8-15 cm high reference grass plane with ample water supply. Most farm crops may differ from this in two ways:

- In the early crop stages, where LAI of the farm crop is lower than the LAI of the reference grass crop, the evapotranspiration of the farm crop is less then the calculated reference evapotranspiration. This is accounted for in the Kristensen & Jensen ET calculation, since a crop LAI is used as input. **Therefore, for most field crops it is therefore not necessary to specify Kc values below 1 in the early crop stages.**
- In the crop mid-season the opposite situation may occur where crop potential evapotranspiration is larger than the calculated reference evapotranspiration of the reference grass crop. This is not handled in the ET calculations, and Kc values above 1 may therefore be relevant for some crops in the mid-season during the period where crop leaf area index is at its maximum.

A Kc value of 1 means that the maximum evapotranspiration rate will equal the reference evapotranspiration rate.

If pan evaporation data are used in place of reference evapotranspiration data in the model input, it often necessary to apply site specific pan coefficients to convert the pan evaporation to reference evapotranspiration. Pan coefficients are normally in the range 0.5 – 0.85.

16.1.5 Irrigation Parameters

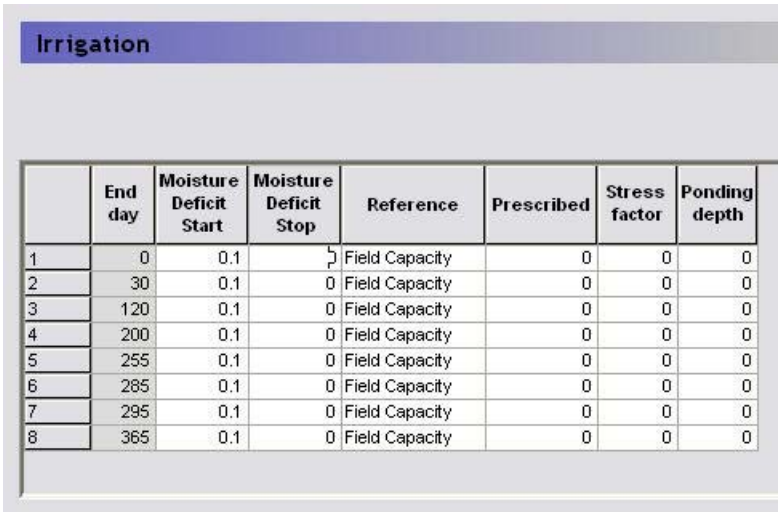
In the irrigation module the amount of irrigation applied can be driven by the amount of water demanded by the crop. That is, in drier periods more irrigation water is required, so more irrigation is applied.

In the Irrigation demand dialogue, if you specify that the Vegetation Property file should be used for the demand calculations, then the demand values will be read from the vegetation file specified in the Vegetation *(p. 249)* dialogue.

Further, in the irrigation module, you specify the type of demand calculation:

- User specified
- Maximum allowed deficit
- Crop stress factor
- Ponding depth

In the Irrigation dialogue of the Vegetation properties file,



Irrigation

	End day	Moisture Deficit Start	Moisture Deficit Stop	Reference	Prescribed	Stress factor	Ponding depth
1	0	0.1	0	Field Capacity	0	0	0
2	30	0.1	0	Field Capacity	0	0	0
3	120	0.1	0	Field Capacity	0	0	0
4	200	0.1	0	Field Capacity	0	0	0
5	255	0.1	0	Field Capacity	0	0	0
6	285	0.1	0	Field Capacity	0	0	0
7	295	0.1	0	Field Capacity	0	0	0
8	365	0.1	0	Field Capacity	0	0	0

you must specify a value for each of these demand types. Although, those that will not by used, may be left at the default values.

Maximum allowed deficit - If irrigation is handled automatically based on the actual moisture content in the soil, the soil moisture deficits are the deficits at which irrigation is going to start and stop. The soil moisture deficit is defined relative to the plant available water content in the root zone, which is the difference between a reference moisture content and the moisture content at wilting point. If, for example, the reference moisture content is the moisture content at field capacity and irrigation should start when 60 % of the available water in the root zone is used and cease when field capacity is reached, the value in the start column should be 0.6, the value in the stop column should be 0 and the reference input should be “field capacity”.

User specified - Alternatively, the irrigation amount applied in each crop stage can simply be prescribed.

Crop stress factor - The crop stress factor is the minimum allowed fraction of the reference ET that the actual ET is allowed to drop to before irrigation starts . That is, the minimum allowed (Actual ET)/(Reference ET) relationship This should be a value between 1 and 0.

Ponding depth - When using this option, the demand will be equal to the difference between the actual ponding depth and specified ponding depth. The option is typically used for modelling irrigation of paddy rice.

Thus:

- **Moisture Deficit Start** - the maximum allowable moisture deficit below the specified reference level. Irrigation will start at this level.

- **Moisture Deficit Stop** - This is the minimum allowable moisture deficit below the specified reference level. If irrigation takes place it will stop at this level;
- **Reference** - The **reference** moisture content. It can be chosen as either saturation or field capacity.
- **Prescribed** - This is the value used for the irrigation demand when 'User specified' is chosen in the Irrigation Demand *(p. 265)* dialogue.
- **Stress Factor** - This is the minimum allowed (Actual ET)/(Reference ET) relationship This should be a value between 1 and 0.
- **Ponding Depth** - Irrigation will stop when the ponding depth reaches this value.

16.2 Example database

There is an example database (MIKE_SHE_vege.ETV) in the installed examples.

As the growing season of a given crop differs significantly depending on climatic region, you may have to adjust the vegetation development to local conditions.

17 UZ Soil Properties Editor

To solve Richards equation two important hydraulic functions are required for all soil types which characterise the individual soil profiles within the model area:

- the Soil Moisture Retention Curve and
- the Hydraulic Conductivity Function.

This information, along with the following parameters, is stored in the soil property database:

- soil moisture at saturation (θs) [-]
- soil moisture at effective saturation (θeff) [-]
- capillary pressure at field capacity (pF_{fc})
- capillary pressure at wilting point (pF_w)
- residual soil moisture content (θr) [-]
- saturated hydraulic conductivity (K_s)

The soil moisture at effective saturation θeff is the maximum achievable soil moisture content.

17.1 Definitions of soil properties

pF scale

pF is a log scale for representing soil matric potential. Thus,

$$pF = \log10(-100\psi) \quad (17.1)$$

where ψ is the matric potential in metres of water. Notice that ψ is always negative under unsaturated conditions.

For example, if your measured wilting point is -15 bar (-152.96 m water), then the pF value is 4.2.

Saturated moisture content - θ_s

This is the maximum water content of the soil, which is equal to the porosity. A typical saturated moisture content is about 0.25 to 0.3.

Effective saturation

This is the maximum water content achieved under field conditions, which is usually slightly less than the saturated moisture content because of entraped air.

Field capacity - θ_{fc}

This is the water content of a free draining soil. In other words, it is the water content in the soil 2-3days after a heavy soaking rain. A typical field capacity is about 0.1 to 0.2.

pFfc - This is the suction pression of a soil when it is at field capacity. The pF_{fc} (field capacity) is used as the initial condition in the unsaturated flow module. A typical value is about 2.0.

Wilting point - θ_{wp}

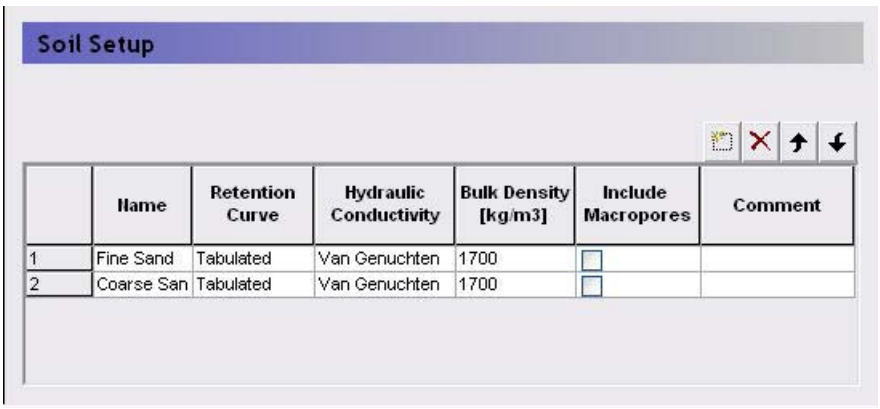
This is the water content when plants can no longer remove water from the soils and start to wilt. It is the minimum water content of the soil in the absence of evaporation. A typical value is about 0.01 to 0.05

pFw - This is the suction pression of the soil when it is at the wilting point. The pF_w (wilting) is typically about 4.2.

Residual moisture content - θ_r

This is the minimum water content at very high suction pressures. A typical residual moisture content is on the order of 10^{-3}.

17.2 Soils Database



In the main soils database dialogue, you can add any number of soils. MIKE SHE only reads information on the soils being used, so there is no penalty for having large databases that include soils used in different projects.

Name - The Name field is used to identify the soil when loading the soil information in MIKE SHE's UZ Soil Profile Definitions *(p. 293)* dialogue.

Retention Curve and Hydraulic Conductivity - The Soil Moisture Retention Curve *(p. 415)* and the Hydraulic Conductivity Function *(p. 418)* are described in the next sections.

Bulk Density - The bulk density is used for calculating sorption during water quality simulations.

Include Macropores - If Macropore Flow *(p. 291)* has been specified, then the macropore data will be required. If this item is checked, then an additional property page will be added to the data tree - Macropore Parameters *(p. 421)*.

Comment - This field can be used to provide meta data for the soil, such as the literature source for the values, or the project that it was last used in.

Curves for the calculated Retention and Conductivity are shown for each defined soil.

17.3 Soil Moisture Retention Curve

The relationship between the water content, θ, and the matric potential, ψ, is known as the soil moisture retention curve, which is basically a function of the texture and structure of the soil. The amount and type of organic material may also have an influence on the relationship. Characteristically, the pressure head decreases rapidly as the moisture content decreases. Hysteresis is also common, that is the relationship between θ and ψ is not unique, but depends on whether the moisture content is increasing or decreasing. MIKE SHE allows for any shape of the soil moisture retention curve, but does not take hysteresis into account (i.e. a unique relation between θ and ψ is assumed).

Typically, the soil moisture curve is measured in a laboratory or assumed based on typical values for similar soils. If laboratory data is available, the measured θ-ψ values can be input directly into MIKE SHE as tabular data. Intermediate values are then calculated by MIKE SHE, using a cubic spline method, and stored internally in the code. Alternatively, the measured values can be fitted to commonly used functional relationships. The appropriate function parameters can be input directly or more refined tabular data may be generated externally to MIKE SHE (e.g. in MS Excel) and input as tabular data.

Several parametric forms of the soil moisture retention curve have been developed over the years. The MIKE SHE interface allows the user to specify two of the most common parametric forms.

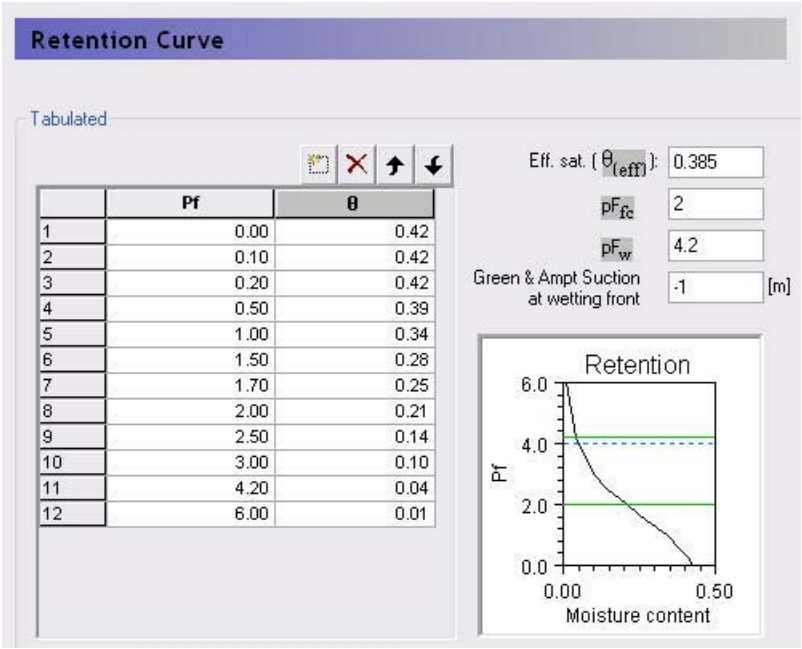
Common data

The upper part of the dialogue contains the common parameters for describing the soil-moisture relationship, which are described in the Definitions of soil properties *(p. 413)* section. The number of parameters in this section changes depending on the retention curve definition that was selected.

Green and Ampt suction - This is the empirical suction [metres of water] ahead of the wetting front in the Green and Ampt empirical relationship. It is

essentially a calibration parameter, but should be about the same as the soil suction value just below field capacity. In other words, when the soil has been freely drained, plus had some water removed by evapotranspiration.

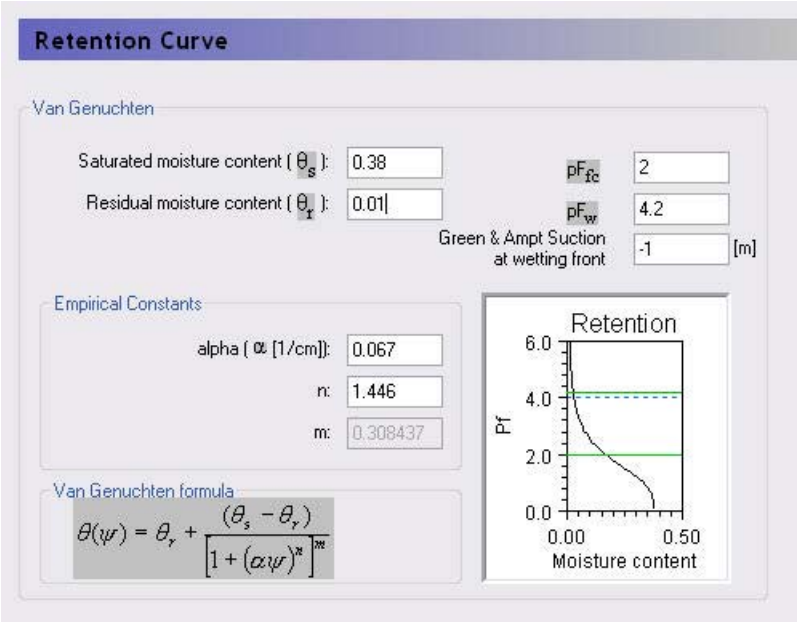
Tabulated



The data points describing the pressure conductivity curve can be given as a table of pF versus θ (moisture content) values. The table should be specified starting with the lowest value of pF (wettest condition) and given in increasing order of pF. (see pF scale *(p. 413)*)

To get a smooth retention curve MIKE SHE adopts a cubic spline curve fitting procedure. As a minimum, you should specify the conductivity at saturation, the field capacity and the wilting point. However, we strongly advise against this because the cubic spline function is unlikely to be able to fit an appropriate function to only 3 points.

Van Genuchten Function



The Van Genuchten formula is the most widely used soil moisture-pressure relationship. The Van Genuchten formula was first published by M. Th. van Genuchten in 1980, in his classic paper. "A closed-form equation for predicting the hydraulic conductivity of unsaturated soils". Soil Science Society of America Journal 44 (5): 892–898. The Van Genuchten formula is

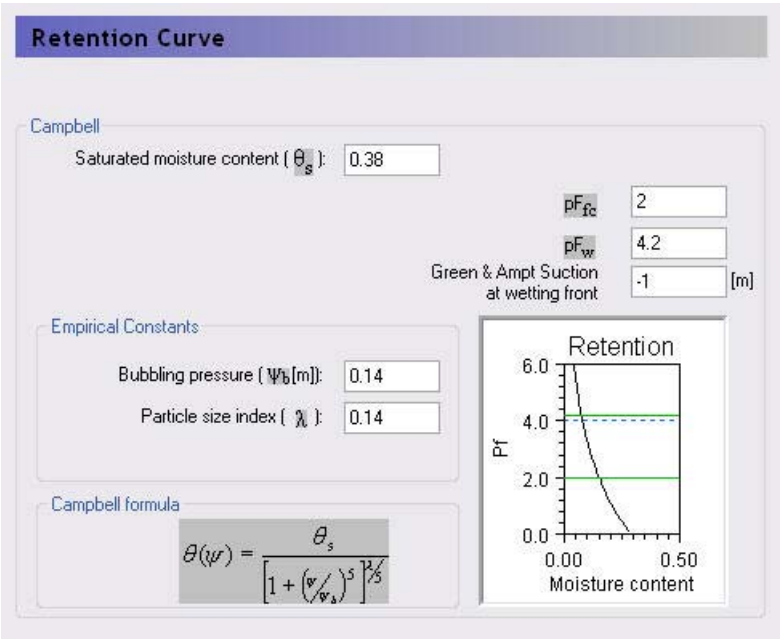
$$\theta(\psi) = \theta_r + \frac{(\theta_s - \theta_r)}{[1 + (\alpha \cdot \psi)^n]^m} \qquad (17.2)$$

where the exponent *m* is related to the empirical constant *n* by $m = 1 - \frac{1}{n}$. and α is the other Van Genuchten empircial constant.

Both *n* and *a* can be optimized using AUTOCAL program.

Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

Campbell Function



Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

17.4 Hydraulic Conductivity Function

The Governing Equation for the unsaturated flow requires information about two hydraulic functions: The hydraulic conductivity function, K(θ) and the soil moisture retention curve ψ(θ) 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(θ)* function is rather difficult and not very reliable. Alternatively procedures have been suggested to derive the function from more easily measurable characterizing properties of the soil or simply to rely on empirical relationships.

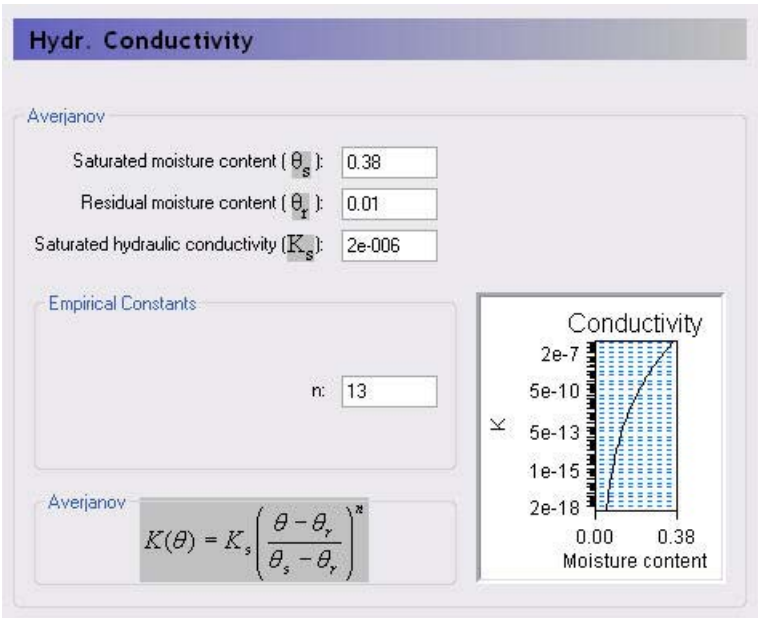
Reviews of various methods for predicting the conductivity function can be found in the literature.

Tabulated

The data points describing the pressure conductivity curve can be given as a table of pF versus K values. The table should be specified starting with the lowest value of pF (wettest condition) and given in increasing order of pF.

To get a smooth hydraulic conductivity curve, MIKE SHE adopts a cubic spline curve fitting procedure. As a minimum, you should specify the conductivity at saturation, the field capacity and the wilting point. However, we strongly advise against this because the cubic spline function is unlikely to be able to fit an appropriate function to only 3 points.

Averjanov Function



In the Averjanov method, the hydraulic conductivity, *K,* is described as a function of the effective saturation, *Se*:

$$K_{(E)} = K_{sat}(S_e)^n \tag{17.3}$$

where

$$S_e = (\theta - \theta_r)/(\theta_s - \theta_r) \tag{17.4}$$

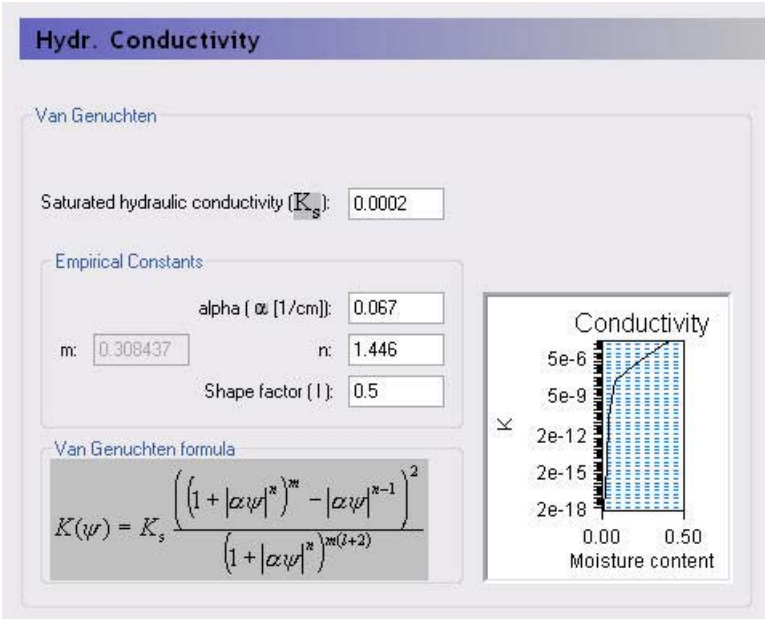
in which θs, θ and θr are saturated, actual and residual moisture contents, respectively.

The full knowledge about the hydraulic conductivity function is seldom available, and the parameter n has to be estimated by calibration.

As a guideline the exponent n is usually small for sandy soils (2-5) and large for clayey soils (10-20). It is important to note that the value of the exponent n will influence the percolation rate in the soil and thereby influence the actual evaporation rate.

Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

Van Genuchten Function

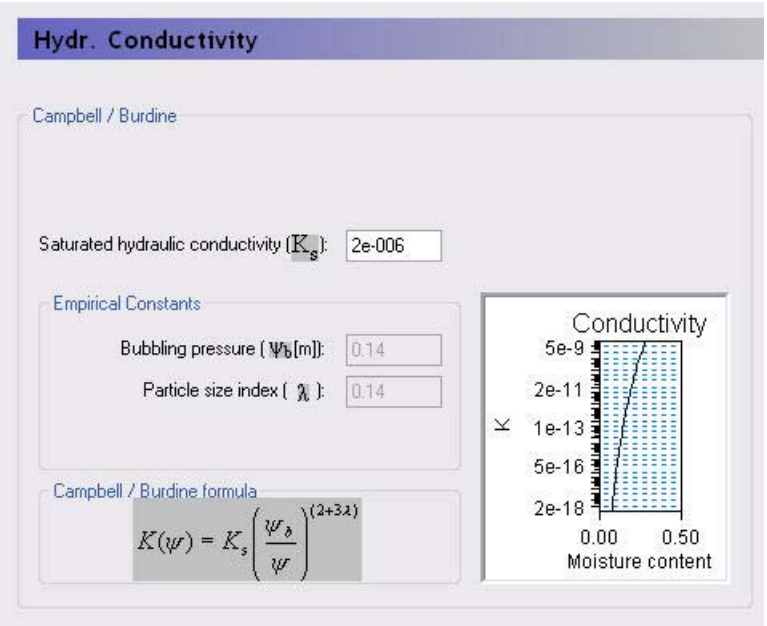


In addition to the tabulated values, parametric functions are available using the Van Genuchten and the Campbell/Burdine formulations. It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input.

The van Genuchten Shape Factor, *l*, is soil texture dependent, with a minimum allowed value of -4.

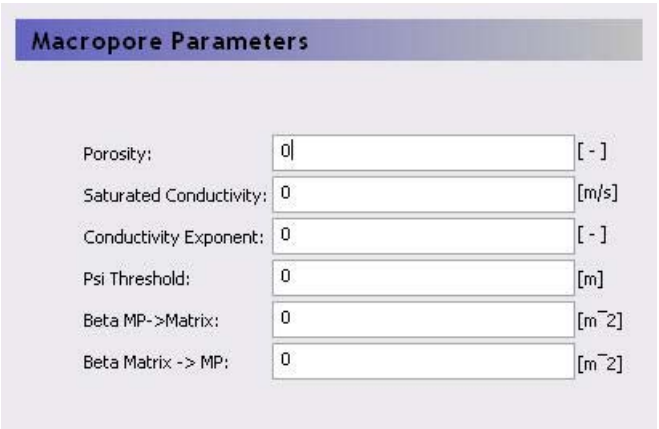
Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

Campbell/Burdine Function



Note It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input using about 100 points.

17.5 Macropore Parameters



The macropore parameters are used for the full macropore flow in the unsaturated zone.

Porosity - is the macropore porosity.

Saturated conductivity - is the saturated conductivity of the macropores.

Conductivity exponent - is the exponent, n, in Equation (31.35), which represents the simple power function assumed for the conductivity-water content relationship.

Psi threshold - is the capillary pressure in the matrix above which flow in the macropores can occur. Pressures below this value would result in water being absorbed into the matrix via capillarity.

Beta - is the mass transfer rate between the macropores and the matrix.

18 Water Balance Editor

The water balance utility is a flexible, post-processing tool for generating water balance data for MIKE SHE simulations. Output from the water balance utility can include area normalized flows (storage depths), storage changes, and model errors resulting from convergence problems. Water balance data can be generated at a variety of spatial and temporal scales and in a number of different formats.

To extract the water balance data, you must specify which simulation you are going, then specify the area of your model that you want the water balance for, and, finally, extract the MIKE SHE water balance data from the results files.

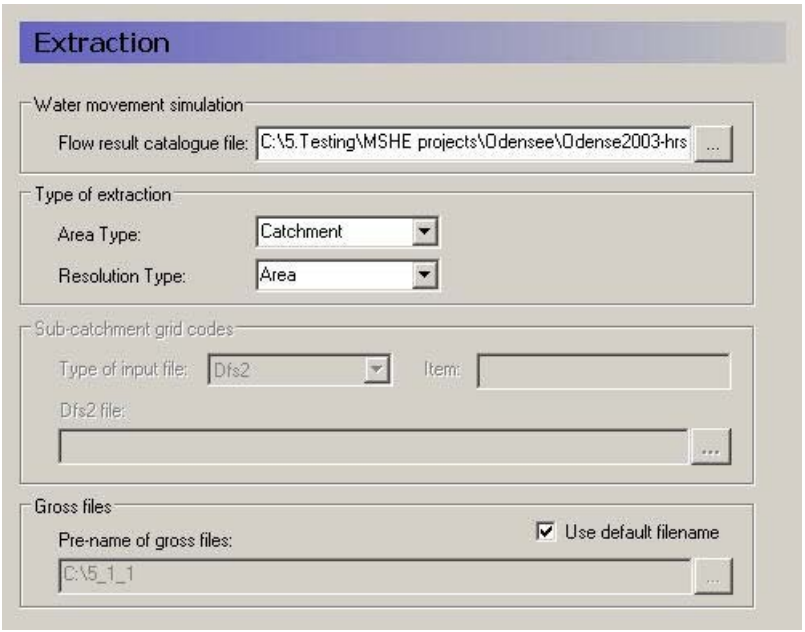
Once you have created a new water balance document, the following three tabs will be displayed

- Extraction *(p. 424)*
- Postprocessings *(p. 425)*
- Postprocessing Detail *(p. 426)*
- Results *(p. 428)*

Related items

- Using the Water Balance Tool *(p. 99)*

18.1 Extraction



Flow result catalogue file

A MIKE SHE simulation generates various output files depending on the options and engines selected for the MIKE SHE simulation. The .sheres file is a catalogue of all the various output files generated by the current MIKE SHE run. When you select the .sheres file, you are not specifying the particular output, but actually just a set of pointers to all the output files.

The extraction process reads all of the output files and makes itself ready to produce specific water balances. In the extraction dialogue, you specify the .sheres file for the simulation that you wish to calculate the water balance for. The .sheres file is located in the same directory as your results.

Note Although, this is an ASCII file, you should be careful not to make any changes in the file, or you may have to re-run your simulation.

Type of Extraction

You can choose to calculate the water balance on the entire model domain or in just a part of the domain. By default the calculation is for the entire domain, or catchment. If you choose the subcatchment area type, they you will be able to use a dfs2 integer grid code file to define the areas that you want individual water balances for.

If you use an area resolution, then the water balance will be a summary water balance for either the entire catchment or the sub-areas that you define.

If you use a single-cell resolution, you will be able to generate dfs2 maps of the water balance.

Sub-catchment grid codes

The subcatchment integer grid code file is only used if you have selected the sub-catchment water balance type. You can specify a delete value to exclude areas from the water balance. The grid spacing and dimensions in this dfs2 file must match exactly the model grid.

You can also specify a polygon shape file to define the sub-catchment areas. The shape file may contain multiple polygon, with the same or different codes. Further, the shape file length units do not have to be the same as the model length units (e.g. feet vs. meters).

Gross files

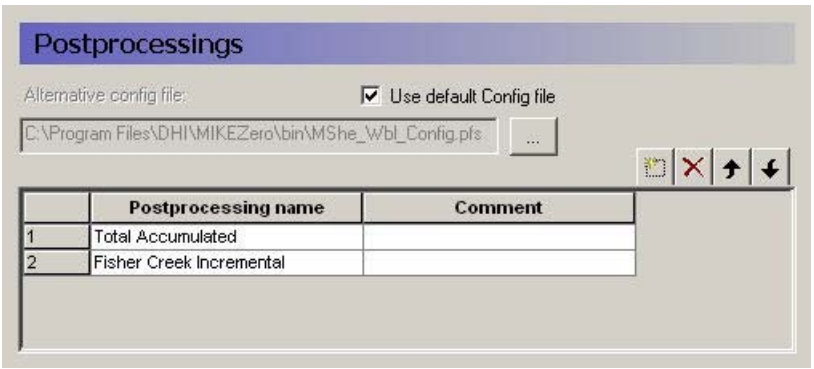
The pre-processor extracts the water balance data from the standard MIKE SHE output files and saves the data in a set of “gross” files. The file names of the gross files is built up from the project name and prefix specified here. The default value is normally fine.

Related items

- Using the Water Balance Tool *(p. 99)*

18.2 Postprocessings

After you have extracted the water balance data from the MIKE SHE results files, then you can switch to the post-processing tab. Here you can create any number of individual water balances by simply clicking on the Add item icon and specifying the water balance parameters in the parameter dialogue.



A single Postprocessing item is created by default when the water balance file is created. The default Postprocessing name can be change to a more appropriate name. Postprocessing items that are no longer needed can be deleted using the Delete button.

Use default Config file

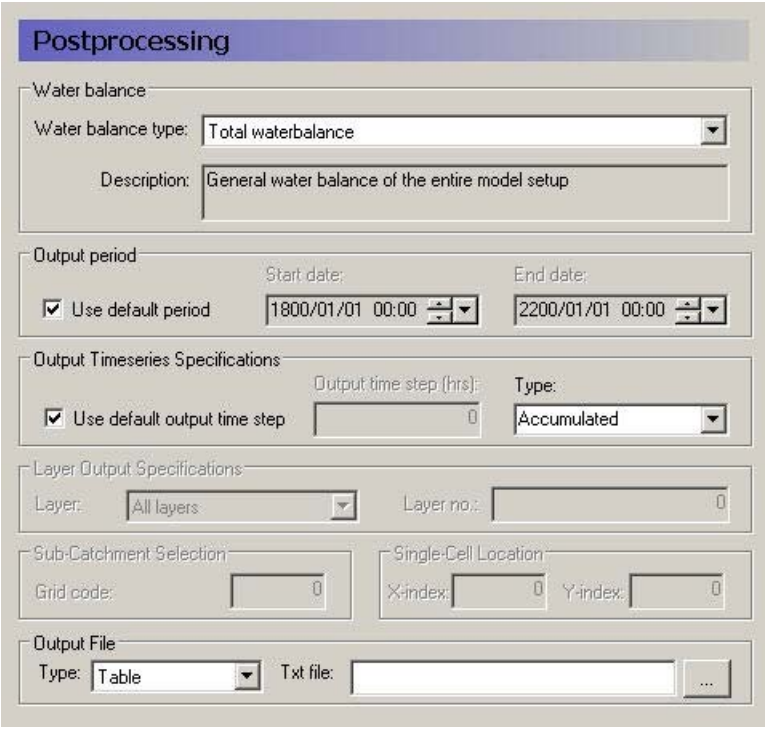
Unchecking the Use default Config file checkbox, allows you to specify the location of a custom water balance Config file. Development of custom water balance configuration files is described in detail in Making Custom Water Balances *(p. 136)*.

Related items

- Using the Water Balance Tool *(p. 99)*

18.2.1 Postprocessing Detail

For each item in the Postprocessing list above, a new item will be added to the data tree. If you expand the data tree, each will have the following dialogue.



Water Balance

Multiple postprocessings can be run on each water balance extraction. More detail on the types of available water balances data are discussed in the Standard Water Balance Types *(p. 131)* section. In brief, the available types include

- The total water balance of the entire model catchment or sub-catchments in an ASCII table, a dfs0 file, a dfs2 map file, or a graphical chart (also by layer),

- Model errors for each hydrologic component (overland, unsaturated zone, etc.) in an ASCII table, a dfs0 file, or a dfs2 map file (also by layer),
- The snow melt and canopy/interception water balance in an ASCII table, or a dfs0 file,
- An abbreviated or detailed water balance for overland or unsaturated flow in an ASCII table, or a dfs0 file, and
- An abbreviated or detailed water balance by layer for saturated flow in an ASCII table, or a dfs0 file.

Output Period

An output period different from the total simulation period can be specified by unchecking **Use default period** and setting the **Start date** and **End date** to the period of interest

Output time series Specification

Incremental or **Accumulated** water balances can be calculated. An incremental water balance is calculated (summed) for each output time step in the Output period. An accumulated water balance each output time step is accumulated over the Output period

Layer Output Specifications

If you are using water balance types that calculate data on a layer basis, you can specify whether you want **All layers** or just the **Specified layer**, where you also must specify a layer number.

Sub-catchment Selection

If you extracted sub-catchment data from the WM results, then you must specify a subcatchment number or the name of the polygon for which you want the water balance for. The combobox contains a list of valid ID numbers or polygon names.

Single Cell Location

If you extracted the WM data by cell, then if you are not creating a map output, then you have to specify a cell location for which you want a water balance.

Output File

If you are creating a table or time series water balance, then you can write the output to either a dfs0 file or to an ASCII file for import to MSExcel, or other post-processing tool. If you are creating a map, then the output will be to a dfs2 file, with the same grid dimensions and spacing as the model grid. If you are creating a chart, then the output will be written to an ASCII file, with a special format for creating the graphic.

Related items

- Using the Water Balance Tool *(p. 99)*

18.3 Results

The data tree for the results tab lists all of the calculated water balances. The dialogue for each item, includes the file name and an Open button, that will open an editor for the file. For ASCII output, this will be your default ASCII editor - usually Notepad. For dfs0 and dfs2 files, the DHI Time Series Editor or Grid Editor will be opened. For the chart output, the graphic will be displayed by the program WblChart.

Related items

- Using the Water Balance Tool *(p. 99)*



THE MIKE SHE REFERENCE GUIDE

19 Reference Guide Overview

The Reference Guide includes detailed descriptions of the tools and dialogues that you are likely to encounter as you develop a MIKE SHE model. If you click F1 in any MIKE SHE dialogue, you will land in one of the sections of this manual. Likewise, if you click F1 in any other MIKE Zero dialogue, you will land in an appropriate section of the on-line help.

The first part of the document follows the general organization of the MIKE SHE user interface. These sections are the places you land, when you press the F1 key in MIKE SHE and include the

- Setup Data Tab *(p. 179)*
- Preprocessed Data Tab *(p. 363)*, and
- Results Tab *(p. 369)*

This is followed by detailed descriptions of the various MIKE SHE specific editors, including the

- Well editor *(p. 401)*
- Vegetation Properties Editor *(p. 407)*
- UZ Soil Properties Editor *(p. 413)*, and
- Water Balance Editor *(p. 423)*

In turn, this is followed by the MIKE SHE Reference Guide. This include both Technical Reference sections and detailed User Guides sections. For the Water Movement module, this includes a

- Evapotranspiration - Technical Reference *(p. 437)*
- Working with Evapotranspiration - User Guide *(p. 457)*
- Snow Melt - Technical Reference *(p. 461)*
- Working with Freezing and Melting - User Guide *(p. 467)*
- Overland Flow - Technical Reference *(p. 515)*,
- Working with Overland Flow and Ponding- User Guide *(p. 541)*
- Channel Flow - Technical Reference *(p. 471)*
- Working with Rivers and Streams - User Guide *(p. 473)*
- Unsaturated Zone - Technical Reference *(p. 575)*
- Working with Unsaturated Flow - User Guide *(p. 607)*
- Saturated Flow - Technical Reference *(p. 623)*
- Working with Groundwater - User Guide *(p. 651)*, and
- Coupling of MIKE SHE and a MIKE 1D Sewer Model *(p. 653)*

For the Water Quality module, this includes

- the Advection Dispersion - Reference *(p. 673)*,
- the Reactive Transport - Reference *(p. 699)*, and
- the Particle Tracking-Reference *(p. 737)*.

Finally, there is a chapter that describes all the Extra Parameter options for MIKE SHE that allow you to control model features that are not yet included in the user interface.

- Extra Parameters *(p. 749)*

There is a separate document for the MIKE SHE Toolbox.



REFERENCE MANUAL FOR WATER MOVEMENT

20 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 *(p. 437)*
  - Kristensen and Jensen method *(p. 439)*
  - The 2-Layer Water Balance Method *(p. 447)*
- Snow Melt - Technical Reference *(p. 461)*
- Overland Flow - Technical Reference *(p. 515)*
  - Finite Difference Method *(p. 515)*
  - Overland drainage *(p. 525)*
  - Simplified Overland Flow Routing *(p. 533)*
- Channel Flow - Technical Reference *(p. 471)*
- Unsaturated Zone - Technical Reference *(p. 575)*
  - Richards Equation *(p. 576)*
  - Gravity Flow *(p. 587)*
  - Two-Layer Water Balance *(p. 589)*
- Saturated Flow - Technical Reference *(p. 623)*
  - 3D Finite Difference Method *(p. 623)*
  - Linear Reservoir Method *(p. 638)*

21 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.

21.1 ET from Snow

ET from snow is a sum of the ET from wet snow and dry snow storages.

$$ET_{snow} = ET_{wetsnow} + ET_{drysnow} \tag{21.1}$$

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 *(p. 238)* before being reduce by the Crop Coefficient, k_c, that is specified in the Vegetation Development Table *(p. 409)*.

However, if there is insufficient wet snow storage, then ET will also be removed from dry snow as sublimation using

$$ET_{drysnow} = ET_{ref} \cdot S_f \cdot \Delta t \quad (21.2)$$

where S_f is the sublimation reduction factor found on the Snow Melt *(p. 243)*, 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.

21.2 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,

$$I_{max} = C_{int} \cdot LAI \quad (21.3)$$

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.

21.2.1 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.

21.3 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.

21.3.1 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,

$$E_{at} = f_1(LAI) \cdot f_2(\theta) \cdot RDF \cdot ET_{ref} \tag{21.4}$$

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 21.1)

$$f_1(LAI) = C_2 + C_1 LAI \tag{21.5}$$

where C_1 and C_2 are empirical parameters [-].

$f_2(\theta)$

The second function, *$f_2(\theta)$*, is given by

$$f_2(\theta) = 1 - \left(\frac{\theta_{FC} - \theta}{\theta_{FC} - \theta_W} \right)^{\frac{C_3}{E_p}} \tag{21.6}$$

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 21.2, 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.

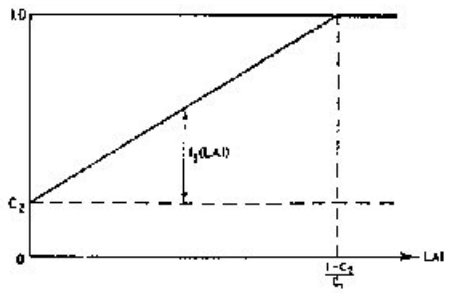


Figure 21.1 The function f*1* versus LAI.

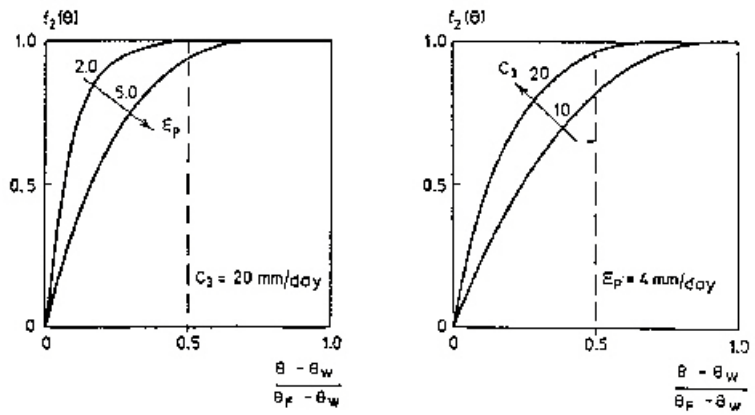


Figure 21.2 The soil moisture function f*2*(θ) for constant C*3* (20 mm/day) and varying E*p* (left), and for constant ET*ref* (4 mm/day) and varying C*3* (right).

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 21.3.

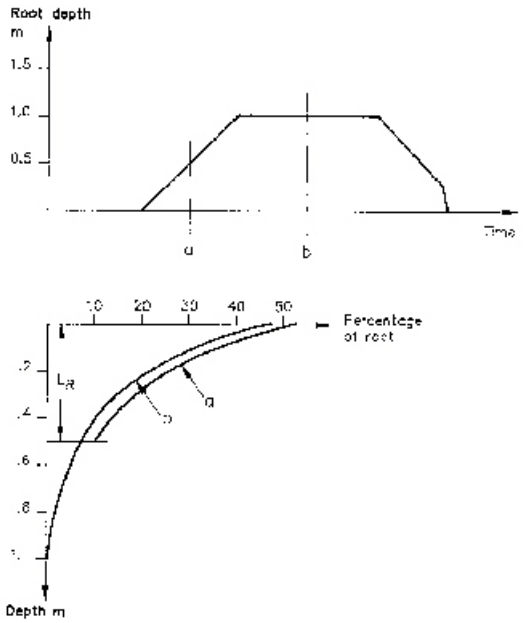


Figure 21.3 Root distribution in time and with depth.

The root extraction is assumed to vary logarithmically with depth by (see Figure 21.3)

$$\log R(z) = \log R_o - AROOT \cdot z \tag{21.7}$$

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,

$$RDF_i = \int_{z1}^{z2} R(z)dz \; / \; \int_{o}^{L_R} R(z)dz \qquad (21.8)$$

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 21.4 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 21.5 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.

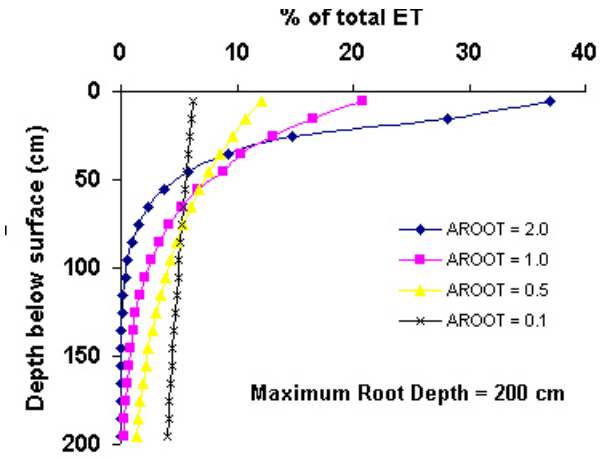


Figure 21.4 Fraction of ET extracted as a function of depth for different values of AROOT.

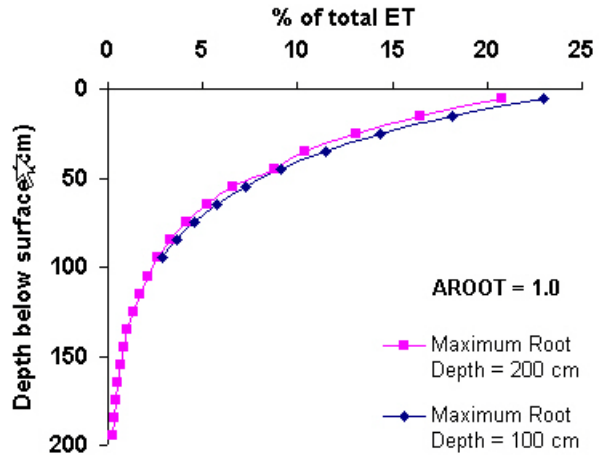


Figure 21.5 Fraction of ET extracted as a function of depth for different maximum root depths.

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.

21.3.2 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:

$$E_s = ET_{ref} \cdot f_3(\theta) + (ET_{ref} - E_{at} - ET_{ref} \cdot f_3(\theta)) \cdot f_4(\theta) \cdot (1 - f_1(LAI)) \qquad (21.9)$$

where ET_{ref} is the reference evapotranspiration, E_{at} is the actual transpiration (Eq. (21.4)), $f_1(LAI)$ is from Eq. (21.5) and the functions $f_3(\theta)$ and $f_4(\theta)$ are given by

$$f_3(\theta) = \begin{cases} C_2 & \text{for } \theta \geq \theta_W \\ C_2 \dfrac{\theta}{\theta_W} & \text{for } \theta_r \leq \theta \leq \theta_W \\ 0 & \text{for } \theta \leq \theta_r \end{cases}$$

$$f_4(\theta) = \begin{cases} \dfrac{\theta - \dfrac{\theta_W + \theta_{FC}}{2}}{\theta_{FC} - \dfrac{\theta_W + \theta_{FC}}{2}} & \text{for } \theta \geq \dfrac{(\theta_W + \theta_F)}{2} \\ 0 & \text{for } \theta < \dfrac{(\theta_W + \theta_F)}{2} \end{cases} \qquad (21.10)$$

In the absence of vegetation $f_1(LAI)$ can be set to zero and E_{at} in Eq. (21.9) goes to zero. This allows us to see how E_s varies in relation to ET_{ref} for different values of q. Thus, Eq. (21.9) can be simplified to

$$\frac{E_s}{ET_{ref}} = f_3(\theta) + f_4(\theta) - f_3(\theta) \cdot f_4(\theta) \qquad (21.11)$$

which is plotted in Figure 21.6.

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.

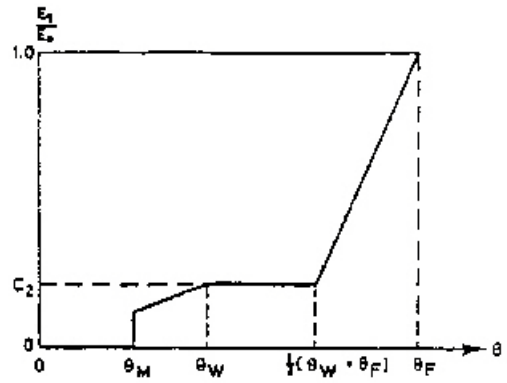


Figure 21.6 Soil evaporation E_s in relation to ET_{ref} as a function of q in the top layer when f(LAI) = 0.

21.3.3 Evapotranspiration Coefficients C_1, C_2 and C_3

The equations for actual transpiration, Eq. (21.4), and soil evaporation, Eq. (21.9), 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.(21.5)). C_3 is also part of Eq. (21.4), but is only found in the soil moisture function, Eq. (21.6).

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 21.7. 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 21.8. 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 21.8, 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).

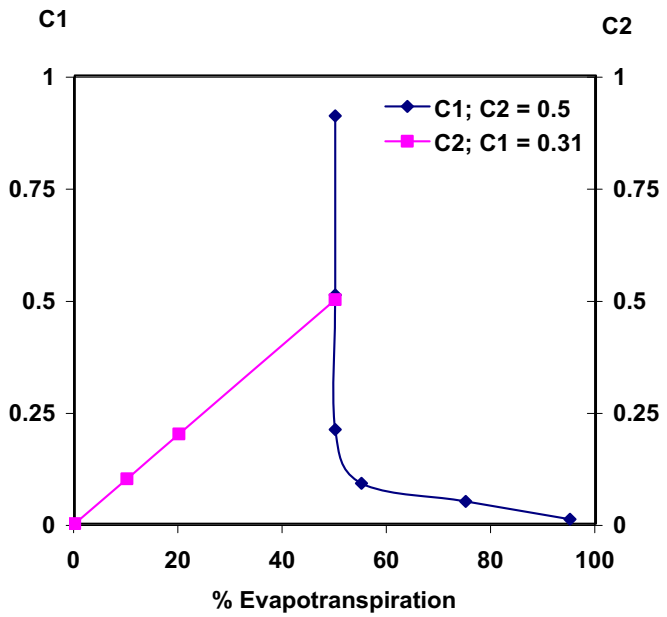


Figure 21.7 The influence of the C_1 and C_2 on the ratio between soil evaporation and transpiration. The values were obtained from model runs assuming C_{int} = 0, the moisture content above field capacity, and LAI = 5

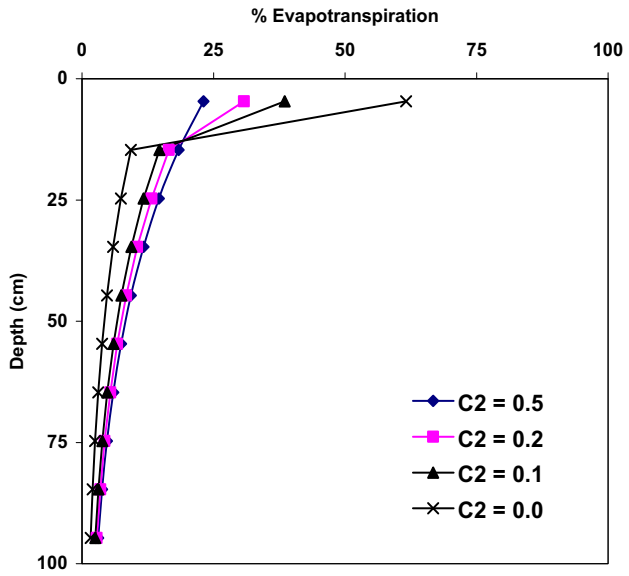


Figure 21.8 Distribution of actual evapotranspiration (in per cent) over depth for different values of C_2. C_2=0 corresponds to pure transpiration.

21.4 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.

21.4.1 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 *(p. 763)* 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.

21.4.2 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.

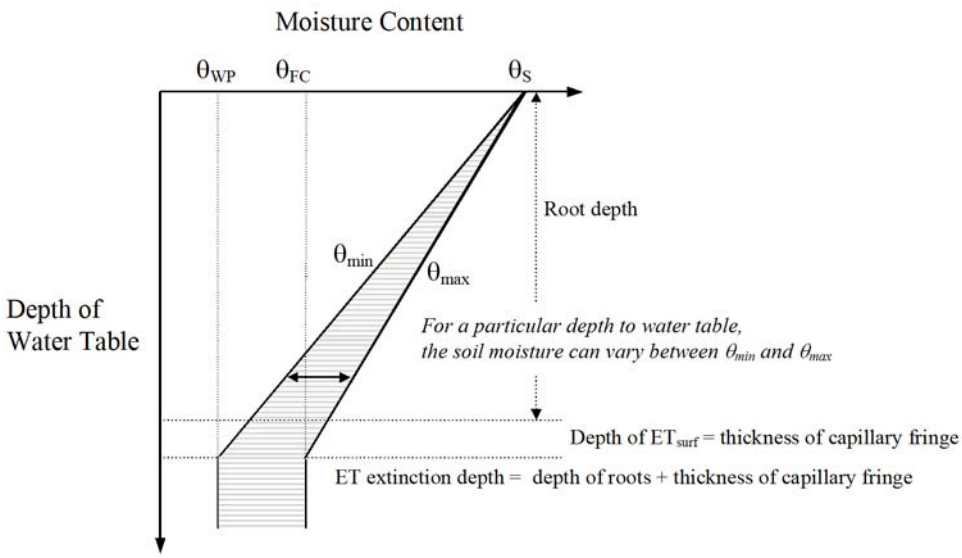


Figure 21.9 Allowable range for soil moisture in the upper ET layer as a function of the depth to the water table.

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 21.9). 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

$$D_{uz} = (\theta_{max} - \theta_{act}) \cdot z_{wt} \tag{21.12}$$

and the total storage available for ET, S_{UZ} is

$$S_{uz} = (\theta_{act} - \theta_{min}) \cdot z_{wt} \tag{21.13}$$

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.

21.4.3 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,

$$S_{intmax} = C_{int} \cdot LAI \tag{21.14}$$

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.

21.4.4 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 $S_{int\ max}$ 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 *(p. 765)*). In the second case, the infiltration rate will be reduced if you have specified an Surface-Subsurface Leakage Coefficient *(p. 276)* that is lower than the specified leakage coefficient in the 2-Layer UZ soil properties *(p. 300)* 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) *(p. 764)*.

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.

21.4.5 Evapotranspiration

MIKE SHE uses the Crop Reference ET rate for all calculations of ET. The Crop Reference ET rate is

$$ET_{rate} = ET_{crop} = ET_{ref} \cdot k_c \quad (21.15)$$

where ET_{ref} is the Reference Evapotranspiration *(p. 238)* and k_c is the Crop Coefficient specified in the Vegetation Development Table *(p. 409)* that adjusts the Reference ET rate for different vegetation types. The maximum amount of ET that can be removed in one time step is

$$ET_{max} = ET_{rate} \cdot \Delta t \quad (21.16)$$

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 *(p. 763)* 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.

$$ET_{snow} = ET_{wetsnow} + ET_{drysnow} \quad (21.17)$$

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 *(p. 238)* before being reduce by the Crop Coefficient, k_c, that is specified in the Vegetation Development Table *(p. 409)*.

However, if there is insufficient wet snow storage, then ET will be removed from dry snow as sublimation using

$$ET_{drysnow} = ET_{ref} \cdot S_f \cdot \Delta t \quad (21.18)$$

where S_f is the sublimation reduction factor found on the Snow Melt *(p. 243)* 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

$$ET_{canopy} = ET_{rate} \cdot \Delta t \quad (21.19)$$

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,

$$ET_{ponded} = ET_{rate} \cdot \Delta t \quad (21.20)$$

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

$$F_{ETUZ} = \frac{min(\theta_{act}, \theta_p) - \theta_{min}}{\theta_p - \theta_{min}} \quad (21.21)$$

where θ_p is water content when ET begins to be restricted given by

$$\theta_p = \theta_{min} + F_p \cdot (\theta_{max} - \theta_{min}) \quad (21.22)$$

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 21.9 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 (21.21) and (21.22) 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.

- A typical value for this θ_p is 0.5. This means that the ET will be removed from the UZ at the full rate until the deficit reaches half the maximum deficit.
- A θ_p of 0 means that ET will be start being reduced as soon as the water content falls below θmax.
- A θ_p of 1 means that ET will not be reduced and ET will be removed at the maximum rate until the wilting point is reached.

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,

$$ET_{UZ} = ET_{rate} \cdot F_{ETUZ} \cdot \Delta t \quad (21.23)$$

However, if Transpiration during ponding *(p. 763)* 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,

$$ET_{left} = (ET_{max} - ET_{snow} - ET_{canopy}) \cdot F_{antol} \quad (21.24)$$

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

$$ET_{SZ} = ET_{rate} \cdot F_{ETSZ} \cdot \Delta t \quad (21.25)$$

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 *(p. 763)* 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 surface. Also, the maximum remaining amount of ET that can be removed is also reduced by the anaerobic tolerance factor, F_{antol}. That is,

$$ET_{left} = (ET_{max} - ET_{snow} - ET_{canopy} - ET_{UZ}) \cdot F_{antol} \quad (21.26)$$

Actual ET

Finally, the actual evapotranspiration is the sum of the above contributions

$$ET_{actual} = ET_{snow} + ET_{canopy} + ET_{ponded} + ET_{UZ} + ET_{SZ} \quad (21.27)$$

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.

21.4.6 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 *(p. 200)* 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.

22 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.

22.1 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 22.1 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.

22.2 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.

23 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.

23.1 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

$$T_{cell} = T_{ref} + (H_{cell} - H_{ref}) \cdot \beta_{dry} \tag{23.1}$$

and under wet conditions (when it is raining or snowing) is

$$T_{cell} = T_{ref} + (H_{cell} - H_{ref}) \cdot \beta_{wet} \tag{23.2}$$

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.

23.2 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

$$P_{cell} = P_{ref} + (H_{cell} - H_{ref}) \cdot \beta_p \tag{23.3}$$

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.

23.3 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

$$W_{frac} = \frac{S_{wet}}{S_{wet} + S_{dry}} \tag{23.4}$$

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

$$M_T = C_T \cdot (T_{air} - T_0) \tag{23.5}$$

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,

$$M_R = -C_{rad} \cdot R_{sw} \tag{23.6}$$

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.

$$M_E = C_E \cdot P \cdot (T_{air} - T_0) \tag{23.7}$$

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.

$$M_{total} = M_T + M_R + M_E \tag{23.8}$$

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

$$M_{total} = M_T + M_R \tag{23.9}$$

In this case, the temperature melting will be positive (i.e. freezing) and radiation melting will be negative.

23.3.1 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

$$A_f = \frac{S_T}{S_{min}} \tag{23.10}$$

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.

23.3.2 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 *(p. 243)* 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 *(p. 238)* and the Crop Coefficient specified in the Vegetation Properties Editor *(p. 407)*.

24 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.

24.1 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.

24.1.1 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(23.1) and (23.2). 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.

24.1.2 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.

24.2 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.

24.3 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.

To achieve this a time varying surface-subsurface leakage coefficient can be specifyed in the Surface-Subsurface Leakage Coefficient dialog.

By default, the Time Series Types *(p. 155)* is Instantaneous, but there 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.

Parameter Name	Type	Value
mean step accumulated leakage coefficient	Boolean	On

24.4 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.

25 Channel Flow - Technical Reference

The hydrologic components of MIKE SHE are directly coupled to DHI's river hydraulic engine MIKE 1D. The MIKE SHE-MIKE 1D 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 river network.
- the full, dynamic coupling of surface and sub-surface flow processes in the river and MIKE SHE.

For technical information on MIKE 1D, please refer to either the MIKE+ River Network Modelling user guide or the MIKE 1D reference manual. Both can be found under the MIKE+ documentation index.

Note: As of release 2024, MIKE SHE can no longer be coupled to the MIKE 11 engine.

25.1 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 separating 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 a MIKE 1D River Model *(p. 477)*:

- **Groundwater exchange with Rivers** *(p. 485)* 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 a MIKE 1D river model to MIKE SHE using Flood Codes** *(p. 492)* 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 Rivers** *(p. 493)* 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.

26 Working with Rivers and Streams - User Guide

26.1 Channel Flow

26.1.1 MIKE 1D Overview

MIKE 1D is a comprehensive 1D channel flow modelling engine 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 1D 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 1D plays a critical role in MIKE SHE Both the overland flow and groundwater flow modules are linked directly to MIKE 1D. The MIKE SHE-MIKE 1D 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 river network.
- the full, dynamic coupling of surface and sub-surface flow processes in the river and MIKE SHE.

26.2 Building a MIKE 1D river model

Integrating a MIKE SHE and a MIKE 1D river model is not very different from establishing a stand-alone MIKE 1D HD river model and a stand-alone MIKE SHE model. A MIKE 1D model can be built in MIKE+ or, using and older version of MIKE software, in MIKE HYDRO River or MIKE 11. If using MIKE+, the setup needs to be converted to the MIKE 1D format .m1dx using the Export function in MIKE+. The .m1dx file can then be specified in MIKE SHE as the river setup. In release 2024, it's still possible to using MIKE 1D setups built using MIKE HYDRO River and MIKE 11 by linking to the .mhydro and .sim11 extensions, respectively. MIKE HYDRO River and MIKE 11 were discontinued after the 2023 release of MIKE software.

In principle, there are three basic set-up steps:

1. Build a stand-alone MIKE 1D 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 1D by defining branches (reaches) where the river HD model should interact with MIKE SHE. Modify your MIKE SHE and MIKE 1D models so that they work together properly. For example, by removing the specified groundwater table in MIKE 1D and adjusting your SZ drainage elevations if you used these in Step 2.

In the above scheme, the first step in coupling MIKE 1D to MIKE SHE is to create a normal MIKE 1D HD river model without coupling it with MIKE SHE. In this regard, a few things should be emphasised:

- In a normal 1D 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 1D 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.

26.2.1 MIKE 1D River Model Network Limitations

There are a few features of MIKE 1D that do not relate well to MIKE SHE.

Short branches

In MIKE 1D there is no restriction on how short your branches are. If you are trying to simulate discontinuous lakes or structures on the flood plain, for example. you may have very short branches. However, MIKE SHE does not allow 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+, 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.
In a dense river network results may also depend on the order the river branches are being processed. Since version 2021 the branches will always be processed in the same order, regardless of sorting in the river model setup. To do the processing in the order of in which they have been specified in the river model GUI instead, use the extra parameter 'disable initial branch sorting (see Order of creating river links *(p. 754)*).

Long coupling links

MIKE SHE links to MIKE 1D River branches. However, when two branches are connected, water is passed between the branches directly. The link has no 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 1D 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 1D 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 1D. 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 1D will likely be unstable.

Long distances between calculation nodes

This is not the same as long distances between cross-sections. MIKE 1D 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.

26.2.2 River Cross-sections

Whenever there is a significant change in the bed slope there should, in principle, be a cross-section at that location. 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 1D is a calculation node. The time step in MIKE 1D 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 1D.

Thus, if you have very short river model 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 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 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.

26.3 Coupling of MIKE SHE and a MIKE 1D River Model

The coupling between MIKE 1D 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 1D 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 26.1 shows part of a MIKE SHE model grid with the MIKE SHE river links, the corresponding MIKE 1D coupling reaches, and the river h-points (points where MIKE 1D calculates the water levels).

The location of each of MIKE SHE river link is determined from the co-ordinates of the river model points, where the river points include both digitised points and h-points on the specified coupling reaches. Since the MIKE SHE river links are located on the edges between grid cells, the details of the river model 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 river water 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 river model 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.

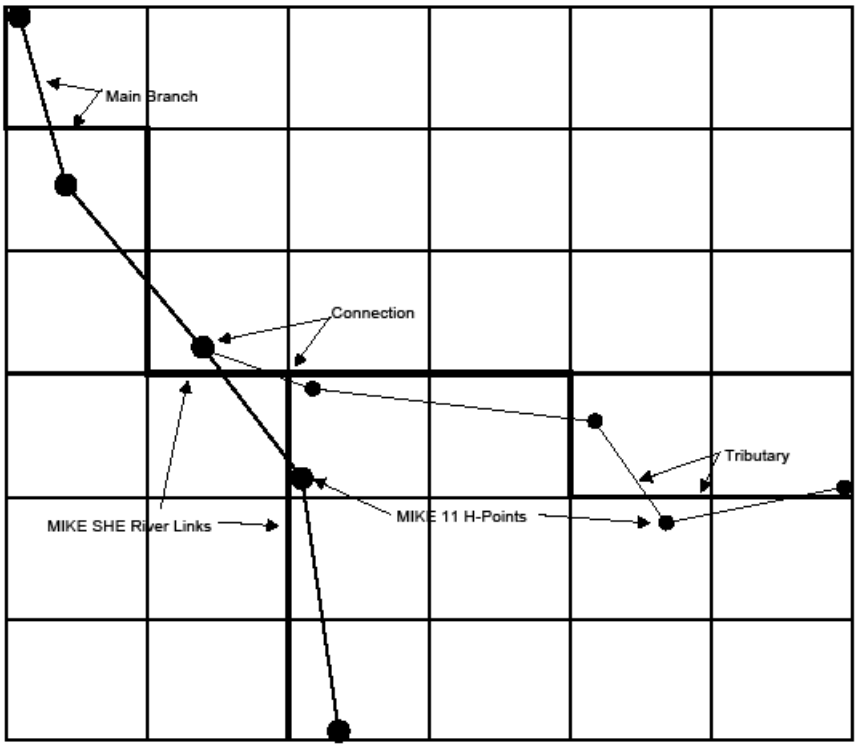


Figure 26.1 MIKE 1D River Branches and h-points in a MIKE SHE Grid with River Links

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 the river model 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 river model 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 1D 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.

26.3.1 MIKE SHE Branches vs. MIKE 1D River Branches

A **MIKE 1D branch** is a continuous river segment defined in MIKE 1D. A MIKE 1D branch can be sub-divided into several coupling reaches.

A **MIKE SHE branch** is an unbroken series of coupling reaches of one MIKE 1D branch.

One reason for dividing a MIKE 1D 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 river model, 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 1D output directly.

Furthermore, the MIKE 1D/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 26.1, 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 1D 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.

26.3.2 The River-Link Cross-section

The MIKE 1D (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 the river and MIKE SHE is calculated based on the river-link cross-section.

The river-link uses a simplified, triangular cross-section interpolated (distance weighted) from the two nearest 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 river cross-section (the elevation of Marker 2 in the cross-section). The left and right bank elevations in MIKE 1D (cross-section markers 1 and 3 in the cross sec-

tion editor) are used to define the left and right bank elevations of the river link (See Figure 26.2).

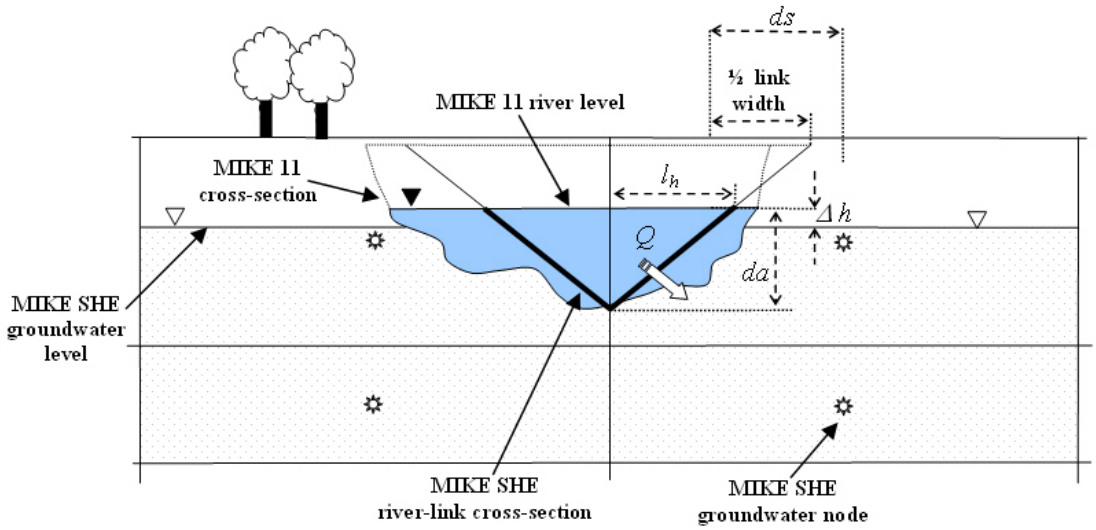


Figure 26.2 A typical simplified MIKE SHE river link cross-section compared to the equivalent MIKE 1D cross-section

If the MIKE 1D 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 a MIKE 1D river model to MIKE SHE using Flood Codes *(p. 492)* option or the Direct Overbank Spilling to and from Rivers *(p. 493)* 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 Rivers *(p. 485)*.

For more detail on flooding and overland exchange with a river model, see Overland Flow Exchange with Rivers *(p. 490)*

26.3.3 Connecting Rivers to MIKE SHE

In MIKE 1D, 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 1D 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 1D calculates the flow, which must be constant between the h-points.

The water levels at the 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 26.3, 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 1D 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.

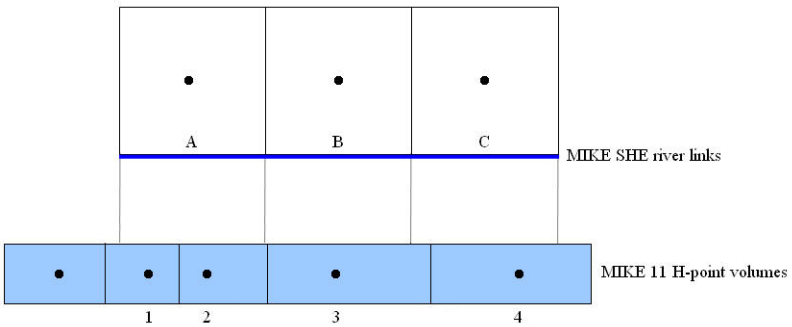


Figure 26.3 Sharing of river model H-point volumes with MIKE SHE river links

The water levels and flows at all river model h-points located within the coupling reaches can be retrieved from the MIKE SHE result file.

However, since the 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 1D Q-points. Rather, the flows stored in the MIKE SHE result file are the estimated flows at the h-points. That is, the flows in the MIKE SHE result file have been linearly interpolated from the calculated flows at the Q-point locations to the H-point locations on either side of the Q-point. If the exact Q-point discharges are needed, they must be retrieved or plotted directly from the MIKE 1D result file.

26.3.4 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

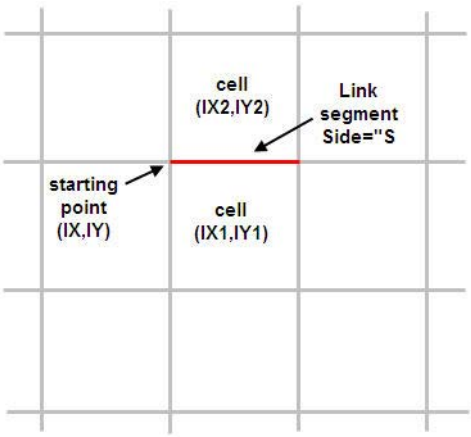
'==>' at start of line indicates that the bank is more than 0.010 m above ground level of a non-flood cell
'***' at start of line indicates that the bank is more than 0.010 m above ground level of a flood (inundation) cell

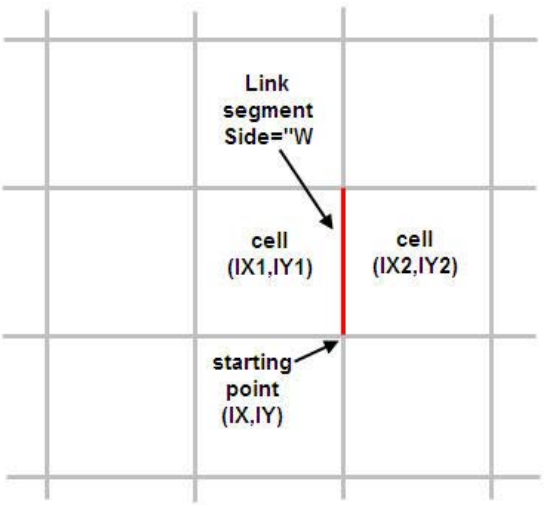
Link	IX	IY	Side	IX1	IY1	Topo1	Bank1	IX2	IY2	Topo2	Bank2	Bed	Width	Leak-opt	Leak-coeff	Spill
1	3	2	S	3	1	0.20	-0.08	3	2	0.22	-0.18	-0.97	71.66	Aq+Bed	0.1000E-04	On
2	3	2	W	2	2	0.12	-0.04	3	2	0.22	-0.14	-0.90	198.97	Aq+Bed	0.1000E-04	On
3	3	3	S	3	2	0.22	-0.10	3	3	0.24	0.00	-0.84	336.07	Aq+Bed	0.1000E-04	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 Exchange attributes table of the MIKE+ Groundwater couplings dialogue. The three possible options are "Aq+Bed", "Aq only", and "Bed only". See Groundwater exchange with Rivers *(p. 485)* and Figure 26.4.

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 Rivers *(p. 485)* and Figure 26.4.

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 26.4.

WeirCoeff: The Weir coefficient value used in the coupling reach in which the river link is contained. See Figure 26.4.

HExpo: The Head exponent value used in the coupling reach in which the river link is contained. See Figure 26.4.

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 26.4.

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 26.4) and the MIKE SHE cell size.

Chainage: Chainage of the 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 river branch. Branches are sorted alphabetically.

26.3.5 Groundwater exchange with Rivers

Groundwater-river exchange will be calculated in all cells adjacent to the river unless they are flooded. All layers that the river cuts into can be included, or just the top layer (see SZ Computational Control Parameters *(p. 201)*).

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.

$$Q = C \cdot \Delta h \tag{26.1}$$

Note that Eq. (26.1) 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 26.2, the head difference between a grid cell and the river is calculated as

$$\Delta h = h_{grid} - h_{riv} \tag{26.2}$$

where h_{grid} is the head in the grid cell and h_{riv} is the head in the river link, as interpolated from the river h-points.

If the ground water level drops below the river bed elevation, the head difference is calculated as

$$\Delta h = z_{bot} - h_{riv} \tag{26.3}$$

where z_{bot} is the bottom of the simplified river link cross section, which is equal to the lowest point in the river cross-section.

If the river water level drops below the bottom of the layer, the head difference can be calculated in two different ways, controlled by theSZ Computational Control Parameters *(p. 201)* dialogue option "Do not limit driving head in layers above river water level". If this option is checked, then the head difference will be calculated as in equation (26.2). If it is unchecked (recommended) then the head difference will instead be

$$\Delta h = z_{grid} - z_{low} \tag{26.4}$$

where z_{low} is the bottom of the layer.

In Eq. (26.1), the conductance, *C*, between the cell and the river link can depend on

- the conductivity of the aquifer material only. See Aquifer Only Conductance *(p. 486)*, or
- the conductivity of the river bed material only. See River bed only conductance *(p. 487)*, or
- the conductivity of both the river bed and the aquifer material. See Both aquifer and river bed conductance *(p. 488)*.

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 26.2, the conductance, *C*, between the grid node and the river link is given by

$$C = \frac{K \cdot da \cdot dx}{ds} \tag{26.5}$$

where *K* is the horizontal hydraulic conductivity in the grid cell, *da* is the thickness 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 26.2, the conductance, *C*, between the grid node and the river link is given by

$$C = L_c \cdot w \cdot dx \tag{26.6}$$

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. (26.6), the wetted perimeter, *w*, is assumed to be equal to the sum of the vertical and horizontal areas available for exchange flow. From Figure 26.2, 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 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 26.2, the conductance, *C*, between the grid node and the river link is given by

$$C = \frac{1}{\frac{ds}{K \cdot da \cdot dx} + \frac{1}{L_c \cdot w \cdot dx}} \tag{26.7}$$

where *K* is the horizontal hydraulic conductivity in the grid cell, *da* is the thickness 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. (26.6), the wetted perimeter, *w*, is assumed to be equal to the sum of the vertical and horizontal areas available for exchange flow. From

Figure 26.2, 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 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.

26.3.6 Steady-state groundwater simulations

For steady-state groundwater models, any coupled river model is not actually run. Rather the initial water level in the river model 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.

Earlier versions had a "canyon option" (extra parameter) for steady state models to limit the head difference used in the river exchange calculation.

This limiting is now the default. For backwards compatibility it is possible to revert to the previous calculation. In this case the limited head difference Δh used in equation(26.4) will be replaced by the unlimited calculation

$$\Delta h = h_{grid} - h_{riv} \tag{26.8}$$

The option can be selected in the SZ Computational Control Parameters *(p. 201)* dialogue.

26.4 Overland Flow Exchange with Rivers

The exchange between overland flow and MIKE 1D can be calculated in three different ways. If the flooding from the 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 the river. If you want to simulate flooding from the river to MIKE SHE then the water can be transferred from the river to MIKE SHE using "Flood Codes" or via direct overbank spilling using a weir formula. In principle, the flood code option does not impact the solution time significantly, 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 topography 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 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 1D 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 the 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 river cross-sections stretch across the lake. MIKE 1D 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 the 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.

26.4.1 Lateral inflow to rivers 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 the 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.

26.4.2 Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes

The MIKE SHE/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 1D 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 the river model (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 1D 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 river water level to the surface topography in the model grid. A grid cell is flooded when the river water level is above the topography. The 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 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 1D 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 the river model. Nor, does MIKE SHE calculate overland exchange to the river, if the cell is flooded by the 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 the river as lateral inflow or outflow to the corresponding H-point in the next river model time step.

In terms of the water balance, the surface water in the inundated areas belongs to the MIKE 1D 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 water in the river model. As long as the element is flooded, any exchange to or from the surface water is managed by MIKE 1D 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.

26.4.3 Direct Overbank Spilling to and from Rivers

If you want to calculate 2D overland flow on the flood plain during a storm event, then you cannot use the Flooding from a MIKE 1D river model to MIKE SHE using Flood Codes *(p. 492)* 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

$$Q = \Delta x \cdot C \cdot (H_{us} - H_w)^k \cdot \left[1 - \left(\frac{H_{ds} - H_w}{H_{us} - H_w}\right)^{k}\right]^{0.385} \quad (26.9)$$

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 the river model 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 *(p. 520)* for overland flow.

26.5 Unsaturated Flow Exchange with Rivers

Direct exchange between rivers 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 rivers 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.

26.6 Water balance with Rivers

The water balance tool in MIKE SHE (Using the Water Balance Tool *(p. 99)*) includes the exchange with rivers, but it does not include the water balance within the river model. In other words, once water enters the 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 rivers.

Water exchanges within the river model can be evaluated using the MIKE View tool. In some cases, this may require you to include the additional output for MIKE 1D, which is selected in the Additional Output tab in the MIKE+ river HD editor.

Note: output in MIKE 1D 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 the 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.

26.7 MIKE+ User Interface

The following section provides additional information for the MIKE+ dialogues that are commonly used with MIKE SHE.

26.7.1 MIKE SHE Coupling Reaches

Each river branch that exchanges water with MIKE SHE is called a coupling reach. A MIKE 1D branch can be sub-divided into several coupling reaches. A reason for doing so could be to allow different riverbed leakage coefficients for different parts of the river.

The upper half of the dialogue displays the properties of the current coupling reach. While, the bottom half of the dialogue is a table listing all of the coupling reaches defined.

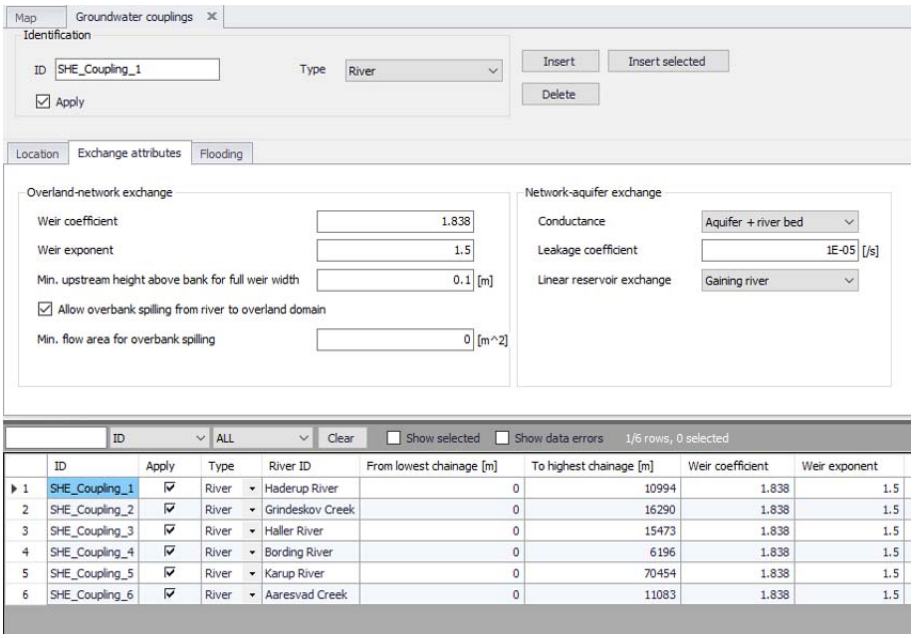


Figure 26.4 MIKE SHE River Links dialogue in the tabular view of the MIKE+ Network Editor

Include all branches button

If the Include all branches button is pressed all the branches in the MIKE+ 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 1D 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 1D 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 *(p. 486)*.

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 *(p. 487)*.

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 *(p. 488)*.

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 *(p. 195)* dialogue. If the weir option is chosen in MIKE SHE, then all MIKE 1D 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 (26.9). 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 (26.9), 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 the 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 river water level goes above the topography of the cell. In this case, water from the 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 river model 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 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 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 river h-points. You must click on the Flood Code checkbox in Figure 26.4, 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 river model 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.

26.8 Common MIKE 1D River Error Messages

There are a number of common error messages that you are likely to encounter when using a river model 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 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 river model is unstable. The slot is a numerical trick that keeps a very small amount of water in the 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.

27 Sewers - Technical Reference

Setting up an integrated MIKE SHE/sewer system model is not very different from establishing a stand-alone sewer model and a stand-alone MIKE SHE model. In principle, there are three basic set-up steps to create a coupled MIKE SHE- sewer model:

1. Establish a MIKE+ collection system 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 components of the hydrologic cycle that are relevant for the study area.
3. Define the locations where the models should interact and then couple the models.

A coupled model can only be run from MIKE SHE; it's not possible to run a coupled setup from MIKE+. Time-stepping of the collection system model is then controlled by MIKE SHE. After MIKE SHE simulates a time step, it will call the MIKE 1D collection system simulation engine and ask it to perform a time step. If the end of the MIKE SHE time step has not yet been reached, MIKE SHE will ask the MIKE 1D engine to calculate the next time step. The sewer model will run normally if it is launched directly from MIKE+.

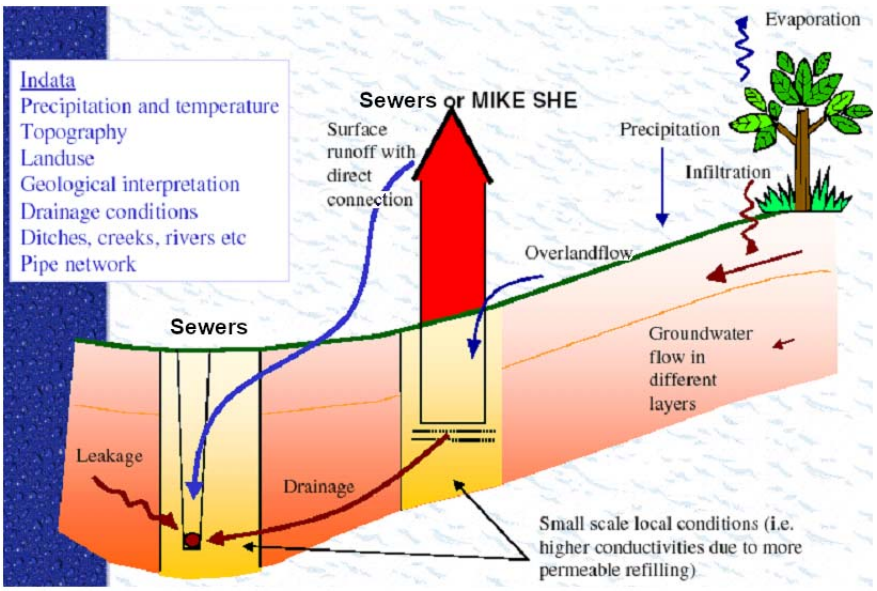


Figure 27.1 MIKE SHE to sewer coupling linkages

All exchange between sewers and MIKE SHE is calculated based on the following equation

$$Q = C \cdot (H_{SHE} - H_{MOUSE})^{k} \quad (27.1)$$

where Q is the exchange between the sewer model and MIKE SHE, C is the conductance, k is a head difference exponent and

$$H_{SHE} = Max(H_{cell}, Z_T, Z_M) \quad (27.2)$$

$$H_{MOUSE} = Max(H_{pipe}, Z_T, Z_M) \quad (27.3)$$

where H_{cell} is the head in the MIKE SHE cell, H_{pipe} is the head in the sewer 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:

1. MIKE SHE SZ to Sewer model pipes: Pipe leakage approach
2. MIKE SHE SZ to Sewer model pipes: Pipe leakage and SZ conductivity approach
3. MIKE SHE SZ to Sewer model pipes: Pipe leakage and grouting conductivity approach
4. MIKE SHE Overland flow to Sewer model open channels
5. MIKE SHE Overland flow to Sewer model manholes

MIKE SHE SZ to Sewer model pipes

This is a leakage-based solution in which the head difference exponent is 1 and the conductance in Equation (33.1) for the flow to or from the pipe is calculated by

$$C = L_c \cdot P_w \cdot l \quad (27.4)$$

where L_c is the leakage coefficient (see below), P_w is the wetted perimeter for the flow (see below), and l is the length of the sewer pipe (link) in the MIKE SHE cell.

Leakage - The leakage can be defined in three ways.

Option 1 is the simple method, which is to use the specified pipe leakage coefficient.

Option 2 uses a combination of the pipe leakage coefficient and the aquifer hydraulic conductivity. In this case, the effective leakage coefficient is calcu-

lated as a series connection of the pipe leakage coefficient (L_p) and the "average" leakage coefficient of the aquifer grid cell (L_{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 sewer 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,

$$L_{aq} = L_{aq,H} + L_{aq,V} = \frac{K_H}{(\Delta xy)/4} + \frac{K_V}{(\Delta z)/4} \tag{27.5}$$

where K_H and K_V are the horizontal and vertical hydraulic conductivities respectively and Δxy and Δz are the horizontal and vertical cell dimensions.

The final leakage coefficient is then calculated as sum of the inverses of both the aquifer leakage coefficient and the pipe leakage coefficient, where the sum of the inverse leakages can be seen as a series of resistances:

$$\frac{1}{L_c} = \frac{1}{L_{aq}} + \frac{1}{L_p} \tag{27.6}$$

Option 3 accounts for pipe grouting with known thickness and conductivity, combined with the pipe leakage.

The radial form of the Darcy equation is

$$q = \frac{2\pi \cdot K \cdot I}{\ln(r_1/r_2)} \cdot (h_1 - h_2) \tag{27.7}$$

where h_1 and h_2 are the heads at distances r_1, r_2 from the center of the radial flow, e. g. a bore hole. For a grouted pipe the grouting radius r_g and the pipe radius r can be used for r_1 and r_2, and the aquifer head and the head in the pipe for h_1 and h_2.

A horizontal pipe may not be fully submerged in the groundwater. To account for this instead of 2π the radial angle α will be used which depends on the groundwater head and head in the pipe, similar to the wetted perimeter. It is calculated twice, once using the inner radius r and the water level h inside the

pipe, and a second time using the outer radius and the ground water level, then the larger of the two values will be used:

$$\alpha = \begin{cases} h \le P_b : & 0 \\ P_b < h \le P_c : & 2 \cdot \mathrm{acos}\left(\frac{dz_c}{r}\right) \\ P_c < h < P_t : & 2\pi - 2 \cdot \mathrm{acos}\left(\frac{dz_c}{r}\right) \\ h \ge P_t : & 2\pi \end{cases} \quad (27.8)$$

With P_b as the pipe (or grouting) bottom level, P_c as the pipe center level and P_t as the pipe (or grouting) top level. dz_c is the vertical distance between the water level and the pipe center, positive in both directions.

The total conductance is calculated as a series connection of the grouting conductance and the conductance through the pipe perimeter:

$$\frac{1}{C} = \frac{\ln(r_g/r)}{\alpha \cdot K_g \cdot l} + \frac{1}{\alpha \cdot r \cdot L_c \cdot l} \quad (27.9)$$

K_g is the hydraulic conductivity in the grouting zone, optionally it can be limited to

$$K_g = \min(K_g, \max(K_h, K_v)) \quad (27.10)$$

r_g is the radius of the grouting zone and r is the radius of the pipe.

Wetted Perimeter - MIKE SHE uses the inner wetted perimeter if the flow is from the sewer to MIKE SHE. Whereas, it uses the outer wetted perimeter if the flow is from MIKE SHE to the sewer. The wetted perimeters are calculated by MIKE 1D.

MIKE SHE Overland flow to Sewer Model LINKS

If a MIKE 1D 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 conductance in Equation (27.1) is defined as

$$C = L_c \cdot l \quad (27.11)$$

where L_c is the exchange coefficient and l is the length of the sewer pipe (link) in the MIKE SHE cell.

If the exponent Equation (27.1) is 1.0, then this is a simple drain formulation and the exchange coefficient 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 [$m^{1/2}/s$]. Generally, the unit of the exchange coefficient will be [$m^{2\text{-exponent}}/s$] for pipes

MIKE SHE Overland flow to Sewer Model Manholes

If the sewer manholes are not sealed, then MIKE SHE can discharge overland flow into the sewer manholes. In this case, the exchange coefficient in Equation (27.1) is defined as

$$C = L_c \tag{27.12}$$

where L_c is the exchange coefficient with units of [m^2/s].

If the exponent Equation (27.1) is 1.0, then this is a simple drain formulation and the exchange coefficient, L_c, is per length with units of [m^2/s]. If the exponent is 1.5, then this is a weir formulation and the units of the exchange coefficient are [$m^{1.5}/s$]. Generally, the unit of the exchange coefficient will be [$m^{2\text{-exponent}}/s$] for man holes.

28 Working with Sewers - User Guide

Five types of exchange are available when linking a MIKE+ collection system model to MIKE SHE. Three types of exchange are specified in MIKE+ only, while the other two require specification in both MIKE+ and MIKE SHE.

The exchange types are summarized in the table below.

Table 28.1 Specification of exchanges between a MIKE+ collection system model and MIKE SHE

Exchange type	Summary	MIKE+ specification	MIKE SHE specification
MIKE SHE overland flow exchange with MIKE+ manholes	Coupling of overland flow in MIKE SHE to sewer manholes. Not overland drain flow (see below).	Exchange parameters are assigned to manholes under Groundwater couplings.	None. MIKE SHE reads the MIKE+ setup at runtime and identifies the locations of coupled manholes. Exchange takes place at the MIKE SHE grid cell in which the manhole is located.
MIKE SHE overland drainage to MIKE+ manholes	Coupling of overland drainage in MIKE SHE to sewer manholes.	One or more drain codes are assigned to each manhole under Groundwater couplings. Instead of (or in addition to) a specific drain code "-1" can be used to allow automatic linkage	Cells that drain to each manhole are given that manhole's code in the Drain Codes file. The cells must also be given the value 4 in the Drainage Option Distribution file. In other words, the Drainage Option Distribution specification informs MIKE SHE that the cell drains to a manhole, and the Drain Codes specification identifies which manhole. The special drain code "-1" lets the cell drain to the closest manhole with active drainage, drain code "-1" and a level at or below the cell ground level.

Table 28.1 Specification of exchanges between a MIKE+ collection system model and MIKE SHE

Exchange type	Summary	MIKE+ specification	MIKE SHE specification
MIKE SHE saturated drainage to MIKE+ manholes	Coupling of saturated zone drainage to manholes.	Same as for overland drainage to manholes	Same as for overland drainage to manholes.
Exchange between MIKE SHE overland flow and MIKE+ pipes	Coupling of MIKE SHE overland flow and MIKE+ pipe flow. Only relevant for open pipes.	Exchange parameters are assigned to pipes under Groundwater couplings.	None. MIKE SHE reads the MIKE+ setup at runtime and identifies the locations of coupled pipes. Exchange takes place at the MIKE SHE grid cell in which the pipe is located.

More details about linking MIKE SHE drain flows to a collection system model are provided below.

MIKE SHE SZ drain flow to Sewer model manholes

If drain flow is specified in MIKE SHE, then the drainage can be discharged to a sewer 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 *(p. 331)*) and the cells that drain to a manhole must have an option value of 4 (see Option Distribution *(p. 337)*). The references between the MIKE SHE drain codes and the sewer manholes are defined in MIKE+. A manhole that has the drain code -1 (either as the only drain code or among other drain codes) will be used as a drain target if it is the closest one to a cell that has a drain code -1 and is at or above the level of that manhole.

MIKE SHE Overland Drainage to Sewer model manholes

If the Overland Drainage option is used in MIKE SHE, then the draining water can be discharged to a sewer manhole. Drain flow is calculated by MIKE SHE based on parameters defined in the ponded drainage specification.

In MIKE SHE the distributed drainage option must be chosen (see Overland Drainage *(p. 280)*) and the cells that drain to a manhole must have an option value of 4 (see Distributed drainage options *(p. 333)*). The references between the MIKE SHE drain codes and the sewer manholes are defined in MIKE+. A manhole that has the drain code -1 (either as the only drain code or among other drain codes) will be used as a drain target if it is the closest one to a cell that has a drain code -1 and is at or above the level of that manhole.

Sewer Outlets to MIKE SHE

Sewer outlets cannot directly discharge to MIKE SHE's overland flow. To work around this, you can add a dummy manhole to your sewer model pipe and then couple the pipe to the outlet via a small diameter dummy pipe (See Figure 28.1). 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.

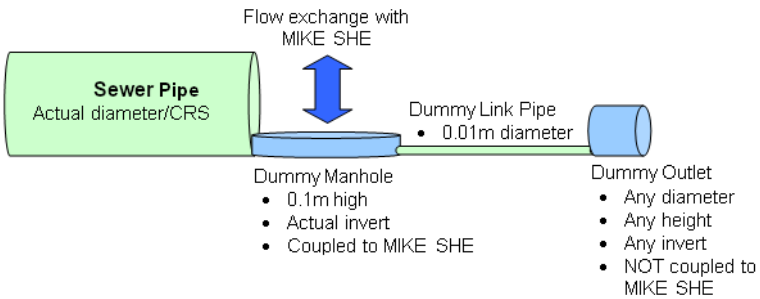


Figure 28.1 Work around for discharging sewer model outlets to MIKE SHE

Miscellaneous parameters

Smoothing of SZ exchange: Ensures a smoother calculation of flows to the sewers when the MIKE SHE time steps are large compared to the MIKE 1D sewer model time step. The sewer coupling is only made at every integer multiple of the MIKE SHE time step. If the Smooth option is not activated, the flows to the sewer can stop after a number of MIKE 1D 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.

Limit inflow: This can be activated so the sewer volume + inflow does not exceed a specified fraction of the maximum sewer volume. This is used to avoid instabilities due to high pressure.

Limit outflow: This can be activated so the sewer volume - outflow doesn't come below a specified fraction of the maximum sewer volume. This is used to avoid instabilities due to drying/negative volume.

28.1 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 28.2 File names and conditions for output for the MIKE SHE-sewer coupling. 'setupname' refers to the name of the model setup file

File Name	Item Name	The file is created when...
.*setupname**setupname*_ SZ2MouseReaches.dfs0	"Reach [n]" (storing reaches cannot be named)	...the MIKE SHE SZ coupling is included and storing reaches have been specified.
.*setupname**setupname*_ OL2MouseReaches.dfs0	"Reach [n]" (storing reaches cannot be named)	…the MIKE SHE Overland coupling is included and storing reaches have been specified.
.*setupname**setupname*_ OL2MouseManholes.dfs0	"<name of manhole>"	...the MIKE SHE Overland flow coupling to manholes is included.
.*setupname**setupname*_ SZDrain2MouseManholes.dfs0	"<name of manhole> <drain code>"	...the MIKE SHE SZ drain coupling to manholes is included
.*setupname**setupname*_ OLDrain2MouseManholes.dfs0	"<name of manhole> <drain code>"	...the MIKE SHE OL drain coupling to manholes is included

28.2 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 the sewer model for one or more MIKE 1D 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 (27.1). 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 the sewers will be less than this value if the flow goes from the sewers 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.

28.3 Water Balance Limitations

The interaction with MIKE SHE is not included in the MIKE 1D Summary HTML file. Thus, the water added from MIKE SHE appears as an error (i.e. 6: Continuity balance in MIKE 1D).

29 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.

29.1 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.

29.1.1 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

$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x}(uh) + \frac{\partial}{\partial y}(vh) = i \tag{29.1}$$

and the momentum equation gives

$$S_{fx} = S_{Ox} - \frac{\partial h}{\partial x} - \frac{u}{g}\frac{\partial u}{\partial x} - \frac{1}{g}\frac{\partial u}{\partial t} - \frac{qu}{gh} \tag{29.2}$$

$$S_{fy} = S_{Oy} - \frac{\partial h}{\partial y} - \frac{v}{g}\frac{\partial v}{\partial y} - \frac{1}{g}\frac{\partial v}{\partial t} - \frac{qv}{gh} \tag{29.3}$$

where S_f is the friction slopes in the x- and y-directions and S_O is the slope of the ground surface. Equations (29.1), (29.2) and (29.3) 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

$$S_{fx} = S_{Ox} - \frac{\partial h}{\partial x} = -\frac{\partial z_g}{\partial x} - \frac{\partial h}{\partial x} \tag{29.4}$$

If we further simplify Equation (29.4) using the relationship $z = z_g + h$ it reduces to

$$S_{fx} = -\frac{\partial}{\partial x}(z_g + h) = -\frac{\partial z}{\partial x} \tag{29.5}$$

in the x-direction. In the y-direction Equation (29.5) becomes

$$S_{fy} = -\frac{\partial}{\partial y}(z_g + h) = -\frac{\partial z}{\partial y} \tag{29.6}$$

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

$$S_{fx} = \frac{u^2}{K_x^2 h^{4/3}} \tag{29.7a}$$

$$S_{fy} = \frac{v^2}{K_y^2 h^{4/3}} \tag{29.7b}$$

Substituting Equations (29.4) and (29.5) into Equations (29.7a) and (29.7b) results in

$$\frac{u^2}{K_x^2 h^{4/3}} = -\frac{\partial z}{\partial x} \tag{29.8a}$$

$$\frac{v^2}{K_y^2 h^{4/3}} = -\frac{\partial z}{\partial y} \tag{29.8b}$$

After simplifying Equations (29.8a) and (29.8b) and multiply both sides of the equations by *h*, the relationship between the velocities and the depths may be written as

$$uh = K_x\left(-\frac{\partial z}{\partial x}\right)^{1/2} h^{5/3} \tag{29.9a}$$

$$vh = K_y\left(-\frac{\partial z}{\partial y}\right)^{1/2} h^{5/3} \tag{29.9b}$$

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.

29.1.2 Finite Difference Formulation

Consider the overland flow in a small region (see Figure 29.1) of a MIKE SHE model, having sides of length *Dx* and *Dy* and a water depth of *h*(*t*) at time *t*.

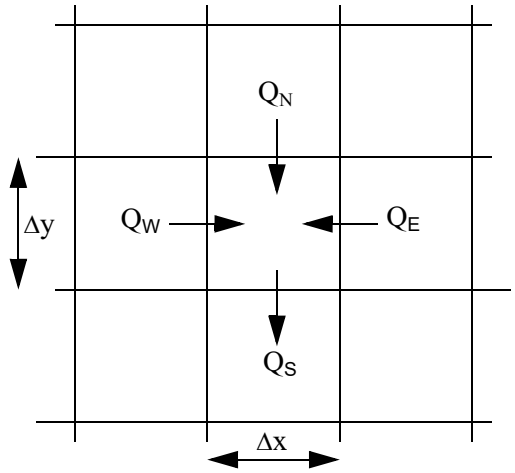


Figure 29.1 Square Grid System in a small Region of a MIKE SHE model

A finite-difference form of the velocity terms in Eq. (29.1) may be derived from the approximations

$$\frac{\partial}{\partial x}(uh) \cong \frac{1}{\Delta x}\{(uh)_{east} - (uh)_{west}\} \tag{29.10}$$

and

$$\frac{\partial}{\partial y}(vh) \cong \frac{1}{\Delta y}\{(vh)_{north} - (vh)_{south}\} \tag{29.11}$$

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

$$\Delta h = h(t+\Delta t) - h(t) = I + \frac{\Sigma Q \Delta t}{\Delta x^2} \tag{29.12}$$

where

$$I = i\Delta x^2, \tag{29.13}$$
$$\Sigma Q = Q_N + Q_S + Q_E + Q_W$$

and where, i is the net input to overland flow in Eq. (29.1) 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 29.2), 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.

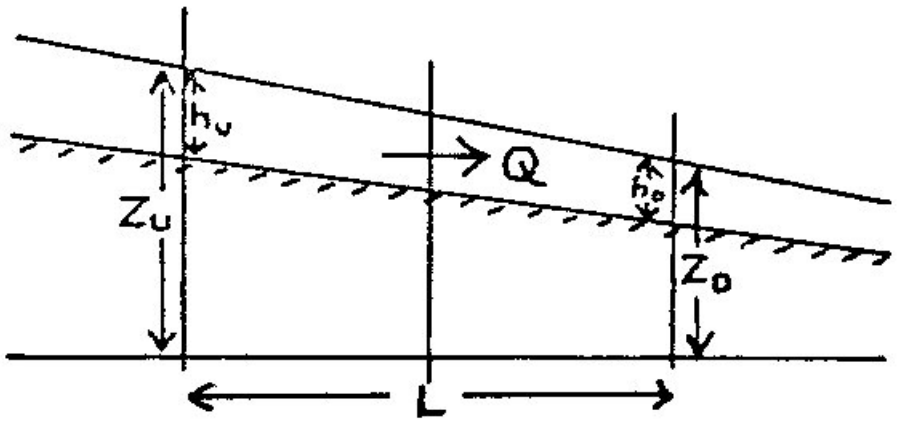


Figure 29.2 Overland flow across grid square boundary.

Equations (29.9a) and (29.9b) may be used to estimate the flow, Q, between grid squares by

$$Q = \frac{K\Delta x}{\Delta x^{1/2}}(Z_U - Z_D)^{1/2}\ h_u^{5/3} \tag{29.14}$$

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 (29.14) 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. (29.14), using the specified boundary water levels.

29.1.3 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. (29.14) 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. (29.12) gives:

$$\Sigma Q \geq \frac{-\Delta x^2 h(t)}{\Delta t} - I \qquad (29.15)$$

where *SQ* is the sum of outflows and inflows.

Splitting *SQ* into inflows and outflows and remembering that outflow is negative, gives:

$$\Sigma \left| Q_{out} \right| \leq \Sigma Q_{in} + I + \frac{\Delta x^2 h(t)}{\Delta t} \qquad (29.16)$$

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 (29.16).

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.

29.1.4 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

$$C = \frac{\frac{dQ}{dA}}{\frac{dx}{dt}} = \frac{1}{dx} \times \frac{dQ}{dh} \times \frac{dt}{dx} \tag{29.17}$$

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,

$$\Delta t = \frac{dh}{dQ} \cdot \Delta x^2 \cdot C \tag{29.18}$$

where Δt is the time step and Δx is the cell width.

The differential term in (29.18) is the inverse of the derivative of the Mannings equation, (29.14), 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.

29.1.5 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:

$$Q = W \cdot C \cdot (H_{US} - H_W)^k \cdot \left[1 - \left(\frac{H_{DS} - H_w}{H_{US} - H_w}\right)^k\right]^{0,385} \qquad (29.19)$$

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 cal-

culated 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.

29.1.6 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 (29.18) 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.

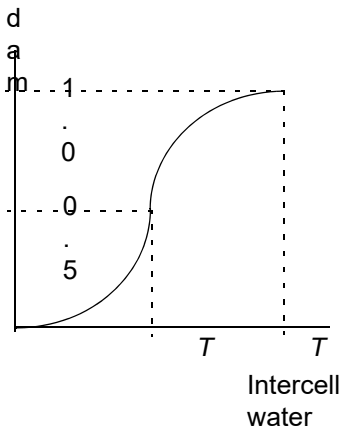


Figure 29.3 Default damping function for numerical stability at low gradients

The default damping function is a pair of parabolic equations (see Figure 29.3). When the flow gradient between cells reaches the threshold the following damping function is applied

$$F_D = 1 - 2\left(\frac{T \cdot dx - dZ}{T \cdot dx}\right)^2 \tag{29.20}$$

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

$$F_D = 2\left(\frac{dZ}{T \cdot dx}\right)^2 \tag{29.21}$$

which goes to zero as the water level difference goes to zero.

In the log-log graph in Figure 29.4, 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.

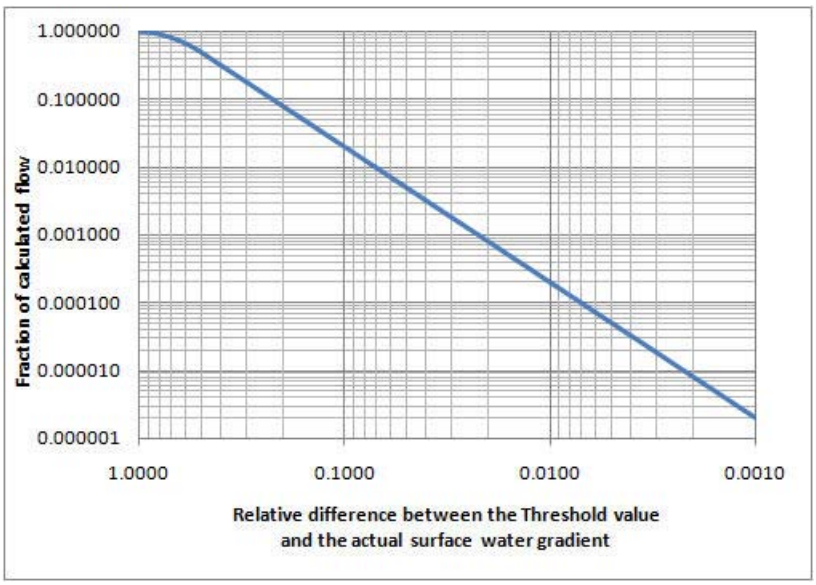


Figure 29.4 Actual overland flow as a function of the difference between the threshold gradient and the surface water gradient based on Equations (29.20) and (29.21).

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 the MIKE+ 2D Overland module.

The alternative function is a single parabolic function (see Figure 29.5)

$$F_D = 1 - \left(\frac{T \cdot dx - dZ}{T \cdot dx}\right)^2 \tag{29.22}$$

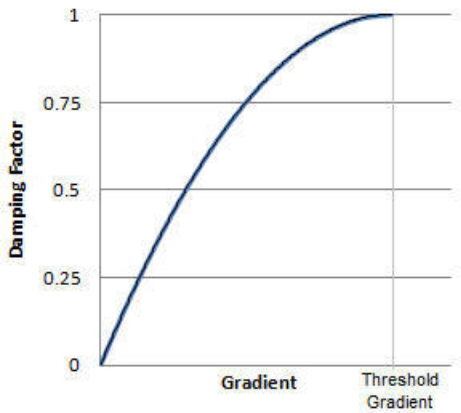


Figure 29.5 Alternative damping function activated by an Extra Parameter.

To activate the alternate function, you must specify the Alternative low gradient damping function *(p. 759)* boolean parameter in the Extra Parameters *(p. 353)* 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 *(p. 197)* for the Overland Flow), below which the damping function becomes active.

29.1.7 Overland drainage

The amount of runoff generated by MIKE SHE depends on the density of the stream network in the river model. 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 Overland drainage function has been designed to facilitate a flexible definition of storm water drainage in both natural and urbanized catchments. In particular, the Overland drainage function supports the following features:

- In urban areas, paving and surface sealing (e.g. roof tops) limits infiltration and enhances runoff.
- Ponded water and precipitation are 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 flow 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.
- Infiltration and overland flow are calculated normally for ponded water that does not drain in the current time step.

Conceptually, the overland drainage function is shown in Figure 29.6.



Figure 29.6 Conceptualization of the overland drainage function

Calculation Order and time step

The overland drainage is calculated explicitly in a particular order with respect to the other hydrologic processes. However, the exact order depends on what UZ solver is being used. In all cases interception is calculated first. When using the Richards- or the gravity-solver, ET will be calculated before OL drainage, when using the 2-layer UZ solver ET will be calculated after drainage.

After the drainage calculation any remaining precipitation will be added to the remaining ponded water.

If there is any ponded water remaining on the cell after the drainage has been removed, then it will be subject to other processes in the following order:

1. Infiltration to the unsaturated zone;
2. If the groundwater table is at/above ground: direct OL SZ exchange and
3. Finally, Lateral Overland Flow will be calculated to the adjacent cells.

The overland 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 overland drainage is calculated, the water depths in all the cells are updated - both sources and destinations. The calculation of infiltration and lateral OL flow is based on the updated ponded depths.

Although overland drainage is a fast process occurring at a time scale similar to overland flow itself, it is being calculated on the unsaturated zone time step that usually is significantly longer than the OL time step. The reasons for this are

1. The computational effort is lower
2. This ensures a more realistic share of the precipitation between infiltration and runoff. Overland drainage is being calculated after net rainfall, but before infiltration, so it can remove the appropriate amount of water from the system. If it was run with the overland component instead, the unsaturated zone component would process all net rainfall, leading to an overestimation of infiltration. Dividing the precipitation between drain inflow and the other processes does also to a certain degree depend on the length of the UZ time step. The length of the UZ time step during rain events can be controlled using the Precipitation-dependent time step control parameters (see Controlling the Time Steps *(p. 83)*).

Calculation method

The calculations of flow from ponding/precipitation into the overland drain storage and the flow from the drain storage to a recipient are very similar. They follow a linear reservoir type of equation, the flow is driven by a head difference and inhibited by a resistance, expressed as the in-/outflow time constants.

For flow into the drain the OL ponded storage change equals the difference between flow into OL and OL flow into the drain:

$$\frac{dS_{OL}}{dt} = P - Q_{Dr}(t) \tag{29.23}$$

Where S_{OL} is the OL ponded storage, P is net precipitation (the only source to OL considered for this purpose) and Q_{Dr} is the flow into the drain. The flow

into the drain is proportional to the drainable part of the current ponded volume, the proportion is given by the drain time constant:

$$Q_{Dr}(t) = C_{Dr} \cdot (S_{OL}(t) - zDr_0) \quad (29.24)$$

where C_{Dr} is the drain time constant and zDr_0 is the drain base level, i. e. the bottom level of the drainable part of OL storage. The drain base level will be the highest of a) ground surface, b) drain level, c) detention storage level (which by default is not drainable) and d) water level in the drain (only if option "check water levels" is active). Substituting (29.24) into (29.23) yields:

$$\frac{dS_{OL}}{dt} + C_{Dr} \cdot S_{OL}(t) = P + C_{Dr} \cdot zDr_0 \quad (29.25)$$

Further integrations and substitutions lead to the linear reservoir type of equation for the volume added to the drain:

$$V(t) = P \cdot t - \left(\frac{P}{C_{Dr}} - S_{OL_0} + zDr_0\right) \cdot (1 - e^{-C_{Dr} \cdot t}) \quad (29.26)$$

With S_{OL_0} being the initial overland storage.

In cases where the OL water level falls below or rises above the threshold for drain flow (the drain base level, see above) during a time step, drain inflow will only be calculated for the part of the time step where the threshold is exceeded.

Finally, the runoff coefficient and the maximum flow rate (if specified) will be applied.

Flow out of the drain to the recipient will be calculated in the same way, except there is no handling of cases where only part of the time step produces drain outflow. Because of this, when the recipient water level is above the drain water level, a reverse flow from the recipient into the drain would be calculated. If this happens the flow will be set to zero to prevent any reverse flow.

For drain inflow, if the option "Instantaneous drain inflow from net rainfall" is selected, the calculation will be simplified to

$$V(t) = P \cdot t \cdot Dr_{RC} \quad (29.27)$$

Where Dr_{RC} is the user specified overland drainage runoff coefficient.

Constraints

Several constraints have been added to the formulation to make the overland Drainage function more practical.

- **Paved Area Fraction** - a paved fraction for the cell is used to reduce infiltration in urban areas.
- **Runoff Coefficient** - a fraction for the cell is used to partition the cell into a fraction that can drain to the drainage network, and a fraction where all rainfall and ponding is left to be subject to infiltration and finite difference Overland Flow. Note that the sub-cell volumes are not being tracked separately - subsequent components as well as subsequent drainage time steps will still work on the overall ponded water. So, this makes most sense if all the water is removed by infiltration or OL 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 because the drain level is considered to be the bottom of the drain storage.
- **Maximum drain inflow rate** - a maximum drainage rate ensures that culvert capacities are not exceeded
- **Check water levels** - there is an optional check to include the destination water level in the calculation (for flow into the drain this is the drain water level, for flow out of the drain this is the recipient water level). If the downstream drainage destination does have a higher water level than the drain, then there will be no flow at all.
- **Inflow time constant** - the rate of inflow to the drains is controlled by a time constant/leakage factor.
- **Outflow time constant** - the rate of outflow to the destination is controlled by a time constant. If the outflow time constant is very small, then water may accumulate on a cell-by-cell basis in the “drain” (unless the ‘check water levels’ option is active). There is currently no limit on the amount of water that 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 river nodes, or sewer manholes.

29.1.8 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 29.7.

In the case of detention storage, the volume of detention storage is calculated based on the user specified depth and OL cell area.

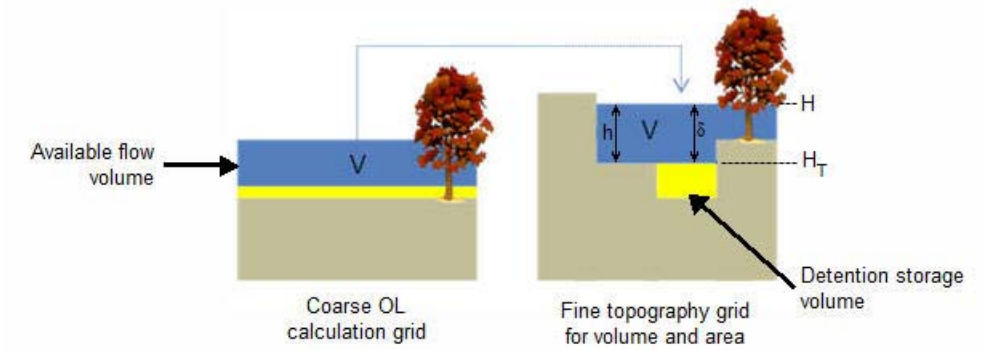


Figure 29.7 The constant volume from the coarse grid is transfered to the fine scale grid.

According to the Gauckler-Manning-Strickler formula, overland sheet flow can be expressed as

$$Q = A_f(h) \cdot M \cdot R(h)^{2/3} \cdot \sqrt{\frac{\Delta h}{\Delta x}} \tag{29.28}$$

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,

$$\frac{\partial Q}{\partial A_f} = \frac{\partial Q}{\partial d} \cdot \frac{\partial h}{\partial A_f} \tag{29.29}$$

which expands to

$$\frac{\partial Q}{\partial A_f} = \frac{\partial h}{\partial A_f} \cdot \left(\frac{\partial A_f}{\partial h} \cdot M \cdot R^{2/3} \cdot \sqrt{\frac{\Delta h}{\Delta x}} + A_f \cdot M \cdot \frac{2}{3} R^{(-1/3)} \cdot \right. \tag{29.30}$$

$$\left. \frac{\partial R}{\partial h} \cdot \sqrt{\frac{\Delta h}{\Delta x}} + A_f \cdot M \cdot R^{2/3} \cdot \frac{1}{2\sqrt{\frac{\Delta h}{\Delta x}}} \cdot \frac{1}{\Delta x} \cdot \frac{\partial \Delta h}{\partial h} \right)$$

For any cross-section of flow, the resistance radius is calculated by

$$\sqrt{R} = \frac{1}{A_f} \cdot \int_0^B h^{\frac{3}{2}} dx \tag{29.31}$$

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,

$$\sqrt{R} = \frac{1}{A_f} \sum_1^n h_i^{\frac{3}{2}} \frac{B}{n} \tag{29.32}$$

where h_i is the local water depth in each sub-grid. Since, A_f is equal to

$$A_f = \sum_1^n h_i \frac{B}{n} \tag{29.33}$$

The resistance radius can be simplified to

$$R = \frac{\left(\sum h_i^{\frac{3}{2}} \right)^2}{\left(\sum h_i \right)^2} \tag{29.34}$$

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

$$R = \frac{\left(\sum \delta_i^{\frac{3}{2}} \right)^2}{\left(\sum \delta_i \right)^2} \tag{29.35}$$

where

$$\delta_i = \begin{cases} \{H - H_T\}_i & H > H_T \\ 0 & H \le H_T \end{cases} \quad (29.36)$$

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 (29.33), the flow area for the x and y flow directions for each coarse grid cell is

$$A_{fx} = A_{fy} = \frac{1}{n}\sum_1^n \sum_1^n \delta_i \frac{\Delta X}{n} = \frac{\Delta X}{n^2}\sum_1^n \delta_i \quad (29.37)$$

and from Equation (29.35), the resistance radius for the x and y flow directions for each coarse grid cell is

$$R_x = R_y = \frac{1}{n}\sum_1^n \frac{\left(\sum_1^n \delta_i^{\frac{3}{2}}\right)^2}{\left(\sum_1^n \delta_i\right)^2} \quad (29.38)$$

Finally, the courant criteria also includes the flow area. Thus, from Equation

$$C = \frac{\frac{dQ}{dA_f}}{\frac{dx}{dt}} = \frac{dQ}{dA_f} \cdot \frac{dt}{dx} \quad (29.39)$$

which implies that the maximum allowed time step, Δt,when using the multi-grid method is

$$\Delta t = \frac{\Delta x}{\frac{dQ}{dA_f}} \cdot C \quad (29.40)$$

where Δx is the coarse overland flow grid size, the denominator is defined by Equation (29.30), 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 rivers 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 the 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 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.

29.2 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.

29.2.1 Theoretical basis

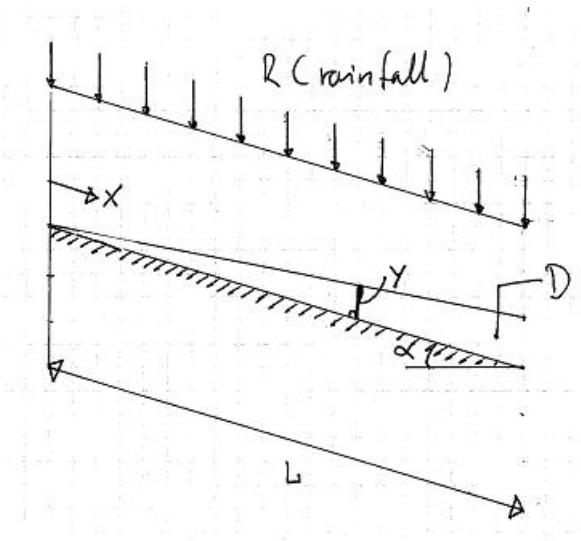


Figure 29.8 Schematic of overland flow on a plane

Figure 29.8 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

$$\frac{\partial q}{\partial x} = R - \frac{\partial y}{\partial t} \tag{29.41}$$

where q is the specific discharge.

For turbulent flow on a plane of infinite width, the Manning equation is

$$q = M \cdot y^{5/3} \sqrt{\alpha} \ [m^2/s] \tag{29.42}$$

where M is the Mannings M.

Now, at equilibrium, the depth no longer changes and the specific discharge approaches the rainfall rate

$$\frac{\partial y}{\partial t} = 0 \Rightarrow \frac{\partial q}{\partial x} = R \Rightarrow q_e = R \cdot x \tag{29.43}$$

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

$$De = \int_o^L y dx = \int_o^L \left(\frac{q_e}{M \cdot \sqrt{\alpha}}\right)^{3/5} dx = \int_o^L \left(\frac{R \cdot x}{M \cdot \sqrt{\alpha}}\right)^{3/5} dx$$

$$De = \frac{5}{8} \cdot \frac{R^{3/5} \cdot L^{8/5}}{M^{3/5} \cdot \alpha^{3/10}} \; [m^3/m] \tag{29.44}$$

The depth, y, near the leading edge of the flow plane can be related to the depth at equilibrium by

$$y = \left(\frac{t}{t_e}\right) \cdot y_e \tag{29.45}$$

where t is the time and t_e is the time until the equilibrium is reached.

Then from (29.42) we can write

$$q = M \cdot \left(\frac{t}{t_e}\right)^{5/3} \cdot y_e^{5/3} \sqrt{\alpha} \; [m^2/s] \tag{29.46}$$

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

$$Q = \int_o^{t_e} q dt = \int_o^{t_e} M \cdot \left(\frac{t}{t_e}\right)^{5/3} \cdot y_e^{5/3} \cdot \sqrt{\alpha} (dt)$$

$$Q = \frac{3}{8} \cdot M \cdot y_e^{5/3} \cdot \sqrt{\alpha} \cdot t_e \; [m^2] \tag{29.47}$$

From (29.46), at equilibrium, ($t = t_e$), the depth of water at the leading edge of the plane (x = L) is

$$q = M \cdot y_e^{5/3} \sqrt{\alpha} = R \cdot L \tag{29.48}$$

which yields

$$y_e^{5/3} = \frac{R \cdot L}{M \cdot \sqrt{\alpha}} \; [m^{5/3}] \tag{29.49}$$

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 (29.47) and (29.44) yields

$$(R \cdot L \cdot t_e) - \left(\frac{3}{8} \cdot M \cdot y_e^{5/3} \cdot \sqrt{\alpha} \cdot t_e\right) = \left(\frac{5}{8} \cdot \frac{R^{3/5} \cdot L^{8/5}}{M^{3/5} \cdot \alpha^{3/10}}\right) \quad (29.50)$$

which, when simplified, yields the time to reach equilibrium

$$t_e = \frac{L^{3/5}}{R^{2/5} \cdot M^{3/5} \cdot \alpha^{3/10}} = \frac{8}{5} \cdot \frac{De}{R \cdot L} \quad (29.51)$$

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

$$y = R \cdot t \quad (29.52)$$

and the relationship between the depth, *y*, and the surface storage at equilibrium, D_e, is given by

$$y = \frac{8}{5} \cdot \frac{De}{L} \ [m] \quad (29.53)$$

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)

$$y = \frac{D}{L}\left(1 + 3/5 \cdot \left(\frac{D}{De}\right)^3\right) \ [m] \quad (29.54)$$

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 (29.54) into the Manning equation (29.42) yields

$$q = M \cdot \sqrt{\alpha} \cdot \left[\frac{D}{L}\left(1 + 3/5 \cdot \left(\frac{D}{De}\right)^3\right)\right]^{5/3} \ [m^2/s] \quad (29.55)$$

29.2.2 Implementation in MIKE SHE

In MIKE SHE the current level of surface detention storage is continually estimated by solving the continuity equation

$$D_2 = D_1 + (\bar{q}_{supply} - \bar{q}) \cdot \Delta t \quad (29.56)$$

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 (29.55) is solved iteratively until a solution satisfies both equations.

29.2.3 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.

29.2.4 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 *(p. 749)*.

29.2.5 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 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 *(p. 749)*.

30 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 *(p. 541)* - this section describes additional information on parameter values;
- Overland Flow Performance *(p. 544)* - this section provides additional information on optimizing your overland simulation;
- Multi-cell Overland Flow *(p. 547)* - this section describe how to include a sub-grid scale topography in your overland flow calculations;
- Overland Drainage *(p. 558)* - 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 *(p. 569)* - this section describes how to limit exchange across the land surface in areas that are compacted or paved.

30.1 Overland Flow Control

The overland flow calculations are controlled by parameters and some basic options.

30.1.1 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 the river. Only ponded water in excess of the detention storage will flow to the river. Also, flooding from the 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 *(p. 760)*.

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 *(p. 756)*.

30.1.2 Separated Flow areas

The Separated Flow Areas *(p. 279)* 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, the 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.

30.1.3 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 the MIKE+ 2D Overland module 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 the MIKE+ 2D Overland module. 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.

30.2 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.

30.2.1 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 the river model 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 Rivers *(p. 490)*.

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 diffi-

cult 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.

30.2.2 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 *(p. 523)*.

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 *(p. 197)* 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 the 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.

30.3 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 30.1.

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 *(p. 529)* 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.

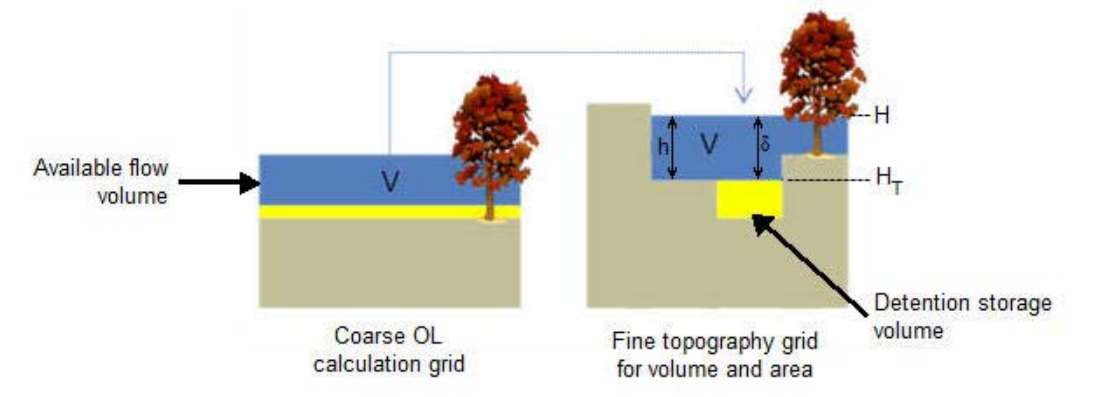


Figure 30.1 The constant volume from the coarse grid is transferred to the fine scale grid

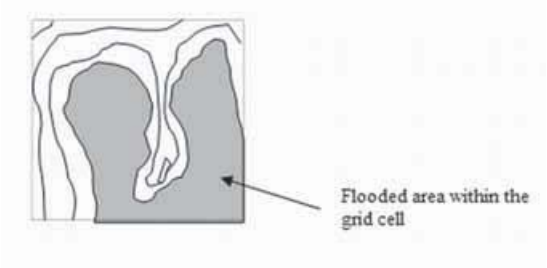


Figure 30.2 Flooded area is a function of the surface water level in the grid cell.

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.

30.3.1 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 *(p. 763)*, 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.

30.3.2 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. 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

$$DepPrec = precipitation \times dt \qquad (30.1)$$

The remaining part of the available water (ponding + precipitation on the ponded part) is scaled to an equivalent water depth in the ponded area:

$$DPonded = (OLDepth + DepPrec) \times (1-PAreaFrac) / PAreaFrac \qquad (30.2)$$

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.

30.3.3 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 *(p. 660)*.

30.3.4 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.

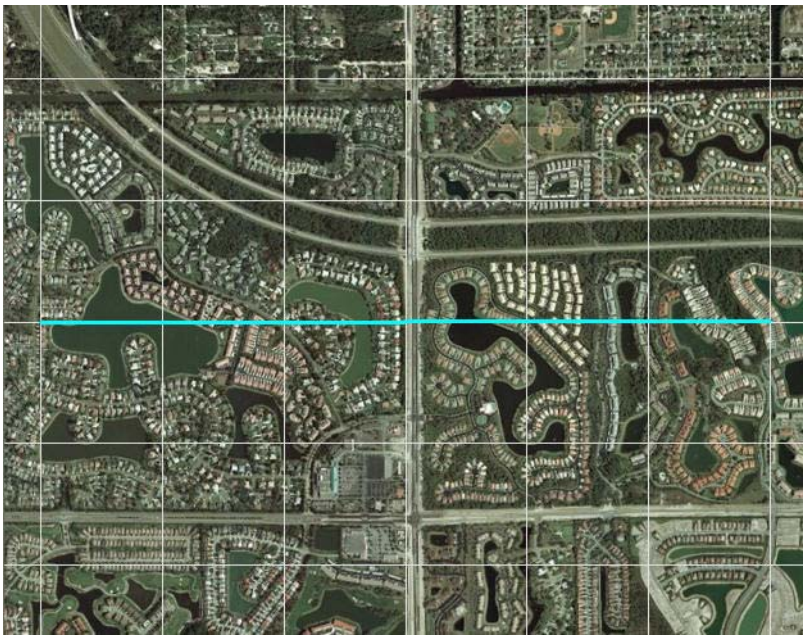


Figure 30.3 Aerial photo of part of the model area

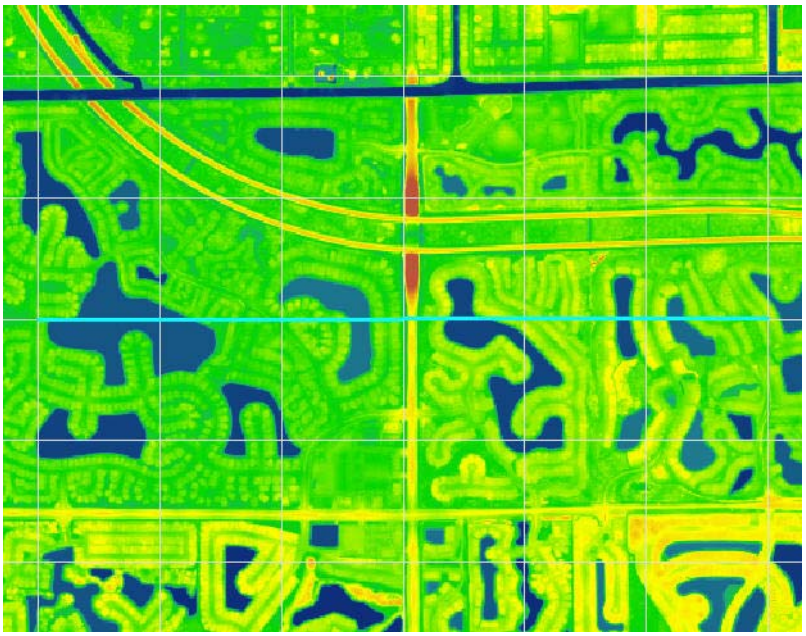


Figure 30.4 5-foot LIDAR data for part of the model area

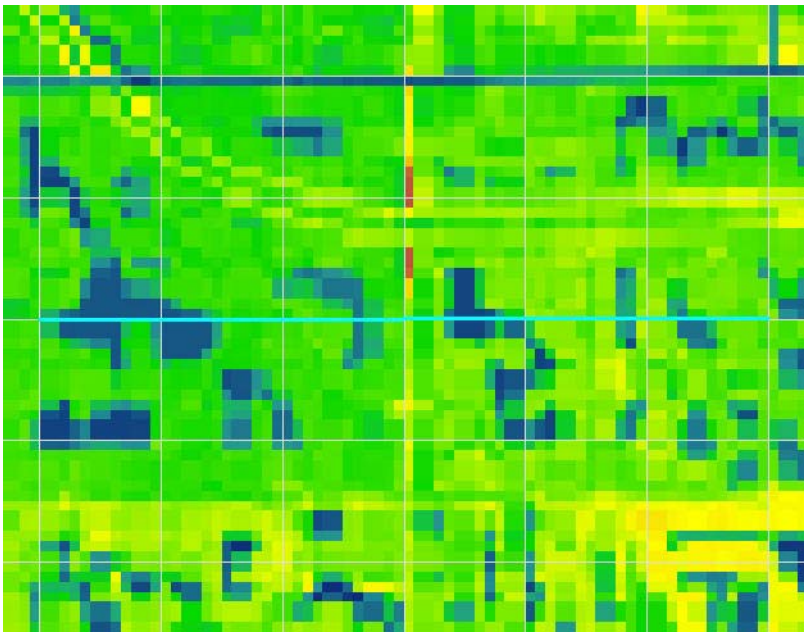


Figure 30.5 Interpolation of the 5-foot LIDAR data to the 125ft model grid

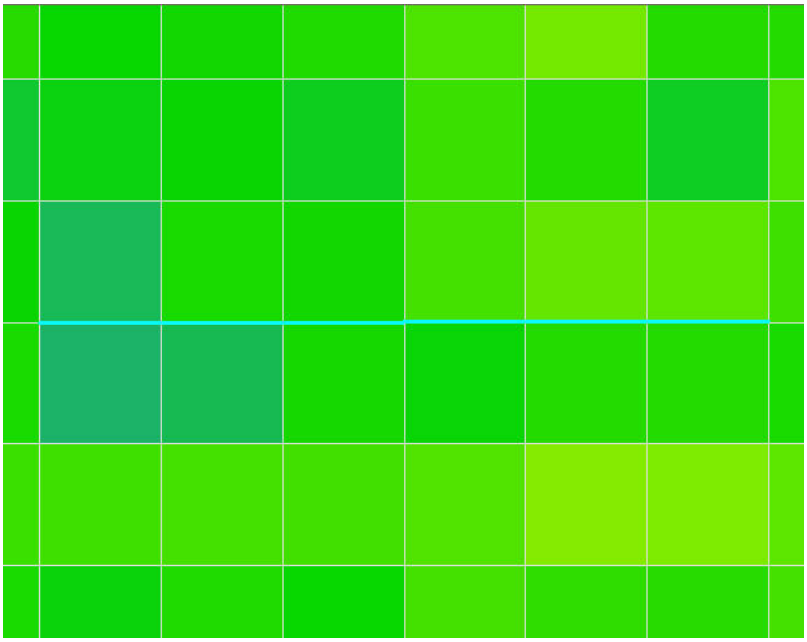


Figure 30.6 Interpolation of the 5-foot LIDAR data to the 1500ft model grid

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 30.7 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 30.8 shows that the multi-cell factor also impacts the run time for the river model. However, this impact is not linear, with the impact on MIKE 1D leveling off after a multi-cell factor of four.

The test model run time is dominated by MIKE 1D. 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 1D. However, as the multi-cell factor increases, the increased resolution of the inflows is not significant above a factor of about four.

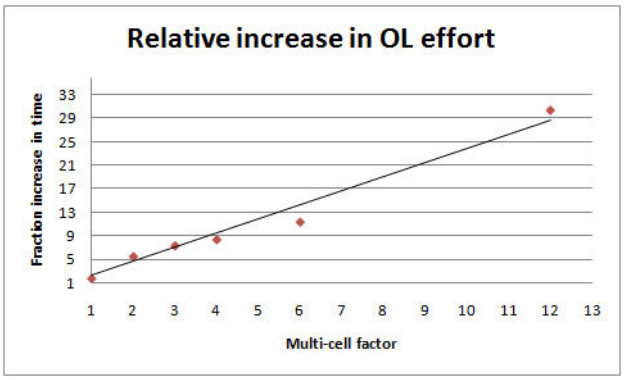


Figure 30.7 Increase in OL run time as a function of multi-cell factor

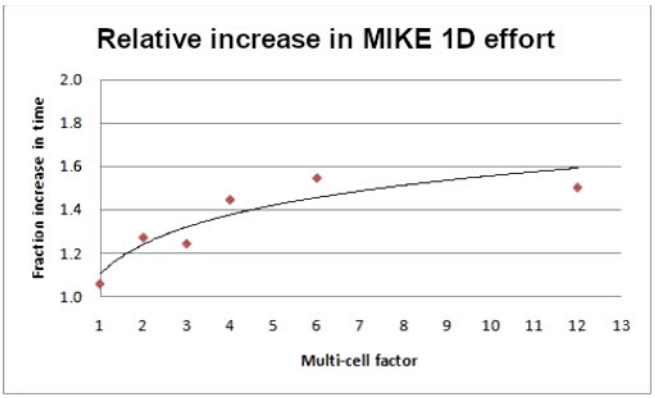


Figure 30.8 Increase in the river model run time as a function of the multi-cell factor.

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 the river model. 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.

30.3.5 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 the river, as well as UZ and SZ, depend only on the coarse scale grid elevations.

Overland flow exchange with rivers

Overland flow exchange with rivers 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 the 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 Rivers *(p. 490)*.

However, if you choose to modify the topography based on a bathymetry file, or the river cross-sections, then this information will be used when calculating the multi-cell elevations.

30.3.6 Setting up and evaluating the multi-grid OL

The multi-cell overland flow method is activated in the OL Computational Control Parameters *(p. 195)* 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.

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 30.9, 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.

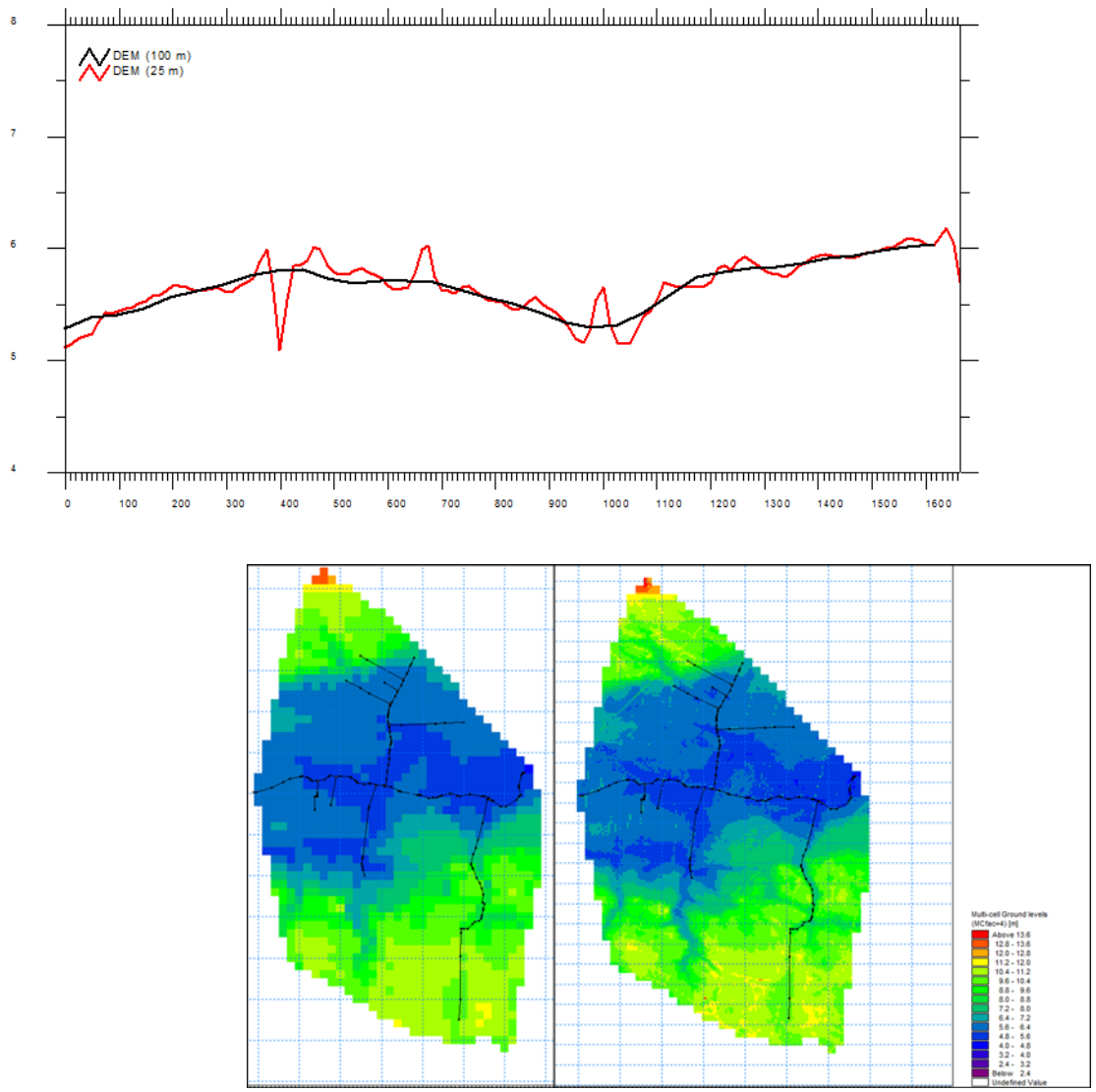


Figure 30.9 Example of preprocessed data - topography using a 100 meter resolution and a sub-scale factor of 4 (25 m sub-scale resolution)

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

30.4 Overland 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 the river model. 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 define the detailed natural drainage network. In urban catchments, the detail of the drainage network may be either unknown or impractical to define.

The overland drainage (OL drainage) option was introduced to alleviate some of these issues. Conceptually, the OL drainage is the similar to the SZ drain-

age 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.

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 located “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 outflow time constant. 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 flows from different cells are not summed together, and there is no physical drainage network to interact with while the water is in storage.

The overland drainage function has been designed to facilitate a flexible definition of storm water drainage in both natural and urbanized catchments. In particular, the overland drainage function supports the following features:

- In urban areas, paving and surface sealing (e.g. roof tops) limits infiltration and enhances runoff.
- Ponded water and rainfall are 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 flow 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.
- Infiltration and overland flow are calculated normally for ponded water that does not drain in the current time step.

Conceptually, the overland drainage function is shown in Figure 30.10.



Figure 30.10 Conceptualization of the overland drainage function

Calculation Order and Time Step

The OL drainage is calculated explicitly in a particular order with respect to the other hydrologic processes. However, the exact order depends on what UZ solver is being used. In all cases interception is calculated first. When using the Richards- or the gravity-solver, ET will be calculated before OL drainage, when using the 2-layer UZ solver ET will be calculated after drainage.

After the drainage calculation any remaining precipitation will be added to the remaining ponded water.

If there is any ponded water remaining on the cell after the drainage has been removed, then it will be subject to other processes in the following order:

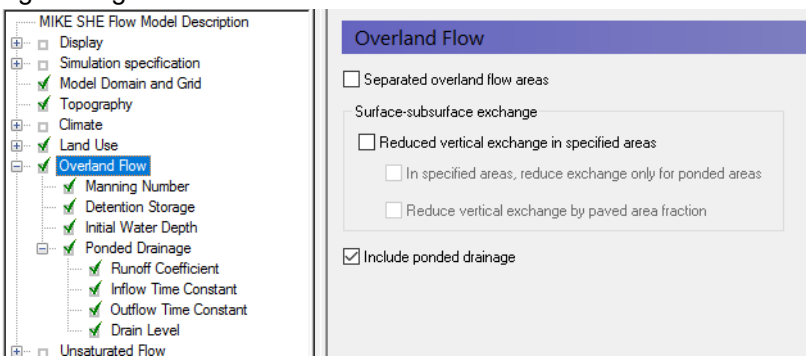
1. Infiltration to the unsaturated zone;
2. If the groundwater table is at/above ground: direct OL SZ exchange and
3. Finally, Lateral Overland Flow will be calculated to the adjacent cells.

The overland 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 water 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.

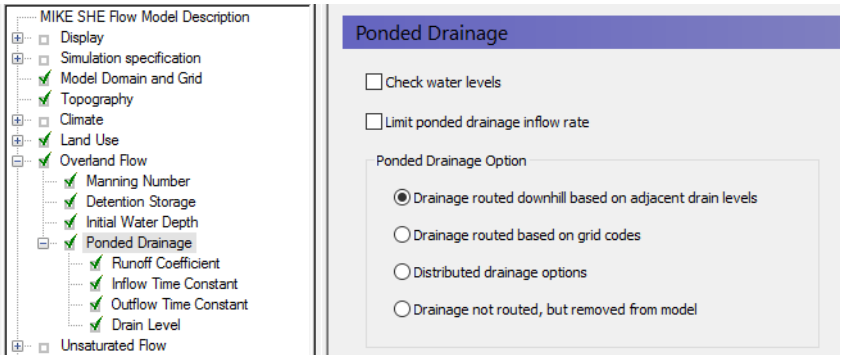
Note: When using OL Drainage together with UZ (Richards or gravity) the time spent for the drainage calculation will be accounted for as UZ time. Without UZ or when using the 2-layer method, the time spend for the drainage calculation adds to OL time.

30.4.1 Specifying Overland Drainage

Overland drainage is specified by selecting the Include overland drainage checkbox in the Overland Flow dialog. This will activate the Overland drainage dialog and the default additional data tree items.



The options for defining OL Drainage are selected in the overland drainage dialog:



There are two check box options for controlling the OL Drainage:

Check water levels

For a flow from the cell into the drain or if the destination of the OL drainage is a river or an internal depression, the target will have a water level. If you chose the option to check water levels and the target water level is higher than the ground level and higher than the level of detention storage (for flow into the drain) or higher than the drain level (for flow out of the drain) then it will inhibit flow, as it reduces the driving head difference. In other words, this option causes the destination water level to be considered when determining the drain base level zDr_0 (see equation (29.24) page 527).

The water level inside the drain is calculated based on the OL Drain level plus the volume of water in the OL Drain storage divided by the full cell area.

If the levels are not considered to represent the physical situation, then this option can be turned off. In this case the target water level will be ignored, and any calculated flow will be added to the target. There is no limit on the amount of water that can be stored inside the drain.

Limit inflow rate to OL Drain Storage

The flow rate into the drain is controlled by the drain inflow time constant. Usually, it is large enough to ensure that all available ponding and rainfall will be routed into the drain in the current time step. However, often the conveyance through the drain is restricted by structures 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.

The maximum inflow rate can be used to control the inflow to the OL drainage system. At every time step, the calculated drainage volume will be checked against the maximum drainage rate. Thus, ponded water will be retained on a grid cell and can be drained later at a controlled rate into the drainage system. While the water is ponded, it will be subject to infiltration, OL flow and

ET. The rate of leakage below the cell can be controlled by the Surface-Subsurface Leakage Coefficient *(p. 276)*.

The way the overland drainage is calculated, it is the UZ time step average of flow and does not necessarily represent the maximum drain flow. If the correct application of the limited drain inflow rate is important, the length of the UZ time step during a rain event can be controlled using the Precipitation-dependent time step control parameters (see 3.4.3 Controlling the Time Steps *(p. 83)*).

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.

30.4.2 Runoff Coefficient

The OL Drainage is calculated from **ponded water** and **net rainfall**. However, the order of operations in the three UZ solvers differs: When using the Richards and Gravity solvers interception and ET are calculated before the OL drainage (i.e. net rainfall = rainfall minus leaf interception minus ET). With the 2Layer Water Balance model interception is calculated first, followed by OL drainage and ET (i.e. net rainfall = rainfall minus leaf interception).

If the option "Instantaneous drain inflow from net rainfall" is not selected, then the drainage runoff coefficient will simply be a scaling factor applied after calculating drain inflow from net rainfall and ponding left over from the previous time step. Conceptually then it can be seen as an area fraction contributing to OL drainage. If at the end of the time step there is still ponded water in the cell, the drainage calculation will be applied to the remainder again in the next time step. Thus, the ponded water will eventually drain away.

However, if the option "Instantaneous drain inflow from net rainfall" is selected, then any ponded water will be neglected, and the specified fraction of net rainfall will be immediately added into the OL drainage storage.

The OL Drainage is also linked to the detention storage. By default, 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 ponded water to the OL Drainage network, then you can use an Extra Parameter found in OL Drainage Options *(p. 760)*.

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 the 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 the 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 time step length, and the calculated flow will be the UZ time step average. This could affect limiting the drain inflow rate (if specified), because the average value in a long time step could be significantly lower than the actual peak during that time. Also, the division of rainfall and ponding between drainage, infiltration and OL flow may be less accurate using long time steps. The length of the UZ time step during rainfall events can be controlled using the Precipitation-dependent time step control parameters (see 3.4.3 Controlling the Time Steps *(p. 83)*).

30.4.3 Time constants

The time constant (or leakage factor) 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. If the Outflow time constant is sufficiently large, the drain storage will not accumulate any water. In this case, inflow to the drain will be instantly discharged to the destination.

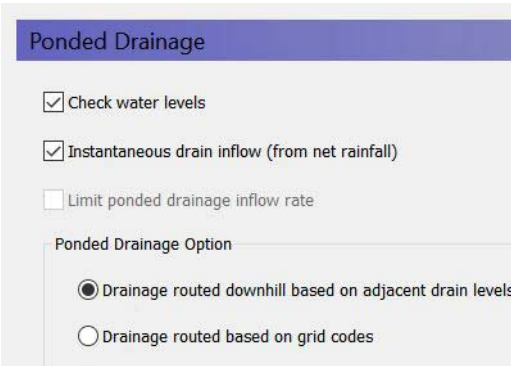
The default value for both time constants is 0.001 1/s. This is quite high and will normally drain nearly all ponded water from the cell. Since the output time constant by default is high, the default storage will be very low. This will normally result in a very sharp peak in your streams in response 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.

30.4.4 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 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. The command line tool MSheFillDepressions.exe in your MIKE installation directory can be used to fill depressions. To get help for that tool call it with the argument '--help'.



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 cell with a river link, 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 an 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

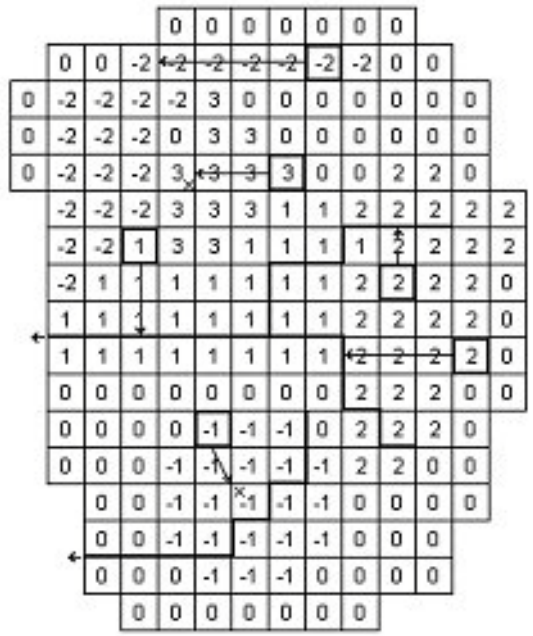
Choosing this method, adds the Option Distribution item to the data tree. With the Option Distribution, you can specify an integer grid code distribution that can be used to specify different drainage options in different areas of your model.

- **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 river branch and chainage. At the moment, this options requires the use of Extra Parameters *(p. 353)* and is described in OL Drainage to Specified River H-points *(p. 761)*.
- **Code = 4** - Drainage in grid cells with a value of 4 is routed to a specified sewer manhole. This option is described in the section in Working with Sewers - User Guide *(p. 509)*.

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.

30.4.5 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 River H-points *(p. 761)*, to generate a table of all the river link-cell references in the PP_Print.log file.

30.4.6 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 time varying parameters. Without this 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.

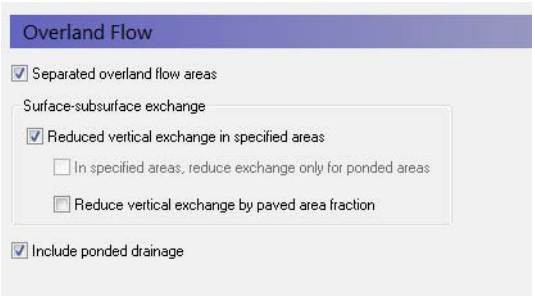
The preprocessor checks this and gives an error if you have specified

- 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).

30.5 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.

30.5.1 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.

30.5.2 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.

30.5.3 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.

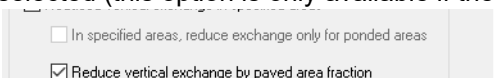
If you now open and existing model with paving, then the drainage will be “correctly” routed. To recreate the previous drainage reference system:

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.

30.5.4 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:

$$QinfAWghtd = Qinfntpd \times (1 - PAreaFrac) + Qinfpd \times (PAreaFrac) \qquad (30.3)$$

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).

30.5.5 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.

30.6 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.

31 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.

31.1 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,

$$h = z + \psi \tag{31.1}$$

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,

$$\Delta h = \frac{\partial h}{\partial z} \tag{31.2}$$

The volumetric flux is then obtained from Darcy's law:

$$q = -K(\theta)\frac{\partial h}{\partial z} \tag{31.3}$$

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:

$$\frac{\partial \theta}{\partial t} = -\frac{\partial q}{\partial z} - S(z) \tag{31.4}$$

where θ is the volumetric soil moisture and S is the root extraction sink term. Combining Eqs. (31.1), (31.3) and (31.4) yields

$$\frac{\partial \theta}{\partial t} = \frac{\partial}{\partial z}\left(K(\theta)\frac{\partial \psi}{\partial z}\right) + \frac{\partial K(\theta)}{\partial z} - S(z) \tag{31.5}$$

The dependent variables, θ and ψ, in Eq. (31.5) are related through the hydraulic conductivity function, $K(\theta)$, and the soil moisture retention curve, $\psi(\theta)$

Eq. (31.5) 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

$$C = \frac{\partial \theta}{\partial \psi} \tag{31.6}$$

which is the slope on the soil moisture retention curve, then the tension-based version of Eq. (31.5) is

$$C\frac{\partial \psi}{\partial t} = \frac{\partial}{\partial z}\left(K(\theta)\frac{\partial \psi}{\partial z}\right) + \frac{\partial K(\theta)}{\partial z} - S \tag{31.7}$$

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. (31.7) 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.

31.1.1 Numerical Solution of Richards Equation

MIKE SHE uses a fully implicit formulation in which the space derivatives of Eq. (31.7) 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:

$$q_{J+1/2}^{n+1} = -K_{J+1/2}^{n+1/2}\left(\frac{\psi_{J+1}^{n+1} - \psi_J^{n+1}}{\Delta Z_{J+1}} + 1\right) \tag{31.8}$$

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 31.1. Similar to Eq. (31.8) the discrete form of Eq. (31.1) gives

$$C_J^{n+1}\frac{\psi_J^{n+1}-\psi_J^n}{\Delta t}= \left|K_{J+\frac{1}{2}}^{n+\frac{1}{2}}\left(\frac{\psi_{J+1}^{n+1}-\psi_J^{n+1}}{\Delta Z_{J+1}}\right)\right. \\ \left.-K_{J-\frac{1}{2}}^{n+\frac{1}{2}}\left(\frac{\psi_J^{n+1}-\psi_{J-1}^{n+1}}{\Delta Z_J}\right)\right|\frac{1}{\frac{1}{2}(\Delta Z_{J+1}+\Delta Z_J)} \\ -S_J^{n+1} \tag{31.9}$$

The soil property *K* is centred in space using the arithmetic mean:

$$K_{J+\frac{1}{2}}^{n+\frac{1}{2}}=\frac{K_{J+1}^{n+\frac{1}{2}}+K_J^{n+\frac{1}{2}}}{2} \qquad K_{J-\frac{1}{2}}^{n+\frac{1}{2}}=\frac{K_J^{n+\frac{1}{2}}+K_{J-1}^{n+\frac{1}{2}}}{2} \tag{31.10}$$

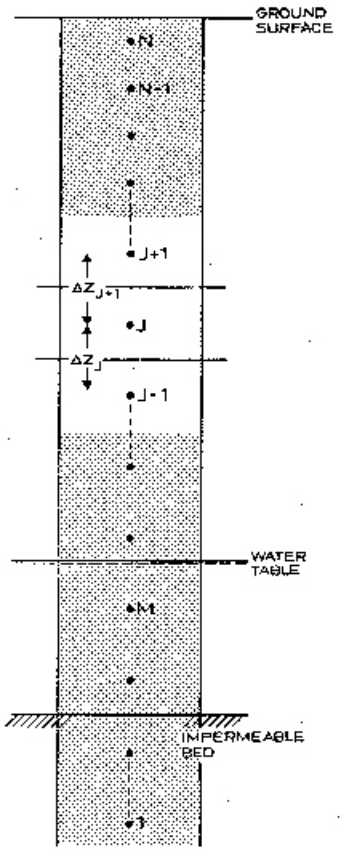


Figure 31.1 Vertical Discretisation in the Unsaturated Zone.

Eq. (31.9) 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 31.1, a system of *N-M+1* equations with *N-M+1* unknowns is obtained. The system of equations forms a tri-diagonal matrix:

$$\begin{bmatrix} B_N & A_N & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ G_{N-1} & B_{N-1} & A_{N-1} & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & G_J & B_J & A_J & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots & \dots & G_{M+1} & B_{M+1} & A_{M+1} \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & G_M & B_M \end{bmatrix} \begin{bmatrix} \psi_N \\ \psi_{N-1} \\ \dots \\ \dots \\ \psi_J \\ \dots \\ \dots \\ \psi_{M+1} \\ \psi_M \end{bmatrix} = \begin{bmatrix} D_N \\ D_{N-1} \\ \dots \\ \dots \\ D_J \\ \dots \\ \dots \\ D_{M+1} \\ D_M \end{bmatrix} \quad (31.11)$$

The *J*'th row in the matrix is

$$A_J^{n+1}\psi_{J-1} + B_J^{n+1}\psi_J + G_J^{n+1}\psi_{J+1} = D_J \tag{31.12}$$

where

$$\begin{aligned}
A_J &= -K_{J-\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_- \\
B_J &= C_J^{n+\frac{1}{2}} / \Delta t + K_{J+\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_+ + K_{J+\frac{1}{2}}^{n-\frac{1}{2}} / \Delta Z_- \\
G_J &= -K_{J+\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_+ \\
D_J &= C_J^{n+\frac{1}{2}}\psi_J^n / \Delta t + \frac{K_{J+\frac{1}{2}}^{n+\frac{1}{2}} - K_{J-\frac{1}{2}}^{n+\frac{1}{2}}}{\frac{1}{2}(\Delta Z_{J+1} + \Delta Z_J)} - S_J^{n+1} \\
\Delta Z_+ &= \frac{1}{2}\Delta Z_{J+1}(\Delta Z_{J+1} + \Delta Z_J) \\
\Delta Z_- &= \frac{1}{2}\Delta Z_J(\Delta Z_{J+1} + \Delta Z_J)
\end{aligned} \tag{31.13}$$

The solution to the matrix system Eq. (31.11) 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

$$\psi_J^{n+1} = E_{J+1}\psi_{J+1}^{n+1} + F_{J+1} \tag{31.14}$$

The E_{J+1} and F_{J+1} can be calculated by combining Eqs. (31.12) and (31.14) as follows:

$$E_{J+1} = \frac{-G_J}{A_J E_J + B_J} \qquad F_{J+1} = \frac{D_J - A_J F_J}{A_J E_J + B_J} \tag{31.15}$$

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. 31.13 and 31.15 for all nodes from bottom-to-top in a *E*, *F*-sweep.
2. ψ is then calculated from Eq. 31.14 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*

$$|\psi^i - \psi^{i-1}| < tolerance\ criteria \qquad if\ |\psi| < 0.5 \tag{31.16}$$

$$\left|\frac{\psi^i - \psi^{i-1}}{\psi^i}\right| < tolerance\ criteria \qquad if\ (|\psi| \geq 0.5) \tag{31.17}$$

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

$$C_J^{i+1} = \frac{\sum^{i} C_J}{\frac{1}{i} + C_J^0}{2} \tag{31.18}$$

$$K_J^{i+1} = \frac{\sum^{i} K_J}{\frac{1}{i} + K_J^0}{2} \tag{31.19}$$

31.1.2 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 *(p. 618)*).

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. (31.9) can be written for the top node N as

$$C_N^{n+\frac{1}{2}}\frac{\psi_N^{n+1}\psi_N^n}{\Delta t} = \left(-R - K_{n-\frac{1}{2}}^{n+\frac{1}{2}}\left(\frac{\psi_N^{n+1}-\psi_{N-1}^{n+1}}{\Delta Z_N}+1\right)\right)\frac{1}{0.5(\Delta Z_{N+1}+\Delta Z_n)} - S_N^{n+1} \tag{31.20}$$

where R is defined negative downwards.

Writing Eq. (31.20) in a similar form to Eq. (31.12) yields

$$A_N\,\psi_{N-1}^{n+1} + B_N\,\psi_N^{n+1} = D_N \tag{31.21}$$

where

$$\begin{aligned} A_N &= -K_{N-\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_- \\ B_N &= C_N^{n+\frac{1}{2}} / \Delta t + K_{N-\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_- \\ D_N &= C_N^{n+\frac{1}{2}}\frac{\psi_N^n}{\Delta t} + \frac{-R - K_{N-\frac{1}{2}}^{n+\frac{1}{2}}}{\frac{1}{2}(\Delta Z_{N+1}+\Delta Z_N)} - S_N^{n+1} \end{aligned} \tag{31.22}$$

If water is ponded on the ground surface, the first node is assumed saturated and the boundary condition simply becomes

$$\psi_N^{n+1} = \psi_N^n = \Delta Z_{N+1} \tag{31.23}$$

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. (31.22) consists of N-M equations. If node M is the first node below the water table, then

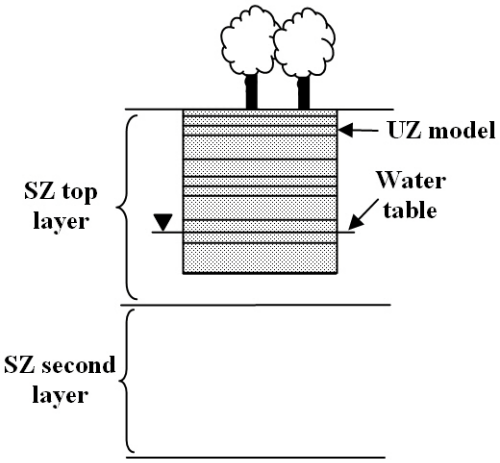
$$E_{M+1} = 0 \quad F_{M+1} = \psi_M^{n+1} = h \tag{31.24}$$

where h is the distance between the water table and node M. Noted that ψ_M is independent of ψ_{M+1} since $E_{M+1} = 0$.

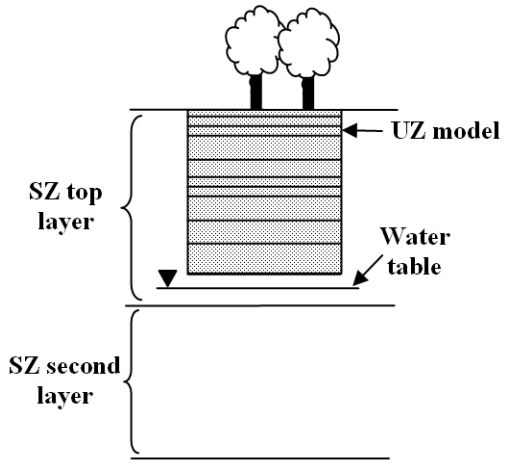
If the UZ model is not coupled to a SZ model, then the lower boundary is automatically converted from a pressure head boundary to a zero flux boundary (Q=0) if the water table falls below the impermeable bed (see Figure 31.1) and, at same time there is an upward flux in the lower part of the profile. 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:

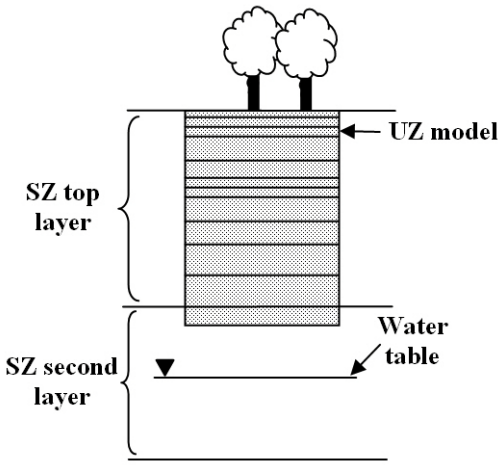
- If the UZ model intersects the water table in the top layer of the SZ model, then the lower boundary is a normal pressure boundary.



- 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.

31.1.3 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 31.2. 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.

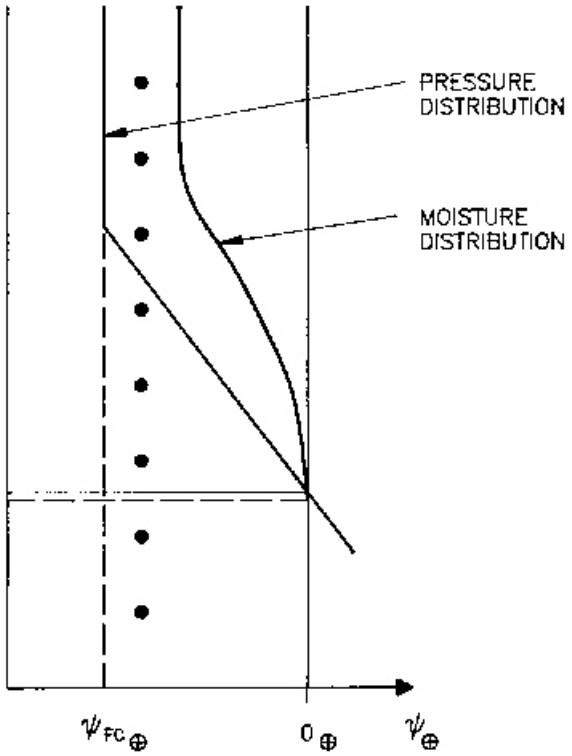


Figure 31.2 Illustration of initial soil moisture profile and pressure head profile.

31.1.4 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.

31.2 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,

$$h = z + \psi \tag{31.25}$$

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,

$$\Delta h = \frac{\partial z}{\partial z} = 1 \tag{31.26}$$

The volumetric flux is then obtained from Darcy's law:

$$q = -K(\theta)\frac{\partial h}{\partial z} = -K(\theta) \tag{31.27}$$

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:

$$\frac{\partial \theta}{\partial t} = -\frac{\partial q}{\partial z} - S(z) \tag{31.28}$$

where S is the root extraction sink term.

31.2.1 Solution method

In the Gravity Flow Module, Equation (31.28) 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 (31.27)).

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.

31.2.2 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 31.2. 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.

31.2.3 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.

31.3 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 *(p. 447)* section.

31.4 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

$$Q_{bypass} = P_{net} \; P_{frac} \sqrt{\alpha_{10} \beta_{50}} / \Delta t \tag{31.29}$$

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 31.3. 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})

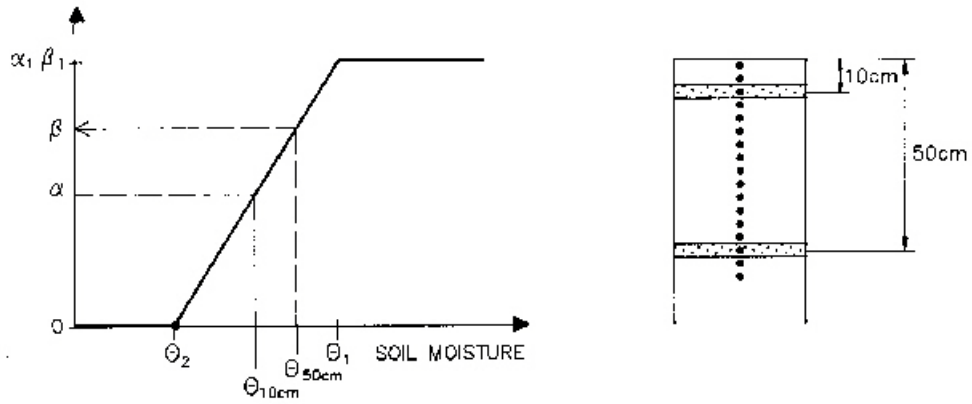


Figure 31.3 α and β as a function of the soil moisture content 10 cm and 50 cm below the ground surface, respectively.

31.4.1 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.

31.5 Full Macropore Flow

The following description is based on Christiansen et al. (2004).

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, ψ_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 lami-

nar and not influenced by capillarity, corresponding to gravitational flow. The vertical volumetric flux (positive upwards) q_{mp} is then given by

$$q_{mp} = -K_{mp}(\theta_{mp}) \tag{31.30}$$

where $K_{mp}(\theta_{mp})$ is the hydraulic conductivity of the macropores depending on the volumetric soil moisture content of the macropores, θ_{mp}. The continuity equation is expressed as

$$\frac{\partial \theta_{mp}}{\partial t} = -\frac{\partial q_{mp}}{\partial z} - S_{mp} \tag{31.31}$$

where S_{mp} is a sink term for water exchange with the surrounding matrix. Combining (31.30) and (31.31) yields the governing equation for the macropores

$$\frac{\partial \theta_{mp}}{\partial t} = \frac{\partial}{\partial z} K_{mp}(\theta_{mp}) - S_{mp} \tag{31.32}$$

The term S_{mp} becomes a source/sink term in Richards equation used in the matrix domain. This term is given by

$$S_{mp} = \beta_{mp} \cdot K(\theta_{matrix}) \cdot (\psi_{mp} - \psi_{matrix}) \tag{31.33}$$

where ψ_{mp} and ψ_{matrix} are the capillary heads in the macropores and in the matrix, respectively, and $K(\theta_{matrix})$ is the hydraulic conductivity in the matrix depending on the volumetric soil moisture content of the matrix, θ_{matrix}. The exchange flow from matrix to macropore is only considered when the capillary head in the matrix, ψ_{matrix}, exceeds the threshold pressure, ψ_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

$$\beta_{mp} = \frac{C_f}{d^2} \tag{31.34}$$

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).

Note: In MIKE SHE there are two values of β_{mp}, depending on the direction of the flow into or out of the micropore.

In the macropores, a simple power law function is assumed to represent the conductivity relation

$$K_{mp}(\theta_{mp}) = K_{s,mp} \cdot \left(\frac{\theta_{mp}}{\theta_{s,mp}} \right)^{n} \tag{31.35}$$

where $K_{s,mp}$ is the saturated hydraulic conductivity of the macropores, $\theta_{s,mp}$ 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 ψ_t, n and β_{mp} that in the present formulation are dependent on soil type. The capillary pressure in the macropores, ψ_{mp}, is supposed to vary linearly with the macropore moisture content θ_{mp} between zero (at $\theta_{mp} = \theta_{s,mp}$) and ψ_t (at $\theta_{mp} = 0$). 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.

31.5.1 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 relationship 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.

31.5.2 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).

31.6 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.

31.6.1 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.

31.6.2 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.

31.7 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 31.4a. 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 31.4a 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 31.4b).

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 *(p. 200)* dialogue.

31.7.1 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 flow rate across the lower boundary in UZ-time step *n* to *n+1* is

$$q_U^{n+1} = \frac{(W^{n+1} - W^n)}{\Delta t^{n+1}} + q_I + q_E \tag{31.36}$$

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:

$$E_{cum}^{n+1} = \left(q_u^{n+1} + \frac{q_G^{n_G} \Delta t_{n_G}}{\Delta t_{n_G+1}} + q_s^{n+1} \right) \Delta t + E_{cum}^{n} \tag{31.37}$$

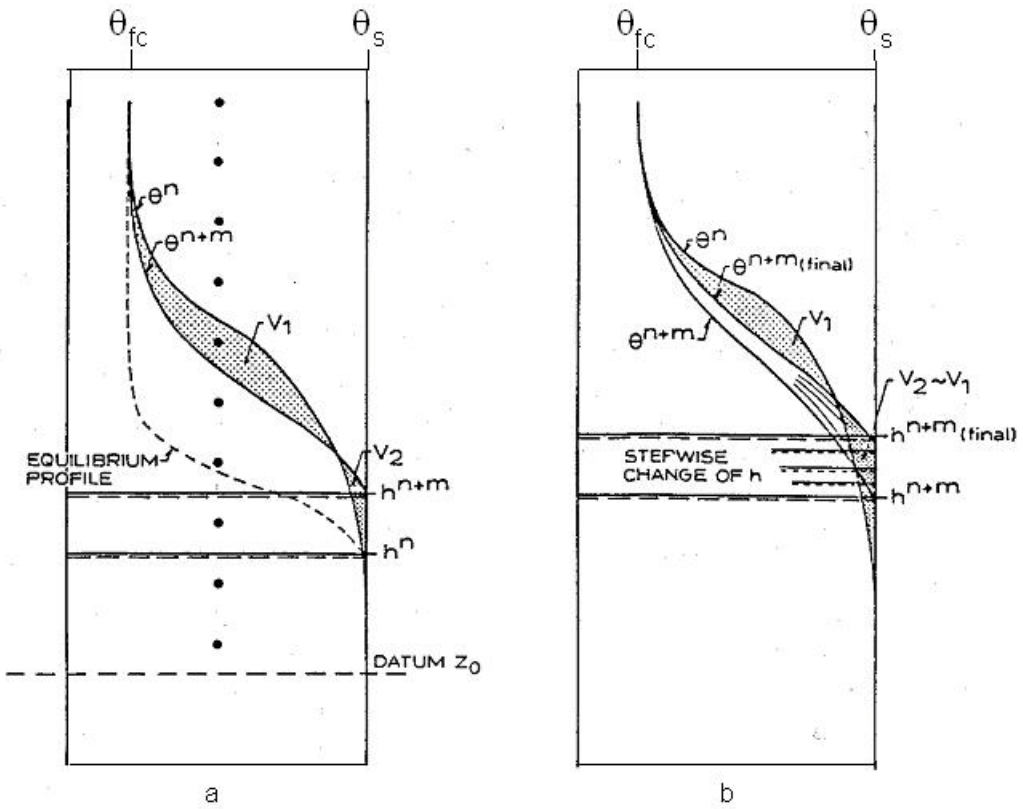


Figure 31.4 a) Soil moisture content at two times n and n+m without corrections, and b) Soil moisture content at time n+m before and after correction.

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.

$$q_u^{n+1*} = q_u^{n+1} - \frac{(h^{n*} - h^n)}{\Delta t} S_y \tag{31.38}$$

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. (31.38)).

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.

32 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).

32.1 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 32.1, which is from a 1963 USGS report, which itself references the figure from a publication from 1934.

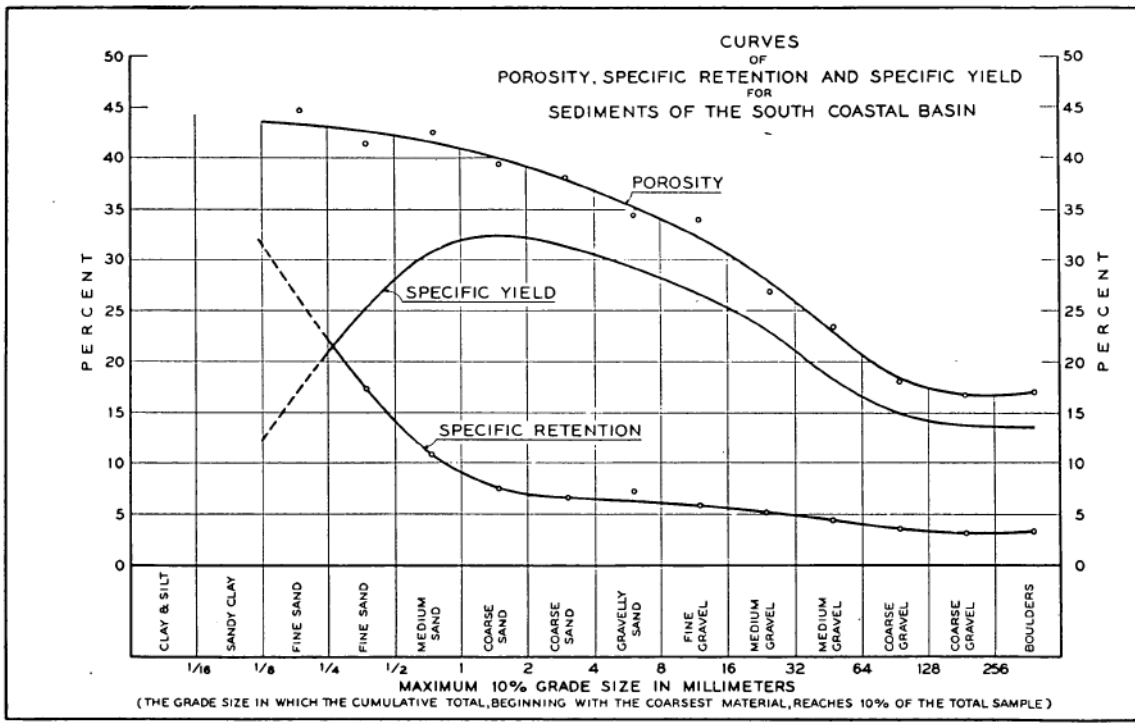


Figure 32.1 Relation between Specific Yield, Porosity and Field Capacity (from USGS OFR 63-59, Figure 3)

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

32.1.1 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 *(p. 415)*.

32.1.2 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 *(p. 418)*.

32.1.3 Typical soil parameters

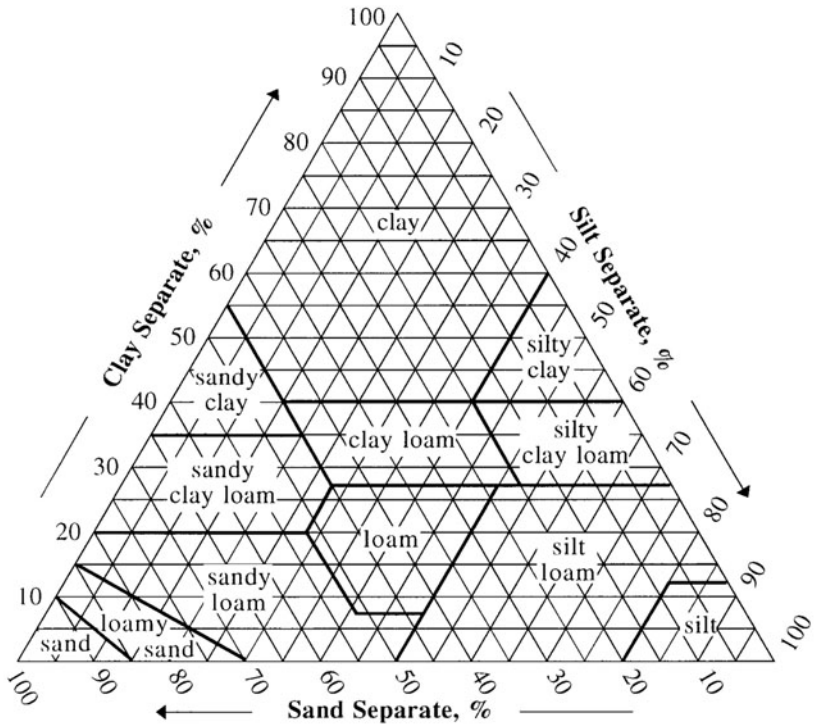


Figure 32.2 USDA Soil texture classification by grain size distribution

32.1.4 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 *(p. 413)*.

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.

32.2 Overview of the UZ User Interface

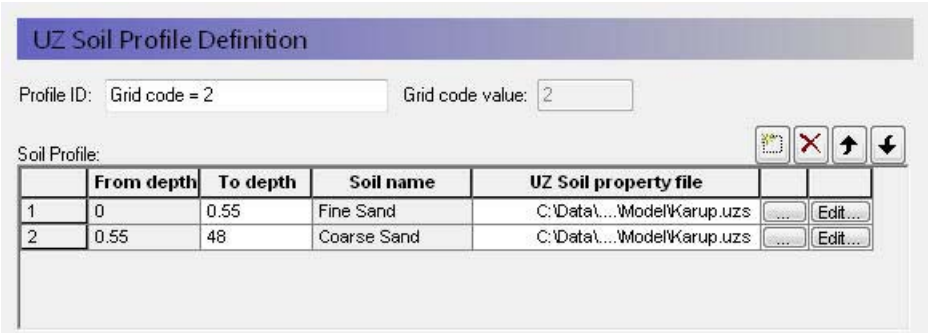
The setup of the UZ model can be divided into two steps: The definition of the soil profile and the definition of the vertical numerical grid. These two steps

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.

32.2.1 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 *(p. 413)*.

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.

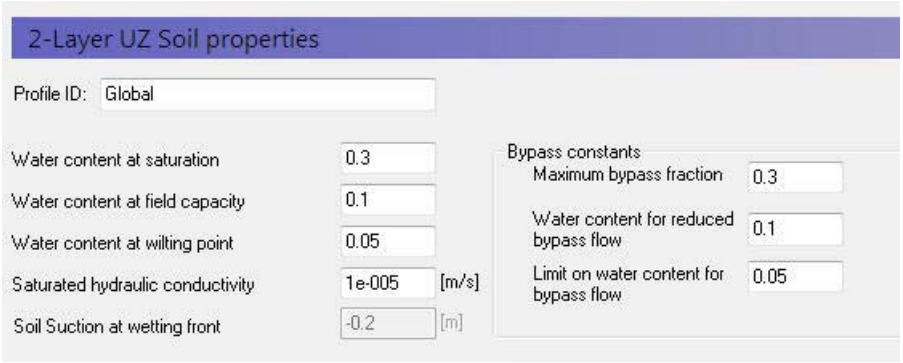
If you specify multiple soils in a soil profile, then the infiltration through the column will depend on the effective hydraulic conductivity function associated with water content, K(θ). This means that zones of high saturation may build up within the soil column. Depending on the situation, this may 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.

32.2.2 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.

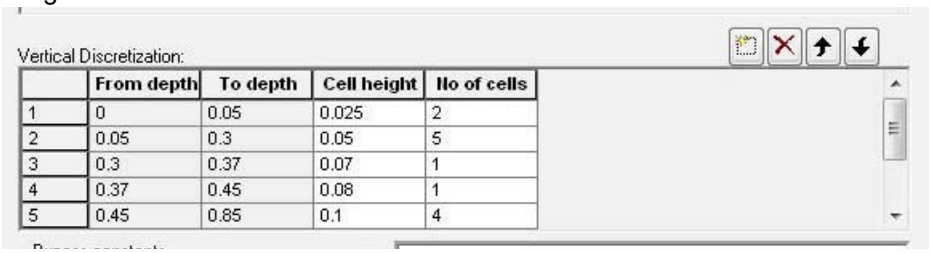
Note: The UZ model only connects to the top layer of the SZ model. All nodes below the water table are ignored. All nodes below the bottom of SZ layer 1 are also ignored. However, results will be output at all of these nodes, which can lead to unnecessarily large output files. There is also a small memory 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 32.3). 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.


Vertical Discretization:

	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.

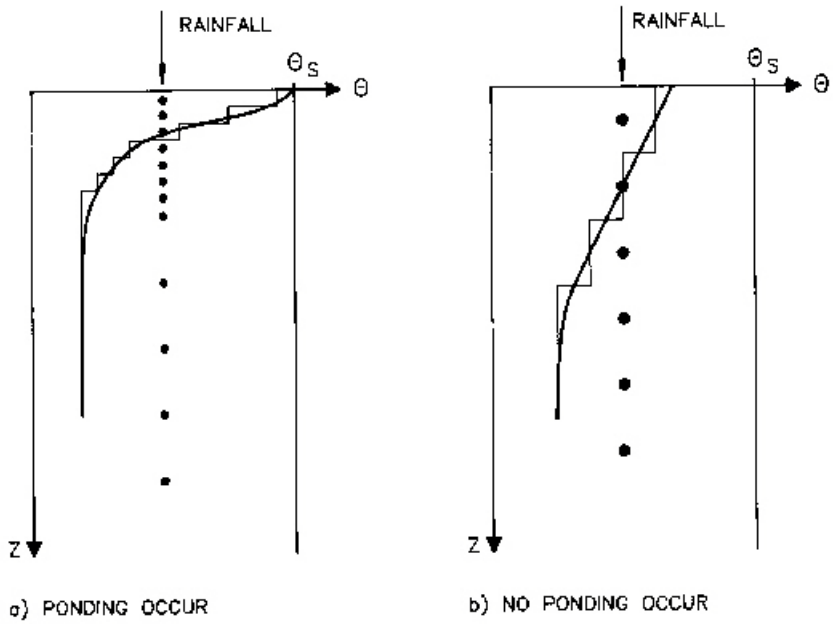


Figure 32.3 Examples of different increments in the soil profile and the resulting water content distribution of 1-3 cm for detailed studies and 3-5 cm for catchment studies. Further down in the profile larger increments can be chosen ranging from 10 cm to 30 cm.

32.2.3 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 *(p. 787)*.

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 the location and value of errors due to a poor setup of the unsaturated zone. Use the "Map output: Unsat. Zone Error" water balance type to produce a map of the error.

To further investigate problems with the UZ solver like water balance errors or excessive time step reductions there are two relevant grid output items:

1. **UZ time step reduction count**: After execution of the 1D UZ solver the resulting water balance will be checked. If the remaining error is too large (see "Maximum water balance error in one node" in UZ Computational Control Parameters *(p. 200)*) the solution will be discarded and restarted with a shorter time step. This could happen a couple of times in a row until an acceptable solution has been found. If there are problems with the setup though time steps can become too short, leading to an excessive run time. This output item can be used to identify the cells leading to the time step reductions. Note that the solver will stop processing new columns after the first column requires a restart. Due to the parallelization there can be a couple of cells signalling a restart at the same time, but not many more than the number of cores used, and usually not all columns will have been touched yet. So there is no strict guarantee that the highest counts in the output contain the most problematic cells, but the probability is high.
2. **UZ iteration count**: This item contains the number of iterations leading to the accepted solution, averaged over the storing time step. If the time step has to be repeated with a shorter length, the iterations in the discarded time step will not be counted.

32.3 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.

32.3.1 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 *(p. 583)*.

32.4 Options for the UZ model

32.4.1 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) *(p. 590)*.

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) *(p. 590)*.

32.4.2 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 initi-

ated 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.

32.4.3 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.

32.4.4 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 ground-

water 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.

33 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.

33.1 3D Finite Difference Method

The governing flow equation for three-dimensional saturated flow in saturated porous media is:

$$\frac{\partial}{\partial x}\left(K_{xx}\frac{\partial h}{\partial x}\right)+\frac{\partial}{\partial y}\left(K_{yy}\frac{\partial h}{\partial y}\right)+\frac{\partial}{\partial z}\left(K_{zz}\frac{\partial h}{\partial z}\right)-Q=S\frac{\partial h}{\partial t} \tag{33.1}$$

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 is the 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.

33.1.1 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

$$Q = \Delta h C \tag{33.2}$$

where *Dh* is the piezometric head difference and *C* is the conductance.

The horizontal conductance in Eq. (33.2) 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

$$C_{i-\frac{1}{2}} = \frac{KH_{i-1,j,k}\ KH_{i,j,k}\ (\Delta z_{i-1,j,k} + \Delta z_{i,j,k})}{(KH_{i-1,j,k} + KH_{i,j,k})} \tag{33.3}$$

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,

$$C_v = \frac{\Delta x^2}{\frac{\Delta z_k}{2K_{z,k}} + \frac{\Delta z_{k+1}}{2K_{z,k+1}}} \tag{33.4}$$

where *Dz* is the layer thickness

Dewatering conditions

Consider the situation in Figure 33.1, where the cell below becomes dewatered.

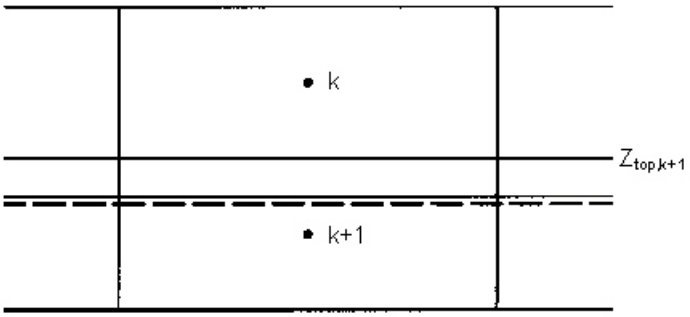


Figure 33.1 Dewatering conditions in a lower cell.

The actual flow between cell *k* and *k+1* is

$$q_{k+½} = Cv_{k+½}(z_{top,\,k+1} - h_k) \tag{33.5}$$

In the present solution scheme the flow will be computed as

$$q_{k+½} = Cv_{k+½}(h_{k+1} - h_k) \tag{33.6}$$

Subtracting Eqs.(33.5) from (33.6) gives the correction term

$$q_c = Cv_{k+½}(h_{k+1} - z_{top,\,k+1}) \tag{33.7}$$

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.

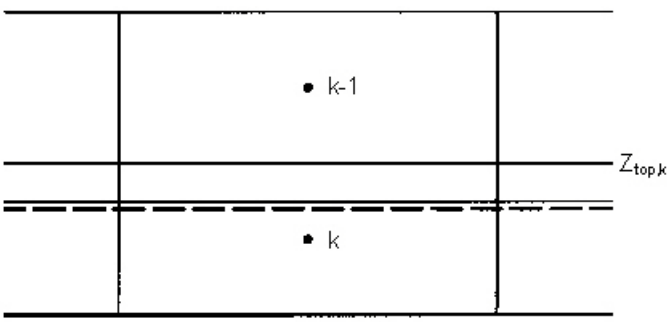


Figure 33.2 Dewatering conditions in the cell above

Thus, from Figure 33.2, the flow from cell *k-1* to *k* is

$$q_{k-½} = Cv_{k-½}(h_{k-1} - z_{top,k}) \tag{33.8}$$

where, again the computed flow is

$$q_{k-½} = Cv_{k-½}(h_{k-1} - h_k) \tag{33.9}$$

Subtracting Eqs. (33.8) from (33.9) gives the correction term

$$q_c = Cv_{k-½}(z_{top,k} - h_k) \tag{33.10}$$

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

$$\frac{\Delta w}{\Delta t} = \frac{S2(h^n - z_{top}) + S1(z_{top} - h^{n-1})}{\Delta t} \tag{33.11}$$

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

$$S = \Delta x^2 \Delta z S_{art} \tag{33.12}$$

and for unconfined cells the storage capacity is given as

$$S = \Delta x^2 S_{free} \tag{33.13}$$

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.

33.1.2 The Successive Overrelaxation (SOR) Solver

Numerical formulation

Equation (33.1) 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:

$$\Sigma Q_{ij}^{n+1} = \Sigma q_z^{n+1} + \Sigma q_x^{n+1} + RH_i \Delta x^2 \tag{33.14}$$

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 33.3 and Figure 33.4 for a description of the geometric relationships between the cells.

The horizontal flow components in Eq. (33.14) are given by

$$q_x^{n+1} = C\Delta h^{n+1} \tag{33.15}$$

where C is the horizontal conductance between any of the adjacent nodes in the horizontal directions.

The horizontal conductance in Eq. (33.15) 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

$$C_{i-\frac{1}{2}} = \frac{KH_{i-1,j,k}\ KH_{i,j,k}\ (\Delta z_{i-1,j,k} + \Delta z_{i,j,k})}{(KH_{i-1,j,k} + KH_{i,j,k})} \tag{33.16}$$

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. (33.14) are given by

$$q_z^{n+1} = \frac{K_v \Delta x^2 \Delta h^{n+1}}{\Delta z_{i,j,k} + \Delta z_{i,j,k+1}} \tag{33.17}$$

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 33.1 and Table 33.2.

Table 33.1 For the case when $K_k < K_{k+1}$.

		Layer *k*	
		confined	**unconfined**
Layer k+1	**confined**	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{\Delta z_{k+1}}{K_{k+1}}}$	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{\Delta z_{k+1}}{K_{k+1}} + \frac{z_{k+1} - h_k}{K_k}}$
	unconfined	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{h_{k+1} - z_{k+1}}{K_{k+1}}}$	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{h_{k+1} - z_{k+1}}{K_{k+1}} + \frac{z_{k+1} - h_k}{K_k}}$
		$\Delta h = h_k - h_{k+1}$	

Table 33.2 For the case when $K_k > K_{k+1}$.

		Layer *k*	
		confined	**unconfined**
Layer k+1	**confined**	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{\Delta z_{k+1}}{K_{k+1}}}$ $\Delta h = h_k - h_{k+1}$	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{\Delta z_{k+1}}{K_{k+1}}}$ $\Delta h = z_{k+1} - h_{k+1}$
	unconfined	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{h_{k+1} - z_{k+1}}{K_{k+1}}}$ $\Delta h = h_k - h_{k+1}$	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{h_{k+1} - z_{k+1}}{K_{k+1}}}$ $\Delta h = z_{k+1} - h_{k+1}$

The transient flow equation yields the finite difference expression

$$S\frac{h_i^{n+1} - h_i^n}{\Delta t} = \Sigma Q_{ij}^{n+1} \tag{33.18}$$

where S is the storage coefficient and Δt is the time step. Eq. (33.18) 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).

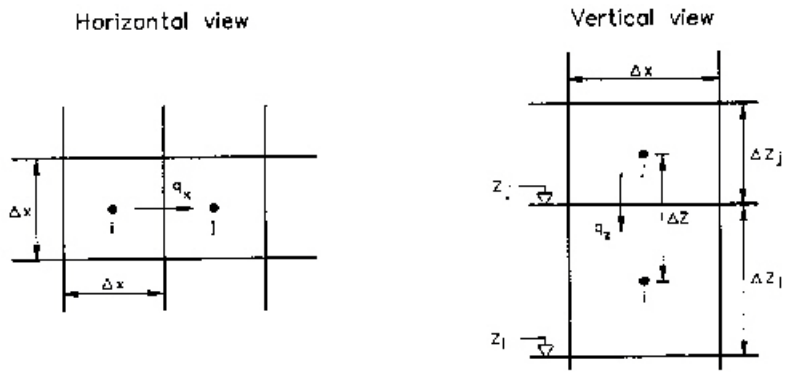


Figure 33.3 Spatial discretisation.

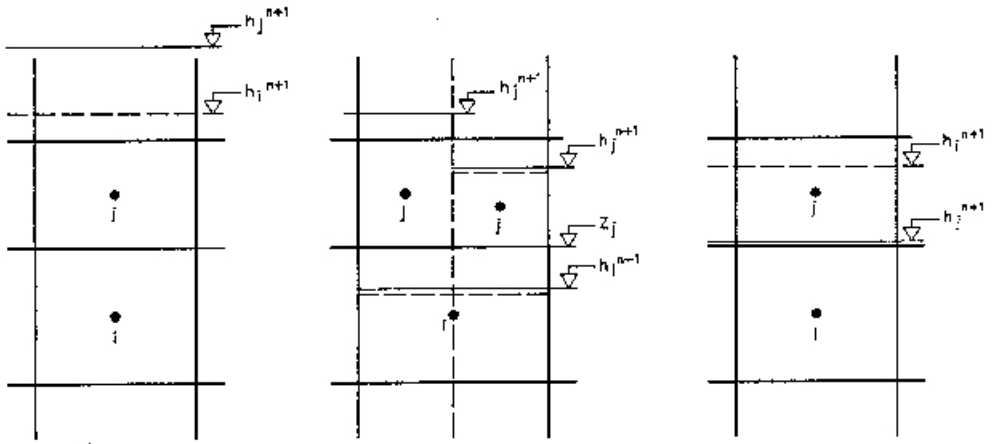


Figure 33.4 Types of vertical flow condition, a) confined conditions in nodes i and j, b) unconfined condition in node i, c) unconfined in nodes i and j, d) dry conditions in node j and confined conditions in node i.

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 33.5 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 33.5. The minimum number of iterations corresponds to the optimal value of *w*.

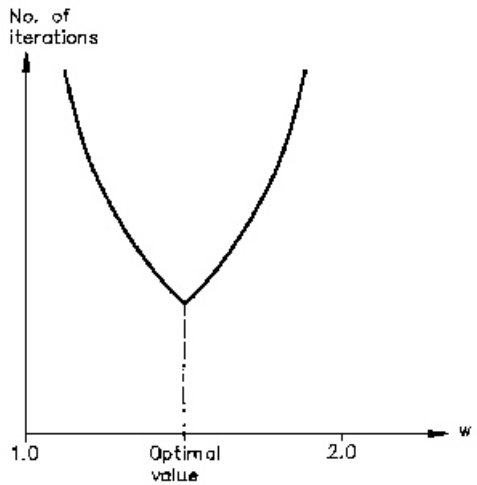


Figure 33.5 Empirically relationship between the relaxation coefficient w and number of iterations for a given model.

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.

33.1.3 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 the river model if the 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).

33.1.4 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 interpolated 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, global specification

The total discharge of the actual time step (constant or time-varying) is distributed over the internal cells weighted by horizontal conductivity and full or saturated thickness (inflow/out-flow, respectively).

Inflow (positive discharge):

$$\text{Cell Discharge } Q_i = Q_{tot} \cdot \frac{\text{Conductivity}_i \cdot \text{layer thickness}_i}{\sum \text{Conductivity}_i \cdot \text{layer thickness}_i} \tag{33.19}$$

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.

Flux boundary, distributed specification

The discharge is specified for each cell, separately for flow along the x- and the y-axis. It is applied to the adjacent internal cells as-is. If there are two internal cells along one axis then the in-/outflow will be distributed equally, without taking saturated thickness or conductivity into account. If there is no adjacent input cell along one axis then the specified flow will be disregarded with no warning. Flow can occur between an internal cell and all its adjacent boundary cells

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:

$$Q = \text{Gradient} \cdot \text{Conductivity} \cdot \text{CellSize} \cdot \text{Saturated thickness}$$

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.

33.1.5 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

$$C_{½} = \frac{\Delta x^2}{\frac{\Delta z}{2K_z}} \tag{33.20}$$

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

$$C_{½} = \frac{\Delta x^2}{\frac{\Delta z}{2K_z} + \frac{1}{K_{leak}}} \tag{33.21}$$

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.

33.1.6 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 *(p. 65)* for further details about routing of drainage water.

There are also some additional Extra Parameter options for drainage routing:

- SZ Drainage to Specified River H-points *(p. 770)*
- SZ Drainage Downstream Water Level Check *(p. 773)*

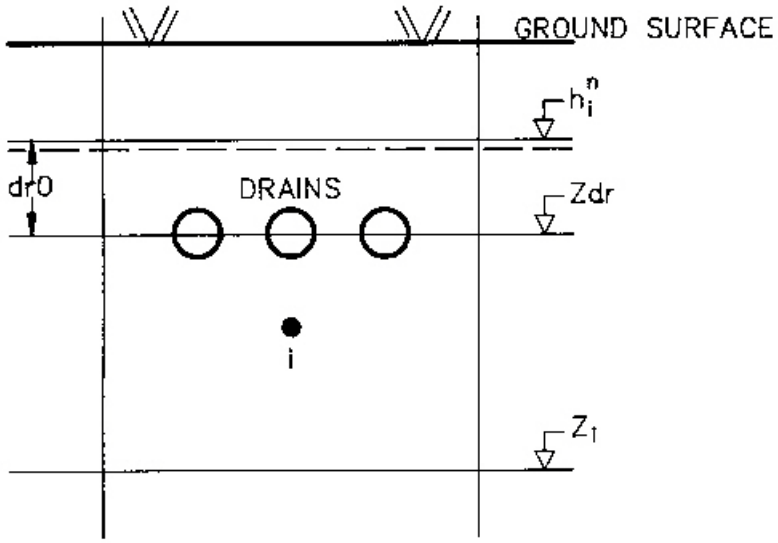


Figure 33.6 Schematic presentation of drains in the drainage flow computations.

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

$$q = (h - Z_{dr})C_{dr} \tag{33.22}$$

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

$$\Delta S = (Q_{dr} + \Sigma q)\Delta t \tag{33.23}$$

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

$$\Delta S = (d_0 - d_t)S_y \tag{33.24}$$

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,

$$Q_{dr} = C_{dr} \cdot \frac{d_0 + d_t}{2} \tag{33.25}$$

where C_{dr} is the drain conductance or time constant.

Substituting (33.24) and (33.25) into (33.23) and rearranging, the water depth at the end of a time step, d_t, can be calculated by

$$d_t = \frac{\left[d_0\left(S_y - \frac{C_{dr}\Delta t}{2}\right) - \Sigma q \Delta t\right]}{S_y + \frac{C_{dr}\Delta t}{2}} \tag{33.26}$$

From which the new water table elevation, h_t, at the end of the time step can be calculated by

$$h_t = Z_{dr} + d_t \tag{33.27}$$

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.

33.1.7 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.

33.1.8 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

$$C_h = \frac{1}{\frac{1}{2 \cdot k_1 \cdot dz_1} + \frac{1}{k_{leak} \cdot dx \cdot dz^*} + \frac{1}{2 \cdot k_2 \cdot dz_2}} \tag{33.28}$$

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

$$C_v = \frac{dx \cdot dx}{\frac{dz_1}{2 \cdot k_1} + \frac{1}{k_{leak}} + \frac{dz_2}{2 \cdot k_2}} \tag{33.29}$$

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].

33.2 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.

33.2.1 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:

$$S = kq \tag{33.30}$$

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

$$\frac{dS}{dt} = I - q \tag{33.31}$$

where t is time, and I is the inflow to the reservoir.

Combining equation (33.8) and (33.9) yields a first-order, linear differential equation which can be solved explicitly

$$\frac{dq}{dt} + \frac{1}{k}q(t) = \frac{1}{k}I(t) \tag{33.32}$$

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

$$q(t+dt) = q_t e^{-dt/k} + I(1 - e^{-dt/k}) \tag{33.33}$$

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

$$S = k_p q_p = k_o q_o + h_{thresh} \tag{33.34}$$

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 (33.34) and (33.32) 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).

$$q_p = q_{p_t} e^{\frac{k_o + k_p}{k_p k_o} dt} + \frac{k_o}{k_p + k_o}\left(I + \frac{h_{thresh}}{k_o}\right)\left(1 - e^{\frac{k_o + k_p}{k_p k_o} dt}\right) \tag{33.35}$$

$$q_o = \frac{k_p q_p - h_{thresh}}{k_o} \tag{33.36}$$

33.2.2 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 33.8, 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 contact with a river is unlikely to be present everywhere. In this case, more than one reservoir can be specified.

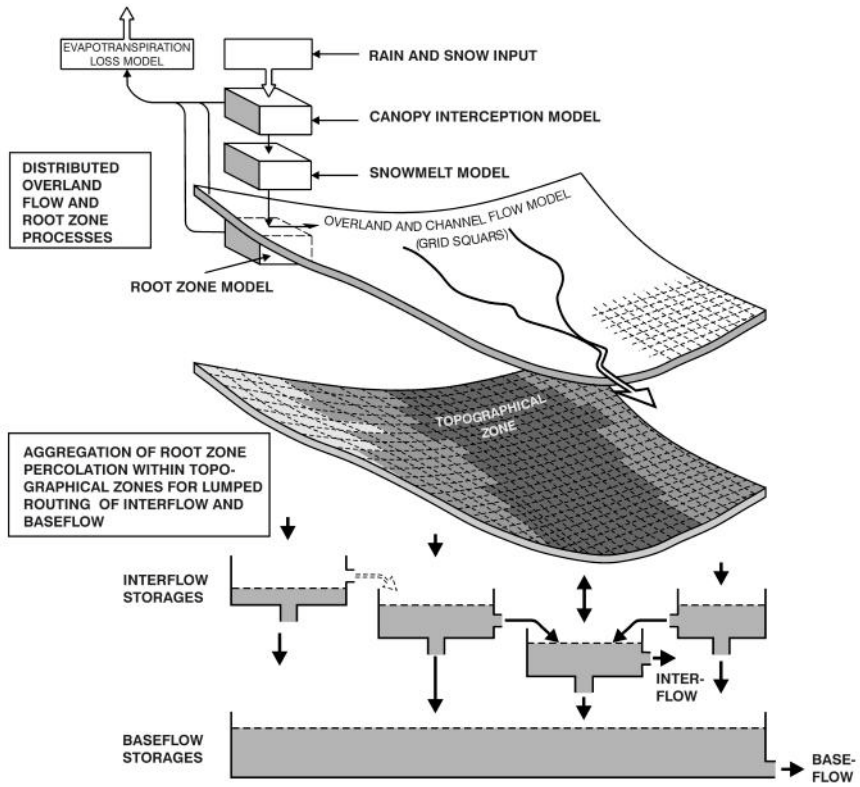


Figure 33.7 Model Structure for MIKE SHE with the linear reservoir module for the saturated zone

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.

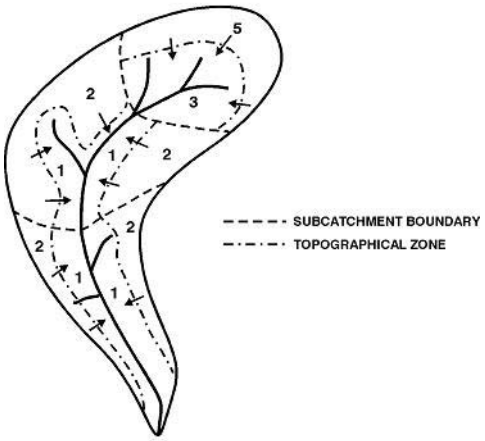


Figure 33.8 Example of Desegregation of a Catchment into Sub-catchments and Topographical Zones

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 overland 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.

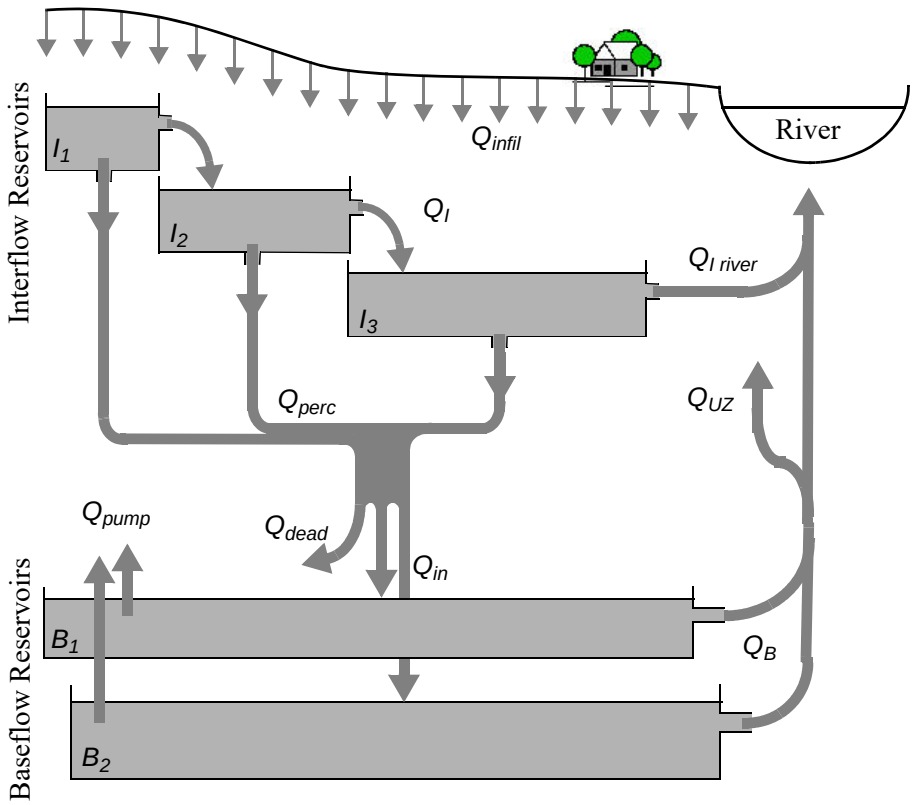


Figure 33.9 Schematic flow diagram for the Subcatchment-based, linear reservoir flow module

33.2.3 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.

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 33.8 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 33.9 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

33.2.4 Calculation of Interflow

Each Interflow Reservoir is treated as A Single Linear Reservoir with Two Outlets *(p. 640)*. Thus, from Eq. 33.30, if the water level in the linear reservoir is above the threshold water level

$$q_I = (h - h_{thresh})/k_i \qquad (33.37)$$

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

$$q_{perc} = h/k_p \qquad (33.38)$$

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 33.37 and 33.38 with the continuity equation

$$\frac{dh}{dt} = \frac{(q_{in} - q_l - q_{perc})}{S_y} \qquad (33.39)$$

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_l and q_{perc} (linear reservoir with two outlets)

$$h_t = h_{t_0} e^{\frac{-(k_i + k_p)dt}{k_i k_p S_y}} + \frac{k_i k_p}{(k_i + k_p)}\left(q_{in} + \frac{h_{thresh}}{k_i}\right)\left(1 - e^{\frac{-(k_i + k_p)dt}{k_i k_p S_y}}\right) \qquad (33.40)$$

In the case where the water level is below the threshold, the formulation for a linear reservoir with one outlet applies, which yields

$$h_t = h_{t_0} e^{\frac{-dt}{k_p S_y}} + k_p q_{in}\left(1 - e^{\frac{-dt}{k_p S_y}}\right) \qquad (33.41)$$

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,

$$q_{out} = -(h_t - h_{t0})S_y/dt + q_{in} \tag{33.42}$$

$$q_I = \frac{k_p}{k_i + k_p}(q_{out} - h_{thresh}/k_p) \tag{33.43}$$

$$q_{perc} = q_{out} - q_I \tag{33.44}$$

and for the single outlet case (no interflow),

$$q_{perc} = -(h_t - h_{t0})S_y/dt + q_{in} \tag{33.45}$$

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.

33.2.5 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

$$q_v = \sum_{i=1}^{N} q_{perc_i} \frac{A_{interflow_i}}{A_{baseflow}}(1.0 - DZ_{frac}) \tag{33.46}$$

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

$$q_{dead} = \sum_{i=1}^{N} q_{perc_i} \frac{A_{interflow}}{A_{baseflow}} DZ_{frac} \tag{33.47}$$

33.2.6 Calculation of Baseflow

From Eq. 33.30, if the water level in the linear reservoir is above the threshold water level

$$q_B = \frac{(h - h_{thresh})}{k_b} \tag{33.48}$$

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. (33.39), for each Baseflow Reservoir

$$\frac{dh}{dt} = \frac{(q_v - q_B - q_{pump})}{S_y} \tag{33.49}$$

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. 639)*. 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),

$$h_t = h_{t_0} e^{\frac{-dt}{k_b S_y}} + k_b (q_v - q_{pump}) \left(1 - e^{\frac{-dt}{k_b S_y}} \right) \tag{33.50}$$

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,

$$h_t = h_{t_0} + \frac{(q_v - q_{pump}) \cdot dt}{S_y} \tag{33.51}$$

If there is no pumping and no baseflow out, then the expression for the water level in the reservoir simply becomes

$$h_t = h_{t_0} + \frac{q_v \cdot dt}{S_y} \tag{33.52}$$

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.

33.2.7 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,

$$FMD = \int_0^{\text{Rootdepth}} (\theta_{FC} - \theta) dz = \sum_{i=1}^{n} (\theta_{FC} - \theta_i) \Delta z_i \tag{33.53}$$

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 baseflow:

$$S_{available} = Q_{B1} S_{y1} \cdot UZ_{frac1} \cdot dt + Q_{B2} S_{y2} \cdot UZ_{frac2} \cdot dt \tag{33.54}$$

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

$$S_{feedback_i} = \theta_i + \frac{S_{available}}{FMD}(\theta_{FC} - \theta_i) \tag{33.55}$$

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 33.9.

33.2.8 Coupling to Rivers

The discharge from the lowest Interflow Reservoir in each subcatchment is distributed evenly over the river model 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 river branches and chainages.

If the 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 riverbed conductivity specified.

Note: The lowest linear reservoir discharges to the river network. However, if the lowest linear reservoir is not connected to a river branch, then the linear reservoir will still discharge. The only difference is that no water will be added to the river. This situation can arise when there are no river branches in small subcatchments. A warning will be written to the log file if a subcatchment is not connected to a river branch.

33.2.9 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 down-stream 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.

34 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.

34.1 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

$$S_y = \frac{S_{y1} \cdot z_1 + S_{y2} \cdot z_2 + S_{y3} \cdot z_3}{z_1 + z_2 + z_3} \quad (34.1)$$

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

$$K_z = \frac{z_1 + z_2 + z_3}{\frac{z_1}{K_{z1}} + \frac{z_2}{K_{z2}} + \frac{z_3}{K_{z3}}} \tag{34.2}$$

  where z is the thickness of the geologic layer within the numerical cell.

34.1.1 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.

34.2 Numerical Layers

34.2.1 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.

34.2.2 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 domain is an “outer boundary”. All other boundary conditions inside the domain are called “internal boundaries”.

34.2.3 Groundwater Drainage

Saturated zone drainage is a special boundary condition in MIKE SHE used to defined natural and artificial drainage systems that cannot be defined in the river model.

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.

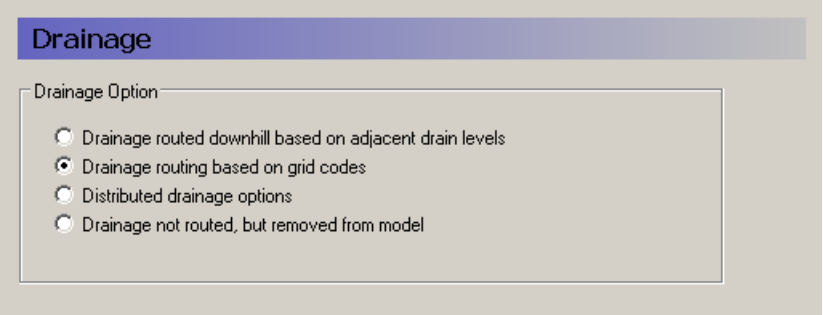
When water is removed from a drain, it is immediately moved to the recipient. In other words, the drain module assumes that the time step is longer than the time required for the drainage water to move to the recipient. This is the same as a “full pipe”. That is, water added to the end of a full pipe 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 river model 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 SZ Drainage to Specified River H-points *(p. 770)* 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.

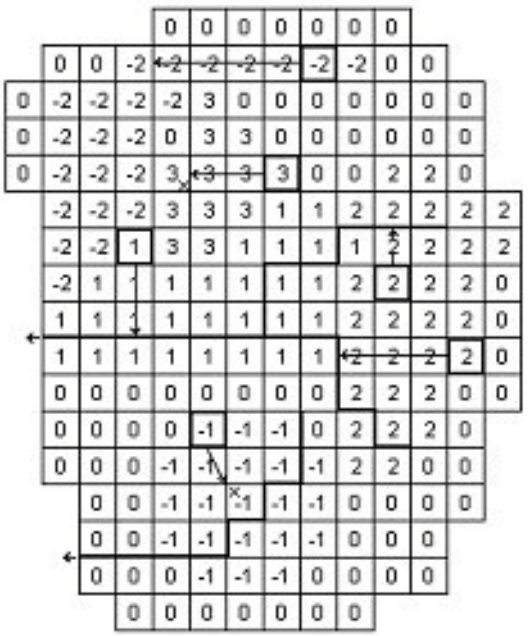
Code = 3 - Drainage in grid cells with a value of 3 is routed to a specified river branch and chainage. At the moment, this option requires the use of Extra Parameters *(p. 353)* and is described in SZ Drainage to Specified River H-points *(p. 770)*.

Code = 4 - Drainage in grid cells with a value of 4 is routed to a specified sewer man hole. This option is described in the section .

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 *(p. 780)*, 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.

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.

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.

34.3 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 1D) are defined independently of the subsurface numerical layers.

The use of grid independent geology and boundary conditions provides a great deal of flexibility in the development of the saturated zone model. Thus the same geological model and many of the boundary conditions can be re-used for different model discretisation and different model areas.

34.3.1 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 34.1 is an example .pfs file for the MODFLOW data extractor program:

Table 34.1 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.
TimeUnit = 'DAYS'	The TimeUnit is not currently used, but must be input. Valid values for TimeUnit are DAYS, HOURS, MINUTES and SECONDS.

Table 34.1 MODFLOW Extraction.pfs file format and description

Line item	Comment
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	



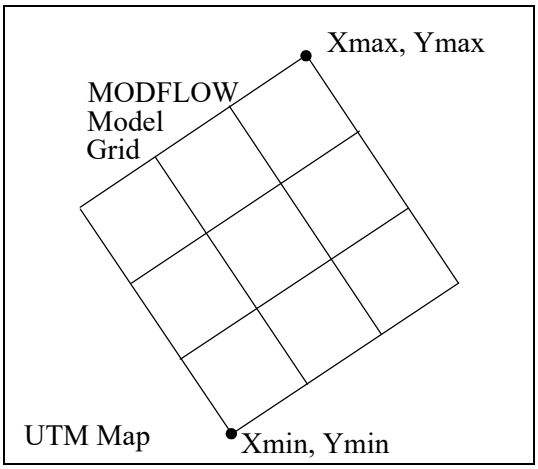
Note: MODFLOW does not have any internal unit checking. The units written in the MODFLOW file are only for display purposes. Also, the units that you define in your MODFLOW user interface may not be the same as those written to the MODFLOW files. So, you need to be careful of units and know what units 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 MOD-

FLOW 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 MODFLOW executable from the command (DOS) line. The executable can be downloaded from the USGS website.



REFERENCE MANUAL FOR WATER QUALITY

35 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 *(p. 673)*
- Reactive Transport - Reference *(p. 699)*
- Particle Tracking-Reference *(p. 737)*

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 *(p. 715)*
- Working with MIKE ECO Lab in MIKE SHE - User Guide *(p. 725)*

36 Advection Dispersion - Reference

36.1 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.

36.1.1 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.

36.1.2 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 36.1 lists the solute exchange possibilities between the compo-

nents, in particular when one or more component is not included in the flow simulation.

Table 36.1 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 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	 Overland flow Unsaturated Zone River	 Overland flow or External boundary (none) Overland flow (none)
River Baseflow to groundwater	 Groundwater	 (none)

An sketch of the different internal boundary conditions is shown in Figure 36.1. Each of these exchanges is detailed in the respective sub-section for each hydrologic component.

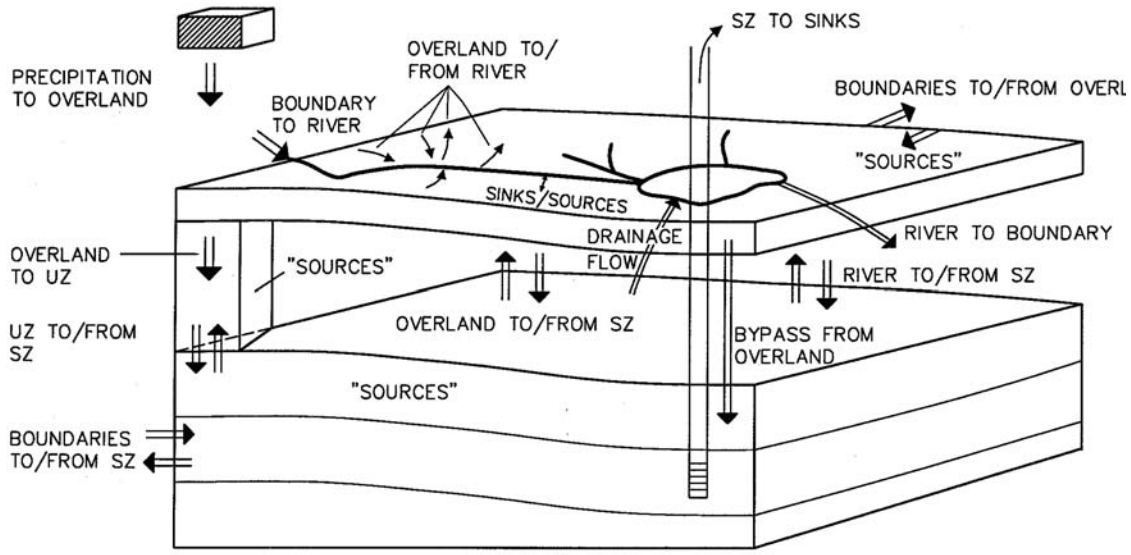


Figure 36.1 Outline of the different transport possibilities between components and boundaries.

36.1.3 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 36.2, 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 the river model, the unsaturated zone component and the groundwater component. Figure 36.2 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 36.2, a SZ source requires that all components have a break when the source starts.

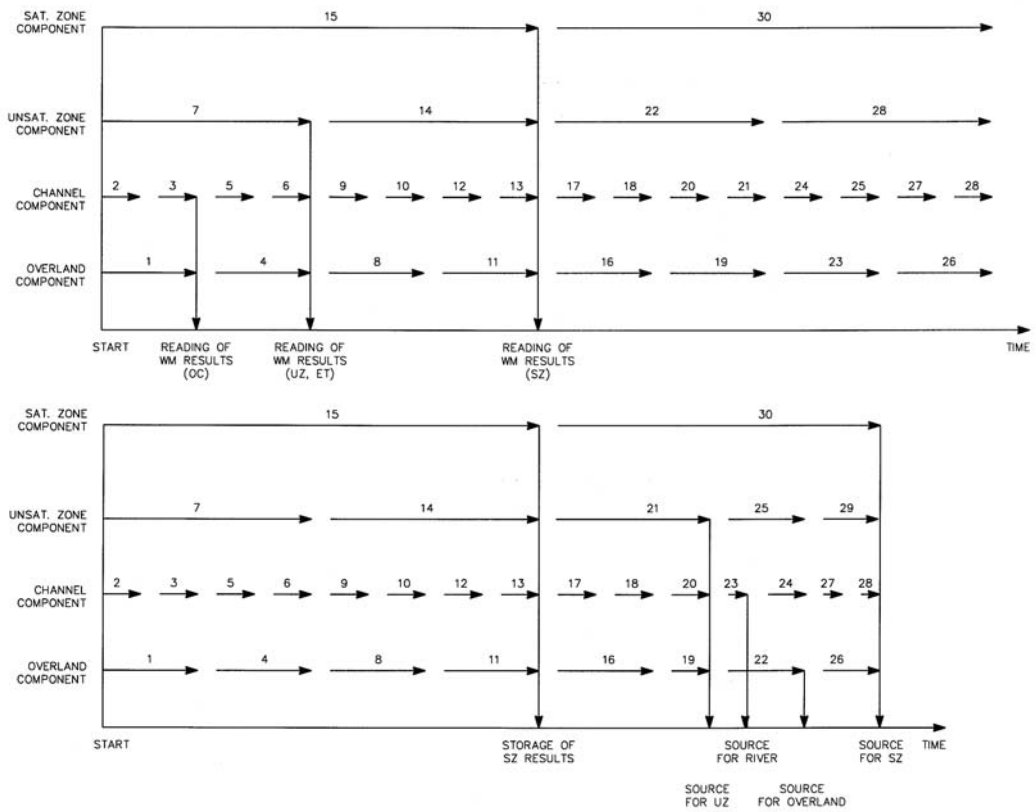


Figure 36.2 An example calculation sequence for solute transport in MIKE SHE.

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:

$$\sigma_x = \frac{v_x \Delta t}{\Delta x} \tag{36.1}$$

and the dispersive Courant number, Γ_x, is defined as:

$$\Gamma = \frac{D_{11}}{\Delta x^2}\Delta t \tag{36.2}$$

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.

36.2 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.

36.2.1 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

$$\frac{\partial c}{\partial t} = -\frac{\partial}{\partial x_i}(cv_i) + \frac{\partial}{\partial x_i}\left(D_{ij}\frac{\partial c}{\partial x_j}\right) + R_c \quad i,j = 1,2,3 \tag{36.3}$$

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

$$v_i = \frac{q_i}{\theta} \tag{36.4}$$

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

$$D_{ij} = a_{ijmn}\frac{v_n v_m}{U} \tag{36.5}$$

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:

$$a_{ijmn} = \alpha_T \delta_{ij} \delta_{mn} + \frac{\alpha_L - \alpha_T}{2}\left(\delta_{im}\delta_{jn} + \delta_{in}\delta_{jm}\right) \tag{36.6}$$

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:

$$
\begin{aligned}
D_{xx} &= \left[\alpha_T\left(V_y^2+V_z^2\right)+\alpha_L V_x^2\right]/U \\
D_{yy} &= \left[\alpha_T\left(V_x^2+V_z^2\right)+\alpha_L V_y^2\right]/U \\
D_{zz} &= \left[\alpha_T\left(V_x^2+V_y^2\right)+\alpha_L V_z^2\right]/U \\
D_{xy} &= \left(\alpha_L-\alpha_T\right)V_x V_y/U = D_{yx} \\
D_{xz} &= \left(\alpha_L-\alpha_T\right)V_x V_z/U = D_{zx} \\
D_{yz} &= \left(\alpha_L-\alpha_T\right)V_y V_z/U = D_{zy}
\end{aligned}
\qquad (36.7)
$$

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:

$$
\begin{aligned}
a_{ijmn} = {} & a_I\,\delta_{ij}\,\delta_{mn} \\
& + a_{II}\left(\delta_{im}\,\delta_{jn} + \delta_{in}\,\delta_{jm}\right) \\
& + a_{III}\left(\delta_{ij}\,h_m\,h_n + \delta_{mn}\,h_i\,h_j\right) \\
& + a_{IV}\left(\delta_{im}\,h_j\,h_n + \delta_{jm}\,h_i\,h_n + \delta_{in}\,h_j\,h_m + \delta_{jn}\,h_i\,h_m\right) \\
& + a_V\,h_i\,h_j\,h_m\,h_n
\end{aligned}
\qquad (36.8)
$$

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. (36.5) and Eq. (36.8) as follows:

$$
\begin{aligned}
D_{xx} &= \alpha_{LHH}\frac{V_x^2}{U} + \alpha_{THH}\frac{V_y^2}{U} + \alpha_{THV}\frac{V_z^2}{U} \\
D_{yy} &= \alpha_{THH}\frac{V_x^2}{U} + \alpha_{LHH}\frac{V_y^2}{U} + \alpha_{THV}\frac{V_z^2}{U} \\
D_{zz} &= \alpha_{TVH}\frac{V_x^2}{U} + \alpha_{TVH}\frac{V_y^2}{U} + \alpha_{LVV}\frac{V_z^2}{U} \\
D_{xy} &= (\alpha_{LHH} - \alpha_{THH})\frac{V_x V_z}{U} \\
D_{xz} &= \left(\frac{\alpha_{LVV}+\alpha_{LHH}}{2} - \frac{\alpha_{TVH}+\alpha_{THV}}{2}\right)\frac{V_x V_y}{U} \\
D_{yz} &= \left(\frac{\alpha_{LVV}+\alpha_{LHH}}{2} - \frac{\alpha_{TVH}+\alpha_{THV}}{2}\right)\frac{V_y V_z}{U}
\end{aligned}
\tag{36.9}
$$

and for symmetrical reasons $D_{xy} = D_{yx}$, $D_{xz} = D_{zx}$ and $D_{yz} = D_{zy}$.

Note that Eq. (36.9) can simplify to Eq. (36.7) if $\alpha_{LHH}=\alpha_{LVV}=\alpha_L$ and $\alpha_{THH}=\alpha_{TVH}=\alpha_{THV}=\alpha_T$

Burnett and Frind /11/ 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.

36.2.2 Solution Scheme

Regular Grid

The numerical solution to the advection-dispersion equation in MIKE SHE AD is based on the QUICKEST method. Leonard /38/ originally introduced this method, which was further developed by Vested et. al. /53/. 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:

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial x}\left(v_x c\right) + \frac{\partial}{\partial y}\left(v_y c\right) + \frac{\partial}{\partial z}\left(v_z c\right) = 0 \tag{36.10}$$

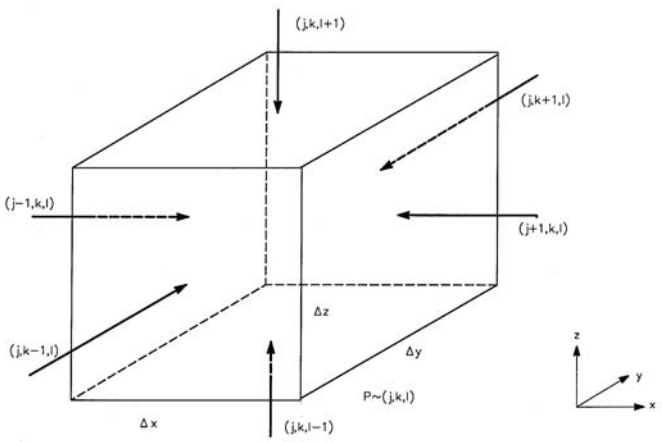


Figure 36.3 Control volume defining an internal SZ grid.

For the control volume shown in Figure 36.3 this equation is written in finite difference form as:

$$c_{j,k,l}^{n+1} - c_{j,k,l}^{n} + \sigma_x \left(c^{*}_{j+\frac{1}{2},k,l} - c^{*}_{j-\frac{1}{2},k,l} \right) + \sigma_y \left(c^{*}_{j,k+\frac{1}{2},l} - c^{*}_{j,k-\frac{1}{2},l} \right) + \sigma_z \left(c^{*}_{j,k,l+\frac{1}{2}} - c^{*}_{j,k,l-\frac{1}{2}} \right) = 0 \quad (36.11)$$

where *n* denotes the time index.

In Eq. (36.11) σ_x, σ_y and σ_z are the directional Courant numbers defined by

$$\sigma_x = \frac{V_x \, \Delta t}{\Delta x}, \quad \sigma_y = \frac{V_y \, \Delta t}{\Delta y}, \quad \sigma_z = \frac{V_z \, \Delta t}{\Delta z} \quad (36.12)$$

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:

$$c^{*}_{j+\frac{1}{2},k,l} = \sum \delta_i c_i$$
$$c^{*}_{j,k+\frac{1}{2},l} = \sum \gamma_i c_i$$
$$c^{*}_{j,k,l+\frac{1}{2}} = \sum \beta_i c_i \quad (36.13)$$

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 36.2.

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 36.4. The other "boundary" concentrations are found in a similar way

.

Table 36.2 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.

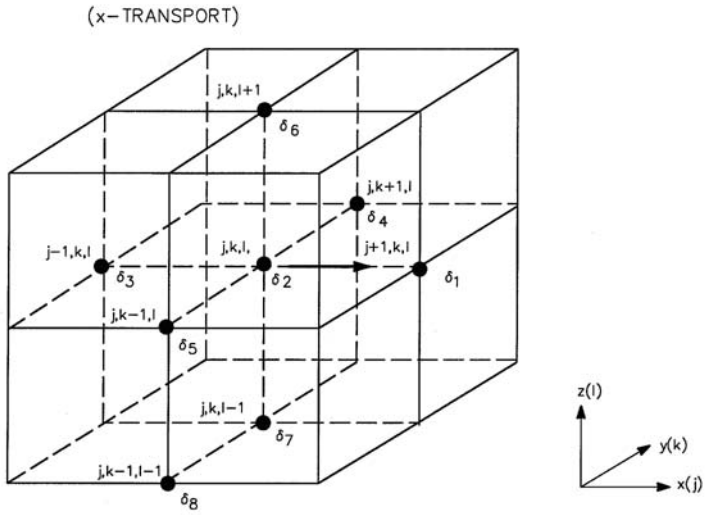


Figure 36.4 Location of interpolation weights for determination of concentrations at the location (j+½,k,l) - a grid boundary.

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:

$$\begin{aligned} TD_{xx,j,k,l} = & -\tfrac{1}{2}\left(D_{xx,j+1,k,l} + D_{xx,j,k,l}\right)\left(c_{j+1,k,l} - c_{j,k,l}\right) / \Delta x^2 \\ & -\tfrac{1}{2}\left(D_{xy,j+1,k,l} + D_{xy,j,k,l}\right) \cdot \\ & \left(c_{j,k+1,l} + c_{j+1,k+1,l} - c_{j,k-1,l} - c_{j+1,k-1,l}\right) / 4\Delta y\Delta x \\ & -\tfrac{1}{2}\left(D_{xz,j+1,k,l} + D_{xz,j,k,l}\right) \cdot \\ & \left(c_{j,k,l+1} + c_{j+1,k,l+1} - c_{j,k,l-1} - c_{j+1,k,l-1}\right) / 4\Delta z\Delta x \end{aligned} \quad (36.14)$$

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.

36.2.3 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.

36.2.4 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 river nodes.

Referring back to Figure 36.1, 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 the 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 the 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.

36.2.5 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 *(p. 753)* 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.

36.2.6 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:

$$\theta_{im}\frac{\partial c_{im}}{\partial t} + \theta_m\frac{\partial c_m}{\partial t} =$$

(36.15)

$$-\frac{\partial}{\partial x_i}\left(\theta_m c_m v_i\right) + \frac{\partial}{\partial x_i}\left(D_{ij}\frac{\partial \theta_m c_m}{\partial x_j}\right) + R_c$$

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:

$$\theta_{im}\frac{\partial c_{im}}{\partial t} = \beta\left(c_m - c_{im}\right)$$

(36.16)

where β is the diffusion coefficient.

Diffusive exchange is included as a distributed source/sink term in the basic advection-dispersion equation.

36.3 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.

36.3.1 Governing Equations

Soil Matrix Transport

For unsaturated solute transport in the soil matrix the advection-dispersion equation is

$$\frac{\partial c}{\partial t} = -\frac{\partial}{\partial z}(cv_z) + \frac{\partial}{\partial z}\left(D\frac{\partial c}{\partial z}\right) + R_c \tag{36.17}$$

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:

$$v_z = \frac{q}{\theta} \tag{36.18}$$

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:

$$D = \alpha_L v_z \tag{36.19}$$

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. .

36.3.2 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

$$R_c = \beta_c \cdot \theta_{matrix} \cdot \frac{\theta_{mp}}{e_{mp}} \cdot (c_{mp} - c_{matrix}) + S_{mp} \cdot c' \tag{36.20}$$

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

$$\beta_c = C_k \cdot \frac{3D_0 \cdot f}{d^2} \tag{36.21}$$

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).

36.3.3 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

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial z}(v_z c) = 0 \tag{36.22}$$

:

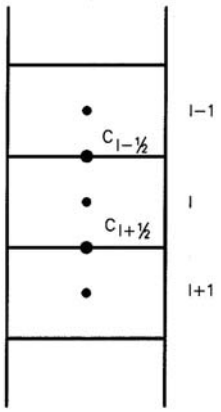


Figure 36.5 Control volume defining an internal grid.

For the control volume shown in Figure 36.5, this equation is written in finite difference form as:

$$c_l^{n+1} - c_l^n + \sigma_z \left(c^*_{l+½} - c^*_{l-½} \right) = 0 \tag{36.23}$$

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

$$\sigma_z = \frac{v_z \Delta t}{\Delta z} \tag{36.24}$$

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

$$c_{l+½} = \sum \beta_i c_i \tag{36.25}$$

However, in the unsaturated zone only three weights need to be determined

$$\beta_1 = \frac{\sigma_z^2}{6} - \frac{\sigma_z}{2} + \frac{1}{3}$$
$$\beta_2 = 1 - \beta_1 - \beta_3$$
$$\beta_3 = \frac{\sigma_z^2}{6} - \frac{1}{6} \quad (36.26)$$

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

$$TD_{z,l} = -\frac{1}{2}(D_{z,l+1} + D_{z,l})(c_{l+1} - c_l)/\Delta z^2 \quad (36.27)$$

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.

36.3.4 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.

36.3.5 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.

36.4 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.

36.4.1 Governing Equations

The transport of solutes on the ground surface is governed by the two-dimensional advection-dispersion equation

$$\frac{\partial c}{\partial t} = - \frac{\partial}{\partial x_i}(cv_i) + \frac{\partial}{\partial x_i}\left(D_{ij}\frac{\partial c}{\partial x_j}\right) + R_c \quad i,j = 1,2 \tag{36.28}$$

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. (36.28) are determined as for isotropic conditions in groundwater as

$$\begin{aligned} D_{xx} &= D_{11} = D_L\frac{V_x^2}{U^2} + D_T\frac{V_y^2}{U^2} \\ D_{yy} &= D_{22} = D_T\frac{V_x^2}{U^2} + D_L\frac{V_y^2}{U^2} \\ D_{xy} &= D_{yx} = D_{12} = (D_L - D_T)\frac{V_xV_y}{U^2} \end{aligned} \tag{36.29}$$

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.

36.4.2 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

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial x}(v_x c) + \frac{\partial}{\partial y}(v_y c) = 0 \tag{36.30}$$

and when written in finite difference form becomes

$$c_{j,k}^{n+1} - c_{j,k}^{n} + \sigma_x \left(c^{*}_{j+\frac{1}{2},k} - c^{*}_{j-\frac{1}{2},k}\right) + \sigma_y \left(c^{*}_{j,k+\frac{1}{2}} - c^{*}_{j,k-\frac{1}{2}}\right) = 0 \tag{36.31}$$

where *n* denotes the time index.

In Eq. (36.31), σ_x and σ_y are the directional Courant numbers defined by

$$\sigma_x = \frac{v_x \Delta t}{\Delta x}, \quad \sigma_y = \frac{v_y \Delta t}{\Delta y} \tag{36.32}$$

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

$$c^{*}_{j+\frac{1}{2},k} = \sum \alpha_i c_i$$
$$c^{*}_{j,k+\frac{1}{2}} = \sum \beta_i c_i \tag{36.33}$$

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 36.3. The other “boundary” concentrations are found in a similar way.

Table 36.3 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 36.3 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 *(p. 681)* 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

$$TD_{xi,j,k} = -\tfrac{1}{2}\left(D_{xx,j+1,k} + D_{xi,j,k}\right)\left(c_{j+1,k} - c_{j,k}\right)/\Delta x^2 - \tfrac{1}{2}\left(D_{xy,j+1,k} + D_{xy,j,k}\right) * \left(c_{j,k+1} + c_{j+1,k+1} - c_{j,k-1} - c_{j+1,k-1}\right)/4\Delta y\Delta x \quad (36.34)$$

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.

36.4.3 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.

36.4.4 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 36.2 the overland transport component exchanges mass with the river, the unsaturated zone and the saturated zone. In the case of exchange to the river, the solute mass is simply added as a source term to

MIKE 1D. 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.

36.5 Solute Transport in MIKE 1D

In MIKE SHE, the solute transport in the river channels is handled by the MIKE 1D Advection-Dispersion (AD) module.

In MIKE 1D, 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 1D exchanges solutes with MIKE SHE's overland and saturated zone flow components.

Detailed information on this module is available as part of the MIKE+ River Network Modelling technical documentation.

37 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 1D 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 (36.3). When processes, such as sorption and decay, are included, the equation is extended to

$$\frac{\delta c}{\delta t} = -\frac{\delta}{\delta x_i}(c v_i) + \frac{\delta}{\delta x_i}\left(D_{ij}\frac{\delta c}{\delta x_j}\right) - \frac{\rho_b}{\theta}\frac{\delta c^*}{\delta t} + \left(\frac{\delta c}{\delta t}\right)_{reac} \tag{37.1}$$

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.

37.1 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.

37.1.1 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:

$$c' = K_d \cdot c \tag{37.2}$$

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.

$$K_d = f_{oc} \cdot K_{oc} \tag{37.3}$$

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

$$R = \frac{v}{v_c} = 1 + \frac{\rho_b}{\theta} \cdot K_d \tag{37.4}$$

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:

$$c' = K_f \cdot c^N \tag{37.5}$$

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 37.1.

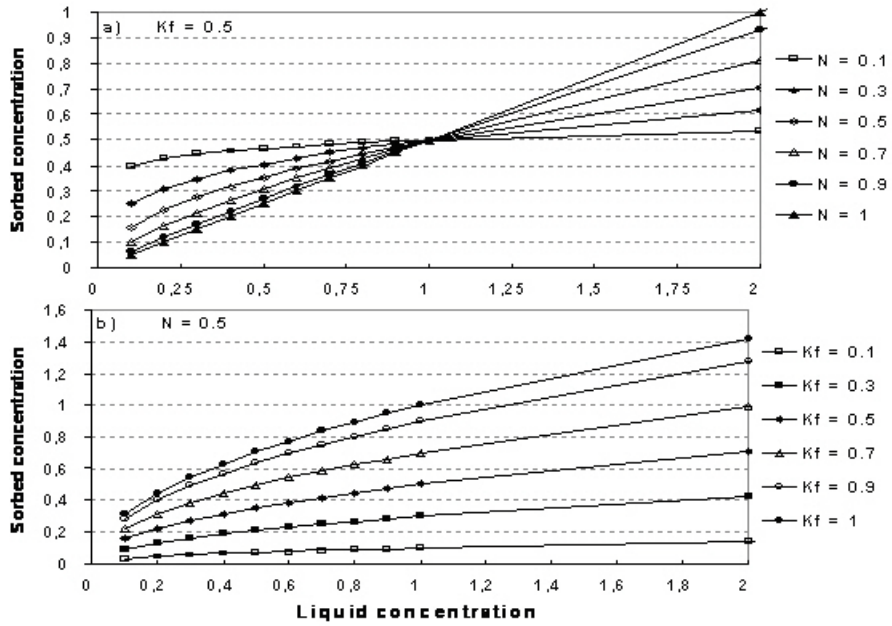


Figure 37.1 Illustration of the Freundlich isotherm. a) effect of change in N, b) effect of change in Kf

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

$$\frac{c}{c^*} = \frac{1}{\alpha\beta} + \frac{c}{\beta} \tag{37.6}$$

or

$$c^* = \frac{c\alpha\beta}{1+\alpha c} \tag{37.7}$$

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 37.2.

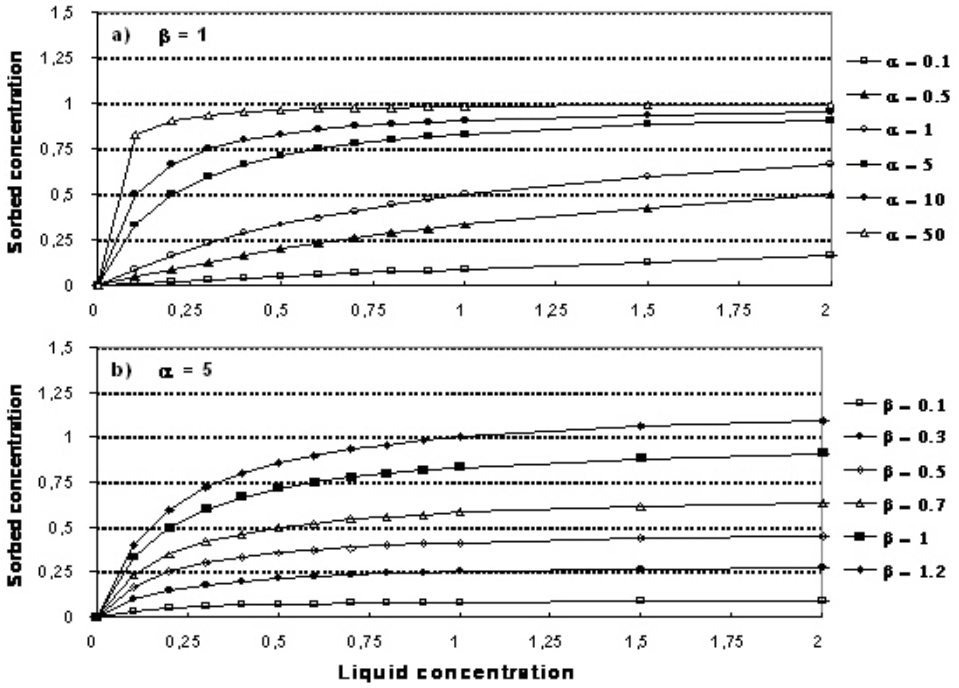


Figure 37.2 Illustration of the Langmuir isotherm. a) effect of change in α, b) effect of change in β.

37.1.2 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:

$$c \leftrightarrow c^{*} \leftrightarrow c^{**} \tag{37.8}$$

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

$$\frac{\partial c^{*}}{\partial t} = K_1 \left(c^{**} - c^{*} \right) \tag{37.9}$$

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.

37.1.3 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:

$$\rho_{ma} = \frac{\theta_{ma} + F_b \theta_{mi}}{\theta_{ma} + \theta_{mi}} \rho_b \qquad for\ 0 \le F_b \le 1$$

$$\rho_{ma} = \frac{\theta_{ma} (1 + F_b)}{\theta_{ma} + \theta_{mi}} \rho_b \qquad for\ -1 \le F_b < 0 \tag{37.10}$$

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. (37.10) is illustrated in Figure 37.3.

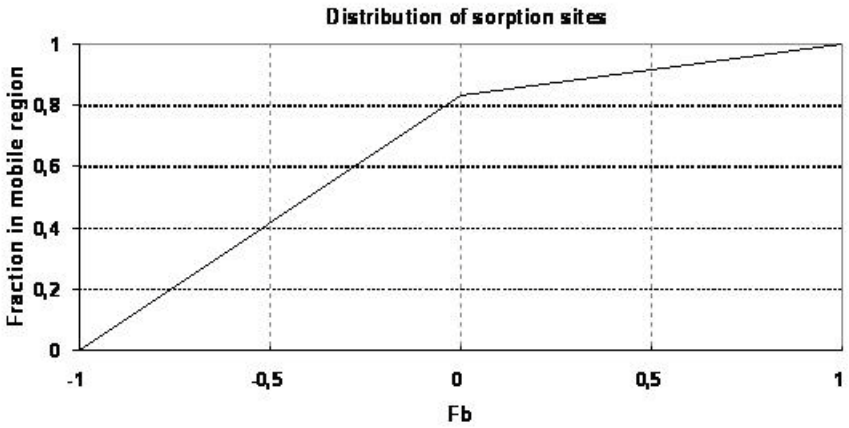


Figure 37.3 Illustration of the fraction of sorption sites located in the macro pore region (ρ_{ma}/ρ_b) as a function of F_b. Mobile porosity = 0.25, immobile porosity=0.05.

37.2 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

$$\left(\frac{\partial c}{\partial t}\right)_{\text{reac}} = \mu_{\text{ref}} c \tag{37.11}$$

where μ_{ref} is the reference degradation rate coefficient calculated by

$$\mu_{\text{ref}} = \frac{\ln 2}{\lambda} \tag{37.12}$$

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

$$\mu = \mu_{\text{ref}} F_w F_t \tag{37.13}$$

where the F_w is the water content function given as

$$F_w = \left(\frac{\theta}{\theta_s}\right)^B \tag{37.14}$$

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:

$$F_t = 0 \qquad for(T_s < 0^oC) \tag{37.15}$$

$$F_t = \frac{T_5}{5} e^{\alpha(5 - T_{ref})} \qquad for(0 \le T_s \le 5^oC) \tag{37.16}$$

$$F_t = e^{\alpha(T_s - T_{ref})} \qquad for(T_s > 5^oC) \tag{37.17}$$

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)

$$T_s = T_{sy} + 0{,}346(T_{air} - T_{sy})e^{-2{,}7028z} \tag{37.18}$$

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 37.4.

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.

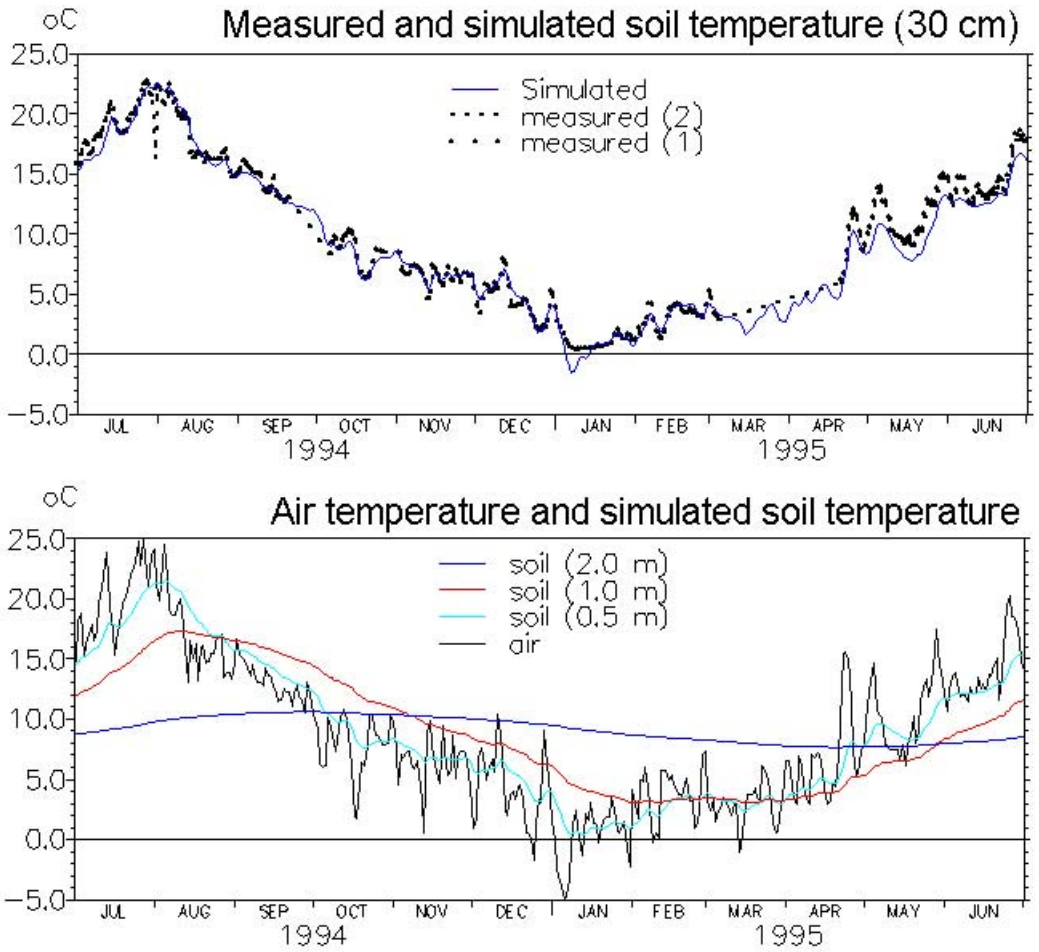


Figure 37.4 Application of the soil temperature function. Top: measured and simulated soil temperature in 30 cm depth (measurements were based on two replicates). Bottom: soil temperatures simulated in different depths based on measured air temperatures.

37.3 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.

$$R_r = f_c S_r c \tag{37.19}$$

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.

37.4 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.

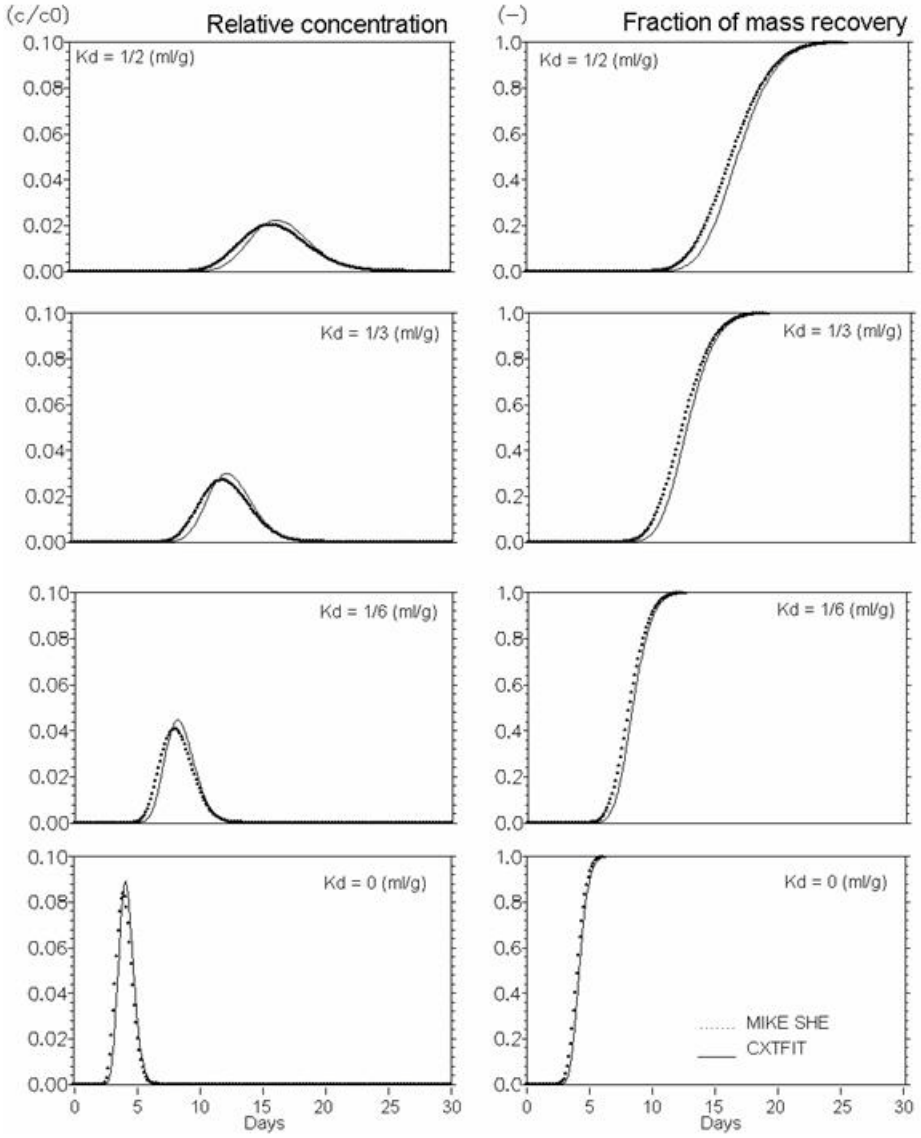


Figure 37.5 Linear equilibrium sorption. Effect of K_d (ml/g)

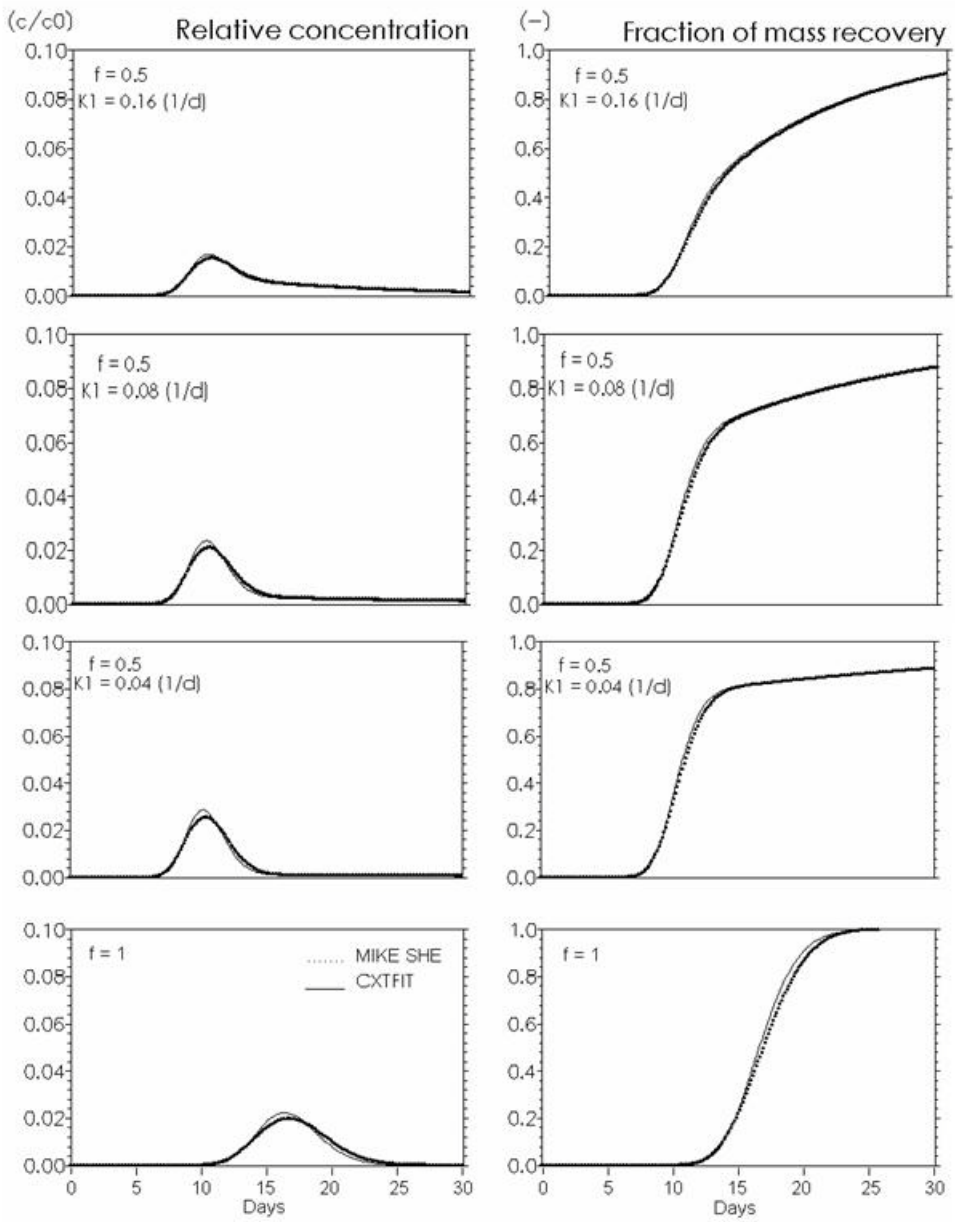


Figure 37.6 Kinetic sorption . Effects of rate constant K1 (d^{-1}) in Eq.(37.9). Input to CXTFIT: f (fraction of equilibrium sorption sites). K_d = 0.5 ml/g. Input to MIKE SHE: Kd = f*K_d (CXTFIT). At f=1 the function is reduced to equilibrium sorption with K_d = 0.5 ml/g.

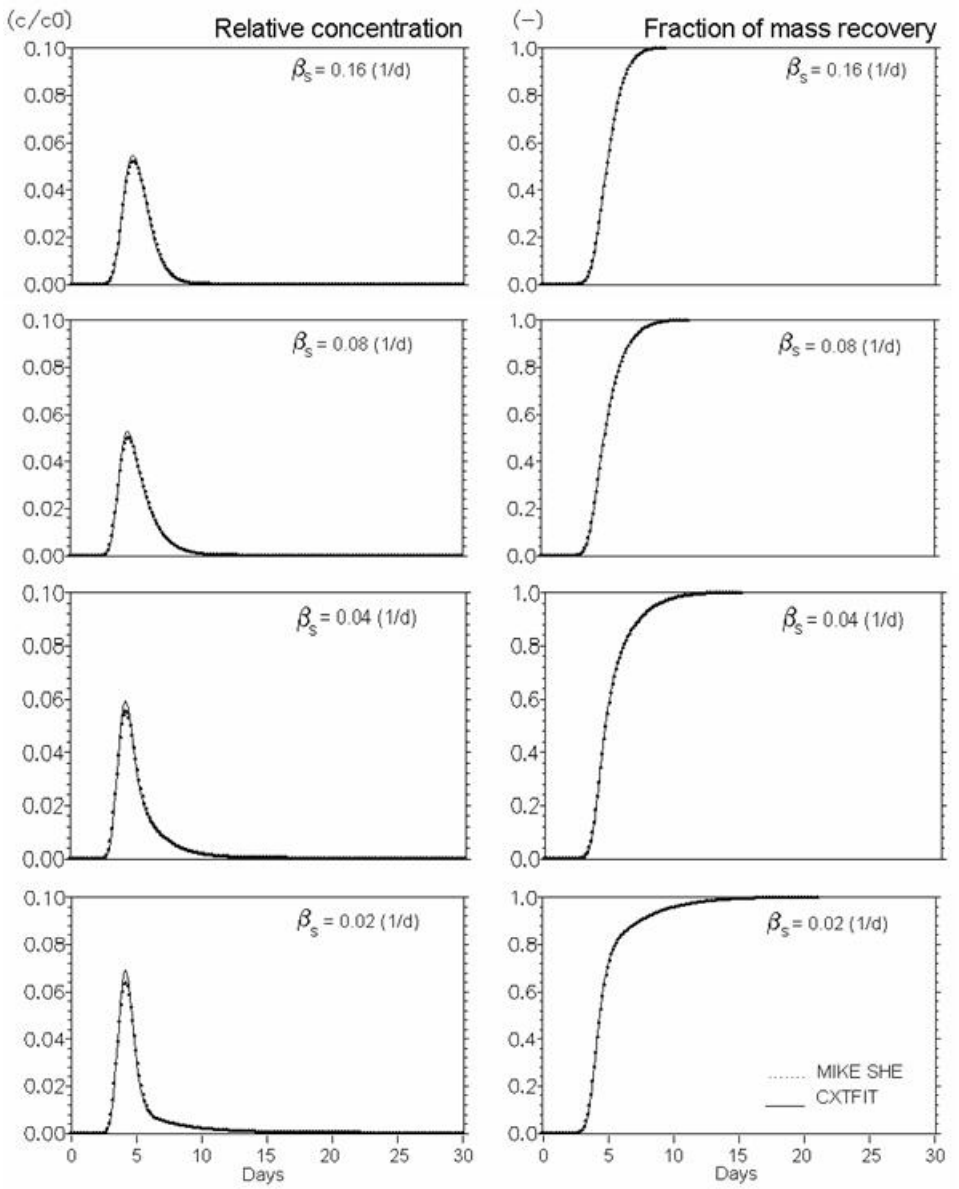


Figure 37.7 Conservative solute transport in dual porosity systems. Mobile porosity = 0.25, immobile porosity = 0.05. Effect of mass transfer coefficient (βs in Eq. (36.16) (d^{-1})).

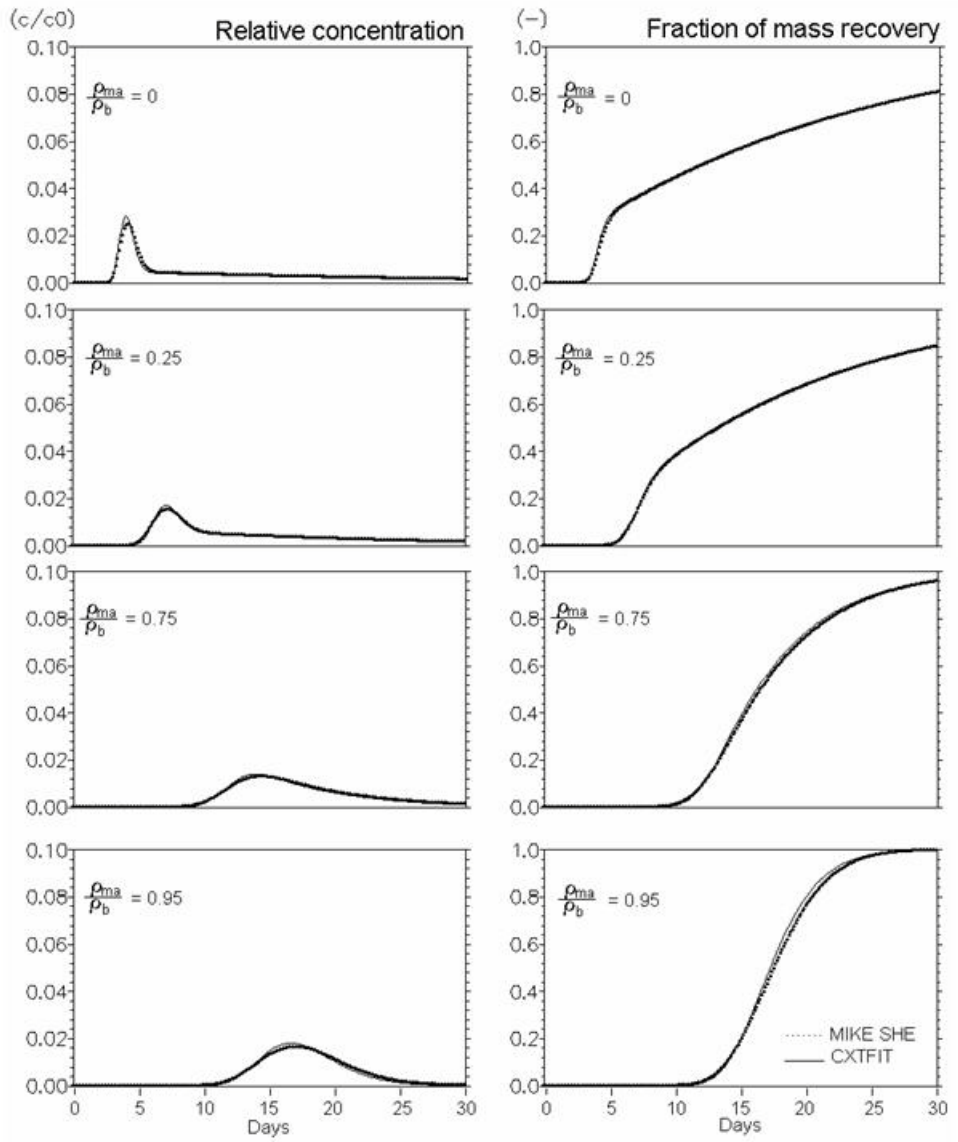


Figure 37.8 Verification of reactive solute transport in dual porosity systems. Mobile porosity = 0.25, immobile porosity = 0.05. Effect of distribution of sorption sites between mobile and immobile regions (ρ_{ma}/ρ_b Eq. 47) (Kd = 0.5 ml/g, mass transfer coefficient (β_s) = 0.08 d^{-1}).

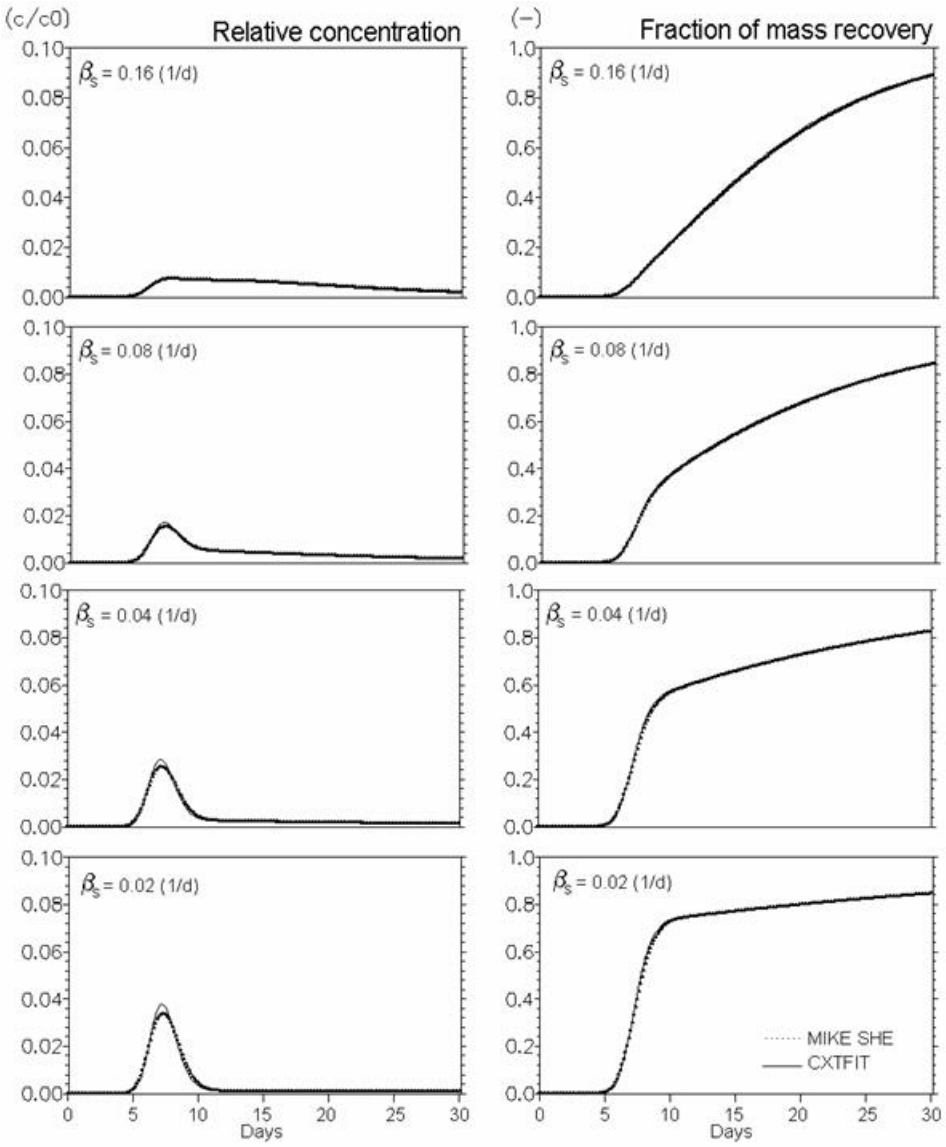


Figure 37.9 Verification of reactive solute transport in dual porosity systems. Mobile porosity = 0.25, immobile porosity = 0.05. Effect of mass transfer coefficient (βs in Eq. (36.16) (d^{-1})). Kd = 0.5 ml/g, ρ_{ma}/ρ_b = 0.25 (Eq. (37.10)).

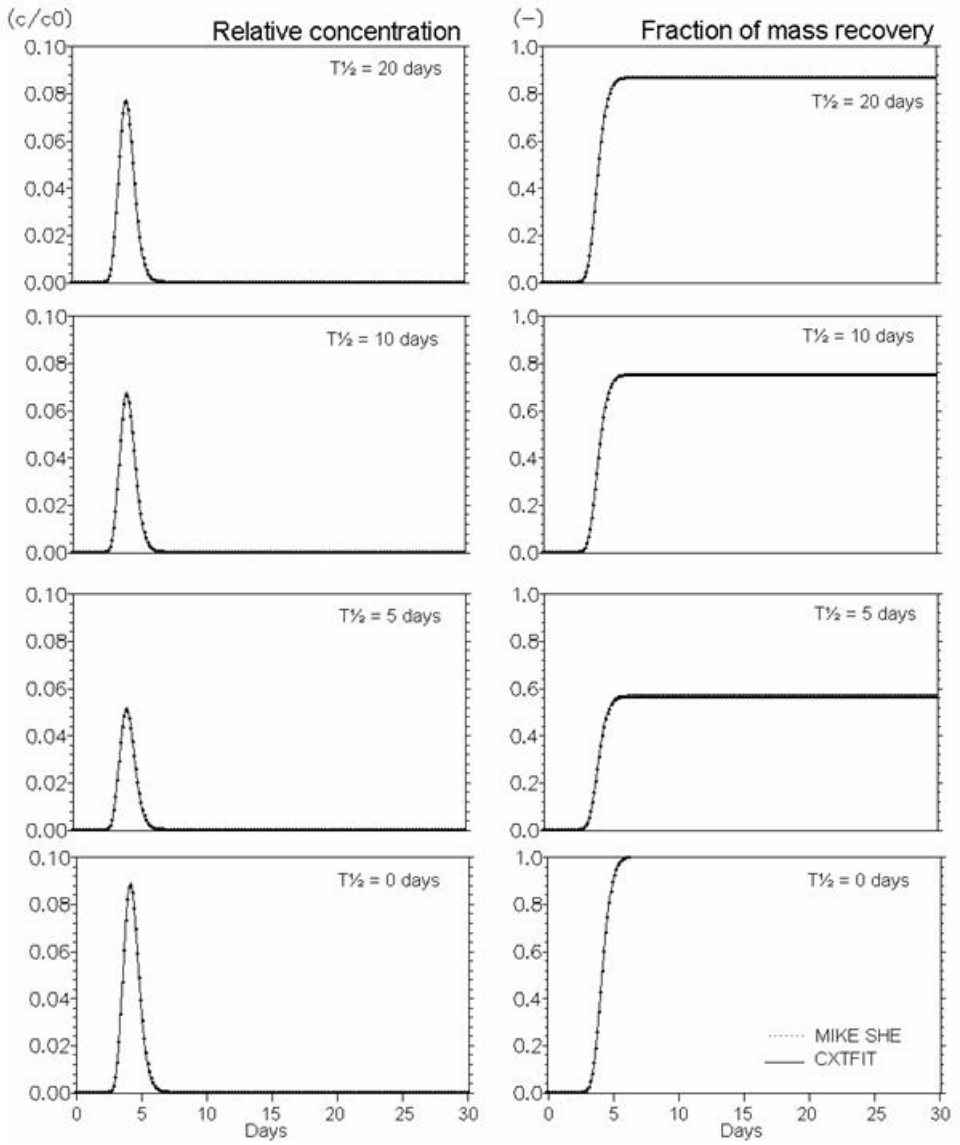


Figure 37.10 Verification of the description of first order degradation. Effect of half life time - T½ (Λ) (days) (Eq. (37.12)).

37.4.1 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 37.11. If plant uptake is included a “concentration factor” (see Eq. (37.19))is needed, indicating to which extent solutes are taken up by the plants.

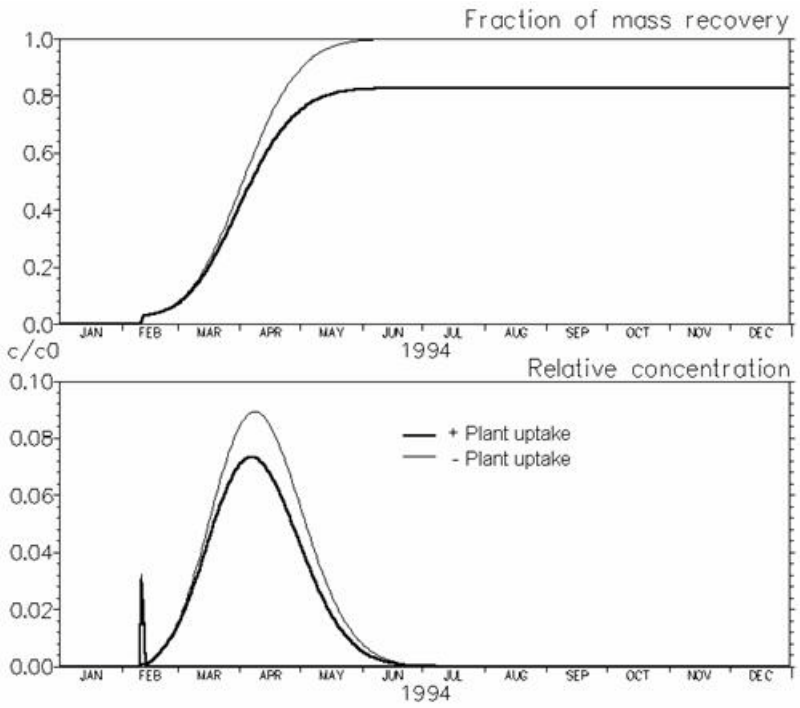


Figure 37.11 Illustration of the effect of plant uptake on solute breakthrough. Plant uptake was simulated with $f_c = 0.5$. The early peak concentrations arise from macro pore transport and were almost alike in the two simulations.

38 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 1D)
- 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

- any solute transfer via irrigation,
- solute migration **from** UZ **to** OL.

In the first case, the solutes will remain in the source location and only the water will be transferred. This will lead to increasing concentrations at the source.

In the second 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.

38.1 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 *(p. 341)*.

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.

38.2 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.

38.3 Simulation and Time Step Control

Simulation time steps are to some extent controlled by the user. Several possibilities for time step control exist to make the execution as fast as possible with no numerical dispersion and instabilities.

The first possibility for controlling the simulation time steps in the different components is simply to define the maximum time step in each component. Note that time steps should be given in increasing order i.e. $dt_{RIVER} \leq dt_{OVERLAND} \leq dt_{UZ} \leq dt_{SZ}$. Also note that this is the MAXIMUM time step. That is, the actual simulation time step is controlled by the stability criterions with respect to advective and dispersive transport given below. Furthermore, time steps for transport cannot exceed the storing time step for the relevant data in the flow result file from a MIKE SHE flow simulation.

Enter the maximum allowable Courant number for each component. The Courant number is defined by *V x dt/dx* (velocity times time step divided by "grid size"). This number should normally not exceed 1.0 for one- and two-dimensional transport (UZ, Overland and Channel Flow) and 0.8 for three-dimensional transport (SZ). The maximum time step will be calculated accordingly.

Enter the maximum allowable dispersive Courant number for each component. The dispersive Courant number is defined by *D x dt/dx2* (Dispersion coefficient times time step divided by "grid size" squared). This number should normally not exceed 0.5. The maximum time step will be calculated accordingly.

The transport limits are used to avoid negative concentrations in cases with extreme gradients (e.g. close to sources) or in areas with highly irregular velocity fields. Enter the maximum allowable transport from a node or grid as a fraction of the storage in the node or grid. A recommended value for all components is 0.9, which ensures that this option is in use (the value 0 determines that this option is not in use).

38.3.1 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.

38.4 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 *(p. 86)*

38.5 Output

The output of the MIKE SHE AD is stored to several dfs0, dfs2 and dfs3 files which can be viewed and processed with the different tools available for these files in MIKE ZERO.

For each species, a concentration file is created for each hydrologic process - a dfs2 file for OL, and a dfs3 file each for UZ and SZ.

For each species, the total WQ mass balance is stored in two dfs0 files. The xx_species_AllItems.dfs0 includes all of the possible water quality mass balance items. The xx_species.dfs0 is a copy of the _AllItems.dfs0 file with all of the non-zero items removed.

The first 20 items in the _AllItems.dfs0 file define the global mass balance (see Table 38.2). 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 38.1 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 38.2 lists all of the rest of the items in the _AllItems.dfs0 organized by the source of the solute.

Table 38.2 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 38.2 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 38.2 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.

38.6 Coupling Water Quality in MIKE SHE and Rivers

Detailed information on the MIKE 1D water quality modules are found in the MIKE+ documentation.

The coupling between MIKE 1D and the rest of MIKE SHE's hydrologic processes is relatively automatic. You must set up a MIKE 1D AD model independently of MIKE SHE using MIKE+ and specify the ,m1dx river setup file in the Rivers and Lakes dialogue. This river setup file must only have the same network geometry as the WM river model setup file. It does not have to be the same river setup.

The MIKE 1D AD 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 1D. 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 1D.
- The overland WQ must be included if you want to simulate water quality coupled to MIKE 1D.
- Recycling of WM results is not supported in MIKE 1D. This means that if you want to simulate the coupling between MIKE 1D and the rest of MIKE SHE, your WQ simulation must be continuous.

39 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.

39.1 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].

39.1.1 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.

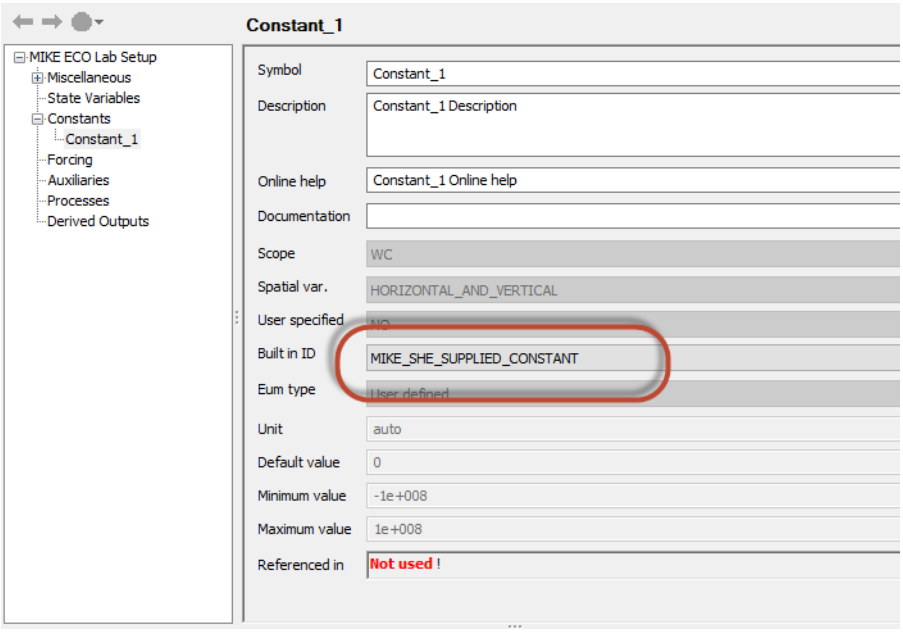
The exception to the exact naming rule is when simulating Dual domain mass transfer. In this case, the State Variable name must use the reserved suffix "_2" for the solute in the secondary porosity. For example, OXYGEN and OXYGEN_2 would be the State Variable names for the species OXYGEN in MIKE SHE.

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)..

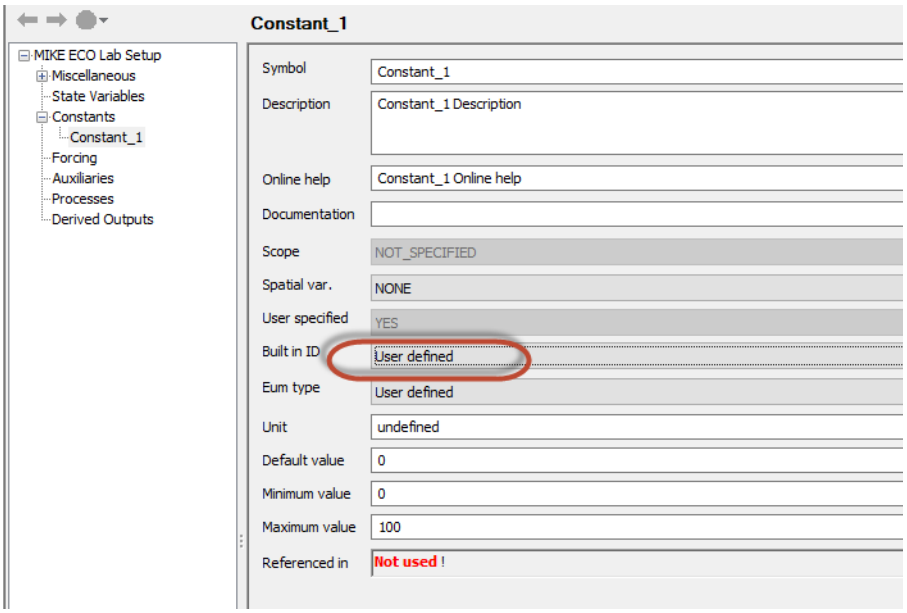
Table 39.1 Available MIKE SHE Supplied Constants

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.

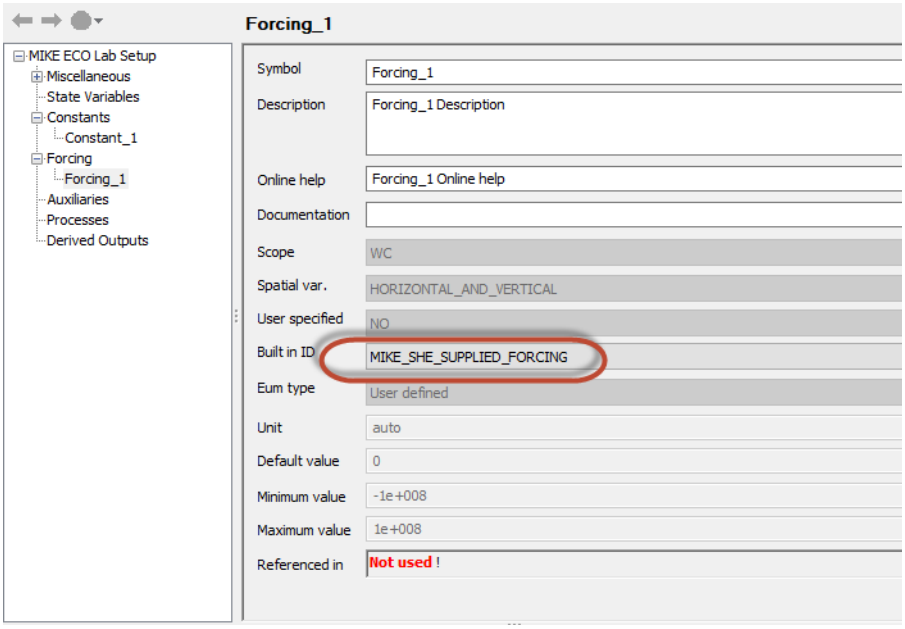
4. Add one or more Forcings

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 39.2 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 1D to define concentrations and mass of solute entering the river.

5. Create Auxiliary Variables, Processes and Derived Outputs

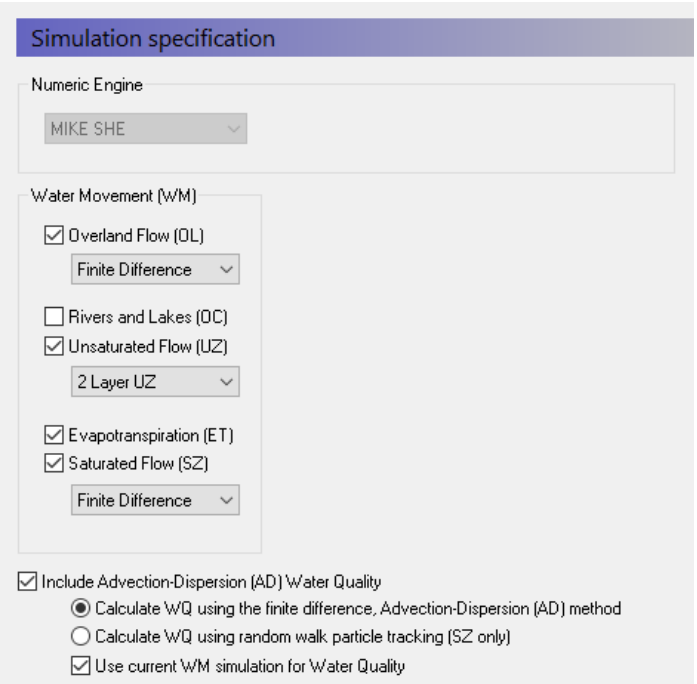
Auxiliary variables are used for intermediate calculations and must be defined as 3D (UZ and SZ) or 2D variables.

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.

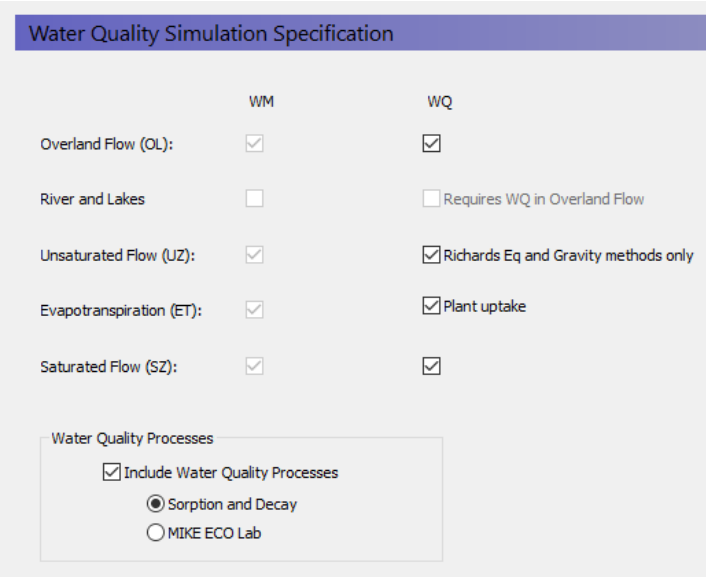
39.1.2 MIKE ECO Lab templates in MIKE SHE

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

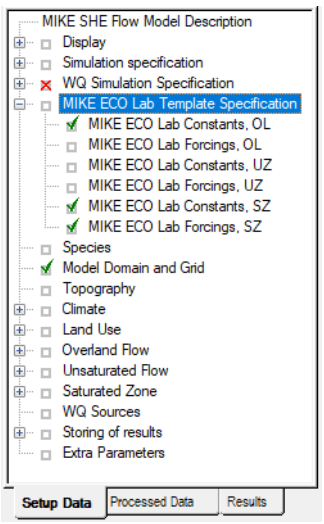


Note also that MIKE ECO Lab will only work with the Finite Difference AD method.

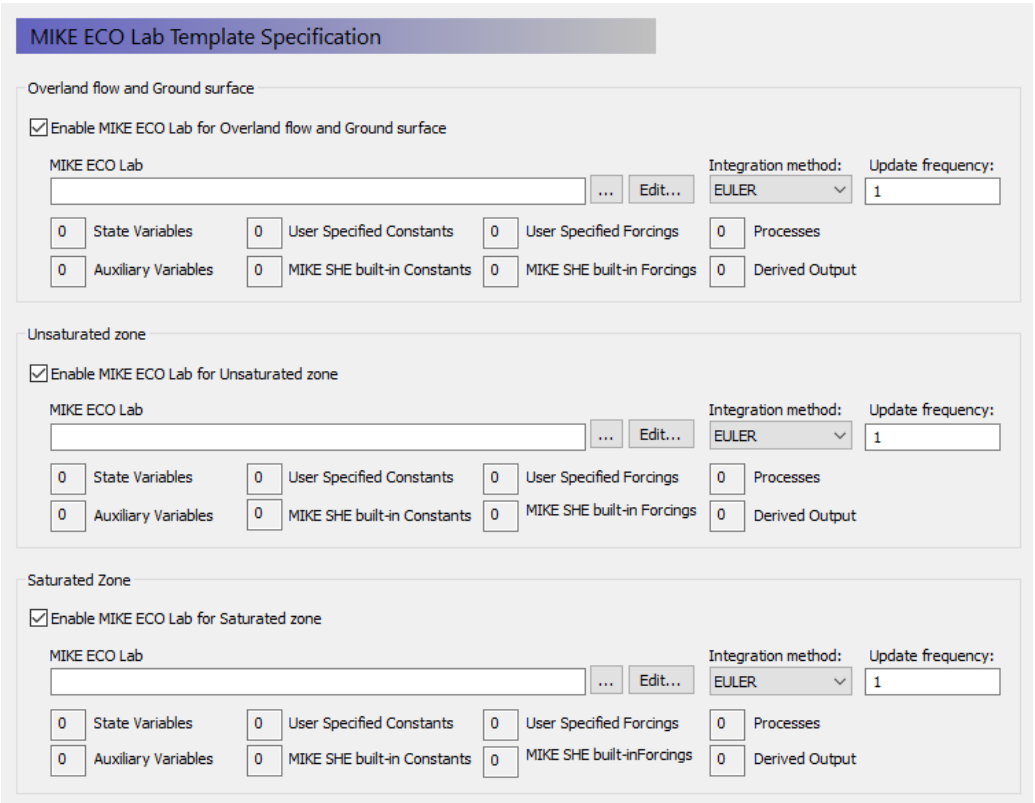
After activating the Water Quality module, the MIKE ECO Lab option must be selected in the WQ Simulation Specification dialogue.



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



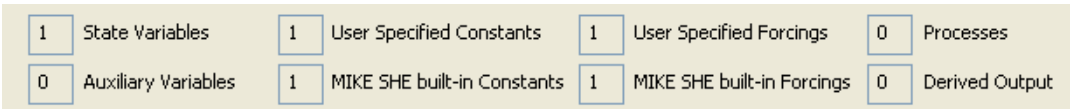
In the MIKE ECO Lab Template Specification dialogue, there is a checkbox for each of the processes. These checkboxes are active if the Water Quality is activated for the process in the WQ Simulation Specification dialogue



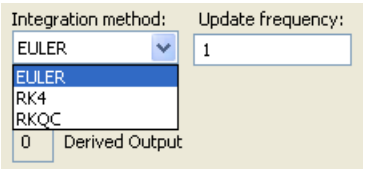
.

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.

39.1.3 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:

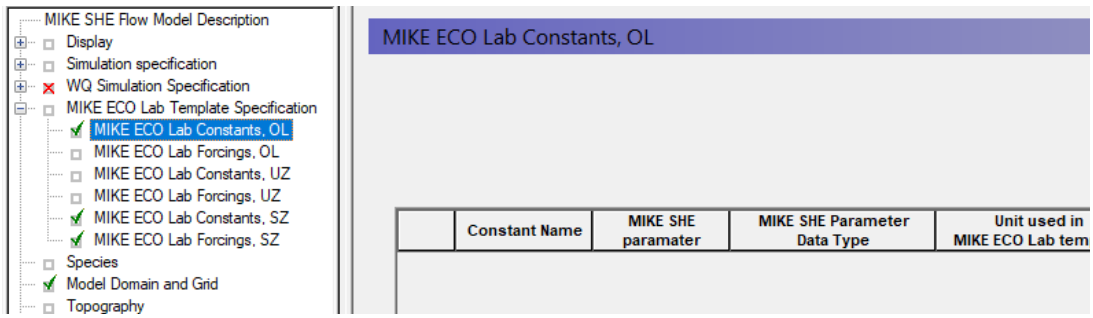
- **Dissolved**
  Dissolved species are mobile in the water. They are active in the subsurface and surface water. Disolved species have a default concentration of [$\mu g/m^3$].
- **Sorbed**
  Sorbed species are only available in the subsurface. They are fixed to the soil matrix and do not move with 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.

39.1.4 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.

39.1.5 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.

40 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

40.1 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:

$$\frac{\partial c}{\partial t} + \nabla \cdot (\underline{u} c) - \nabla \cdot (\underline{D} \cdot \nabla c) = 0 \tag{40.1}$$

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].

$$\underline{X}_p(t_{n+1}) = \underline{X}_p(t_n) + [\underline{u}(\underline{X}_{p,n}, t_n) + \nabla \cdot \underline{D}(\underline{X}_{p,n}, t_n)]\Delta t + \underline{B}(\underline{X}_{p,n}, t_n) \cdot \underline{Z}_{p,n+1} \sqrt{\Delta t} \tag{40.2}$$

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

$$\underline{B} = \underline{R} \cdot \underline{B}^* \tag{40.3}$$

where

$$\underline{\underline{R}} = \begin{bmatrix} \frac{u_x}{|\underline{u}|} & \frac{-u_y}{\beta} & \frac{-(u_y^2 + u_z^2 + u_x u_z)}{\beta|\underline{u}|} \\ \frac{u_y}{|\underline{u}|} & \frac{u_x + u_z}{\beta} & \frac{u_y(u_x - u_z)}{\beta|\underline{u}|} \\ \frac{u_z}{|\underline{u}|} & \frac{-u_y}{\beta} & \frac{u_x^2 + u_y^2 + u_x u_z}{\beta|\underline{u}|} \end{bmatrix} \tag{40.4}$$

$$|\underline{u}| = \sqrt{u_x^2 + u_y^2 + u_z^2} \tag{40.5}$$

$$\beta = \sqrt{|\underline{u}|^2 + 2u_x u_z + u_y^2} \tag{40.6}$$

$$\underline{\underline{B}}^* = \begin{bmatrix} \sqrt{2(\alpha_L|\underline{u}| + D_m)} & 0 & 0 \\ 0 & \sqrt{2(\alpha_T|\underline{u}| + D_m)} & 0 \\ 0 & 0 & \sqrt{2(\alpha_T|\underline{u}| + D_m)} \end{bmatrix} \tag{40.7}$$

α_L and α_T are the longitudinal and transversal dispersion coefficients, respectively and D_m is the neutral dispersion. Using (40.2) repeatedly, the location of a particle at time $t_N = N\Delta t$ can be determined:

$$\underline{X}_p(t_N) = \underline{X}_p(t_o) + \sum_{n=0}^{N} ([\underline{u}(\underline{X}_{p,n}, t_n) + \nabla \cdot \underline{D}(\underline{X}_{p,n}, t_n)]\Delta t + \underline{B}(\underline{X}_{p,n}, t_n) \cdot \underline{Z}_{p,n}\sqrt{\Delta t} \tag{40.8}$$

After applying (40.8) for a large number of particles, (i.e. N_p), the average solute concentration for an arbitrary volume can be calculated using (40.9)

$$c_{V,N} = \frac{1}{V}\sum_{p=1}^{N_p} m_p \delta; \delta = \begin{cases} 1; \underline{X}_{p,N} \in V \\ 0; \underline{X}_{p,N} \notin V \end{cases} \tag{40.9}$$

where m_p is the particle mass. Using this procedure an accurate solution of the advection-dispersion equation (40.1) 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 (40.2) and (40.8) 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 (40.9)) 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:

$$z_{new} = \left(\frac{z_{old} - Bottom_{old}}{Top_{old} - Bottom_{old}}\right) \times (Top_{new} - Bottom_{new}) + Bottom_{new} \quad (40.10)$$

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:

$$n_i = n \times \frac{V_i}{V_{\text{sink}}} \times \frac{V_{\text{sink}}}{V_{\text{sink}} + V_{tot}} \tag{40.11}$$

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 (40.11) 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.

41 Particle Tracking (PT)

The PT module calculates the flow path of a hundreds, thousands, or even millions, of hypothetical particles, which are moved in the three-dimensional, saturated groundwater zone (SZ). The particles are displaced individually in a number of time steps. The movement of each particle is composed of a deterministic part, in which the particle is moved according to the local groundwater velocity calculated by the MIKE SHE water movement (WM) module, and a stochastic part where the particle is moved randomly based on the local dispersion coefficients.

Particle tracking is only calculated for the saturated zone (SZ) and particles that leave SZ are not traced any further. Initially, the user assigns a number of particles to each model grid cell. Particles are added during the simulation from sources, for example solutes in precipitation or from boundary or internal defined concentration cells. Particles leave SZ when they arrive at a boundary or an internal constant concentration cell or when they go to a sink. Possible sinks in the Particle Tracking are wells, rivers, drains, and exchange with the unsaturated zone or overland flow.

All particles are assigned a mass, which means that a number of particles within a specific volume correspond to a solute concentration. The Particle Tracking module can therefore be used for solute transport simulations and is in some cases superior to the conventional numerical solution of the advection-dispersion equation since numerical dispersion is negligible. However, the module is mostly used for delineation of abstraction well capture zones and upstream zones and for determination of groundwater age and conservative solute transport times.

The PT module uses the concept of 'registration' cells. This records particle data when particles enter certain model cells. Registration can be used to delineate capture zones or to observe particles passing through some region of interest, such as a redox layer.

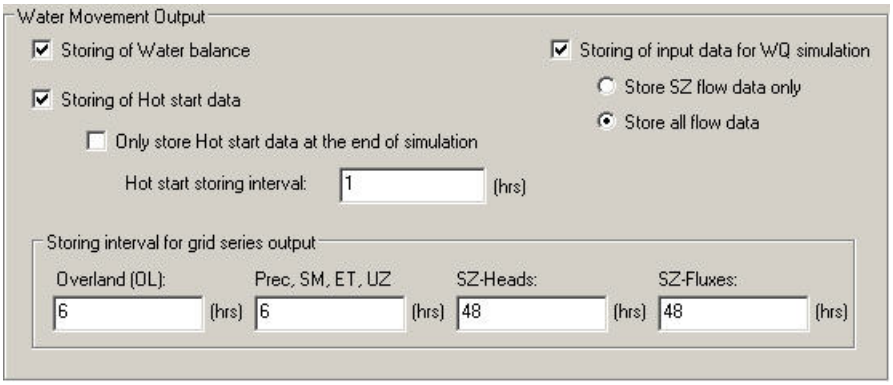
41.1 Requirements in MIKE SHE WM

Prior to running a PT simulation, the MIKE SHE Water Movement (WM) simulation must be run. This section describes what needs to be specified in the WM simulation to run the PT simulation afterwards.

41.1.1 Flow Storing Requirements

Particle transport calculations in MIKE SHE PT are based on the groundwater flows from a MIKE SHE WM simulation. In principle, only groundwater fluxes are needed but to ensure that all the needed WM result data types are stored the user has to specify that results should be stored for an AD run in

the WM input. Thus, you must check the appropriate checkbox under “Storing of Results” in the MIKE SHE WM GUI.

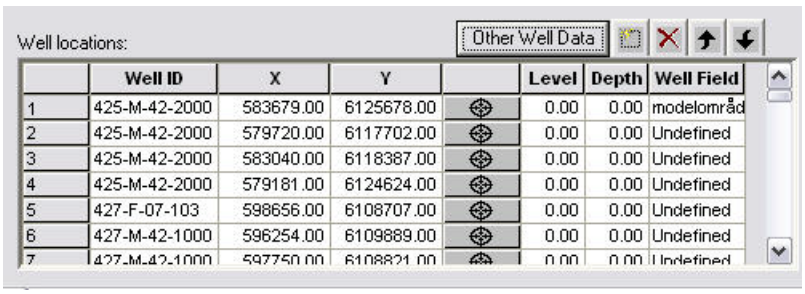


The user can choose between “SZ only” and “All hydraulic components”, however, for PT-simulations “SZ only” will be sufficient, since particle tracking is only calculated for the saturated zone.

The simulated particle distribution is stored with a desired frequency in the MIKE SHE WM GUI under “Storing of results” => “Grid series output”. It is important that the SZ and SZ-flow use the same storing frequency in order to run the following PT simulation. The WM result files to be used in the PT-simulations will be located in a folder with the same name as the *.SHE file.

41.1.2 Specification of Well Fields

To be able to retrieve particle locations based on well fields, it is necessary to specify the well fields in the MIKE SHE well database file.



	Well ID	X	Y		Level	Depth	Well Field
1	425-M-42-2000	583679.00	6125678.00		0.00	0.00	modelområd
2	425-M-42-2000	579720.00	6117702.00		0.00	0.00	Undefined
3	425-M-42-2000	583040.00	6118387.00		0.00	0.00	Undefined
4	425-M-42-2000	579181.00	6124624.00		0.00	0.00	Undefined
5	427-F-07-103	598656.00	6108707.00		0.00	0.00	Undefined
6	427-M-42-1000	596254.00	6109889.00		0.00	0.00	Undefined
7	427-M-42-1000	597750.00	6108821.00		0.00	0.00	Undefined

41.2 Output from the PT simulations

The result files will be located in standard Results directory for your project. The PT result files are:

- **projectname.PTRES**: An ASCII file in pfs-format listing the abstraction wells and the computational cells, where abstraction occurs. Used for retrieval of particle location - see PT Registration Extraction *(p. 377)*.
- **projectname.PTREG** and **projectname.trf**: Two binary files that cannot be opened directly.
- **projectname.PTBIN**: An optional binary file containing all of the particle locations at every saved time step. Individual path lines can be extracted using the Extraction of Particle Path Lines *(p. 746)*.
- **projectname.PTGross.shp**: An optional point theme shape file containing the path line information of every particle at every saved time step. As part of the shape file, a .shx and a .dbf file are also created. The .dbf file can be opened in e.g. Microsoft Excel if it does not exceed the maximum possible number of lines.

Table 41.1 Fields in <projectname>_PTGross.dbf

Field Name	Unit	Description
ID	-	Particle ID
X	As model coordinates	Location in x direction
Y	As model coordinates	Location in y direction
Z	As model coordinates	Location in z direction
IX	Cell index	Cell in x direction
IY	Cell index	Cell in y direction
Layer	Layer index	Cell in z direction
Reg code	-	'Undefined', int code, 'Incell containing Well', blank
Sink/Other	-	Sink description
Well ID	-	ID of the well (sink)
Well Field	-	ID of the well field (sink)
Branch name	-	Name of the river branch (sink)
Chain*	(length)	Chainage in the river branch (sink)
Time stamp	date/time	Registration time
Ttotal*	(time span)	Time since start of simulation
dBirth*	(time span)	Time since particle entered simulation
dTInc*	(time span)	Time since particle has last been registered
PathLen*	(length)	Length of particle path since start of simulation
dPth_Len*	(length)	Length of particle path since has last been registered

*Output in user units, name of unit added to field name. As the length of shp fields is limited to 10 truncation may occur

- **projectname_AD_3DSZ.dfs3**: Temporal and spatially varying SZ concentrations in the mobile phase based on the mass of the particles.
- **projectname_PT_3DSZ.dfs3**: Temporal and spatially varying PT results including:
  - **Number of particles** - this is the actual number of particles in each cell
  - **Accumulated particle count** – this is the number of particles that have entered the cell during the simulation
  - **Number of registered particles** – this is the number of particles that started in this cell that have become registered
  - **Most recent registration zone** – this is the registration zone attached to the last particle to be registered that originated in the cell
  - **Average age** – this is the average age of all the particles in the cell
  - **Average transport time** – this is the average length of time from when the particle was born in this cell until it was registered somewhere

Besides these result files, the program also writes output to two log files. The error log list errors encountered during execution and the print log file contains execution step information, statistics on the run and a mass balance (if requested).

41.3 Extraction of particle registrations

After the particle tracking has been run, the registration information needs to be read from the output files and processed in a useful way. The Results Tab includes an utility to sift through all the particles and their registrations, to find the ones that you want. This is output from this utility is an ArcGIS point-theme, shape file with the starting locations of the all the particles that meet your registration criteria. The extraction utility allows your to filter the results for:

- Destination type:
  - Specific sink types (drain, river, unsaturated zone, well, constant concentration boundary or constant concentration sink)
  - Registration codes specified by the user
  - Wells found in the flow results
  - Well fields found in the flow results*)
- Layer from which the particles originated
- Release (birth) time
- Transport time



Note: To extract particle locations based on well fields requires that different well fields have been defined, see section Specification of Well Fields *(p. 742)*.

The results can be written to either

- a single shape file where the point attributes allow further selection of the particles in ArcView, or
- separate files for each destination type and optionally for each layer e.g. one file for each sink type/layer combination.

More detailed information on the actual extraction mechanics can be found in the PT Registration Extraction *(p. 377)* section.

41.3.1 Running from a batch file

The registration extraction can be run from a command line. To execute the program open a command line and type:

`Ptoutputretrieval.exe` *projectname.she extraction_num*

The extraction_num is the item number in the table of extraction items in the PT Registration Extraction *(p. 377)* dialogue. The extraction will proceed silently - that is without any messages. To run the extraction with the messages, you need to use

`MZLaunch` *project_name.she* `-e Ptoutputretrieval.exe`

which will start the MZLaunch utility

projectname_ptoutputretrieval.err: If errors occur during execution of the program these are written to this log file.

41.3.2 Limitations with the PT registration method

When using registration zones to identify particles that move through certain parts of the model it should be noted that particles can appear more than once in the output. As they move from one zone to the next they are repeatedly registered and are finally also registered when they are removed from the model by a sink. An example would be a particle moving into a registration zone with code 1. The particle is then registered as being in an 'active cell' and the registration zone code and travel time to this zone is memorised. If the particle is at a later time removed by a well it will again be registered but now it will be registered as being removed by the 'Well' sink.

If there are multiple wells within one cell and output for wells is requested then the output can contain the same particle more than once. As the model does not know which of the wells the particle should be assigned to (the program looks at the total well sink for the cell and cannot distinguish individual wells) the particle will be repeated for each of the wells within the cell.

41.4 Extraction of Particle Path Lines

It is too cumbersome to extract and plot the path lines for all of the thousands of particles that can be generated during a PT simulation. The PT Path line Extraction utility allows you to extract the path lines for specific particles if you have saved the intermediate locations in the Storing of Results *(p. 341)* dialogue.

To extract a particle path line you need a Particle ID. These can only be found after the simulation by evaluating the PT output. For example, you can find the particle ID by extracting the particle start locations that end in a specific well and then finding the ID numbers of the particles that you want in the shape file that was created.

In the Results tab, the PT Pathline Extraction *(p. 380)* utility is available to make this extraction.

Running from a batch file

The path line output retrieval program can be run from a command line. To execute the program, open a command line and type:

```
PtBinRetrieval.exe file_name.she extraction_num
```

The extraction_num is the item number in the table of extraction items in the PT Pathline Extraction *(p. 380)* dialogue. The extraction will proceed silently - that is without any messages. To run the extraction with the messages, you need to use

```
MZLaunch file_name.she -e PtBinRetrieval.exe
```

which will start the MZLaunch utility. Particle IDs can be found by using the PT Output Retrieval utility.

ADDITIONAL OPTIONS

42 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 Rivers and MIKE SHE *(p. 750)*
- Negative Precipitation *(p. 753)*
- Precipitation multiplier *(p. 754)*

Rivers

- Order of creating river links *(p. 754)*
- Irrigation River Source Factors *(p. 755)*
- Validation of HD Res file *(p. 756)*

Overland Flow

- Time-varying Overland Flow Boundary Conditions *(p. 756)*
- Simplified Overland Flow options *(p. 758)*
- Explicit Overland Flow Output *(p. 759)*
- Alternative low gradient damping function *(p. 759)*
- OL Drainage Options *(p. 760)*
- OL Drainage to Specified River H-points *(p. 761)*

Unsaturated Zone

- UZ-SZ exchange *(p. 763)*
- Transpiration during ponding *(p. 763)*
- Threshold depth for infiltration (2-Layer UZ) *(p. 764)*

Saturated Zone

- Sheet Pile Module *(p. 765)*
- SZ Drainage to Specified River H-points *(p. 770)*
- SZ Drainage Downstream Water Level Check *(p. 773)*
- Time varying hydraulic conductivity *(p. 773)*
- 2-norm reduction-criteria in the inner iteration loop *(p. 774)*

Water Quality

- Disable SZ solute flux to dummy UZ *(p. 775)*
- SZ boundary dispersion *(p. 775)*

Miscellaneous

- Maximum number of threads *(p. 776)*
- Including OpenMI *(p. 776)*
- Plot control for Detailed Time Series Output *(p. 777)*
- Extra Pre-Processing output *(p. 777)*
- Mean-Step-Accumulated time varying parameters *(p. 777)*

42.1 Climate

42.1.1 Temperature coupling between Rivers and MIKE SHE

Temperature is not a feature that can currently be modelled in MIKE SHE though MIKE 1D 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 the river either as base flow, overland or drain flow. The temperature is in turn received by MIKE 1D 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 OL 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 consistent 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

42.1.2 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.

42.1.3 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.

42.2 Rivers

42.2.1 Order of creating river links

By default MIKE SHE always processes the branches in the same order, regardless of sorting in MIKE+. To do the processing in the same order the branches are stored in MIKE+, set this extra parameter to true. This can be useful when a model is required to give the same results it gave with an older version of MIKE SHE, where the branches would always be processed in the order given by the river model setup:

Parameter Name	Type	Value
disable initial branch sorting	boolean	on

42.2.2 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:

$$Depth = MIN\left(\frac{C_s \cdot \Delta t}{A}, \frac{F_V \cdot V_L}{A} + \frac{F_D \cdot D_L \cdot \Delta t}{A}\right) \quad (42.1)$$

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 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.

42.2.3 Validation of HD Res file

In some cases you may want to turn off the time stamp validation for the MIKE 1D output files. For example, this may be useful if you are linking older existing river models as input for the MIKE SHE WQ.

Parameter Name	Type	Value
disable validation of HD res file	Boolean	On

42.3 Overland Flow

42.3.1 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.

42.3.2 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 river stream network as lateral inflow.

Parameter Name	Type	Value
no simple OL routing	Boolean	On

42.3.3 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 = 0.0001

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.

42.3.4 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 cal-

culated 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 the MIKE+ 2D Overland module.

To activate the alternate function, you must specify the following boolean parameter in the Extra Parameters *(p. 353)* 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 *(p. 523)* in the Reference manual.

42.3.5 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

42.3.6 OL Drainage to Specified River H-points

The Distributed Drainage option allows you to route drainage from the OL drainage directly to 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 river branches that are not defined in the MIKE SHE coupling dialog of MIKE+.

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 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 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 1D.

The drain code references the area that OL drainage is routed to the specified 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 con-

dition 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 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 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 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 Overland Drainage *(p. 280)*.
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 *(p. 285)* item in the data tree under the Overland Flow. 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\MIKE 1D> Reaches For <SZ\OL> 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 *(p. 99)*)

42.4 Unsaturated Zone

42.4.1 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.

42.4.2 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

42.4.3 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.

42.5 Saturated Zone

42.5.1 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	

42.5.2 SZ Drainage to Specified River H-points

The Distributed Drainage option allows you to route drainage from the SZ drains directly to 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 river branches that are not defined in the MIKE SHE coupling dialog of MIKE+.

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 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 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 1D.

The drain code references the area that drainage is routed to the specified 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 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 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 *(p. 331)*.
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\MIKE 1D> Reaches For <SZ\OL> 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 *(p. 99)*)

42.5.3 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

42.5.4 Time varying hydraulic conductivity

This set of extra parameters allows you to define the horizontal and vertical hydraulic conductivity 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

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

By default, the Time Series Types *(p. 155)* 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.

42.5.5 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

42.6 Water Quality

42.6.1 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 *(p. 753)* 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

42.6.2 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.

42.7 Miscellaneous

42.7.1 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

42.7.2 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.

42.7.3 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.

42.7.4 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

42.7.5 Mean-Step-Accumulated time varying parameters

Some parameters in MIKE SHE allow you to specify time varying dfs2 files as input by checking on a tickbox in the Data Tree item.

By default, the Time Series Types *(p. 155)* 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.

Parameter Name	Type	Value
mean step accumulated manning number	Boolean	On (Note: this is for OL)
mean step accumulated leakage coefficient	Boolean	On (Note: this is for OL)
mean step accumulated OL-drainage levels	Boolean	On (Note: this is for OL)
mean step accumulated OL-drainage time constants	Boolean	On (Note: this is for OL)
mean step accumulated horizontal conductivity	Boolean	On (Note: this is for SZ)
mean step accumulated vertical conductivity	Boolean	On (Note: this is for SZ)
mean step accumulated drainage levels	Boolean	On (Note: this is for SZ)
mean step accumulated drainage time constants	Boolean	On(Note: this is for SZ)

Note The code does not check for the time series type.

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/ Allen, G. R., Pereira, L. S., Raes, D and Smith, M. 1998. "Crop Evapotranspiration-Guidelines for computing crop water requirements." FAO Irrigation and Drainage Paper 56. FAO, Rome, Italy, 78-86.

/4/ Ammentorp, H.C., A. Refsgaard (1991): Operationalizing a three-dimensional solute transport model (in Danish). Report M4/7. Landfill project. Danish Environmental Protection Agency.

/5/ 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.

/6/ Bear, J (1979): Hydraulics of groundwater. McGraw-Hill Inc., 567 pp.

/7/ Bear, J. and A. Verruijt (1987): Modeling Groundwater Flow and Transport. D. Reidel Pub. Com., Dordrecht, Holland.

/8/ 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.

/9/ Brouwer, C., & Heibloem, M. (1986). Irrigation water management: irrigation water needs. Training manual, 3. FAO, Rome, Italy.

/10/ 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.

/11/ 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.

/12/ 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.

/13/ Camp Dresser & McKee Inc. (2001): Evaluation of Integrated Sur-

face Water and Groundwater Modeling Tools, Internal report, pp35

/14/ 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

/15/ 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

/16/ 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

/17/ 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.

/18/ FAO, 1979. Yield response to water. FAO irrigation and drainage paper no. 33. FAO, Rome Italy.

/19/ Fetter, C.W., 1992: Contaminant Hydrogeology. Macmillan Pub. Com., New York, New York.

/20/ Fleming G., 1975. *Computer Simulation Techniques in Hydrology,* Elsevier Environmental Science Series

/21/ Freeze, R.A., Cherry, J.A. (1979): Groundwater, Prentice-Hall, Inc., Englewood Cliffs, N.J.

/22/ 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.

/23/ 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.

/24/ 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.

/25/ 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.

/26/ 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.

/27/ 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.

/28/ 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.

/29/ Jensen, K.H., H.C. Ammentorp and T. Sevel (1984): Modelling solute transport in the unsaturated zone. Nordic Hydrology 15(4/5), 223-234.

/30/ Kaiser-Hill Company (2001): Model Code and Scenario Selection Report Site Wide Water Balance Rocky Flats Environmental Technology Site, Internal Report, pp66

/31/ 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

/32/ Klein, M., 1995. PELMO - Pesticide Leaching Model, version 2.01. Users Manual. Frauenhofer-Institut für Umweltche-mie und Ökotoxiko-logie, 57392 Schmallenberg

/33/ Knudsen, J., 1985a. WATBAL User's Guide. Danish Hydraulic Institute, Denmark

/34/ Knudsen, J., 1985b. WATBAL Hydrological Modelling System A Short Description. Danish Hydraulic Institute, Denmark

/35/ Konikow, L.F. and D.B. Grove (1977): Derivation of Equations describing Solute Transport in Groundwater, US.G.S. Water Resour. Invest. 77-19.

/36/ 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

/37/ 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.

/38/ 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.

/39/ 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.

/40/ 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.

/41/ Middlemis, H. (2004): Benchmarking Best Practice for Groundwater Flow Modelling, report to The Winston Churchill Memorial Trust of Australia, pp45

/42/ 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.

/43/ 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)

/44/ 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.

/45/ Refsgaard, J.C. and J. Knudsen, 1996. Operational validation and intercomparison of different types of hy7drological models, *Water Resources Research*, **32**(7): pp2189-2202

/46/ 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.

/47/ 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.

/48/ Scheidegger, A.E. (1961): General theory of dispersion in porous media. Jour. of Geophys. Research, Vol. 66, no. 10.

/49/ 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.

/50/ Thomas, R.G. (1973): Groundwater models. Irrigation and drainage.

Spec. Pap. Food Agricultural Organis. No. 21, U.N., Rome.

/51/ 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.

/52/ 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.

/53/ Vested, H.J., Justesen, P. and Ekebjærg, L. (1992): Advection-dispersion modelling in three dimensions. Appl. Math. Modelling, Vol. 16, pp 506-519.

/54/ 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.

/55/ 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.

/56/ 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

/57/ 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.

APPENDIX A



A.1 MIKE SHE Output Items

The available output from MIKE SHE depends on the processes selected in the Simulation Specification dialogue. Thus, for example, results for Overland Flow only appear when Overland flow is being calculated.

Some of the available output items are calculated as part of another process. For example, the depth of overland water is calculated based on seepage to and from the groundwater and as part of the MIKE 1D water level calculations, even if the overland flow is not directly simulated.

Furthermore, some of the output items require that more than one process be simulated. For example, the leaf area index is only available if both evapotranspiration and unsaturated flow are calculated.

In the absence of an explicit remark, the sign convention for MIKE SHE's output is positive in the positive direction. In other words, all flows in the direction of increasing X, Y and Z coordinates are positive. Thus, vertical downward flows, such as infiltration are negative.

Flows that do not have a direction are positive if storage or outflow is increasing. Thus, all flows leaving the model are positive, and water balance errors are positive if the model is generating water.

Also important to remember is that the output items related to flow are accumulated over the storing time step. In many cases, these values are required for the Water Balance program described in the section Using the Water Balance Tool *(p. 99)*. The values that are part of the water balance are automatically turned on when the water balance option is selected.

However, the output items that are not flows, such as temperature, water depth and Courant number represent the instantaneous value at the end of the storing time step.

Finally, some of the output items are actually input items. For example, precipitation is usually input as a time series for several polygons or grid code areas. The output file is a fully distributed dfs2 version of the input time series files.

The available output items for gridded data and time series data are listed in the following tables, with each table comprising the potential output items in each potential output file. Not all files are output for each simulation and not all Items are in each file.



Note The **Code**, in the Tables is a **Data Type Code** that is needed when importing time series items into the Detailed WM time series output *(p. 344)* dialogue.

List of Output Tables

- A.1.1 Output items for the Simulation Status (Detailed Time Series) *(p. 789)*
- A.1.2 Gridded output items in projectname_ET_AllCells.dfs2 *(p. 789)*
- A.1.3 Gridded output items in projectname_ET_UzCells.dfs2 *(p. 789)*
- A.1.4 Gridded output items in projectname_SnowMelt.dfs2 *(p. 790)*
- A.1.5 Gridded output items in projectname_overland.dfs2 *(p. 791)*
- A.1.6 Gridded output items in projectname_OLMultiCell.dfs2 *(p. 792)*
- A.1.7 Gridded output items in projectname_flood.dfs2 *(p. 792)*
- A.1.8 Gridded output items in projectname_wetland.dfs2 *(p. 792)*
- A.1.9 Gridded output items in projectname_2DUZ_AllCells.dfs2 *(p. 793)*
- A.1.10 Gridded output items in projectname_2DUZ_UzCells.dfs2 *(p. 793)*
- A.1.11 Gridded output items in projectname_3DUZ.dfs3 *(p. 794)*
- A.1.12 Gridded output items in projectname_2DIrrigation.dfs2 *(p. 795)*
- A.1.13 Gridded output items in projectname_3DIrrigation.dfs3 *(p. 795)*
- A.1.14 Time series output items in projectname_Irrigation-Log_<area>.dfs0 *(p. 796)*
- A.1.15 Gridded output items in projectname_2DSZ.dfs2 *(p. 796)*
- A.1.16 Gridded output items in projectname_3DSZ.dfs3 *(p. 796)*
- A.1.17 Gridded output items in projectname_2DSZFlow.dfs2 *(p. 797)*
- A.1.18 Gridded output items in projectname_3DSZflow.dfs3 *(p. 797)*
- A.1.19 Gridded output items in projectname_LR.dfs2 *(p. 798)*
- A.1.19 Gridded output items in projectname_LR.dfs2 *(p. 798)*
- A.1.20 Gridded output items in projectname_SimpleOL.dfs2 *(p. 799)*
- A.1.21 Gridded output items in projectname_ADOC.dfs2 *(p. 799)*
- A.1.22 Gridded output items in projectname_ADUZ3.dfs3 *(p. 800)*
- A.1.23 Gridded output items in projectname_ADSZx.dfs2 *(p. 800)*
- A.1.24 Gridded output items in projectname_ADSZ3D.dfs3 *(p. 801)*
- A.1.25 Gridded output items in projectname_PTSZ3D.dfs3 *(p. 801)*

Table A.1.1 Output items for the Simulation Status (Detailed Time Series)

Output Item	Item Type	Default unit	Time Series Type	Code
SimStatus: Basic time step length	TimeStep	hour	MeanStepAcc	145
SimStatus: SZ time step length	TimeStep	hour	MeanStepAcc	146
SimStatus: No. of SZ iterations / time step	ItemUndefined	IntCode	MeanStepAcc	147
SimStatus: Avg. no. UZ iterations / column / time step	ItemUndefined	IntCode	MeanStepAcc	148
SimStatus: No. of Overland iterations per time step	ItemUndefined	IntCode	MeanStepAcc	149

Table A.1.2 Gridded output items in projectname_ET_AllCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
precipitation rate	PrecipitationRate	millimeter/Day	MeanStepAcc	10
Air temperature	Temperature	degreeCelsius	MeanStepAcc	303
average water content in the rootzone	VolumetricWaterContent	One/One	Instantaneous	128
rooting depth	RootDepth	meter	Instantaneous	11
leaf area index	LAI	One/One	Instantaneous	12
crop coefficient	Fraction	One/One	Instantaneous	182

Table A.1.3 Gridded output items in projectname_ET_UzCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
reference evapotranspiration	ETRate	millimeter/Day	MeanStepAcc	355
ETref x Kc	ETRate	millimeter/Day	MeanStepAcc	340
actual evapotranspiration	ETRate	millimeter/Day	MeanStepAcc	15

Table A.1.3 Gridded output items in projectname_ET_UzCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
actual transpiration	ETRate	millimeter/Day	MeanStepAcc	16
actual soil evaporation	ETRate	millimeter/Day	MeanStepAcc	13
actual evaporation from interception	ETRate	millimeter/Day	MeanStepAcc	17
actual evaporation from ponded water	ETRate	millimeter/Day	MeanStepAcc	18
canopy interception storage	Interception	millimeter	Instantaneous	19
evapotranspiration from SZ	ETRate	millimeter/Day	MeanStepAcc	14

Table A.1.4 Gridded output items in projectname_SnowMelt.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
Total Snow storage	StorageDepth	millimeter	Instantaneous	100
Dry Snow storage	StorageDepth	millimeter	Instantaneous	254
Wet Snow storage	StorageDepth	millimeter	Instantaneous	255
Wet Snow storage Fraction	Fraction	-	Instantaneous	256
Fraction of cell area covered by Snow	Fraction	-	Instantaneous	257
Precipitation + Irrigation added to Snow	Precipitation-Rate	millimeter/Day	MeanStepAcc	258
Total Snow converted to Overland Flow	Precipitation-Rate	millimeter/Day	MeanStepAcc	259
Freezing due to Air temperature	storagechang-erate	millimeter/Day	MeanStepAcc	260
Melting due to Air temperature	storage change rate	millimeter/Day	MeanStepAcc	261
Melting due to SW Solar Radiation	storage change rate	millimeter/Day	MeanStepAcc	262
Melting due to energy in Rain	storage change rate	millimeter/Day	MeanStepAcc	263
Snow evaporation	ETRate	millimeter/Day	MeanStepAcc	99

Table A.1.5 Gridded output items in projectname_overland.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
depth of overland water	WaterDepth	meter	Instantaneous	61
depth of overland water (TS average)	WaterDepth	meter	Instantaneous	374
depth of overland water (TS min)	WaterDepth	meter	Instantaneous	375
depth of overland water (TS max)	WaterDepth	meter	Instantaneous	376
overland flow in x-direction	Discharge	m3/Sec	MeanStepAcc	58
overland flow in y-direction	Discharge	m3/Sec	MeanStepAcc	59
flow from overland to river	Discharge	m3/Sec	MeanStepAcc	342
overland flow velocity (TS average)	Flow velocity	meter/sec	Instantaneous	377
overland flow velocity (TS min)	Flow velocity	meter/sec	Instantaneous	378
overland flow velocity (TS max)	Flow velocity	meter/sec	Instantaneous	379
OL drainage to river or MOUSE	Discharge	m3/Sec	MeanStepAcc	62
OL Drain Stored Volume	WaterVolume	m3	Instantaneous	356
OL Drain Storage Depth	StorageDepth	millimeter	Instantaneous	367
OL Drain Storage Inflow (flow)	Discharge	m3/Sec	MeanStepAcc	357
OL Drain Storage Inflow (flux)	storagechang-erate	millimeter/Day	MeanStepAcc	366
drain flow with OC storing frequency	Discharge	m3/Sec	MeanStepAcc	114
flooded (yes no)	Logical	-	Instantaneous	90
OL flow from flooded areas to river	Discharge	m3/Sec	MeanStepAcc	91
net precipitation rate for AD	Precipitation-Rate	millimeter/Day	MeanStepAcc	60
Overland flow to MOUSE	Discharge	m3/Sec	MeanStepAcc	152
External sources to Overland (for OpenMI)	Discharge	m3/Sec	MeanStepAcc	238
overland water elevation	Elevation	meter	Instantaneous	241
Mean OL Wave Courant number (explicit OL)	Fraction	-	MeanStepAcc	304
Max OL Wave Courant number (explicit OL)	Fraction	-	MeanStepAcc	305

Table A.1.5 Gridded output items in projectname_overland.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
Max Outflow OL-OL per Cell Volume (explicit OL)	Fraction	-	MeanStepAcc	306
External inflow to OL drain (for OpenMI)	Discharge	m3/Sec	MeanStepAcc	359

Table A.1.6 Gridded output items in projectname_OLMultiCell.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
depth of Multi-Cell overland water	WaterDepth	meter	Instantaneous	307
Multi-Cell overland water elevation	Elevation	meter	Instantaneous	317

Table A.1.7 Gridded output items in projectname_flood.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
H Water Depth	WaterDepth	meter	Instantaneous	369
P Flux	FlowFlux	m3/Sec/m	Instantaneous	370
Q Flux	FlowFlux	m3/Sec/m	Instantaneous	371

Table A.1.8 Gridded output items in projectname_wetland.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
Water content in root zone (2-layer UZ)	Volumetric WaterContent	-	Instantaneous	141
Water content below root zone (2-layer UZ)	Volumetric WaterContent	-	Instantaneous	142

Table A.1.8 Gridded output items in projectname_wetland.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
Maximum water content (2-layer UZ)	Volumetric WaterContent	-	Instantaneous	143
Minimum water content (2-layer UZ)	Volumetric WaterContent	-	Instantaneous	144

Table A.1.9 Gridded output items in projectname_2DUZ_AllCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
infiltration to UZ (negative)	Infiltration	millimeter/Day	MeanStepAcc	121
exchange between UZ and SZ (pos.up)	Recharge	millimeter/Day	MeanStepAcc	122
bypass flow UZ (negative)	Infiltration	millimeter/Day	MeanStepAcc	124
UZ deficit	Deficit	millimeter	Instantaneous	57
infiltration to macropores (negative)	Infiltration	millimeter/Day	MeanStepAcc	125
macropore recharge to SZ (negative)	Recharge	millimeter/Day	MeanStepAcc	126
Total recharge to SZ (pos.down)	Precipitation-Rate	millimeter/Day	MeanStepAcc	310
average soil moisture content in top 5 compartments	Volumetric WaterContent	-	Instantaneous	37

Table A.1.10 Gridded output items in projectname_2DUZ_UzCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
rate of change in UZ storage	storagechange rate	millimeter/Day	MeanStepAcc	119
epsilon calculated in UZ	EpsilonUZ	millimeter	Instantaneous	123
groundwater levels used by UZ	Elevation	meter	Instantaneous	140
column mean macropore water content above GW	Volumetric WaterContent	-	Instantaneous	139

Table A.1.10 Gridded output items in projectname_2DUZ_UzCells.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
column total net exchange matrix to macropores	ExchangeRate	millimeter/Day	MeanStepAcc	136
column total exchange matrix to macropores	ExchangeRate	millimeter/Day	MeanStepAcc	137
column total exchange macropores to matrix	ExchangeRate	millimeter/Day	MeanStepAcc	138
groundwater feedback to the unsaturated zone	Recharge	millimeter/Day	MeanStepAcc	45
External sources to UZ columns (for OpenMI)	ExchangeRate	millimeter/Day	MeanStepAcc	334
UZ time step reduction count	IntegerCode	intCode	StepAcc	383
UZ iteration count	IntegerCode	intCode	MeanStepAcc	384

Table A.1.11 Gridded output items in projectname_3DUZ.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
unsaturated zone flow	DarcyVelocity	millimeter/Day	MeanStepAcc	117
water content in unsaturated zone	Volumetric WaterContent	-	Instantaneous	118
pressure head in unsaturated zone	PressureHead	meter	Instantaneous	159
saturation in unsaturated zone	Fraction	-	Instantaneous	380
root water uptake	ETRate	millimeter/Day	MeanStepAcc	129
macropore water content	Volumetric WaterContent	-	Instantaneous	130
macropore flow	DarcyVelocity	millimeter/Day	MeanStepAcc	131
exchange from matrix to macropores	ExchangeRate	millimeter/Day	MeanStepAcc	132
exchange from macropores to matrix	ExchangeRate	millimeter/Day	MeanStepAcc	133
net exchange from matrix to macropores	ExchangeRate	millimeter/Day	MeanStepAcc	134
microbiological clogging UZ conductivity	HydrConductivity	meter/Sec	Instantaneous	127

Table A.1.11 Gridded output items in projectname_3DUZ.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
UZ soil temperature	Temperature	degreeKelvin	MeanStepAcc	210
External sources to UZ nodes (for OpenMI)	ExchangeRate	millimeter/Day	MeanStepAcc	333

Table A.1.12 Gridded output items in projectname_2DIrrigation.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
irrigation: actual water content in root zone	VolumetricWaterContent	-	Instantaneous	20
irrigation: soil moisture deficit in root zone	Deficit	millimeter	Instantaneous	135
total irrigation	IrrigationRate	millimeter/Day	MeanStepAcc	21
irrigation from river	IrrigationRate	millimeter/Day	MeanStepAcc	26
irrigation from wells	IrrigationRate	millimeter/Day	MeanStepAcc	28
irrigation from external source	IrrigationRate	millimeter/Day	MeanStepAcc	22
irrigation index	IrrigationIndex	-	Instantaneous	23
irrigation shortage	IrrigationRate	millimeter/Day	MeanStepAcc	24
irrigation total demand	IrrigationRate	millimeter/Day	MeanStepAcc	25
sprinkler irrigation	IrrigationRate	millimeter/Day	MeanStepAcc	153
drip and sheet irrigation	IrrigationRate	millimeter/Day	MeanStepAcc	154

Table A.1.13 Gridded output items in projectname_3DIrrigation.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
ground water extraction for irrigation	Discharge	m3/Sec	MeanStepAcc	27

Table A.1.14 Time series output items in projectname_IrrigationLog_<area>.dfs0

Output Item	Item Type	Default unit	Time Series Type	Code
Irrigation	WaterVolume	m3	MeanStepAcc	
Demand	WaterVolume	m3	MeanStepAcc	
Shortage	WaterVolume	m3	MeanStepAcc	
Irrigated area	SurfaceArea	m2		
Demand area	Surface area	m2		
Log area	Surface area	m2		

Table A.1.15 Gridded output items in projectname_2DSZ.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
depth to top phreatic surface (negative)	HeightAbove Ground	meter	Instantaneous	106
elevation of top phreatic surface	Elevation	meter	Instantaneous	368
depth to bottom phreatic surface (negative)	HeightAbove Ground	meter	Instantaneous	381
elevation of bottom phreatic surface	Elevation	meter	Instantaneous	382

Table A.1.16 Gridded output items in projectname_3DSZ.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
head elevation in saturated zone	Elevation	meter	Instantaneous	101
saturated thickness of SZ layer	Length	meter	Instantaneous	327
SZ horizontal conductivity (for DA-OpenMI)	HydrConductivity	meter/Sec	Instantaneous	110
SZ vertical conductivity (for DA-OpenMI)	HydrConductivity	meter/Sec	Instantaneous	111

Table A.1.17 Gridded output items in projectname_2DSZFlow.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
seepage flow SZ -overland	DarcyVelocity	millimeter/Day	MeanStepAcc	107
seepage flow overland - SZ (negative)	DarcyVelocity	millimeter/Day	MeanStepAcc	108
SZ flow from flooded areas to river	Discharge	m3/Sec	MeanStepAcc	116
External inflow to SZ drain (for OpenMI)	Discharge	m3/Sec	MeanStepAcc	300
seepage flow Deficit (FEFLOW coupling)	StorageDepth	millimeter	Instantaneous	329
seepage flow SZ-OL application deferred	StorageDepth	millimeter	Instantaneous	63

Table A.1.18 Gridded output items in projectname_3DSZflow.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
3D UZ recharge to SZ (negative)	Recharge	millimeter/Day	MeanStepAcc	113
groundwater flow in x-direction	Discharge	m3/Sec	MeanStepAcc	102
groundwater flow in y-direction	Discharge	m3/Sec	MeanStepAcc	103
groundwater flux in x-direction	DarcyVelocity	millimeter/Day	MeanStepAcc	372
groundwater flux in y-direction	DarcyVelocity	millimeter/Day	MeanStepAcc	373
groundwater flux in z-direction	DarcyVelocity	millimeter/Day	MeanStepAcc	104
SZ head elevation stored with SZ flows	Elevation	meter	Instantaneous	251
groundwater extraction	PumpingRate	m3/Sec	MeanStepAcc	109
SZ exchange flow with river	Discharge	m3/Sec	MeanStepAcc	115
SZ drainage flow from point	Discharge	m3/Sec	MeanStepAcc	112
SZ flow to general head boundary	Discharge	m3/Sec	MeanStepAcc	105
SZ flow to MOUSE	Discharge	m3/Sec	MeanStepAcc	151
External sources to SZ (for OpenMI)	Discharge	m3/Sec	MeanStepAcc	239

Table A.1.19 Gridded output items in projectname_LR.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
recharge to interflow reservoirs	Discharge	m3/Sec	MeanStepAcc	29
interflow from interflow reservoirs	Discharge	m3/Sec	MeanStepAcc	30
percolation from interflow reservoirs	Discharge	m3/Sec	MeanStepAcc	31
interflow reservoir storage	WaterVolume	m3	Instantaneous	32
change in interflow reservoir storage	Discharge	m3/Sec	MeanStepAcc	33
percolation to baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	34
percolation to dead zone of baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	211
river discharge to baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	252
river discharge to dead zone of baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	253
baseflow from baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	35
groundwater feedback from baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	36
pumping from baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	44
pumping for irrigation from baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	248
storage in baseflow reservoir	WaterVolume	m3	Instantaneous	46
dead zone storage in baseflow reservoir	WaterVolume	m3	Instantaneous	212
change in subcatchment storage in baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	38
change in dead zone storage in baseflow reservoir	Discharge	m3/Sec	MeanStepAcc	213

Table A.1.20 Gridded output items in projectname_SimpleOL.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
simple overland water depth	WaterDepth	meter	Instantaneous	155
simple overland exchange to lower zone or river	Discharge	m3/Sec	Instantaneous	156
simple overland recharge	Discharge	m3/Sec	Instantaneous	157
water level in M11 h-point	Elevation	meter	Instantaneous	214
discharge in M11 q-point	Discharge	m3/Sec	MeanStepAcc	215

Table A.1.21 Gridded output items in projectname_ADOC.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
Overland concentration	Concentration	gram/M3	Instantaneous	216
Overland sorbed concentration	Concentration_4	gram/gram	Instantaneous	217
Overland fixed(undef) concentration	ItemUndefined	UnitUndefined	Instantaneous	335
Overland mass per area	MassPerUnitArea	gram/M2	Instantaneous	218
Air temperature	Temperature	degreeCelsius	Instantaneous	219
Overland temperature	Temperature	degreeCelsius	Instantaneous	331
External sources to Overland (for OpenMI)	SpecificSoluteFluxPerArea	gram/M2/Day	MeanStepAcc	266
Overland MIKE ECO Lab Output	ItemUndefined	UnitUndefined	MeanStepAcc	311
Overland Drainage concentration	Concentration	gram/M3	Instantaneous	362
Overland Drainage fixed(undef) concentration	ItemUndefined	UnitUndefined	Instantaneous	365
Overland Drainage mass per area	MassPerUnitArea	gram/M2	Instantaneous	363
Overland Drainage temperature	Temperature	degreeCelsius	Instantaneous	364

Table A.1.22 Gridded output items in projectname_ADUZ3.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
UZ concentration (matrix phase)	Concentration	gram/M3	Instantaneous	220
UZ sorbed concentration (matrix phase)	Concentration_4	gram/gram	Instantaneous	221
UZ fixed(undef) concentration (matrix phase)	ItemUndefined	UnitUndefined	Instantaneous	336
UZ mass flux (matrix phase)	SpecificSoluteFluxPerArea	gram/M2/Sec	MeanStepAcc	224
UZ concentration (macropore phase)	Concentration	gram/M3	Instantaneous	222
UZ sorbed concentration (macropore phase)	Concentration_4	gram/gram	Instantaneous	223
UZ fixed(undef) concentration (macropore phase)	ItemUndefined	UnitUndefined	Instantaneous	337
UZ mass flux (macropore phase)	SpecificSoluteFluxPerArea	gram/M2/Sec	MeanStepAcc	225
UZ soil temperature	Temperature	degreeCelsius	Instantaneous	226
UZ MIKE ECO Lab Output	ItemUndefined	UnitUndefined	MeanStepAcc	312

Table A.1.23 Gridded output items in projectname_ADSZx.dfs2

Output Item	Item Type	Default unit	Time Series Type	Code
External mass flux to SZ drain (for OpenMI)	SpecificSoluteFluxPerArea	gram/M2/Day	MeanStepAcc	302
Drain temperature	Temperature	degreeCelsius	Instantaneous	332

Table A.1.24 Gridded output items in projectname_ADSZ3D.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
SZ concentration (mobile phase)	Concentration	gram/M3	Instantaneous	227
SZ sorbed concentration (mobile phase)	Concentration_4	gram/gram	Instantaneous	228
SZ fixed(undef) concentration (mobile phase)	ItemUndefined	UnitUndefined	Instantaneous	338
SZ concentration (immobile phase)	Concentration	gram/M3	Instantaneous	229
SZ sorbed concentration (immobile phase)	Concentration_4	gram/gram	Instantaneous	230
SZ fixed(undef) concentration (immobile phase)	ItemUndefined	UnitUndefined	Instantaneous	339
SZ soil temperature	Temperature	degreeCelsius	Instantaneous	231
SZ porosity	PorosityCoefficient	-	Instantaneous	232
External sources to SZ (for OpenMI)	SpecificSoluteFluxPerArea	gram/M2/Day	MeanStepAcc	267
SZ MIKE ECO Lab Output	ItemUndefined	UnitUndefined	MeanStepAcc	313

Table A.1.25 Gridded output items in projectname_PTSZ3D.dfs3

Output Item	Item Type	Default unit	Time Series Type	Code
Number of particles	IntegerCode	intCode	Instantaneous	233
Accumulated particle count	IntegerCode	intCode	Instantaneous	250
Number of registered particles	IntegerCode	intCode	Instantaneous	234
Most recent registration zone code	IntegerCode	intCode	Instantaneous	235
Average age	Age	Year	Instantaneous	236
Average transport time to nearest registration cell	Age	Year	Instantaneous	237

INDEX

Symbols

.fif file 71, 363
.frf file 91
.hot.sheres 91
.sheres file 91

Numerics

16-bit colours 180
2-norm reduction-criteria 627
32-bit colours 180
3D Finite Difference Method . . .306, 623
64-bit CPU 32, 88

A

Actual time step 85
ET 193
OL 193
UZ 193
Air temperature 48
elevation correction 461
American units 145
ArcGIS 159
Area Inundation Exchange 492
Calculation - Exchange Flows . 492
AROOT 442
Auxiliary Variables
ECO Lab 726
Average steady-state river conductance . 632
Averjanov 419

B

Baseflow 647
Baseflow Reservoirs 310
Batch files 86
water balance 106
Bathymetry 271
Bilinear interpolation 169
Boundary condition
Available 326, 328
Internal 327
Outer 325
Specification 325
Boundary Conditions
3D Finite Difference Method . . 632
Boundary conditions
Richards Equation 581
boundary conditions
Dirichlet 632
Fourier 633
Neumann 632
Bypass Flow 291

C

Calculation
Baseflow 647
Dead Zone Storage 646
Exchange Flows 492
Interflow 644
Interflow Percolation 646
Calculations
Lumped UZ 620
Calibration
Water Quality 717
Campbell 418
Campbell/Burdine 421
Canopy
ET 454
Canopy Evaporation 438
Canopy Interception 438, 459
Channel Flow 471
Classification
Partial automatic 298
Specified 299
Climate 47
Coefficients
Evapotranspiration 445
Color 180
Column Classification 289
Computational Layers 323
Concatenation 92
Concentration units 221, 726
Conductivity 418
Conjugate Gradient 204
Consecutive simulations 92
Constants
ECO Lab 726
Control Parameters
OL 195
SZ 201
UZ 200
Coupling
Unsaturated Zone - Saturated Zone 601

UZ and SZ 61, 65
Coupling Procedure
Unsaturated Zone - Saturated Zone . 602
CPU Speed 33
Crop Coefficient 453
Cross-section
River Link 480
Current Layer 186

D
Data Tree 40
Database
Example 412
Dead Zone Storage 646
Degree-day Melting or Freezing Coefficient 245
DEM 46
Demo Limits 31
Derived Outputs
ECO Lab 726
Detention Storage 54, 274
dfs2 files 44
Display Properties 180
Drain Codes 335
Drain Level 334
Drain Time Constant 335
Drainage 65, 655
PCG Solver 636
Saturated Zone 635
SOR 636
dual domain mass transfer 727
Dual porosity 719

E
ECO Lab 220, 726
Constants 222
Forcings 222
in MIKE SHE 731
Running with MIKE SHE 736
Templates 726
templates 726
units 221, 726
ET
Actual 456
Canopy 454
Ponded Water 454
Saturated Zone 455
Unsaturated Zone 454
EUM Data Units
change 146
Default units 145
Imperial 145
SI 145
EUM data units 143
Evaporation 457
Evapotranspiration . . 47, 437, 453, 457
Crop Reference ET 48, 238
Parameters 409
Reference ET 238
Evapotranspiration Parameters . . . 251
Exchange Flows 492
Exchange Mechanisms
Surface Water/Aquifer 471
Extra Parameters 353, 749

F
FAO-56 458
fif file 71
Finite Difference 518
Finite Difference Method 515
Fixed species 224, 734
Flood codes 270
Flooded Area 492
Flow Storing Requirements 741
Forcings
ECO Lab 726
Freezing coefficient 245
frf file 91

G
Geologic Model 63, 651
Conductivity values 652
Lenses 64, 653
Geological Layers 313
Geological Lenses 314
Geological Units 312
Google Earth 230
Gradual activation of SZ drainage . . 627
Gravity Flow 587
Green and Ampt 60, 289
Grid editor
2D to 3D Layer Mapping 175
MIKE SHE vs MIKE 3 161

overview 161
simple math 174
Grid series
Output 352
Ground water table 298
Groundwater Table 304

H
Hardware requirements 32
Horizontal Extent 356
Horizontal Interpolation 652
Hot start 44
Hydraulic Conductivity
Horizontal 315
Vertical 316
Hydraulic conductivity
time varying 773
hyper threading 32

I
Image overlays 182
Imperial units 145
Import 345, 348, 404
TAB delimited text file 405
Index
Ecological 735
Infiltration Fraction 237
Infrastructure 573
Initial Conditions
Gravity Flow 588
Initial conditions
3DFinite Difference Method . . 585
initial conditions
saturated zone 637
Initial Potential Head 324
Integer grid codes 167
polygons 168
Interactive Map 401
Interflow calculation 644
Interflow Percolation 646
Interflow Reservoirs 308
Interpolation 169, 172, 173
Interpolation Method
ECO Lab 220
Inverse distance 173
Irrigation
Application methods 263
Demand 265
Parameters 410
Priorities 267
Sources of water 259
Water sources 258
Irrigation Command Areas
Water Source Types - Single Well . 261

K
Kristensen and Jensen method 439

L
LAI 50, 458
Land use 49
Lapse rate 467
Layer numbering 161
Leaf Area Index 458
Leaf Area Index (LAI) 50
Leakage Coefficient 276
Lenses 64
Linear Reservoir
Limitations 649
One Outlet 639
Subcatchments 643
Two Outlets 640
Linear Reservoir Method 305, 638
Description 640
Local depressions 564, 656
Log files 92
Low gradient damping function 523
Lower Level 355

M
Macropore Flow 591
Macropores
transport in 689
Manning number 273
Manning's M 53
Maps
background 40
Maximum Residual Error
PCG 626
SOR 631
Maximum Wet Snow Fraction 246
Melting coefficient 245
Melting Coefficient for Solar Radiation 245

MIKE 1D 471
MIKE FLOOD 57
MIKE Hydro River
  Unsaturated zone 494
MIKE+ 94
minimum flow area 522
Minimum Snow Storage 246
Model chains 92
Model Domain 228
Model domain 44
Model Grid 228
Model grid 44
Model Limits
  Demo version 31
Model limits
  Licensed version 30
MODFLOW 62, 663
Multi-cell overland flow 547
Multi-core 32
multi-core CPUs 32
Multiple simulations 92

N

Net Rainfall Fraction 236
NON-UTM 230
Numerical formulation
  SOR 628
Numerical layers 65
Numerical Solution 519, 577

O

OpenMI 88, 776
  Temperature 750
Option Distribution 337
Output
  file types 91
  items 92
  log files 92
  multiple simulations 92
  results concatentation 92
Overland Flow 271, 286, 515
  Boundary conditions 54
Overland flow
  alternative damping function . . . 759
  boundary conditions 522
  catchment based options 758
  damping function 523
  detailed output option 759
  detention storage 54
  Diffusive wave approximation . 515
  embankments 54, 543
  evaluating 546
  extra parameter options 756
  Mannings M 53
  multi-cell 547
  Multi-cell limitations 533
  Multi-cell method 529
  no flow boundary 543
  performance 544, 547
  seperated flow areas 543
  Simplified method 533
  stagnant water 545
  Stickler coefficient 517, 541
  SZ exchange 634
  Threshold gradient 525, 546
  time varying boundaries 756
  turning off 545
  velocities 543
  velocity 93
  water balance correction 520
Overland Flow Routing
  Coupling 537
  Directly to the river 758
  Theory 534
Overlays 180
Over-relaxation 205

P

Parallelization 88
Particle Tracking 741
Paved areas 570
Paved Runoff Coefficient 281
PCG Steady State Solver 632
Penman-Monteith 48, 238
Penman-Montieth 458
pF scale 413
Plant Transpiration 439
Ponded Water
  ET 454
Potential flow
  PCG 624
potential flow 624
Precipitation 47, 233

elevation correction 233, 462
infiltration fraction 237
net rainfall 236
Rate 234
time series 234
Precipitation Lapse Rate 236
Pre-conditioned Conjugate Gradient . 623
Preprocessed Data 363
Pre-processed data 363
.fif file 363
editing 364
land use 73
MIKE Hydro 73
SZ drainage 75
UZ 74
viewing 364
Processes
ECO Lab 726
Properties Editor 413
PT Simulations
Output 742
Pumping Wells 338

R

Rainfall
time series 234
RAM 33
Recharge 94
Redistribution of ponded water 758
Reference ET 458
Relaxation Coefficient
SOR 631
Release 2011 662
Results 369
detailed time series output 69
Storage 341
Results Viewer
Gridded Data 371
Richards Equation 576
River Links 231
River links 477
River Overlays 185
River-Aquifer Exchange 485
Full Contact 486
Reduced Contact 487, 488
Root depth 50
Root Distribution 441
Root Mass Distribution Parameter . . 255

S

Saturated Flow 623
Saturated flow 62
OL exchange 634
Saturated Zone 304
ET 455
Saturated zone
drainage 635, 655
Saturation
definition 413
Separated Flow 278, 279
Service packs 34
Shape overlays 184
Sheet Pile Module 638, 765
Input File 766
Leakage Coefficient 765
Location 765
Sheet piling 765
sheres file 91
Short wave radiation 239
shp file 45
SI 145
Simplified ET 447
Simplified Macropore Flow 590
Simulation
hot start 189
start/end dates 188
Simulation Period 188
Simulation Specification 186
Simulation Title 188
Snow melt 48, 233
detailed description 461
Snow water equivalent 469
Snowmelt Constants 243
Soil Evaporation 444
Soil Moisture 449
Soil Moisture Retention Curve 415
Soil Profile - Definitions 293
Soil profiles 58
Soil properties
2-Layer UZ 300
Soils database 58
Solute Transport
Requirements 715
Results 716

Solver parameters 43
Sources and sinks 586
Spatial data
  stationary real 161
  time varying 162
Spatial resolution 616
Species
  Type 224, 734
Specific yield 62, 64, 74, 617
State Variables
  ECO Lab 726
steady-state 489
Steady-state river water depth 632
storage 626
Storage Coefficient 317
Storage terms
  PCG 626
Storing time step 85
Storing time steps 193
Subcatchments 230
Sublimation 439, 453
Successive Overrelaxation 628
Surface Depth
  ET 302
Surface Drainage 331
Surface water storage 573
Surfer data 46
Suspended species 224, 734
SZ
  conceptual model 63
  drainage 65
  finite difference 63
  lenses 64
  linear reservoir 63
  specific yield 62, 64, 75

T
Temperature
  air 48
  elevation correction 233
Temperature Lapse Rate 241
Templates 726
Threshold gradient 525, 546
Threshold Melting Temperature 244
Time Series 370
Time series
  detailed output control 777
Time series output 344
Time series output, WQ 347
Time step
  Control 190
  Maximums 191
  Storing time steps 193
Time step control 43
Time varying
  Hydraulic conductivity 773
Time varying data 162
Topography 232
Transpiration 458
Triangular interpolation 172
Two-Layer Water Balance 589
  Simplified ET 447
Type
  Species 224, 734

U
Unsaturated Flow 575
Unsaturated flow 57
Unsaturated Zone 288
  ET 454
Upper Level 355
UTM Zones 229
UZ
  bypass flow 59
  discretization 59
  dry soils 60
  Green and Ampt 60
  initial conditions 59
  macropore flow 59
  mass balance errors 61
  soil profiles 58
  soils database 58
  specific yield 62, 64, 75
  sub-grid variability 59
UZ Classification 60, 297
UZ Coupling 648
UZ-SZ
  Coupling 61
  limitations 62, 618
UZ-SZ Coupling
  Evaluation 617

V
Van Genuchten 417, 420

Vegetation 50, 249
Development Table 409
Properties 257, 407
Stages 408
Velocities 543
Velocity
overland flow 93
Veritical Interpolation 653

W

Water Balance
units 106
Water balance 99
limitations 130
standard types 131
types 108
Water Balances
batch files 106
custom 136
Water Depth - Initial 275
Water infrastructure 573
Water Quality
Calibration 717
Limitations 715
Requirements 715
Results 716
Time step control 716
Water quality 42
mass balance 719
turning on 187
weir formula 522
Well Database Overlays 185
Well editor 401
Well Fields 742
Well Filters 403
Well Locations 402
Wells
Layers 404
Wet Lapse Rate 242
WM Requirements 741