

MODFLOW with ModelMuse - getting started

2D vertical synthetic model development and analysis

1 Outline

This will be our attempt to create a more realistic (GW) flow model matching the very basic River Bank Filtration scenario. We will be using exactly the same codes those we used for the 2D horizontal model: [MODFLOW-2005](#) (v. 1.12.00) and the Graphical User Interface (GUI) [ModelMuse](#) (v. 5.4.0.0). Additionally, we will work with [MODPATH](#) (v. 6.0.01) for exploring streampath to the wells.

Our domain will still be a rectangular shaped (for simplicity), but a 2-dimensional (2D) vertically oriented one with Dirichlet boundaries in the left (inflow) and right (outflow) ends. The top will include a river (Cauchy - RIV) and a well (Neumann) boundaries. We will again consider a confined aquifer. In addition, we will include a variety of domain complexities or conditions such as multiple conductivities (K_s), Recharge zones (R_s). The (*final*) conceptual setup of our domain will be as is in the figure below. As was in the 2D Horizontal model, we will develop this set-up in a step-wise manner.

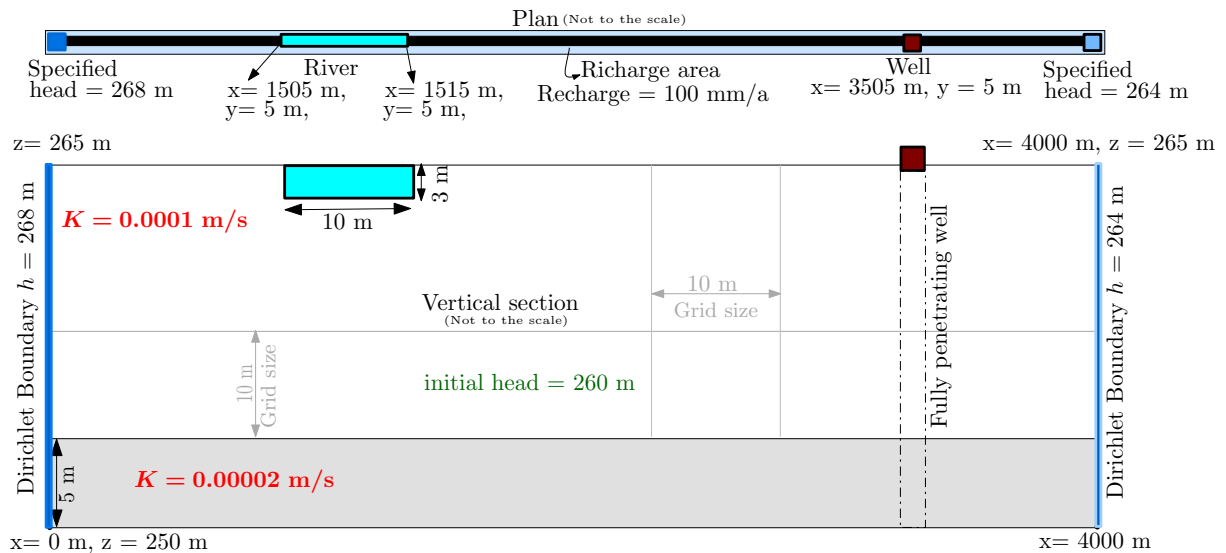


Figure 1: The conceptual setup for the 2D vertically oriented model. The figure at the top shows the plan view and the bottom one the vertical section

The overall goal of this exercise will be to learn to tackle the vertically oriented 2D domains. In addition, to observe the impact of different boundary (well and river) and domain (different K_s) conditions on the GW flow. The gained knowledge can be extended to more practical 2D domains and developing the more complete 3D groundwater flow models.

2 Model geometry and domain discretization

- ☐ Most of the steps that we follow in this model setup will be similar to that we followed for the 2D horizontal model. The horizontal model documentation can be used as a complementary document.
- ☐ Double-click on the ModelMuse icon  on your desktop - an initial ModelMuse Window opens
- ☐ Select „Create new MODFLOW model“ - a new Window opens
 - ☐ in the new Window: let the „Length unit“ be **Meters** and the „Time unit“ be **Seconds**
 - ☐ press „Next“ until you can **specify the grid**
- ☐ Specify initial grid
 - ☐ select „**MODFLOW-2005**“ as the MODFLOW Version
 - ☐ set $x = 0$, $y = 10$, $z = 250$ for grid origin
 - ☐ set further model geometry data: „number of columns“ = **40**, „number of rows“ = **1**, „number of layers“ = **1**, „model top“ = **265**, and „upper aquifer“ = **250**
 - ☐ Also, set „Column width“ = **10** and Row width = **10**
 - ☐ press „Finish“ to close the Window
 - ☐ select „File“/„Save as“ in the menu bar to save the model input data giving a file name (e.g.,) „mysecondmodel“ by using the file format **mmZLib**
- ☐ Set vertical layer properties
 - ☐ select „Model“ in the main menu and then the „Model Layer Groups“
 - ☐ click the „basics“ tab and give an aquifer name, e.g., „Confined_Aquifer_1“ or keep the default name
 - ☐ let the „Layer type“ be „**Confined**“ as we have a confined aquifer
 - ☐ click on the „Discretization“ tab and in the „Vertical discretization“ enter **5**
 - ☐ select „Uniform spacing“ and keep the default values. The last two steps divides the single layer aquifer into five sublayers with uniform spacing.
 - ☐ press „OK“ to close the Window

Important!

!! „Model Layer Group“ is a very important feature particularly for the vertically oriented models.



!! An attempt to learn about different options should be made

3 Model Structure and Parameters


- ☐ Enter basic aquifer data
 - ☐ select „Data“ from the menu bar and then „Edit Data Sets“/„Required“
 - ☐ select „Layer definition“ on the left side, then check „Model_Top“ and „Upper_Aquifer_Bottom“ (values should be 265 and 250, resp.)

Note 1

▣ **The name of the aquifer bottom may change if it was changed in the „MODFLOW Layer Groups“**

- Select „Hydrology“:
 - select „Kx“ and set „Kx“ = 0.0001 (K_x unit is m/s by default)
 - select „Modflow_Initial_Head“, then set initial head = **260** (in the box)
 - press „Apply“ and „Close“ to close the Window
- Setting the low permeable zone
 - select „Create rectangle object“ in the graphical menu (the square symbol  two rows below „Customize“)
 - click on lower left corner of grid in „front view“ (below the plan view) with left mouse button, press left mouse button again, keep it down, move cursor to (4000, 255), and release the mouse button
 - name the object (e.g., „small_k“)
 - tick box „Set values of intersected cells“
 - select tab „Data Sets“, then move to „Required“/„Hydrology“/„Kx“
 - replace the value = 0.0001 by 0.00002 and press „OK“
- Verify the setup of low permeable zones
 - click on „Data visualisation“ icon ( below the „Help“ menu in the menu bar)
 - click on „Color grid“
 - in „Data set or boundary condition“ navigate to and select: „Data sets“/„Required“/„Hydrology“/„Kx“
 - Press „Apply“ - the **top layer** should appear red and the **bottom layer** blue in the **front view** of the model

4 Model Boundary Conditions

- Setting the defined model boundary conditions
 - select „Model“/„MODFLOW Packages and Programs“ in the menu bar
 - select „Boundary conditions“/„Specified head“ on the left side, then tick the sub-item „CHD“ and press „OK“
 - select „Create point object“ in the graphical menu (point symbol  that is two rows below „Navigation“ menu)
 - click on **leftmost** cell in **top view** with left mouse button, and doubleclick
 - The „Object Properties“ Window opens: name the **point object**, e.g. „head_1“
 - select the tab „MODFLOW Features“ and then tick on „CHD“
 - in the table: set „Starting time“ = -1, „Ending time“ = 0, „Starting head“ = **268**, „Ending head“ = **268**, and press „OK“ to close the Window
 - click on **rightmost** cell in **top view** with left mouse button, move cursor to lower right cell, and double-click - the „Object Properties“ Window opens

- name point object, e.g. „head_r“
- select „MODFLOW Features“ and „CHD“
- in the table: set „Starting time“ = -1, „Ending time“ = 0, „Starting head“ = **264**, „Ending head“ = **264**, and press „OK“

5 Simulation and visualization - I

- Simulate the model
 - select „File“/„Save“ (no more model archive to be created)
 - select „Run MODFLOW-2005“ by clicking on the **green triangle** below „Grid“ menu
 - confirm the **file name** to save MODFLOW input files (*.nam where „*“ stands for the model name)
 - check information from ModelMonitor (a new Window opens) for a „green smileys“ (hopefully!), indicating the model terminated successfully.
 - close the **ModelMonitor** Window
- Verify simulation results
 - the **listing file** (*.lst) is opened (in notepad) once the **ModelMonitor** Window is closed
 - check **budget** in output listing (it is located near the end of the file)
 - check at the closure if the **Volumetric budget** terms are reasonable - i.e., **Percent Discrepancy** is close to 0)
 - close output listing and the dark „command prompt“ Window
- Plotting and analyzing results
 - select „Import and display model results“ (coloured symbol below „Data“ menu)
 - select model file *.fhd and click „open“
 - select „Contour grid“ and press „OK“
 - select „Update the existing data sets with new values“ (not required for the first time)
 - check hydraulic head isolines - the resulting plot should appear as in: [Figure 2](#)

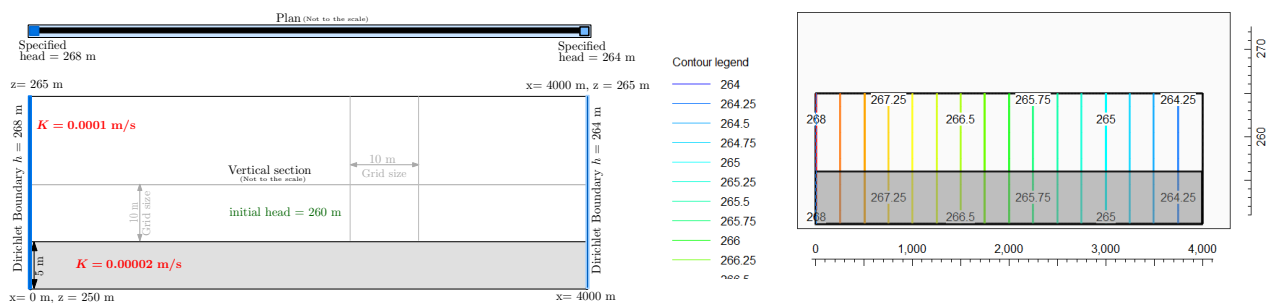



Figure 2: Left: The model domain and, Right: Obtained hydraulic head isolines for the front view

- The head isolines sequentially and uniformly decreases from the left boundary at 268 m to 264 m at the right boundary. The sequential distribution of head is largely due to symmetry of the domain and also that of the low permeable zone at the bottom

Food for thought I

- ⊙. In the data set $K_z = \frac{K_w}{10}$ is set by default. This default value should be the target of further exploration.
- ⊙. How about having an irregular low permeable zone in the domain?


6 Applying recharge (Neumann) boundary conditions

- Set Recharge Boundary Condition
 - select „Model“ / „MODFLOW Packages and Programs“ in the menu bar
 - select „Boundary conditions“/„Specified flux“ on the left side, then tick the sub-item „RCH“ and press „OK“
 - select „Create square object“ using a symbol  for recharge zone containing inflow and outflow boundaries
 - click on **upper left corner** of grid in **top view** with left mouse button; press left mouse button again, keep it down, move cursor to (4000, 0), and release the mouse button
 - name object (like „recharge“) in the „Object Properties“ Window
 - tick box „set values of intersected cells“
 - select the tab „MODFLOW Features“ and „RCH“
 - set „Starting time“ = -1, „Ending time“ = 0, and select „F()“ below „Recharge rate“
 - type the following expression into the input field: 100/1000/365.25/86400 [no „=“ required] (unit for recharge rate is according to space/time dimension of the model (m and s by default), i.e. conversion from mm/a is needed),
 - press „OK“, and press „OK“ again

Note II

- ▣ It can be useful to color-code different objects in the domain
- ▣ This can be done using the „object properties“ Window

7 Simulation and visualization - II

- Follow the steps from [Section 5](#) up to the steps „verify simulation results“
- Verify simulation results
 - the **listing file** (*.lst) is opened (in a notepad Window) once the **ModelMonitor** Window is closed
 - check **budget** in output listing (it is located near the end of the file)
 - check at the closure if the **Volumetric budget** terms are reasonable - i.e., **Percent Discrepancy** is close to 0)
 - close output listing and the black „command prompt“ Window
- Plotting and analyzing results
 - select „Import and display model results“ (coloured symbol  below „Data“ menu)
 - select model file *.fhd and click „open“

- select „Contour grid“ and press „OK“
- select „Update the existing data sets with new values“
- check hydraulic head isolines - the resulting plot should appear as in [Figure 3](#)

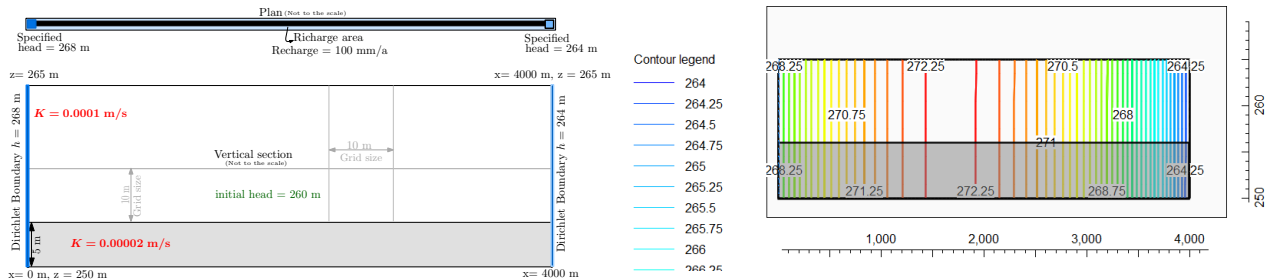



Figure 3: Left: The model domain and, Right: Obtained hydraulic head isolines for the front view

- The distribution of isolines in the [Figure 3](#) suggest that recharge of the domain distorts the uniformity of isolines (observed in [Figure 2](#)) towards the lower head (right) boundary


Food for thought II

- ⊙ How will the head-isoline change if there are multiple recharge zones with different recharge capacities?
- ⊙ Will changing the porosity (from the default 0.25) impact the groundwater flow condition in the domain?


8 Applying river (Cauchy) boundary condition

- River with uniform depth of 3 m is to be added to the domain. This is to appear on the top-most sub-layer of the front view. Following steps are to be performed to add river in the domain:
 - select „Model“/„MODFLOW Packages and Programs“ in the menu bar
 - select „Boundary conditions“/„Head-dependent flux“ on the left side, then tick the sub-item „RIV“ and press „OK“
 - select „Create point object“ using the  symbol for creating a river object
 - in **top view** with left mouse button, move cursor to (1500, 5), and double-click
 - in the new (properties) Window that opens - name the **point object** e.g., „river“
 - select „MODFLOW Features“ submenu and tick on the „RIV“
 - in the **table** in the Window, set „Starting time“ = -1 and „Ending time“ = 0, „River stage“ = **263**, „Conductance per unit length or area“ = **1e-6**, „River bottom“ = **260**. This fixes the river depth uniformly to 3 m.
 - press „OK“

Note III

- The width of the river in the model is equivalent to the width of space discretization (Δx) = 10 m - see conceptual model-setup: [Figure 1](#)
- The width of the river can be extended using line object  symbol

9 Simulation and visualization - III

- ☐ Follow the steps from [Section 5](#) or [Section 7](#) up to the steps „Plotting and analyzing results“
- ☐ Plotting and analyzing results
 - ☐ select „Import and display model results“ (coloured symbol  below „Data“ menu)
 - ☐ select model file ***.fhd** and click „open“
 - ☐ select „Contour grid“ and press „OK“
 - ☐ select „Update the existing data sets with new values“
 - ☐ check hydraulic head isolines - the resulting plot should appear as in [Figure 4](#)

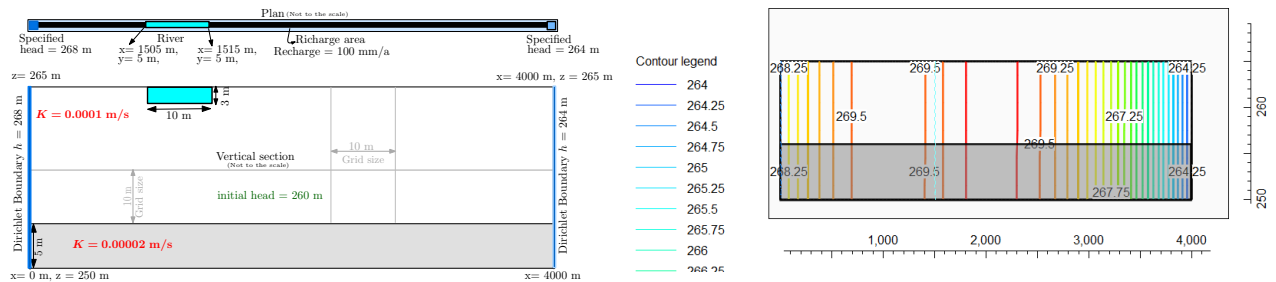


Figure 4: Left: The model domain with a river, and Right: Obtained hydraulic head isolines for the front view displaying the influence of river on the head


- ☐ River provides additional source of water to the model and its impact is very distinct by comparing [Figure 3](#) and [Figure 4](#)

Food for thought III

- ⊙. How will the head isline change if river has a non-uniform depth?
Hint: \rightsquigarrow e.g., use linear interpolation to uniformly vary river depth
- ⊙. How about changing the conductance of the riverbed? Will it impact the head isolines? How?


10 Adding a pumping well in the domain

The combined functioning of river and well is a typical feature of many Engineering solutions, e.g., Riverbank filtration, Managed Aquifer Recharge. Since Wells operates, generally, vertically the vertically oriented (2D) model are often among the most optimum exploration methods for such Engineering solutions. Following steps are to be undertaken to place a well in the vertically oriented 2D domain:

- ☐ Select „Model“/„MODFLOW Packages and Programs“ in the menu bar
 - ☐ select „Boundary conditions“/„Specified flux“ on the left side, then tick the sub-item „WEL“
 - ☐ press „OK“ to exit the Window
- ☐ Select „Create point object“ (the point symbol  two rows below „Navigation“) for placing a well in the domain
 - ☐ click on cell at (3505,5)
 - ☐ An „Object properties“ Window opens - name the object suitably, e.g., „well“

- select „MODFLOW Features“ tab in the Window and click on „WEL“ (on the left side)
- In the **table** that appears set „Starting time“ = -1 and „Ending time“ = 0, „Pumping rate per unit length and area“ = $-5e-5$. Note that, the „-“ sign refers to pumping well, and the unit for the pumping rate is m^3/s .

11 Simulation and visualization - IV

- Follow the steps from [Section 5](#) or [Section 7](#) up to the steps „Plotting and analyzing results“
- Plotting and analyzing results
 - select „Import and display model results“ (coloured symbol  below „Data“ menu)
 - select model file ***.fhd** and click „open“
 - select „Contour grid“ and press „OK“
 - select „Update the existing data sets with new values“
 - check hydraulic head isolines - the resulting plot should appear as in [Figure 5](#)

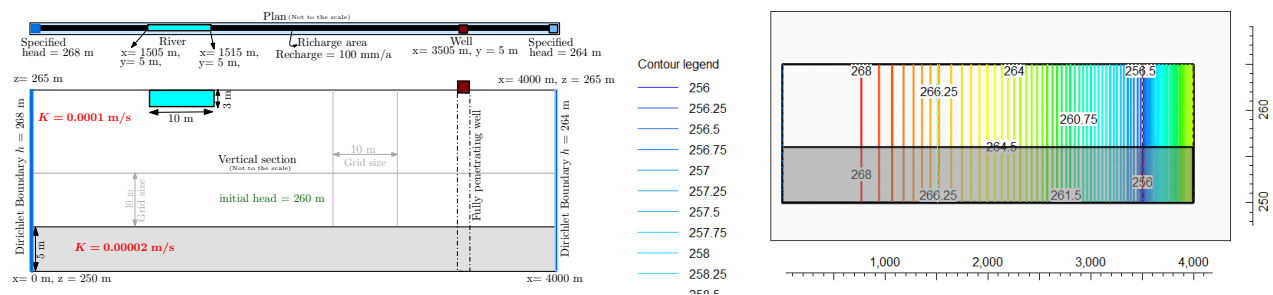


Figure 5: Left: The model domain with a river and a well, and Right: resulting hydraulic head isolines displaying the combined influence of river and a well







- The pumping well forces flow towards itself. In fact, water is also pulled from the outflow (right) boundary. Comparing [Figure 3](#) and [Figure 4](#) basically displays that. The path-plot should more clearly demonstrate these.

Food for thought IV

- ⊙ The well in the model is fully penetrating, what happens if the well is partially penetrating?
- ⊙ What will be the scenario if the well is a recharging one instead of a pumping one used in the example?

12 Additional post-processing (using MODPATH)

- select „Model“/„MODFLOW Output Control“/„Head“, set „external file type“ = **binary**, and press „OK“
- select „Model“/„MODFLOW Packages and Programs“ in the menu bar, then go to „Post processors“/„MODPATH“
 - set: „MODPATH version“ **6**; „Output mode“ = „Pathlines“, „Reference time for simulation“ (ReferenceTime) = 0, „Tracking direction“ = **backward**

- go to submenu „Version **6 & 7** options and set: „End of particle tracking (StopOption)“ to **stop at termination points (steady-state)**
- go to submenu „Output times“ and set „Method of specifying times (TimePointOption)“, to „Specified times; set „Number of times (TimePointCount)“ to 4 and define suitable times (time unit is **seconds**) e.g., 864000, 2592000, 4320000, 8640000 (which corresponds to: 10, 30, 50, and 100 days)
- press „OK“
- Carrying out **Particle Tracking** using MODPATH in the well („well“)
 - zoom-in (using symbol ) near well locations, select „Select objects“ (the red cursor ) , and double click on point object „well1“
- Select „MODFLOW Features“/„MODPATH“
 - a new Window opens
 - select „initial particle placement“ and „cylinder“
 - set number of particles around cylinder = **20**
 - press „OK“
- select „File“/„Save“ (to save MODPATH inputs no model archive needs to be created)
- Simulating MODPATH
 - select „Run MODFLOW-2005“ by clicking the „green triangle“ symbol . This is required when MODPATH input is applied, so that required input (binary) files can be created
 - Click the **downward black triangle**  symbol beside the „Run MODFLOW-2005“ symbol, and select „Export MODPATH Input Files“
 - confirm the file name to save MODPATH input files (***.mpn**) and run MODPATH
 - check and close output „listing“ file, close the dark **command prompt** window
- Visualizing MODPATH results
 - select „Data visualization“ (coloured symbol  to the left of symbol  in the second row of symbols)
 - select „MODPATH Pathlines“/„Basic“
 - set „MODPATH pathline file“ = ***.path**
 - select „Options“
 - set Color by = „None“, press „Apply“, and press „Close“
 - check pathlines - it should appear as in [Figure 6](#) below
 - select „File“/„Save“ (no model archive needs to be created)

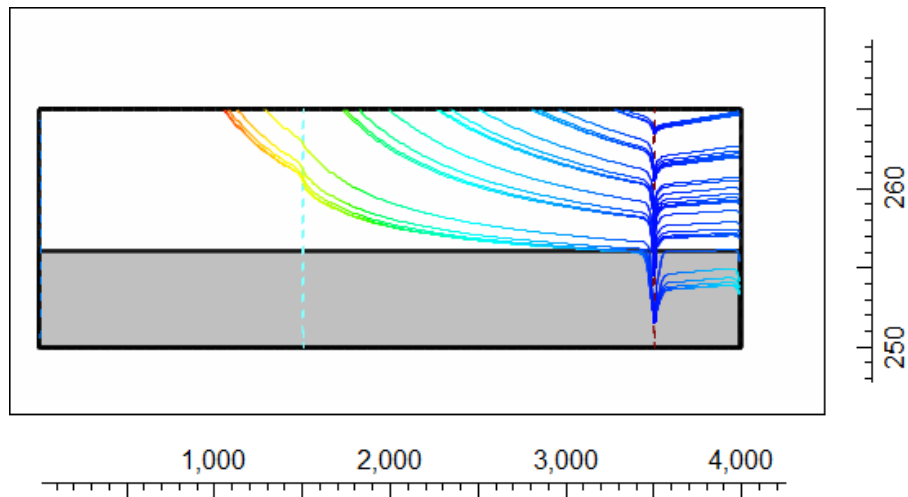


Figure 6: The pathlines observed at the two different wells.

Food for thought V

☉. Why not change domain porosity and check its impact on flow path.

🌀 **HINT:** Use the following steps to change the porosity: „Edit“/„Data Sets“/„MODPATH“ (the default value is 0.25).

Acknowledgements

This document was originally prepared by Dr. Martin Binder and others (at Technische Universität Dresden). The current version is significantly modified by Dr. Prabhas K Yadav and MSc Anton Köhler (both from Universität Tübingen) and Dr. Moulshree Tripathi. Dr. Yadav is supported by DAAD (funding Nr.: 91968853) and DFG (funding Nr.: YA 945/1-1). MSc Köhler is funded through DFG (funding Nrs.: DI 833/27-1 und GR 971/37-1)