



## MODEL SETUP MIT MODELMUSE (LAST CHANGE 2021 08 25)

(ORIGINAL DOCUMENT DEVELOPED BY M. BINDER / INSTITUTE FOR GROUNDWATER MANAGEMENT)

### 2D FLOW within a synthetic model scenario

#### 1) DISCRETIZATION SPACE / TIME

- Create New Model
  - Start MODEL Muse and "Create new MODFLOW model" and press "Next" until you can specify the grid
  - specify initial grid (difficult to be changed later!!!)
    - note: grid origin is at the upper left(!) corner, and length unit is meters (default)
    - set  $x = 0$ ,  $y = 2500$ ,  $z = 250$  for grid origin
    - set further input data: number of columns = 40, number of rows = 25, number of layers = 1, model\_top = 265, upper aquifer = 250
    - Press "Finish"
  - select "File" / "Save as" in the menu bar, and then save input data under a file name like "myfirstmodel" (or so) by using the format mmZLib
- Import background image

#### 2) STRUCTURE & PARAMETERS

- Enter Basic Aquifer Data
  - select "Data" / "Edit Data Sets" / "Required" in the menu bar
  - [select "Layer definition" on the left side, then check "Model\_Top" and "Upper\_Aquifer\_Bottom" (values should be 265 and 250, resp.)]
  - select "Hydrology":
    - select "Kx" and set  $Kx = 0.005$  (unit of hydraulic conductivity is m/s by default)
  - select "Create rectangle object" in the graphical menu (= square two rows below "Customize")
    - Click on the upper right corner of the grid in the top view with the left mouse button
    - Press the left mouse button again, keep it down, move the cursor to (2500, 1500), and release the left mouse button
    - name the object (like "small\_K")
    - select tab "Data Sets," then move to "Required" / "Hydrology" / "Kx."
    - replace 0.005 by 0.002 and press "OK."

#### 3) BOUNDARY CONDITIONS

- Set Defined Head Boundary Condition
  - 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")
  - Click on the upper-left cell in the top view with the left mouse button, move the cursor to the lower-left cell, and double-click
    - name straight-line object, e.g., "inflowhead."
    - 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"

- Click on the upper right cell in the top view with the left mouse button, move the cursor to the lower right cell, and double-click
  - name straight-line object, e.g., "outflowhead."
  - select "MODFLOW Features" and "CHD."
  - set starting time = -1 and ending time = 0
  - Click on the symbol "F()" below "Starting head."
  - type the following expression into the input field (above "Logical operators"): interpolate (y, 256, 0, 254, 2500), press "OK"
  - repeat for "Ending head."
- 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 rectangle object" for 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
    - name object (like "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 "=" 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
  - select "Create rectangle object" for recharge zone containing outflow boundary
    - Click on the upper right corner of the grid in the top view with the left mouse button; press the left mouse button again, keep it down, move the cursor to (2000, 0), and release the left mouse button
    - name object (like "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" again
- Set River Boundary Condition
  - 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 polyline object" (symbol consisting of three line segments, two rows below "View") for river
    - Click on the lower-left corner in the top view with the left mouse button, move the cursor to (2500, 1500), and double-click
    - name polyline object (like "river")
    - select "MODFLOW Features" and "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 m<sup>2</sup>/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 water depth of 5 m.)
- Define Wells
  - 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" and press "OK"
- select "Create point object" (dot symbol two rows below "Navigation") for the first well
  - click on the cell containing the well location
  - name point object (like "well 1")
  - select "MODFLOW Features" and "WEL"
  - 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" (Please do not forget the negative sign! Input corresponds to a pumping rate of 2000 m<sup>3</sup>/d.)
- select "Create point object" for the second well
  - Click on the cell containing the well location
  - name point object (like "well 2")
  - select "MODFLOW Features" and "WEL"
  - 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: -5000/86400, and press "OK" (Please do not forget the negative sign! Input corresponds to a pumping rate of 5000 m<sup>3</sup>/d.)

#### 4) PROPERTIES / SIMULATION

- Define Initial Head
  - select "Data" / "Edit Data Sets" / "Required" in the menu bar
  - select "Modflow\_Initial\_Head", then set initial head = 257, press "Apply" and press "Close"
- RUN Simulation / MODFLOW
  - select "File" / "Save" (no model archive needs to be created)
  - select "Run MODFLOW-2005" by clicking on the green triangle below "Grid"
  - confirm the file name to save MODFLOW input files (\*.nam where "\*" stands for the model name)
  - check information from ModelMonitor ("green smileys" – hopefully ...)
  - close ModelMonitor window

#### 5) POST PROCESSING

- Check model run
  - Once MODELMONITOR is closed, the listing file is opened; check budget in output listing (it's located near the end of the file – check closure of water balance and if the budget terms are reasonable)
  - close output listing and the black "command prompt" window
- Plot heads
  - select "Import and display model results" (colored symbol below "Data")
  - select model file \*.fhd
  - select "Contour grid" and press "OK"
  - select "Update the existing data sets with new values" (not needed in very first try)
  - check hydraulic head isolines
- Prepare Particle Tracking
  - 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 7; "Output mode" = Pathlines & time series, "Reference time for simulation" = 0, "Tracking direction" = backward, and press "OK"

- got to submenu “Version 6 & 7 options and set: End of particle tracking (StopOption) stop at termination points
- go to submenu “Output times” and set Method of specifying times to “Specified times; set “Number of times” to 4 and define suitable times (time unit is seconds) like 864000, 2592000, 4320000, 8640000 (10, 30, 50, and 100 days)
- Carry out Particle Tracking Using MODPATH
  - zoom in near well locations, select “Select objects” (the red cursor), and double click on point object “well1”
    - select “MODFLOW Features” / “MODPATH”
    - select “initial particle placement” and “cylinder”
    - set number of particles around cylinder = 20 and press “OK”
  - double click on point object “well2”
    - select “MODFLOW Features” / “MODPATH”
    - select “initial particle placement” and “cylinder”
    - set number of particles around cylinder = 20 and press “OK”
  - select “File” / “Save” (no model archive needs to be created)
  - select “Run MODFLOW-2005” by clicking on the green triangle below “Grid” (only necessary for the first run to generate the necessary binary head file)
  - select “Run MODFLOW-2005” by clicking on the **black(!)** triangle below “Grid” 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
  - select “Data visualization” (colored symbol to the right of the middle 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
  - select “File” / “Save” (no model archive needs to be created)
  - (Please note that another MODFLOW run needs to be carried out before MODPATH is started if other parameters, which affect the flow behavior, were changed.)

## Optionally

- [optionally Refine Grid]
  - learn about buttons “Delete grid lines”, “Drag grid lines”, “Add vertical grid line”, “Add horizontal grid line” and “Subdivide grid cells” (left group in second row of symbols)
  - zoom in near well locations, select “Subdivide grid cells”, mark both cells containing a well, set “subdivide each column into” = 2, “subdivide each row into” = 2, and press “OK” [note: The “layers” section has to remain unchanged!]
  - repeat the previous step several times to split a subsequently increasing number of columns and rows
  - zoom in near well 1, select “Delete grid lines” and click on the intersection of grid lines at the well location
  - select “Subdivide grid cells”, mark the cell containing well 1, set “subdivide each column into” = 3, “subdivide each row into” = 3, and press “OK”
  - zoom in near well 2, select “Delete grid lines” and click on the gridline passing through the well location
  - select “Subdivide grid cells”, mark the cell containing well 2, set “subdivide each row into” = 3, and press “OK”
- choose “Restore default 2D view” and learn about the symbol “Show or hide 2D gridlines” (on the very right in the second row of symbols)