

MODFLOW with ModelMuse - getting started

2D horizontal synthetic model development and analysis

1 Outline

This will be our first groundwater (GW) flow model using [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 flowpath to the wells.

Our domain will be of a rectangular shape (for simplicity), i.e., a 2-dimensional (2D) horizontally oriented domain, with Dirichlet boundaries in the left (inflow) and right (outflow) ends. The top and the bottom boundaries will be Neumann (no-flow) boundaries. 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 set-up of our domain will be as is in the figure below. We will develop this setup in a step-wise manner.

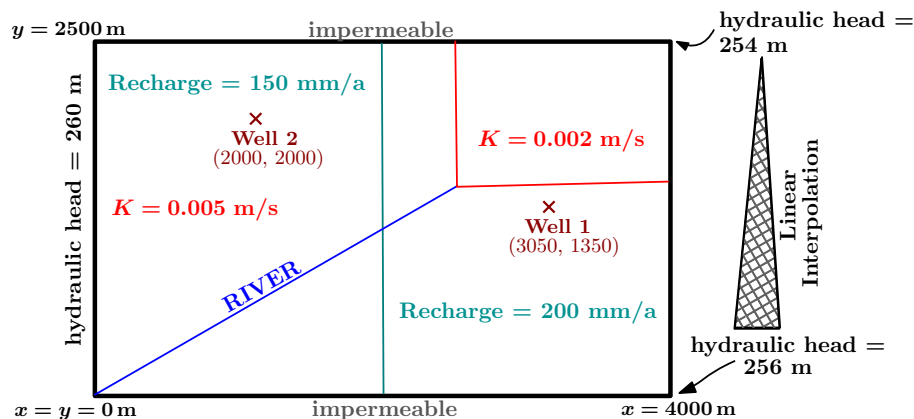


Figure 1: Conceptual setup of the 2D horizontally oriented model

The conceptual setup ([Figure 1](#)) will be simulated in order to gain insight of the impact of domain properties (K_s , R_s etc.) on the hydraulic head (h), consequently of the groundwater flow. Through visual analysis, from [MODPATH](#) simulations we will show the path of the streams towards the pumping-out well.


The overall goal of this exercise will be to get you acquainted with the basics of MODFLOW methods using ModelMuse as the user interface. This is to be achieved through the implementation of the most common boundary types in the domain.

Important!


!! You should always draw the conceptual setup and check it with your model before any simulation run

!! Save your model frequently so as to not loose your changes

2 Model geometry and domain discretization

- ☐ Double-click on the ModelMuse icon  in your computer desktop
- ☐ Select „Create new MODFLOW model“
- ☐ In the new Window: let the „Length unit“ be **Meters** and the „Time unit“ be **Seconds**
- ☐ Press „Next“ until you can specify the grid

Warning!!

- X.** Domain size data will be difficult to change at the later steps
- X.** Make sure to periodically save the mode by clicking on the  diskette symbol below the „Grid“ menu

- ☐ Specify initial grid
 - ☐ select „MODFLOW-2005“ as the MODFLOW Version
 - ☐ set $x = 0, y = 2500, z = 250$ for grid origin
 - ☐ set further model geometry data: „number of columns“ = **40**, „number of rows“ = **25**, „number of layers“ = **1**, „model top“ = **265**, and „upper aquifer“ = **250**
 - ☐ press „Finish“ and then select „File“/„Save as“ in the menu bar to save the model input data giving a file name (e.g.,) „myfirstmodel“ by using the file format **mmZLib**

Note 1

- ☐ **Grid origin in MODFLOW is at the upper left(!) corner**
- ☐ **The columns are counted from left to right and rows from back to front**
- ☐ **The values in „column width“ and „row width“ - they are the grid size**
- ☐ **Saving *.mmZLib format requires less disk space but *.gpt is a more convenient option**

3 Model structure and parameters

- ☐ Applying basic model parameters
- ☐ Select „Data“/„Edit Data Sets“ in the **menu bar** and click „Required“
 - ☐ select „Layer definition“ on the **left side**, then check „Model_Top“ - the value should be **265** and the „Upper_Aquifer_Bottom“ - the value **250**
 - ☐ select „Hydrology“:
 - ☐ select „ K_x “ and set its value: $K_x = 0.005$ (Unit: m/s)
 - ☐ select „Modflow_Initial_Head“, then set initial head = 257 (in the box)
 - ☐ Press „Apply“ and then „Close“
- ☐ Select „Create rectangle object“ in the graphical menu (☐ symbol two rows below „Customize“ menu)
 - ☐ click on upper **right corner** of grid in top view with left mouse button
 - ☐ press left mouse button again, keep it down, move cursor to (2500, 1500), and release left mouse button

- name the object (e.g., „small_k“)
- select tab „Data Sets“, then move to „Required“/„Hydrology“/„Kx“
- replace $K_x = 0.005$ by $K_x = 0.002$ and press „OK“

Note 2

- ▣ You can color the object in the „Properties“ tab and choosing color provided in „Color object interior“ or „Color object line“
- ▣ The exact dimension of the object (e.g., „small_k“) can be fixed in the tab „Vertices“

4 Setting the model boundary conditions

- Set **left** or the **inflow** boundary (Dirichlet BC) in the Model
 - 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 straight-line object“ in the **graphical menu** (↵ stairs-like symbol two rows below „Customize“ menu)
 - click on **upper left** cell in **top view** with left mouse button, move cursor to **lower left** cell, and **double click**
 - name straight-line object, e.g. „inflowhead“
 - you may color the line in the properties tab (see [Note 2](#))
 - select the tab „MODFLOW Features“ and then „CHD“
 - in the table: set „Starting time“ = -1, „Ending time“ = 0, „Starting head“ = 260, „Ending head“ = 260, and press „OK“



Important!

- !! Compare the ModelMuse output with your conceptual setup ([Figure 1](#))
- !! The non-assigned boundary are No-flow (Neumann BC) by default

- Set **right** or the **outflow** boundary (Dirichlet BC) in the Model
 - perform the steps as mentioned above for setting the left boundary (up to „Create straight-line object“)
 - click on **upper right** cell in **top view** with left mouse button, move cursor to **lower right** cell, and **double-click**
 - name straight-line object, e.g. „outflowhead“
 - select the tab „MODFLOW Features“ and then „CHD“
 - set starting time = -1 and ending time = 0
 - click on the symbol „F()“ below the „Starting head“
 - type the following **expression** into the bottom input field (above „Logical operators“): **interpolate(y, 256, 0, 254, 2500)** (no = is required in the input field)
 - press „OK“


- repeat the last (above) two steps for setting the „Ending head“

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 file (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

Important!

!! Verifying water budget is still important even after the simulation terminates successfully.

- 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 needed for the first time)
 - check hydraulic head isolines - the resulting plot should appear as in [Figure 2](#):

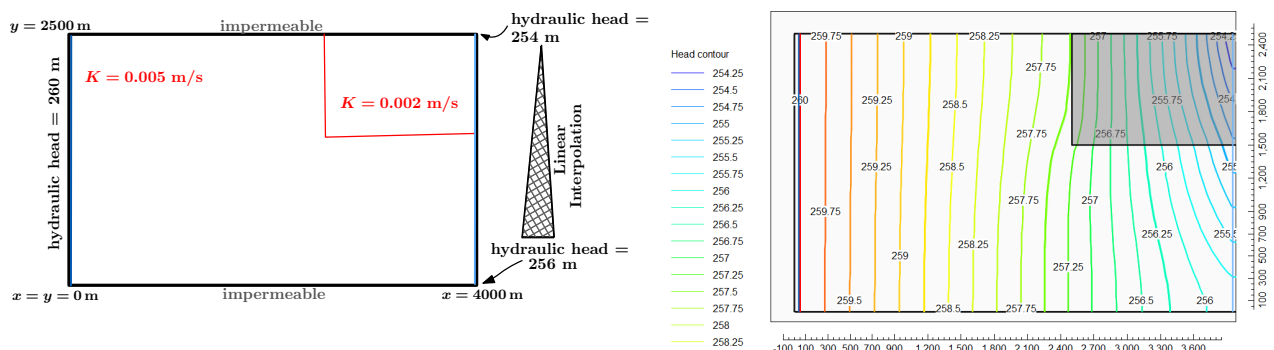



Figure 2: Left: The model domain and, right: the obtained hydraulic head isolines

- As observed in the [Figure 2](#) the head isolines gets distorted in the low conductive zone. This is the expected result.



Food for thought I

- ⦿ Try different combination of K_s values in [Section 3](#) and analyze the resulting isolines
- ⦿ Click on „Export Image“  symbol and learn different options available for visualizing and exporting results

6 Applying recharge (Neumann) boundary conditions


- 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“
 - Press „OK“, which closes the Window
- Select „Create rectangle object“ (see [Section 3](#)) for the recharge zone containing inflow boundary
 - 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 **(2000, 0)**, and release left mouse button
 - The „Object Properties“ Window opens - name the object (e.g., „recharge_left“)
 - select „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**: 150/1000/365.25/86400 (no „=“ is required). The unit for recharge rate is now converted to m/s from the given data in mm/a, see [Figure 1](#))
 - Press „OK“, and „OK“ again

Note 3

- ▣ The objects in the model can be selected using the **red cursor**  symbol, which is below the „Customize“ menu
- ▣ Clicking on the tree  symbol opens a Window with „Objects“ in the model. The Objects from there can be selected/deselected
- ▣ **Starting time** = -1 and **Ending time** = 0 are default for steady-state simulation

- Repeat the above steps for applying the recharge on the **right side** of the domain (based on conceptual setup - [Figure 1](#))
 - select „Create rectangle object“ for recharge zone containing outflow boundary
 - click on **upper right corner** of grid in top view with left mouse button; press left mouse button again, keep it down, move cursor to **(2000, 0)**, and release left mouse button
 - name object (e.g., „recharge_right“)
 - select „MODFLOW Features“ and „RCH“
 - set **Starting time** = -1 and **Ending time** = 0
 - select „F()“ below „Recharge rate“, type the following expression into the input field: 200/1000/365.25/86400
 - press „OK“, and press „OK“ to go back to the main Window.

7 Simulation and visualization - II

- ☐ Follow all the steps provided in [Section 5](#) up to the “Plotting and analyzing results” steps
 - ☐ Press “Yes” when the “save as” Window raises “**XY.nam** already exists, Do you want to replace it?” (where **XY** is the name of your model)
- ☐ 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](#):

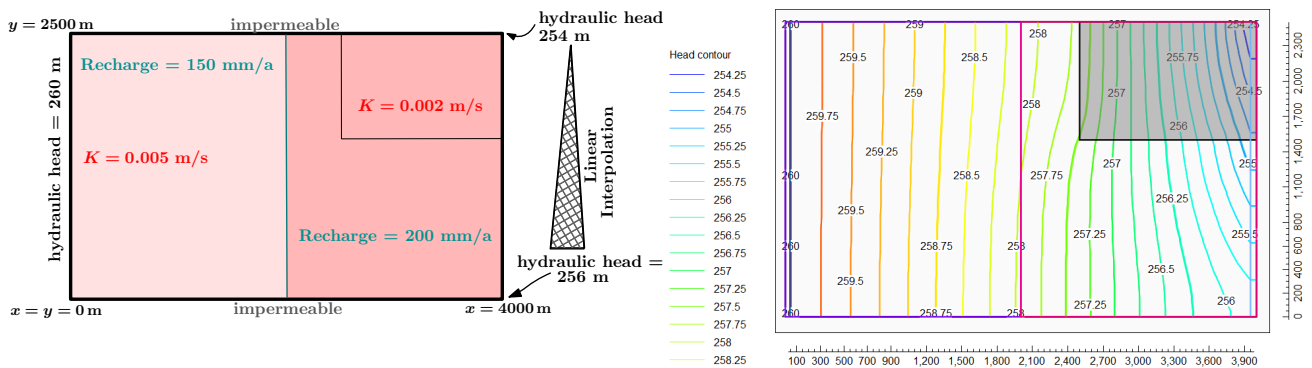



Figure 3: Left: The model domain with recharge and, Right: Obtained hydraulic head isolines with recharge

- ☐ As expected, only small change in the contour pattern is observed due to maximum recharge of 200 mm/a in the domain. The addition of water in the domain from recharge leads to slight increase in the Hydraulic gradient compared to that observed without the recharge in [Figure 1](#).

Food for thought II

- ☐ How about using the annual precipitation rate in India as the recharge rate. How much impact that will make to the hydraulic heads?
- ☐ How about seasonal (monsoon) precipitation impact - what type of model is needed for such modelling?

8 Applying river (Cauchy) boundary conditions


- ☐ 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“
 - ☐ press „OK“ - this Window closes and we are back to main model Window
- ☐ Select „Create polyline object“ (symbol consisting of three line segments , two rows below „View“ menu) for the river
 - ☐ click on **lower left corner** in top view with left mouse button, move cursor to **(2500, 1500)**, and double-click

- In the „Object Properties“ Window, name the polyline object (e.g., „river“)
- select „MODFLOW Features“ and tick „RIV“
- set **starting time** = -1 and **ending time** = 0
- select „F()“ below „River stage“, type the following expression into the input field: **interpolate(x, 260, 0, 258, 2500)**, and press „OK“
- select „F()“ below „Conductance per unit length or area“, type the following expression into the input field: **1000/86400**, and press „OK“ (This corresponds to a streambed conductance of 1000 m²/d)
- select „F()“ below „River bottom“, type the following expression into the input field: **interpolate(x, 255, 0, 253, 2500)**, press „OK“, and press „OK“ again (This corresponds to a river depth of 5 m.)

Warning!!

- X.** Make sure to re-simulate MODFLOW (see [Section 5](#)) after adding any new object in the domain. The file (model.nam) name should not be changed but be overwritten.

9 Simulation and visualization - III

- Follow all the steps provided in [Section 7](#) up to the „Plotting and analyzing results“ steps
 - Press „Yes“ when the „save as“ Window raises „XY.nam already exists, Do you want to replace it?“ (where **XY** is the name of your model)
- Plotting and analyzing results
 - steps are similar to that used in [Section 7](#)
 - 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](#):

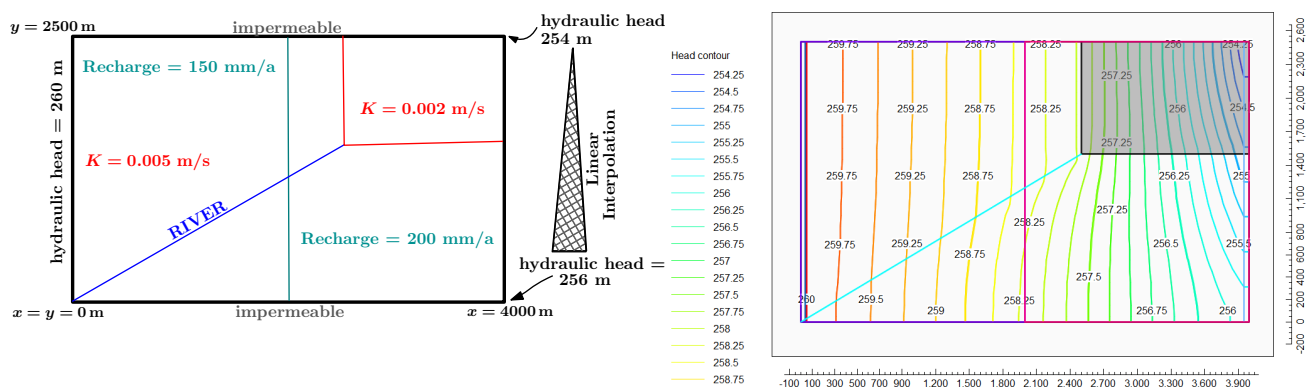



Figure 4: Left: The model domain with recharge and a river and, right: the observed hydraulic head isolines due to combined impact of recharge zones and the river

- The impact of river on the domain can be observed upon comparing [Figure 2](#), [Figure 3](#) and [Figure 4](#). The river appears to significantly distort the head isolines pattern, and consequently it will influence the groundwater flow behaviour.

Food for thought III

- ⊙. Why not make the model more practical by replacing the outflow boundary with the river boundary?
 - ⊙. How about plotting flow lines or imposing flow lines on head isolines?
- 🌀 **HINT:** MODFLOW produces three graphics data files - *.fhd - head data, *.fdn - drawdown data, and *.cbc - flow data.

10 Adding pumping well (Neumann boundary) in the domain


- Select „Model“/„MODFLOW Packages and Programs“ in the menu bar - a new Window opens
 - select „Boundary conditions“/„Specified flux“ on the **left** side, then **tick** the sub-item **WEL** and
 - press „OK“ - the Window closes
- Select „Create point object“ (**dot symbol**  two rows below „Navigation“) for the well
 - click on cell containing the well location (**3050, 1350**)
 - name point object (e.g., „well 1“)
 - set **Starting time** = -1 and **Ending time** = 0
 - select „**F()**“ below „Pumping rate per unit length or area“, type the following expression into the input field: **-2000/86400** and press „OK“. This input corresponds to a pumping rate of 2000 m³/d

Warning!!

- X.** Do not forget the (negative) „-“ sign in the Well input; this means it is a pumping well.

- Create (e.g.,) the second well
 - follow the steps as mentioned for the setting the first well.
 - place the well (naming it, e.g., „well2“) at the location: (**2000, 2000**)
 - let the pumping-out rate in this well be 5000 m³/d

11 Simulation and visualization - IV

- Follow all the steps provided in [Section 7](#) or [Section 9](#) up to the „Plotting and analyzing results“ steps
 - press „Yes“ when the „save as“ Window raises „**XY.nam** already exists, Do you want to replace it?“ (where **XY** is the name of your model)
- Plotting and analyzing results
 - steps are similar to that used in [Section 7/Section 9](#)
 - 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](#). The conceptual setup of the model in [Figure 5](#)(left) matches with the original conceptual model setup.

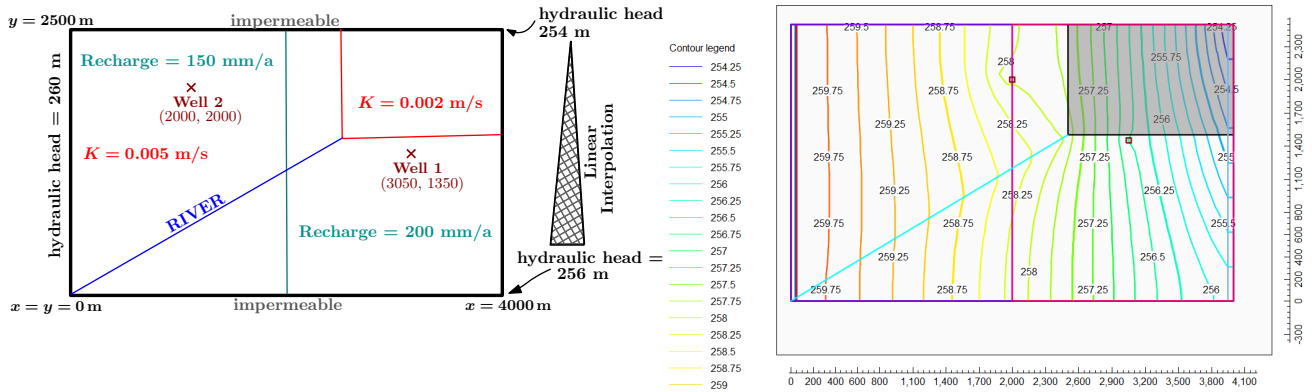


Figure 5: Left: The conceptual setup of the domain, right: the observed hydraulic head isolines due to combined impact of recharge zones, the river and two wells

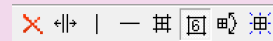
- As can be observed in [Figure 5](#), the wells with pumping rate of maximum 5000 m^3 significantly impacts the hydraulic isolines. The impacts are particularly significant in the well's vicinity.

Food for thought IV

- ⊙ Why not add a few more wells at different locations in the domain?
- ⊙ Are obtained results accurate? Perhaps the used grid size is too large or small. Why not check that.

🌀 **HINT:** ModelMuse provide several options to manipulate grid size.

Those options can be accessed from the symbol list -









12 Additional post-processing (using MODPATH)

A very short discription on [MODPATH](#):

MODPATH is a [USGS](#) particle-tracking code that uses groundwater velocities generated by MODFLOW to compute **advective flow paths** and **travel times**. It moves **massless particles** through the reconstructed seepage velocity field using a semi-analytical [Pollock method](#), ensuring fast and accurate cell-to-cell tracking. MODPATH supports both **forward tracking**, which shows where groundwater flows, and **backward tracking**, which identifies recharge areas and capture zones. It works under both steady-state and transient flow conditions.

- 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**
- got to submenu „Version **6 & 7** options and set: „End of particle tracking (StopOption)“ to **stop at termination points (steady-state)**
- go to sub-menu „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 first well („well1“)
 - 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“
- Using MODPATH in the second well („well2“) - procedure similar as above
 - double click on the point object „well2“
 - select „MODFLOW Features“/„MODPATH“
 - 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“, close the black 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)

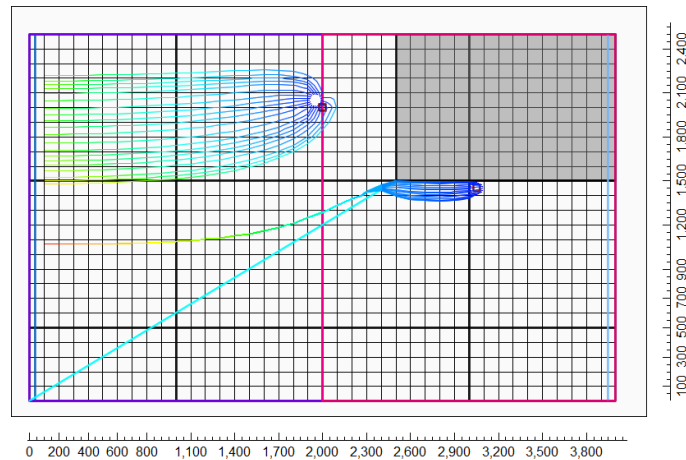


Figure 6: The pathlines observed at the two different wells.

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)