

## A simple MicroFEM course: Ten lessons for finite element groundwater modelling starters

Kick Hemker  
hemker@microfem.nl

---

### ===== INSTALL MICROFEM-LT =====

Before you can start with the lessons you have to download and install the MicroFEM software. This requires the following steps:

- Go to the website: [www.microfem.nl](http://www.microfem.nl) (or [www.microfem.com](http://www.microfem.com)).
- Click "Download" in the left side menu bar. This will show a page with the "download table". The first item (microfem-lt.zip) is the free MicroFEM software.
- Download this zip-file. Its size is smaller than 2 MB.
- Unzip the zip-file. It contains only one file: MicroFEM LT Setup.exe
- Click the setup file to install MicroFEM. This is all straight forward and easy.

Of all the other files in the download table it is recommended to also download the Tutorial (microfem-tutorial.pdf). It contains a lot of useful information for MicroFEM starters, but we will not use this Tutorial in this course.

---

Contents	page
Lesson 1A: A simple finite element grid	2
Lesson 1B: Create and run a simple model	5
Lesson 2A: Drawing contour maps	8
Lesson 2B: Finite element head computation	11
Lesson 3A: Grid refinement and Well radius	14
Lesson 3B: Labels and Water balances	17
Lesson 4A: Hexagonal FemMesh grid	19
Lesson 4B: Stopping criterion	21
Lesson 5A: Profiles and Leaky aquifers	22
Lesson 5B: Boundary conditions	25
Lesson 6A: FemMesh grid generation	27
Lesson 6B: Two-aquifer system and Grid manipulation	30
Lesson 7A: Marking nodes and Heterogeneity	33
Lesson 7B: Using formulas and the Xtra worksheet	36
Lesson 8A: FemGrid grid generation	39
Lesson 8B: Water table aquifer	42
Lesson 9A: 3D-flowlines	45
Lesson 9B: Batch-file	48
Lesson 10A: Water divide and Capture area	51
Lesson 10B: Transient modelling	55

===== MicroFEM Lesson 1A: A simple finite element grid

In these lessons a line starts with @ when your action (mouse or keyboard) is required.  
All other text lines are explanations.

@ Double click the yellow MicroFEM icon.

A window opens with, from top to bottom:  
- the blue title bar with the text "MicroFEM 4.10 LT"  
- a Menu bar (Files, Project, View, .... etc. )  
- a large black rectangle: "the Map"  
- a grey Status line, with no information yet  
- a Toolbar with only grey buttons, left and right  
and at the right hand side: an empty Table.

@ Menu bar: Help / Info / About MicroFEM ... / [OK]

This means that you have to do a number of actions:

- 1 - go to the Menu bar and click on "Help"
- 2 - then move to "Info" in the pull down menu
- 3 - then click on "About MicroFEM ... "
- 4 - click on the OK button to close again the About box pop-up window.

In this way you display the About box, so you can check the present MicroFEM version.

Any [text] between square brackets means that you have to click on a button.  
Clicking means "Left clicking". Right clicking is seldom required.

=== Step 1: Create an extremely simple grid (mesh, network) of only six equilateral elements

@ Menu bar: Files / New grid ... / select Create New grid / Irregular boundary grid (FemGrid) / [OK]

@ Enter the Model name in the upper cell: "Lesson 1" / [OK]

You may enter more information (Project, Location, etc.), but only the Model name is required to continue.

The pop-up window disappears and we are now in one of the two "Grid generation modes"  
At the right-hand side an empty table (with the tabs: Fixed nodes, Segments and Regions) has appeared.

Also, in the lower left corner, a Toolbar with some coloured buttons has appeared.

Place the arrow cursor above each of these buttons one-by-one, and read the texts shown in the small yellow pop-up windows

We call such texts the "hint" of a button.

One button is much larger than the others. It shows the hint: "Next generation step (F8)"

Find the button with the text "Start with a hexagon (F5)"

I call this button "F5". Clicking on this button has the same effect as pressing the F5-key on the keyboard.

@ Toolbar: [F5] / [OK]

@ Toolbar: [F8] / [F8] / [F8] / [F8] / [F8] / [F8]

Note that each time we click on the F8 button (or press the F8 key), the text on this button changes.

A simple finite element grid has been built on the map.

Click with the cursor on each of the seven nodes in the map and look at the table on the right hand side.

Also, click on the cells in the table and look at the corresponding nodes in the map.

How large is the grid from West to East? And from North to South?

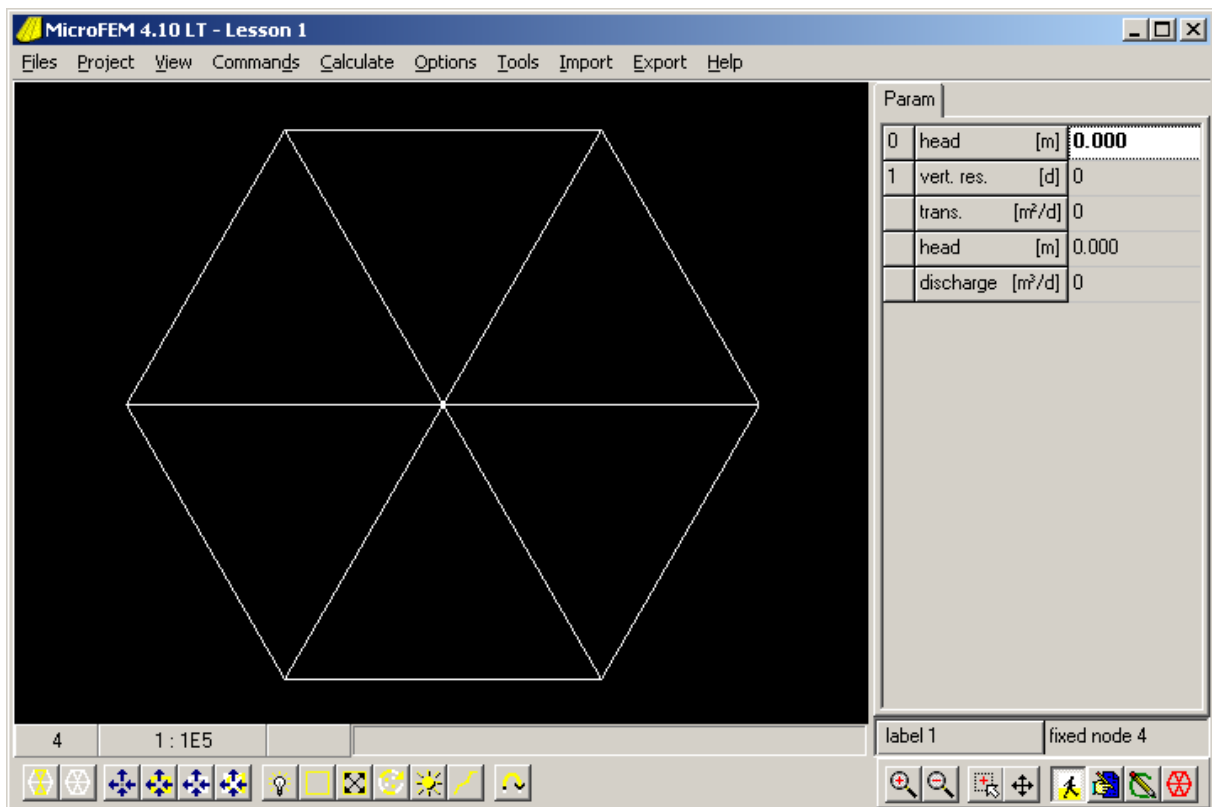
@ Toolbar: [F9]

This saves all information of the generated grid to a Finite Element Network data file (a "fen-file")

We have finished with the grid in "Grid generation mode" and now can move on.

@ Toolbar: [F10] / [OK]

During this step we have to choose the number of aquifers of our model and accept the suggested number of aquifers = 1



=== Intermission: Stop, save and start

This is the end of Lesson 1A. Suppose you want to stop for today.

@ Menu bar: Files / Exit

A pop-up window appears with the following question:

"Do you want to save changes to Base model?"

Click [Yes] this allows you to save a file "Lesson 1.fem"

After that, a similar pop-up window will ask:

"Do you want to save changes to Project? (Lesson 1.fpr)"

Again, click [Yes].

When you want to start again, there are different possible ways to continue:

- You can click on the "Model 1.fpr" icon. The icon looks like a white paper with the MicroFEM logo.
  - Or you can click on the "Model 1.fem" icon. The icon looks similar, with a small blue 3D cube.
  - Or on the "Model 1.fen" icon. The icon looks similar again, with a small red hexagon.
- In all cases MicroFEM will be started first.

The fpr-file (project file) tells MicroFEM to open all model files: in this case it is only the Base-model: "Lesson 1.fem"

A Base-model is a relatively simple model. All its data can be saved in a single file, the fem-file.

The fem-file (model file) contains data about the grid, transmissivities, heads, discharges, boundary conditions, etc.

The fem-file information is always required, for any MicroFEM model.

Many MicroFEM models also require additional information, saved in additional files. For example, transient models require Storage coefficients.

The fen-file (network file) only contains grid information.

When you start with a fen-file you are asked to enter the number of aquifers of the model.

A fen-file can also be used to repeat the grid generating process with modified data.

In the latter case, the file must be loaded from the Menu bar:

Menu bar: Files / New grid ... / Open existing network file with / Appropriate generator / [OK] / Open

## ===== MicroFEM Lesson 1B: Create and run a simple model

### === Step 2: The MicroFEM GUI

Now that we have a model, the GUI (Graphical User Interface) looks quite different.

The Map, the Status bar and the Table (on the right hand side) now all show information.

Map: Shows the model grid, just as in "Grid generating mode".

- Move with the cursor from node to node. Check the information on the Status bar.
- Click on each node. Clicking on a node makes that node the "current node". The current node is blinking (alternating white and black)

Table: All (five) cells in the table contain zero values. Click on any cell to make it the "active cell".

Toolbar: There are 4 (zoom and pan) + 4 (mode) buttons on the right hand side.

Read the hints.

- Try the 4 zoom and pan buttons. Note the scale on the Status bar.
- Click on the Mode buttons: Input mode (mode 2), Drawing mode (mode 3), Alter-grid mode (mode 4), and back to Walking mode (mode 1).
- Note that there is a separate Toolbar for each mode.
- Note that the colour of the active cell in the Table changes with the mode.

### === Step 3: Model data input

Select Input mode

@ Toolbar [Input mode] (the second of the lower right four buttons)

All data of the model are still zero (default value).

The Table shows the model data of the "current node".

You can modify model data in "Input mode" only.

We start by entering transmissivities of 2000 m<sup>2</sup>/d for all nodes.

@ Table: Make the "trans. [m<sup>2</sup>/d]" cell (3rd from the top) the active cell.

@ Click on the blue cell (the cell with the zero). The cell opens for input (white cell, blue text).

@ Enter 2000, and then press F5 on the keyboard (or [all nodes (F5)] on the Toolbar.

We will now enter a well discharge of 1000 m<sup>3</sup>/day for the centre node only.

@ Map: click on the node in the centre of the grid. This node will become blue (blinking blue and black)

On the Status bar the position of the node is given by "x=0.00 y=0.00"

@ Table: Make the "discharge [m<sup>3</sup>/d]" cell (bottom of the list) the active cell.

@ Open the cell, enter 1000 and press this time the "Enter" key on the keyboard.

By using the Enter key you assign the entered value to the current node only.

All required data have been entered now, but we still have to set the boundary condition to "fixed heads".

#### === Step 4: Mark all boundary nodes

"Marking nodes" is an important procedure in MicroFEM.

It is the way to select a particular part (subset) of the nodes.

You mark nodes when you want to do something with that group of nodes (i.e. that particular part of the model) only.

In this case we want to do something with all boundary nodes (in this case all nodes, except the centre node).

@ Select Walking mode by clicking the [Walking mode] button (the first of the lower right 4 buttons), or "Alt+1" on the keyboard.

The Toolbar for the Walking mode is displayed. Read the hints of all buttons of this toolbar. Note that the keyboard equivalents from left to right are: Ins, Del, F2, F3, .... , F12

@ Menu bar: Commands

You will see that the available commands correspond one-to-one with the toolbar buttons.

@ Toolbar: [Delete marks (Del)] / [Mark model boundary (F7)]

Marked nodes are connected by yellow lines.

Click on each of the nodes and watch the Status bar.

The "current node" is on the left side of the Scale (1:50000 or something similar), the cursor node (the node closest to the cursor) is on the right.

A yellow square is added when the node is marked.

#### === Step 5: Specify fixed head boundary conditions

All boundary nodes are marked. The centre node is not marked.

By default a model boundary is a no-flow boundary in MicroFEM.

By default the top of the model is also a no-flow boundary.

The base of the model is always a no-flow boundary.

When we do not change the boundary condition, there is no groundwater boundary flow possible. When a well discharges groundwater in such a model, there is no steady-state solution possible.

We will now set the model boundary (all boundary nodes) to a "fixed head" boundary condition.

@ Select Input mode by clicking the [Input mode] button (the second of the lower right four buttons).

The Toolbar for the Entering mode is displayed. Read the hints of all buttons of this toolbar.

Note that the keyboard equivalents from left to right are: Ins, Del, F2, F3, F4, two buttons without a keyboard equivalent, F5, .... , F12

When you click on Menu bar: Commands, you will see that the available commands correspond one-to-one with the toolbar buttons.

@ Table: Make the "head [m]" cell the active cell (you do not have to open this cell).

@ Toolbar: [Fixed heads for marked nodes (f9)]

Nothing seems to happen, but when you click on each node, and look at the table, you may see the results of our last action.

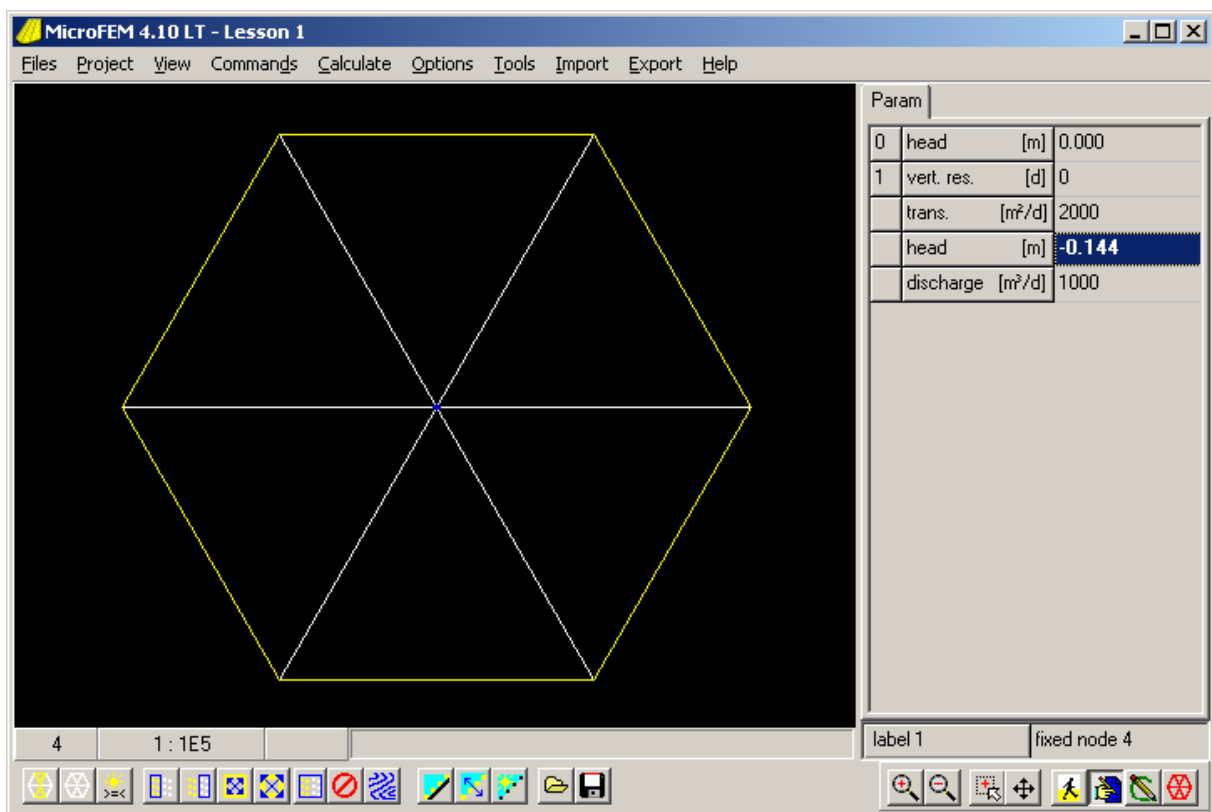
All the boundary nodes show the same information in the Table: Transmissivity = 2000 m<sup>2</sup>/d, Head = 0.00 m and fixed, Discharge = 0 m<sup>3</sup>/d

Only the centre node is different: Transmissivity = 2000 m<sup>2</sup>/d, Head = 0.00 m (no "fixed" text), Discharge = 1000 m<sup>3</sup>/d.

=== Step 6: Compute model heads

@ Menu bar: Calculate / Go calculate

Check that the head of the centre node has changed to -0.144 m



You may move (drag) the bar between the Table and the Map a little to the left.

This increases the size of the Table, but also shows a more accurate result: -0.14434

In the next lesson it will be explained why this result must be equal to:  $\text{square\_root}(3)/12 = 0.1443376$ .

=== Step 7: Save results

@ Menu bar: Files / Save all

If you forget to save your files, MicroFEM will remind you when you close the program by clicking the upper right cross button.

===== MicroFEM Lesson 2A: Drawing contour maps

@ Start MicroFEM and load the "Lesson 1" model by clicking the project file "Lesson 1.fpr"

MicroFEM always starts in Walking mode, with the full model grid on the Map and the current node in the centre.

We found that the head of the centre node is computed as -0.144 m, or -0.14434 to be more precise.

=== Step 8: Draw contour lines of the computed heads

@ Toolbar [Drawing mode] (the third of the lower right 4 buttons).

The grid is automatically removed, and only the nodes are shown.

The Toolbar for the Drawing mode is displayed.

It is easily recognized by the 16 colours of the Palette.

Read the hints of all buttons of this toolbar.

The last button (Draw section (F12)) is grey and cannot yet be used.

Note that the keyboard equivalents from left to right are: Ins, Del, F2, F3, F4, ..., F12

When you click Menu bar: Commands, you will see again that the available commands correspond one-to-one with the toolbar buttons.

There is a little black star in the White box of the Palette. White is the current drawing colour. A drawing colour can be chosen by clicking one of the coloured squares.

@ Click a [Blue] square of the Palette / [Draw grid (Ins)]

@ [Red] / [F5] / [Green] / [F6] / The Spacing = 5000 / [OK]

The map grid are the green 5000-by-5000 m squares on the Map.

All drawing objects are plotted on top of each other, e.g. the red model boundary on top of the blue element boundary lines.

@ Table: Make the "head [m]" cell (4th cell from the top) the active cell.

@ [Yellow] / [F7] / [OK]

A pop-up window with the Minimum value and the Maximum value of all nodal heads is displayed, together with a suggested Interval of 0.02 m

We accepted this Interval and as a result some kind of a "yellow spider web" is displayed on the Map.

@ [F7] / Interval = 0.01 / [OK]

You can choose almost any non-negative interval.

When zooming and panning, the coloured figure is redrawn.

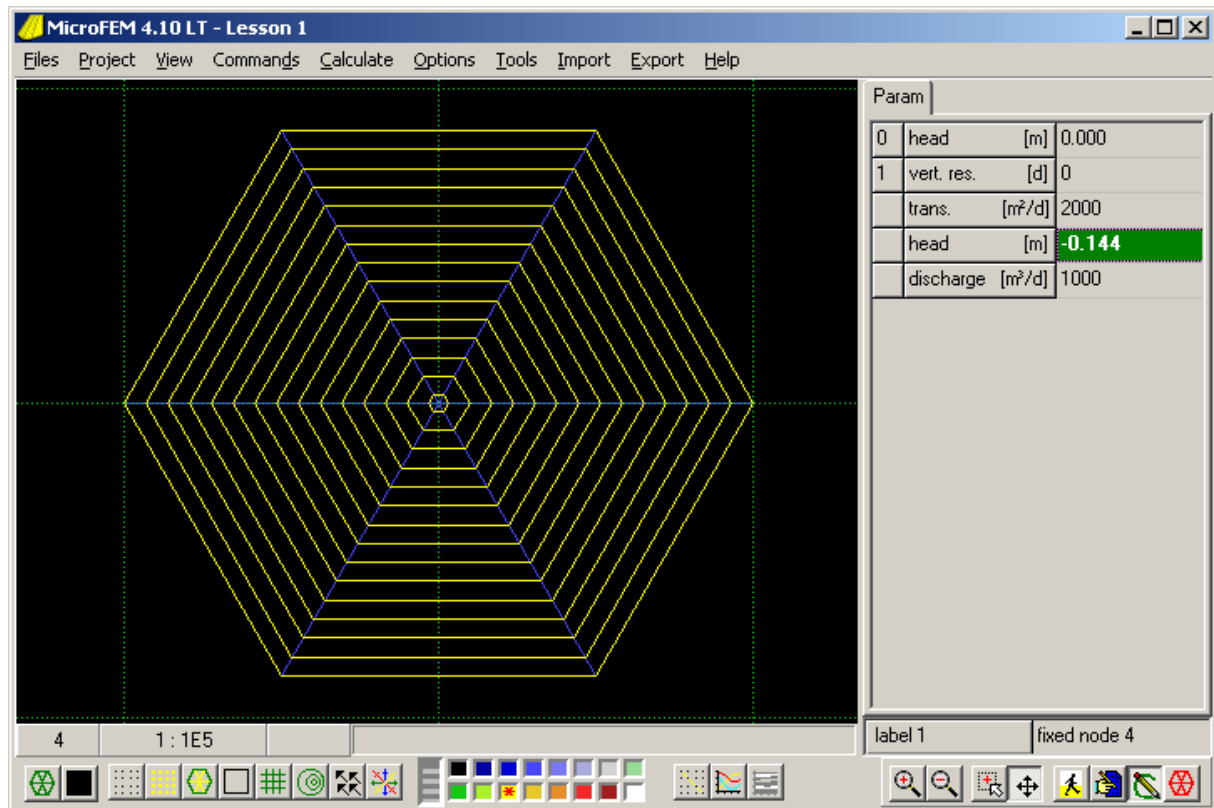
Only when the model becomes extremely small, the very dense map grid is not displayed.

@ Right-click in the Map: a pop-up menu appears. Click on "Fit to scale".



@ Menu bar: Options / Map ... / Uncheck the “Model boundary” box / [OK]

In this way you have (temporarily) removed the red Model boundary drawing object from the drawing list.



From the spider web figure it may be clear that the head contours are drawn by linear interpolation within each element.

When a numerical model is computed, the unknown heads of the nodes are found by solving a set of water balance equations. In MicroFEM these equations are based on the assumption that the head gradient is constant within each element. It is also assumed that the transmissivity is constant within each element. This implies that the flow is uniform (has same magnitude and the same direction) within each element.

In the upper element the direction of flow is 180 degrees (South).

We can compute the gradient here as  $\Delta h / \Delta y = 0.14434 / 4330.13$

We can read  $y=4330.13$  from the Status line, when the cursor is near one of the northern nodes.

Because the elements are equilateral, all element angles are 60 degrees and  $4330.13 = 5000 * (1/2) * \text{square\_root}(3)$

In all other elements the direction of the gradient is different, but its magnitude is the same.

=== Step 9: The drawdown cone

The spider web is a 2D representation of our (very simple) drawdown cone.

In this case the drawdown cone looks like an upside down six-sided pyramid.

@ Menu bar: Tools / 3D Viewer

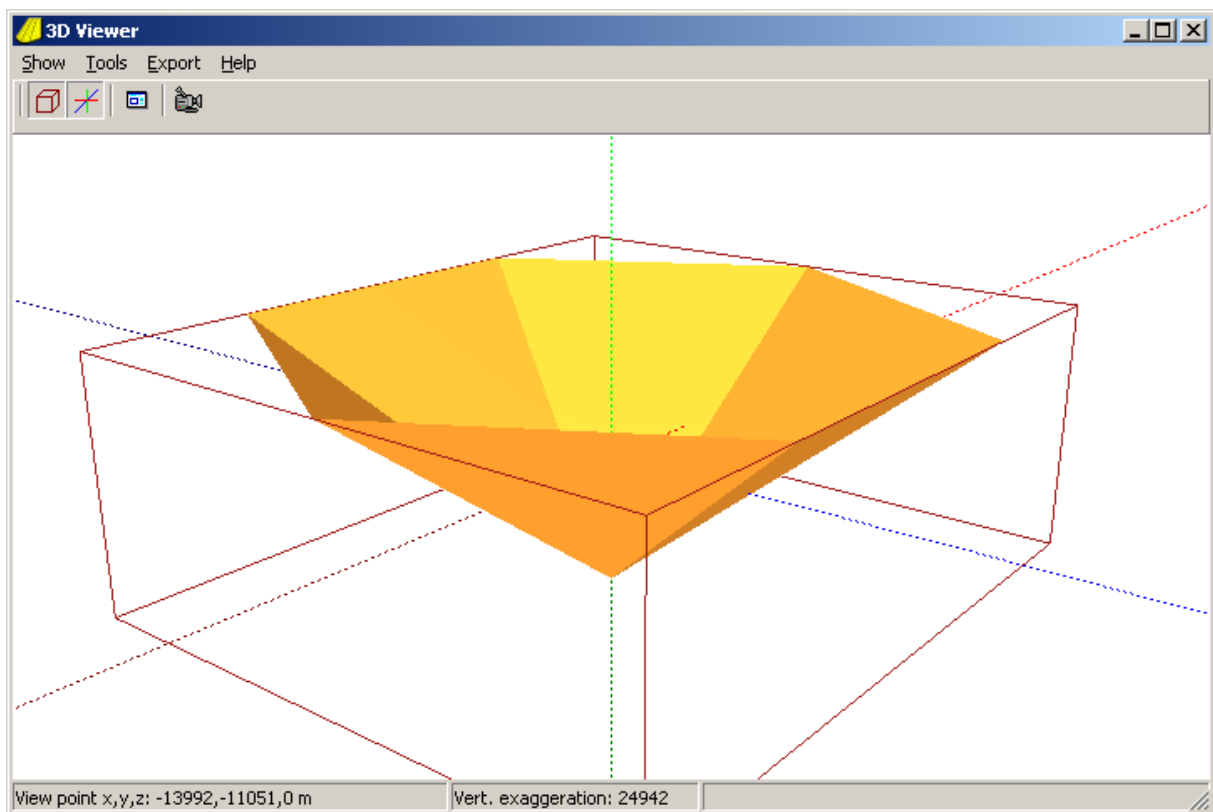
Use the mouse wheel to adjust the size

To rotate the view, move the mouse cursor with the left button pressed.

@ 3D Viewer menu bar: Tools / Options ... / Heads / Uncheck the box next to the green square / [OK]

This removes the green "base" of the pyramid.

The six-sided symmetry of our "drawdown cone" and the constant gradient for each side (element) are obvious.



@ Close the 3D Viewer by clicking on the [x] in the upper right corner of the 3D Viewer screen.

===== MicroFEM Lesson 2B: Finite element head computation

@ Start MicroFEM and load the "Lesson 1" model by clicking the project file "Lesson 1.fpr"

To make sure that all heads are computed, we will run the model first.

@ Menu bar: Calculate / Go calculate

=== Step 10: Water balance

@ [Walking mode] / [Draw grid (Ins)]

@ Make the centre node the current node.

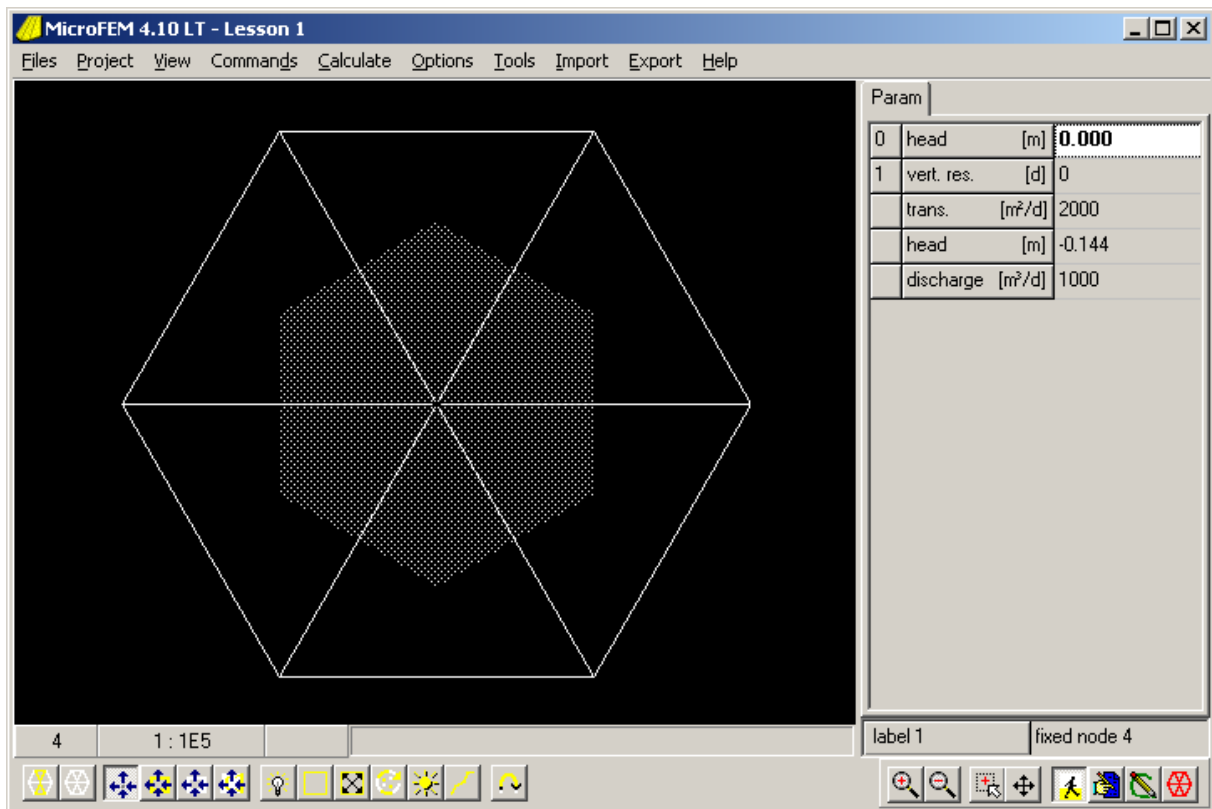
@ [Waterbalance node (F2)]

An area around the current node is shaded.

This area intersects the nodal connections of the current node exactly in the middle.

This area is also exactly one third of each neighbouring element of the current node.

It is called the "Nodal area".



Also a pop-up window with a water balance is displayed

The size of the shaded area appears to be 21650650 m<sup>2</sup>

The outflow (by the well) is 1000 m<sup>3</sup>/d and the lateral inflow is the same.

The well outflow was given (we entered a well discharge of 1000 m<sup>3</sup>/d).

The lateral inflow is computed, based on the location of the nodes, the transmissivity and the nodal heads.

@ [Water balance node (F2)] or Close the Water Balance pop-up window.

=== Step 11: Head computation based on water balance of nodal area

@ Table: Make the "head [m]" cell (4th cell from the top) the active cell.

From now on we will use codes for all cells in the table.

From top to bottom the codes of these cells are: H0, C1, T1, H1, Q1.

The "C" and "Q" are the often used notations for "vertical resistance (days)" and "well discharge (m<sup>3</sup>/day)" respectively.

The "1" is used here because these codes refer to the uppermost aquifer (aquifers are numbered top-down).

H0 and C1 are used in MicroFEM to specify the top boundary condition.

H0 and C1 are zero for all nodes in this model. This implies that the aquifer is fully confined.

Other top boundary conditions will be discussed in another lesson.

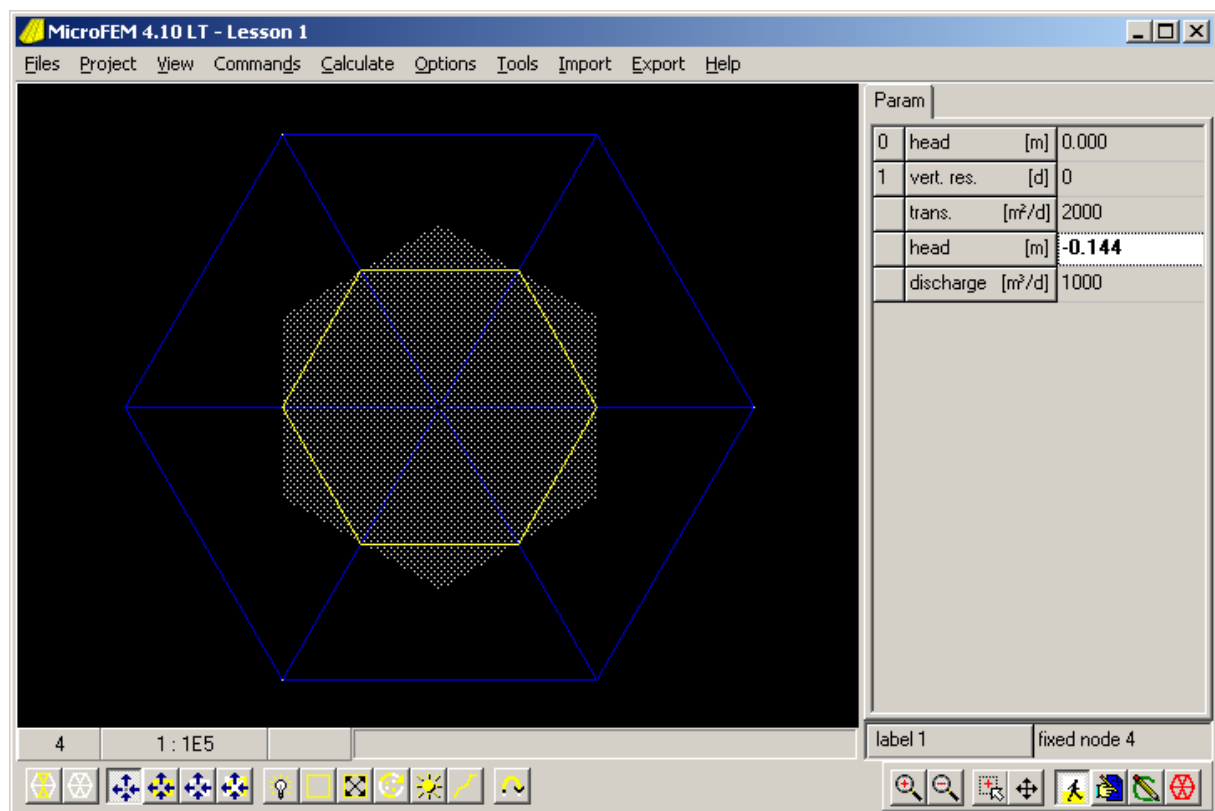
@ Toolbar [Drawing mode] / [Blue] / [Draw grid] / [Yellow] / [Draw contours (F7)] / Maket Interval = 0.072 / [OK]

In this way we created a yellow six-sided polygon around the centre node.

This polygon connects the middles of the nodal connections.

@ Make the centre node the current node.

@ [Walking mode] / [Water balance node (F2)]



The lateral inflow into the shaded area is the same as the flow over the yellow polygon.

The net lateral inflow (inflow minus outflow) of the shaded areas outside the yellow polygon is zero for each element (because these are closed areas within an area with uniform flow.)

The length of each side of the polygon is 2500 m.

The gradient is perpendicular to these sides (in this model).

Using Darcy the inflow over each side = Length of the side \* Transmissivity \* Gradient

Total inflow of the nodal area =  $6 * 2500 * 2000 * \Delta h / (2500 * \sqrt{3})$ .

$\Delta h$  is the head difference between the model boundary and the centre node.

Total outflow = Well discharge = 1000 m<sup>3</sup>/d.

Total inflow = total outflow, so  $\Delta h = \sqrt{3}/12$

Since all boundary heads are equal to zero, the head in the well node =  $-\sqrt{3}/12$

= - 0.144337567 m

With a higher head (less negative) the gradient in all elements would be lower and the lateral inflow would be less than the well discharge.

Similarly, a more negative head and associated higher gradient would produce more lateral inflow than the well discharges.

There is only one head for the centre node that makes the total inflow equal to the outflow.

Please note:

1 - For this simple 6-elements grid, the computed head in the well is independent of the element size !

2 - In reality the drawdown in a well is dependent on the well radius.

We will continue with this in the next lesson.

@ Menu bar: Files / Save all

This last command is routine only.

Actually we did not have to save the model, because we did not change the model data.

===== MicroFEM Lesson 3A: Grid refinement and Well radius

@ Start MicroFEM and load the "Lesson 1" model by clicking the project file "Lesson 1.fpr"

The very simple model we build so far was only good enough to explain some basic principles. A realistic well flow model requires much more nodes between the well and the model boundary.

In MicroFEM it is simple to refine the grid stepwise.

During a refinement step a node is added on each node connection.

In this way each element is split up into four smaller ones.

=== Step 12: Refine grid

@ Menu bar: Project / Information ... / read Information / [OK]

The pop-up window shows that the number of nodes and elements (7 and 6).

@ [Walking mode] / [Delete marks (Del)] / [Toggle all marks (F9)]

In this way all nodes become marked.

@ [Alter-grid mode] / [Add marked nodes (F5)] / [Yes] / [Walking mode]

Check the Project Information again: the number of nodes and elements = 19 and 24.

Check that the new nodes on the boundary are automatically set to "fixed head".

These nodes have the transmissivities of their neighbouring nodes: 2000 m<sup>2</sup>/d

The head of the centre node is still -0.144 m.

All new nodes halfway the well and the boundary have interpolated heads of -0.072 m.

@ Menu bar: Calculate / Go calculate

The head in the well is lowered to -0.192, and the nodes halfway the boundary are -0.048 m.

Note that the difference between the well node and its neighbouring nodes remains 0.144 m

@ [Alter-grid mode] / [F5] / [Yes] / [Walking mode]

Check that there are now 96 elements and 61 nodes.

@ Menu bar: Calculate / Go calculate

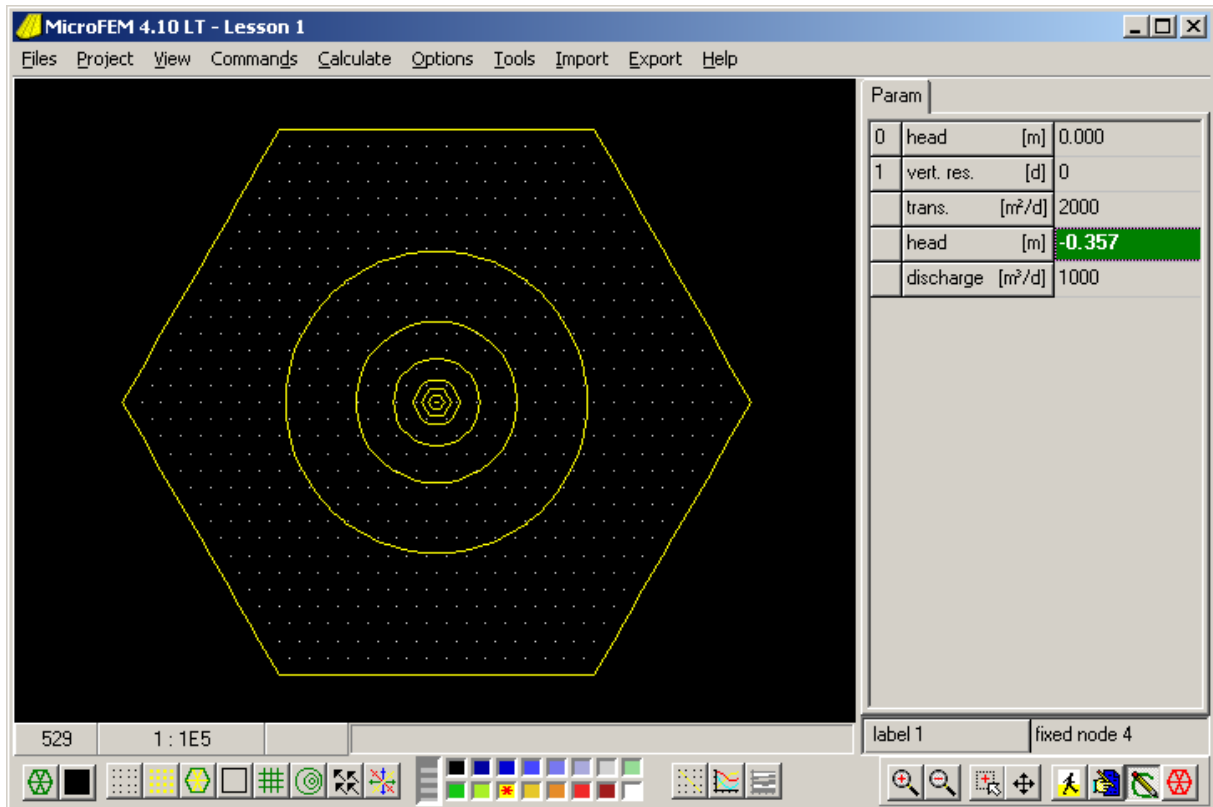
The head in the well is lowered to -0.247

@ Do two more refinement steps and calculate.

The grid consists of 1536 elements and 817 nodes. Well head is -0.357

@ [Walking mode] / [Del]

@ [Drawing mode] / [Del] / [Grey] / [Draw all nodes (F2)] / [Yellow] / select H1 / [F7] / [OK]

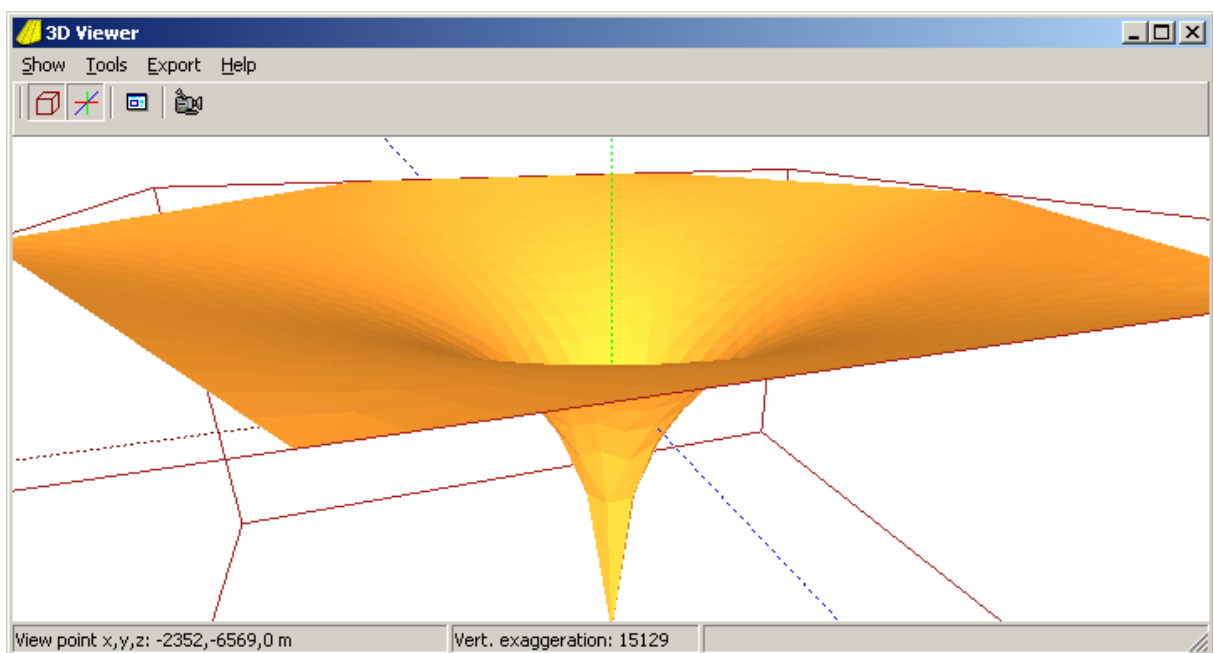


The contour lines show the drawdown cone of the well.

@ Menu bar: Tools / 3D Viewer / Increase the size of the 3D Viewer window.

@ 3D Viewer menu bar: Tools / Options ... / Heads / Uncheck the box next to the green square / [OK]

The drawdown cone is nicely represented. Linear interpolation within the elements is only obvious close to the well.



Apart from rotating and zooming (Step 9 Lesson 2A) more mouse movements are available:

- Changing vertical exaggeration: Move the mouse up-down with the left button and the Ctrl-key pressed.
- Panning the image: Move the mouse with the left button + Shift-key pressed.
- Changing perspective: Move the mouse with the right button + Shift-key pressed.

Use the Grey icon on the Viewer Toolbar to reset the position.

@ Close the 3D Viewer.

=== Step 13: Relation between element size and well radius

It is obvious that the drawdown (s) in the well is a function of element size.  
The real drawdown in the well depends on the well radius (r\_w).

With the Thiem equation  $s = Q/(2 \cdot \pi \cdot T) \ln(r_1/r_2) = 1000/(2 \cdot \pi \cdot 2000) \ln(5000/r_w)$  we can compute for each drawdown (and element size) the corresponding well radius that produces the same drawdown as the finite element model. Here r\_1 is the distance to the model boundary where the drawdown is assumed zero.

	element	
s	size	r_w
0.144	5000	815
0.192	2500	448
0.247	1250	224
0.302	625	112
0.357	312.5	56

The drawdown values (s) are computed with our MicroFEM model. The r\_w values are the well radii that produce the same drawdown when using the Thiem equation.

There appears to be a fixed ratio of about 6 between the element size in the model and the well radius that produces the same drawdown in the well.  
From this we conclude that when we want to compute the drawdown in a well with a MicroFEM model, we have to choose the element size around the well 6 times larger than the well radius.

Since we modified the original model we save it with a new name:

@ Menu bar: Files / Save as ... / "Lesson 3" / [Save]



## ===== MicroFEM Lesson 3B: Labels and Water balances

@ Start MicroFEM and load the "Lesson 3" model by clicking the project file "Lesson 3.fpr"

Of all 817 nodes, there is only one node that represents the well.

Since the well is exactly in the centre of the model, the current node at the start is also the well node.

We recognize the well node by its well discharge (Q1 in the Table) of 1000 m<sup>3</sup>/d.

It is often useful to assign names to nodes. We will label this well node as "well".

Make sure that the current node (the blinking node in Walking mode) is the well node.

### === Step 14: Assigning labels

@ Input mode / Make "Label 1" the active cell.

The "Label 1" cell can be found at the lower right hand side, just above the four Mode buttons.

The text in this cell is "fixed node 4" which was inherited from the grid generation procedure and not important at the moment.

@ Open the cell / type "well" / press this time the Enter-key on the keyboard.

Note that a click on a cell makes that cell the "active cell". Clicking on an active (=selected) cell in Input mode (the cell is blue) opens that cell for input.

When you prefer the keyboard, use Tab key and the Up and Down arrow keys to select a cell, and use the Enter key to open a cell.

By pressing the Enter-key the text "well" is assigned to the current node only.

### === Step 15: Water balance

A water balance is a simple, yet powerful way to look at (and check) model results.

@ Walking mode / [Water balance all nodes (F5)]

The model on the Map is shaded.

The pop-up window shows the model water balance in three columns: "Inflow", "outflow" and "In-Out" (Inflow minus Outflow).

The Inflow is Boundary flow: this should be a value close to 1000 m<sup>3</sup>/d, but slightly different, for example 999.58 m<sup>3</sup>/d.

The outflow is Well flow: 1000 m<sup>3</sup>/d

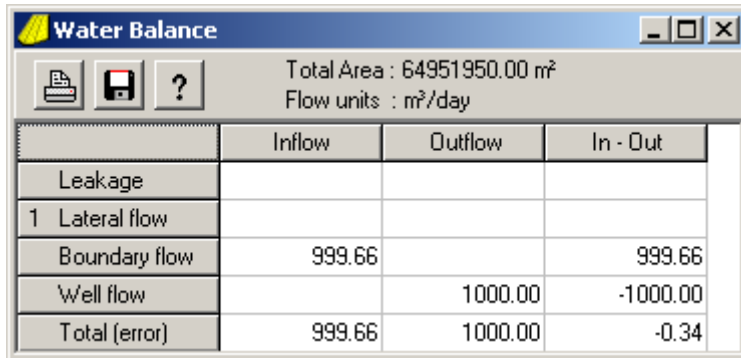
For a steady state model the Inflow must be equal to the Outflow: any difference is an error. Small errors (< 1 per mille of the Inflow) are usually acceptable.

The reason for this small water balance error is that the heads of the nodes have small errors. The present nodal heads are obtained from the file when we loaded the "Lesson 3" model. Heads were saved in the fem-file with a limited accuracy. When we want to improve the accuracy, we have to run the model (again).

@ Menu bar: Calculate / Options ... / [Calculate] / [Stop]

In the black window "Calculation of Heads" the iterative calculation process is displayed. Within a few milliseconds 20 iterations are performed.

When we close the calculation windows and check the water balance, it is clear that the error is reduced, but still larger than zero.



	Inflow	Outflow	In - Out
Leakage			
1 Lateral flow			
Boundary flow	999.66		999.66
Well flow		1000.00	-1000.00
Total (error)	999.66	1000.00	-0.34

@ Walking mode / [Water balance all nodes (F5)]

Apparently we need to do more iterations to obtain a more accurate result.

@ Menu bar: Calculate / Options ... / Minimum number of iteration = 200 (just add a zero) / [Calculate] / [Stop] / [OK]

@ Walking mode / [Water balance all nodes (F5)]

The water balance error is now given as 0.00 m<sup>3</sup>/d.

Since all values in the water balance table are rounded off, this implies that the error is now less than 0.005 m<sup>3</sup>/d.

Save the model.

@ Menu bar: Files / Save as ... / "Lesson 3" / [Save]

=== Step 16: On-line Help system

@ Menu bar: Help / Help F1 / Contents tab

Many years ago, when MicroFEM was still DOS software (before 1998) we used to supply MicroFEM with a 190 p. Manual. All this information (and more) has been reworked for the present MicroFEM on-line Help system. It works like a book with Chapters and Sections.

Also, on many pop-up windows you will find a [Help] button. For example:

@ Menu bar: Project / Information ...

or

@ Menu bar: Calculate / Options ...

When you click such [Help] buttons, a Help screen is displayed related to the information on that window, the so called "Context-sensitive help".

## ===== MicroFEM Lesson 4A: Hexagonal FemMesh grid

From the previous lesson we learned that a small spacing between nodes is required when we want to model the head inside a pumping well. For example, when the well diameter is 0.4 m, its radius is 0.2 m and the nodal spacing around the well should be 1.2 m. Such a small local spacing can best be obtained with the "FemMesh" grid generator.

In this lesson we will build a new model. The purpose is to compute the drawdown inside and around the well accurately. We will first consider a fully confined aquifer, and change this to a leaky aquifer in the next lesson.

=== Step 1: Create a grid for well flow

@ Menu bar: Files / New grid ... / Create New grid / Large contrast grid (FemMesh) / [OK]

@ Enter the Model name: "Lesson 4" / [OK]

@ Toolbar: [F5] / Radius = 10000 / [OK]

The hint of this F5 button reads: "Start with a hexagon", similar to the button we used in Lesson 1 with the "FemGrid" generator.

With this F5 button we automatically filled the Table (right-hand side) with all the required information to generate our grid.

This Table is somewhat different from what we saw in Lesson 1, when using the FemGrid generator.

The "fixed nodes" tab shows that there are 7 fixed nodes.

Do not confuse "fixed nodes" of the grid generator with "fixed head" boundary conditions.

The positions of fixed nodes are given in the table.

These positions remain fixed during the grid generation process.

The coordinates of the fixed nodes are given in the first two columns.

The 3rd column shows the (approximate) spacing near that node when the grid will be generated. This spacing is 2000 m at the boundary and 100 m for the centre node.

@ Toolbar: [F8] / [F8]

When we clicked [F8] the second time, this button showed the text "Nodes on segments".

As a result a number of nodes appear in the shape of 6 triangles.

The nodal spacing towards the centre becomes gradually smaller.

The 6 triangles are defined on the Triangles tab.

@ Table: Triangles tab

@ Click on each of the 6 lines of the Triangles Table

or click on one, and then use the Up & Down-arrow keys

Each triangle is defined by three fixed nodes.

@ Table: Quadrangles tab

There are no quadrangles defined for this grid.

@ Toolbar: [F8] / [F8]

Read the texts on the button before you click on it.

@ Toolbar: [F8]

All nodes within the triangles are shifted (one by one, but very quickly) to the centre of their neighbouring (connected) nodes.

Since the neighbouring nodes are also shifted, this process is repeated a few times.

The purpose of this process is to improve the shape of the elements.

Grids with equilateral elements produce the best (most accurate) results.

Elements with obtuse angles are preferably avoided.

@ Toolbar: [F8]

The “renumbering nodes” step shows no visible action.

Zoom in on the centre node ( $x = 0.00$   $y=0.00$ ) until the surrounding grid is clearly visible.

The position of the node near the cursor is given on the Status bar.

Click on the node just right of the centre node and check that the spacing between these nodes = 94.38 m. In our model we want this spacing much smaller, only 1.2 m.

@ Table: Fixed nodes tab

@ Table: Enter a spacing of 1.2 for the centre node (line 4, column 3)

@ Toolbar: [F8] / [F8] / [F8] / [F8] / [F8] / [F8]

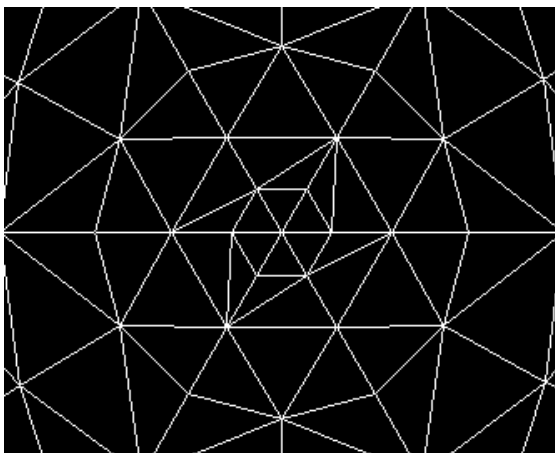
The Status line shows that the grid now consists of 675 nodes.

When you want to repeat the grid generation process, you can start by entering a new (or the same) value in any cell of the table, or you press the Esc-key

Zoom in on the centre node. Click on the node just right of the centre node and check that the spacing here is 1.18 m.

Save the grid and continue with Lesson 4B.

@ Toolbar: [F9] / Save fen-file / [F10] / [OK]



===== MicroFEM Lesson 4B: Stopping criterion

=== Step 2: Enter model data, boundary conditions and compute heads

We will here repeat some actions of Lessons 1B and 1C (steps 3-6)

@ [Input mode] / Select T1 / Open cell / 2000 / [F5]

Zoom in and make sure that the blinking current node is the centre node

@ Select Q1(Discharge) / Open cell / 1000 / [Enter]

@ [Walking mode] (or "Alt+1" on the keyboard).

@ Toolbar: [Ins] / [Del] / [F7]

@ [Input mode] / Select H1

@ Toolbar: [Fixed heads for marked nodes (F9)] / [Del]

Check that all boundary nodes are fixed (at 0.00 m)

@ Menu bar: Calculate / Go calculate

@ [Walking mode] / [F5]

Check the water balance.

The model water balance shows an error: the boundary inflow is less than the well outflow.

@ Menu bar: Calculate / Options ... / set "Error of heads" = 1E-12 / [Calculate]

The "Error of heads" is a stopping criterion for the iterative procedure that computes all unknown heads. During one iteration a new value for all unknown (not fixed) heads is found, based on the heads of the previous iteration. In this way all heads are systematically improved (i.e. closer to the theoretical solution). The largest head improvement of all nodes is displayed on the screen at the end of each iteration. When this largest improvement is smaller than 1E-12 (the stopping criterion we used) the calculation process stops. To obtain the required accuracy takes about 450 iterations.

@ [Stop] / [OK] / [F5] / Check water balance / [F5]

@ [Zoom area] / Zoom in until the centre node is clearly shown / [Zoom area] (switch it off)

Check that the head in the well is -0.840 m

We can check this result with the analytical solution (Thiem equation, see Lesson 3: Step 13). With a well radius of 0.20 m and an approximate average distance to the boundary of 9000 m, the steady state drawdown in the well is computed as 0.8526 m

Save this model for the next lesson.

===== MicroFEM Lesson 5A: Profiles and Leaky aquifers

Load the model of Lesson 4

=== Step 3: Drawdown cone in profile

@ Right-click in the Map / Fit to scale

@ [Drawing mode] / [F10] / [F11]

First a yellow straight line is displayed in the map: the Profile line.

Then the profile itself is displayed, showing the drawdown cone (yellow curve).

Shift the profile window out of the map area. Adjust the size if necessary.

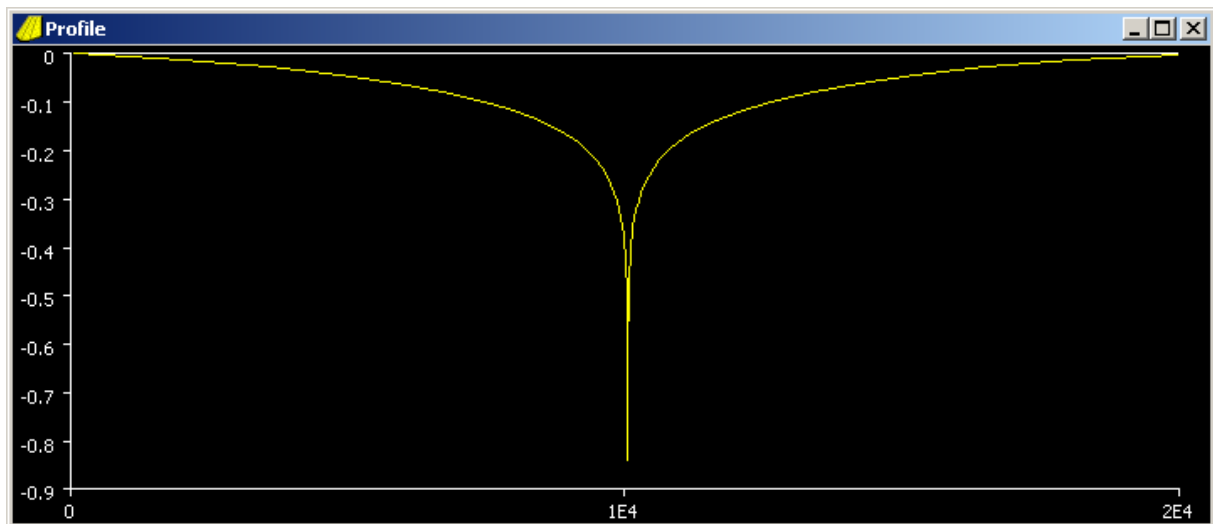
The default position of the profile line in the Map is West-East through the centre of the model.

A left mouse click at a location in the map sets the position of the left end of the line.

Similarly a right-mouse click positions the right end of the line.

The graph in the profile is updated immediately.

The end points can be placed anywhere, also outside the model boundaries.



Enlarge the Profile window.

The x-axis always starts with zero at the left-hand side.

Right-click in the Profile for a pop-up menu.

You can zoom-in to display details.

Use "Fit to scale" to reset the scale.

When "Zoom In" is unchecked (right-click in profile to uncheck), you can also left-click in the Profile.

The "Profile options" window appears with a "Scale" and "Colors" tab.

I assume that these options do not require further explanation.

The same "Profile options" window can be obtained from the Main menu:

@ Menu bar: Options / Profile ...

The Profile graph can be exported to the Clipboard

@ Menu bar: Export / Copy to clipboard / Profile

Now you can paste the Profile graph in PowerPoint, Word, etc.

The Profile window and the Profile line can be removed with the same [F10] and [F11] buttons (or keys).

=== Step 4: Leaky aquifer

For a leaky aquifer we have to allow vertical flow in an aquitard above the aquifer. This requires a fixed head boundary at the top of the model and a resistance between the aquifer head and this fixed head.

The fixed head is assigned to H0, the vertical resistance to C1.

The vertical resistance of an aquitard is defined as the saturated thickness of a (homogeneous) aquitard divided by its vertical conductivity. It is expressed in days (and the reciprocal of leakance).

The vertical flux in the aquitard is proportional to the difference in head (H0-H1). More precisely, it is computed as (H0-H1)/C1. The downward leakage (m3/d) for a nodal area is computed as  $a \cdot (H0-H1)/C1$ , where "a" is the nodal area (m2).

The resistance of a completely impervious aquitard is infinitely large.

IMPORTANT

=====

One of the MicroFEM "rules" is that when a resistance is infinite, we enter the value "0" (zero).

=====

That explains why in our original fully-confined model we kept the C1 at zero. When C1 is zero, there will be no vertical flow in the top aquitard and the value of H0 has no effect on the computed fluxes or heads.

In our model we assume a vertical resistance of 1500 days.

When we keep H0 at zero (the same as the fixed head aquifer boundary) the computed heads in the aquifer show the drawdown by the well.

@ [Input mode] / C1 / 1500 / [F5]

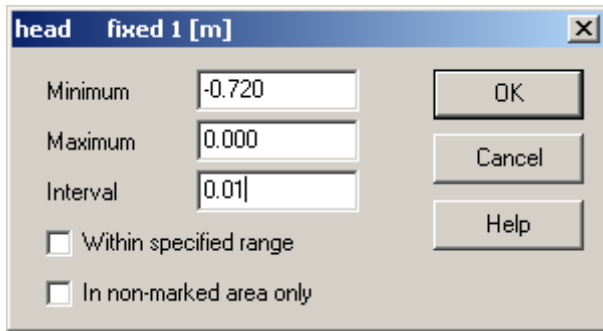
@ Menu bar: Calculate / Go calculate

@ [Walking mode] / [F5] / Draw your conclusions / [F5]

The model water balance shows that the boundary inflow is reduced to 35 m3/d.

Almost all water discharged by the well now comes from leakage through the aquitard.

@ [Drawing mode] / H1 / [F7] / interval = 0.01 / [OK]



Repeat the previous actions when you did not notice that the Minimum head of all nodes is -0.720 m

Apparently, changing from fully confined to semi-confined reduced the drawdown in the well from 0.840 to 0.720 m.

=== Step 5: No-flow boundaries

In step 2 we set fixed-head boundaries for the confined aquifer.

Since the leaky aquifer is (mainly) recharged by the top aquitard, we can (alternatively) choose a no-flow boundary condition.

@ [Walking mode] / [Ins] / [Del] / [F7]

@ [Input mode] / H1 / [Nonfixed head for marked nodes (F8)] / [Del]

Calculate heads, check the water balance and find the head in the well.

	Inflow	Outflow	In - Out
Leakage	999.24		999.24
1 Lateral flow			
Boundary flow			
Well flow		1000.00	-1000.00
Total (error)	999.24	1000.00	-0.76

Draw your conclusions, then go to Lesson 5B.



## ===== MicroFEM Lesson 5B: Boundary conditions

When we chose a fixed-head boundary, the head is specified (given) by us, while the boundary flux (and the head gradient) is computed by the model.

When we chose a no-flow boundary, the boundary flux (and the gradient) is given by us (zero), while the head is computed by the model.

From the latest computations it appears that the model boundary condition (fixed head or no-flow) has so little effect on the drawdown in the well that we find a well head of -0.720 in both cases.

When the model boundary is chosen far enough from the well, the boundary condition has a negligible effect on the model results.

The drawdown at the no-flow boundary is 0.000 m at the corners of the model boundary, and -0.001 m in the middle of the North and South boundary sides. When you drag the bar between the Table and the Map a little to the left (as we did in Lesson 1C, Step 6) these values appear to be -0.00040 and -0.00066 m respectively (when "Error of heads" is still set at 1E-12 on the Calculation options window)

@ Menu bar: Files / Save all

When discussing boundary conditions this usually refers to the outer boundary of the model. However, this is not necessarily always the case. It is perfectly permitted to define an inner fixed-head boundary. The heads of such nodes remain fixed when the model heads are computed. This implies that water will flow into or out of the model via such inner open boundaries.

Example: In our present single layer well flow model we discharge 1000 m<sup>3</sup>/d and compute a head in the well of -0.71975 m. We can consider this as a constant (or given) boundary flux condition. Alternatively, we can set the well discharge to zero and make this single node an open boundary with a fixed head of -0.71975 m

@ Zoom in on the well node (e.g. scale 1:100)

@ [Walking mode] / [Del] / [Automarker (F10)] / Click the well node / [F10]

The well node is now marked, but this is only obvious from the yellow bordered node numbers on the Status line. To recognise a single marked node in the map you can "toggle" all marks with the [F9] button. This button invokes that all marked nodes become unmarked, and all unmarked nodes become marked. Only marked nodes are connected with yellow lines. Click [F9] again to restore the situation with the well node marked only.

We will now set the fixed-head boundary condition:

@ [Input mode] / H1 / [F9] / [Del]

and remove the well discharge (set Discharge to zero):

@ Select Q1 / Open cell / 0 / [Enter]

The well node head is now fixed at -0.71975 and the well discharge is zero.

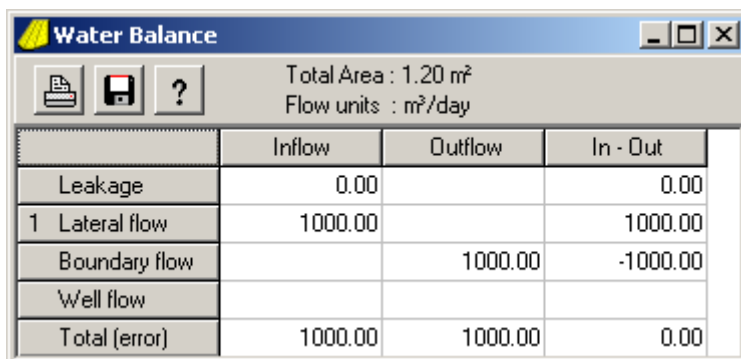
@ Menu bar: Calculate / Go calculate

@ [Walking mode] / [F5]

The boundary outflow appears to be 1000 m<sup>3</sup>/d, the same as the original well discharge.

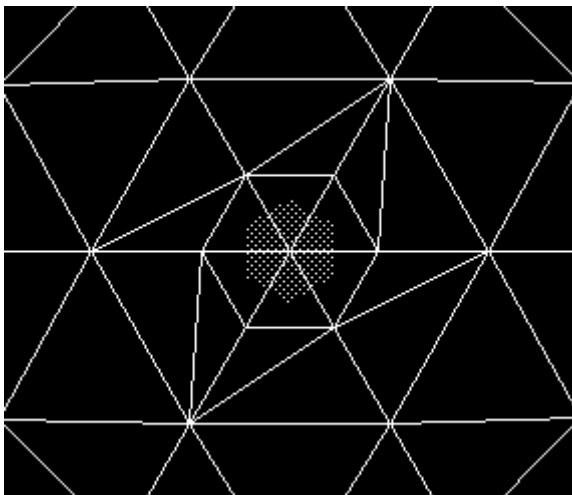
We can now specify any fixed-head well level and compute the discharge. Drawdown and discharge are linearly related. Check this with well levels of -0.36 m (50%), -1.4395 (200%) and when the well level is reset to -0.71975.

@ [Walking mode] / [F2]



	Inflow	Outflow	In - Out
Leakage	0.00		0.00
1 Lateral flow	1000.00		1000.00
Boundary flow		1000.00	-1000.00
Well flow			
Total (error)	1000.00	1000.00	0.00

The lateral flow component is the (horizontal) flow across the internal boundary of the water balance area (i.e. the boundary between the shaded water balance area and the other (non-shaded) parts of the model). In this case the lateral flow is the inflow over the six-sided boundary of the shaded area.



The inflow from leakage (in this small = 1.2 m<sup>2</sup> area) is given as 0.00 m<sup>3</sup>/d. When we compute this manually, we find a very small value indeed:

$$\text{Leakage} = a \cdot (H_0 - H_1) / C_1 = 1.2 \cdot (0.71975) / 1500 = 0.000576$$

This ends lesson 5.

The model has already been saved.

===== MicroFEM Lesson 6A: FemMesh grid generation

One of the advantages of finite element models above finite difference models is that f.e.models are far more flexible with respect to the grid. The irregularly shaped elements allow you to take care of all kinds of linear or curved inner and outer boundaries (aquifer boundaries, faults, meandering rivers, etc.). Nodes can be located at the exact position of pumping and observation wells.

This flexibility also has disadvantages, since it takes more effort to create a well-designed grid. However, when our ambitions are not very high, we can do with a type of grid typical for f.d.models (like ModFlow). In such models the shape is a simple rectangle and the grid is more refined somewhere in the middle part because that is the area of interest.

To create such a grid is very simple with the FemMesh generator. It requires the location of 8 nodes: 4 boundary nodes and 4 nodes that define the four-sided more refined inner area. For each of these 8 nodes we also have to specify the element sizes near that node. Around the four-sided inner rectangle we define 4 trapezoidal quadrangles. These five quadrangles cover the full model area. In the following example we will create such a simple FemMesh grid.

The model area is about 7500 by 5500 m, while the centre part with the finer spacing is 1500 by 1500 m.

@ Start MicroFEM /

@ Menu bar: Files / New grid ... / Create New grid / Large contrast grid (FemMesh) / [OK]

@ Enter the Model name in the upper cell: "Lesson 6" / [OK]

@ Table: Fixed nodes tab / Enter 8 in the cell above the table / [Change number to]

Clicking the [Change number to] button increases the size of the Fixed nodes table to 8 rows.

@ Enter the following data one-by-one in the table of 8\*3 cells.

The easiest way is to open the upper left cell and then only use the Keyboard: enter 0 / [Enter] / 0 / [Enter] / 400 / [Enter] / 7500 / etc.

0	0	400
7500	0	400
7500	5500	400
0	5500	400
3000	2000	50
4500	2000	50
4500	3500	50
3000	3500	50

When there are tabs (not spaces) between the numbers, you can also paste all data into the table. In that case right-click the upper-left cell when it is blue (not open) and select "Paste" in the pop-up menu.

We do not use Triangular regions (skip that Tab) since our grid will be based on five quadrangular regions.

@ Table: Quadrangles tab

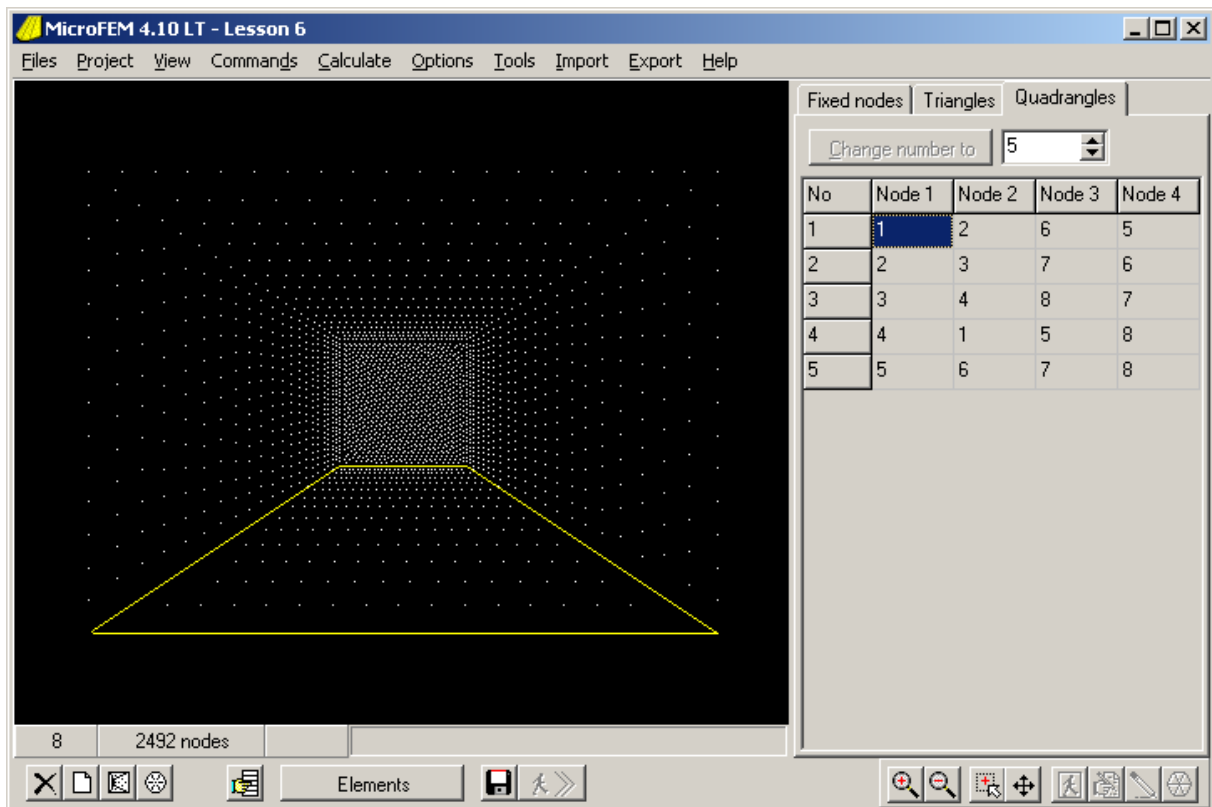
@ Change the number to "5" / [Change number to]

@ Enter the following data one-by-one in the table of 5\*4 cells (pasting doesn't work here)

1 2 6 5  
2 3 7 6  
3 4 8 7  
4 1 5 8  
5 6 7 8

@ Toolbar: [F8] / [F8] / [F8]

When you are still unsure how the five quadrangles are defined, click on each row of the Quadrangles table.



Finish the grid generating procedure.

@ Toolbar: [F8] / [F8] / [F8]

Of course it is not required to create such a symmetrical grid.

@ Change data fixed node 3 into: 9000 7000 700

@ And fixed node 7 into: 4600 3600 70

@ Generate the grid (six times [F8])

Since the maximum number of regions is 100 quadrangles and 100 triangles a quite complex grid can be generated with FemMesh.

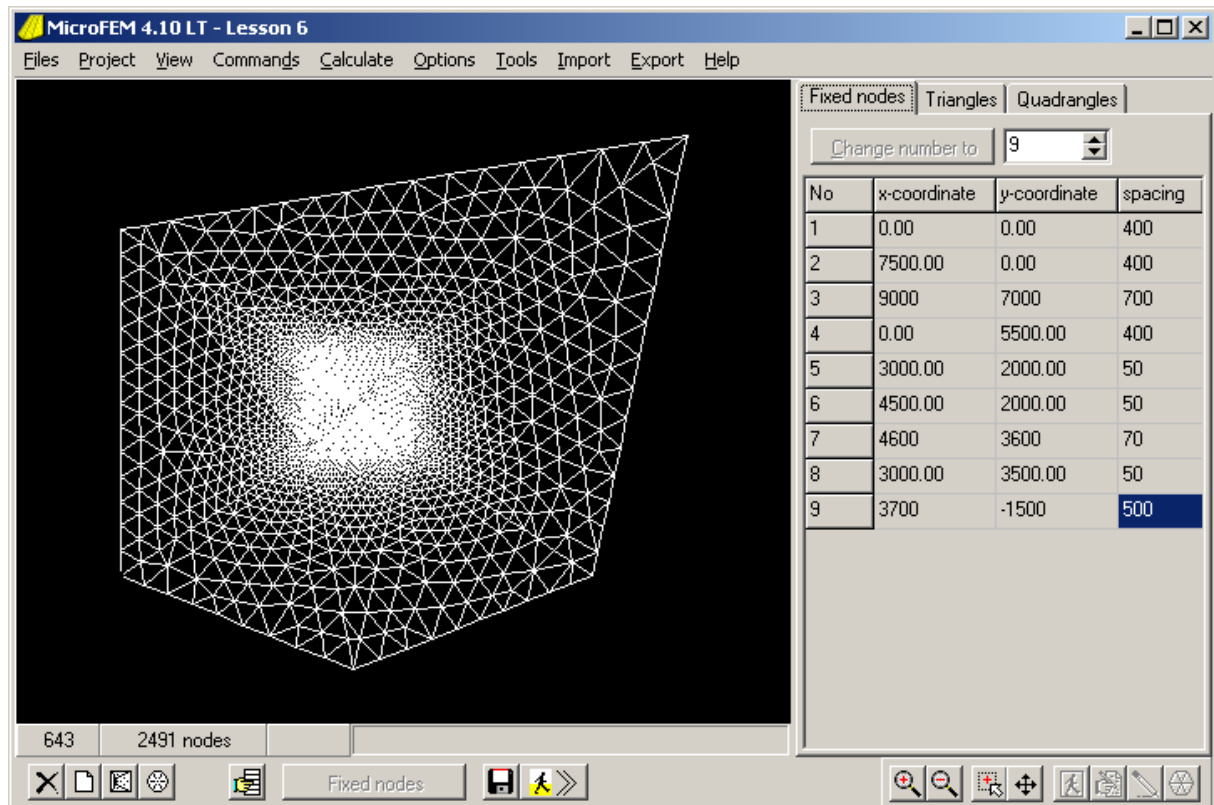
Let us make the grid slightly more complex;

@ Add a Fixed node (no 9): 3700 -1500 and spacing 500

@ Add three triangles: 1,5,9; 5,6,9 and 9,6,2.

@ Delete the first Quadrangle (Click on one of its cells / Right-click / Delete / Quadrangle

@ Generate the grid.



When you are not satisfied with the results, you may press the [Esc] key anytime to interrupt, modify input data and then restart the grid generation process.

@ Save the grid [F9]

The shape of the triangles and quadrangles is free to choose as long as they do not overlap, and together cover the whole model area. Actually, small regions within the model area may be left undefined. In such regions no nodes will be added during the “Nodes within regions” step.

@ Load the Example input [F4] and check the above remark about small undefined region.

===== MicroFEM Lesson 6B: Two-aquifer system and Grid manipulation

=== Step 2: A two-aquifer system

Assume that a well pumps from the top aquifer of a leaky two-aquifer system.

@ Start by clicking the “Lesson 6.fen” file and set the number of aquifers to 2 / [OK]

When we have a single aquifer model on the screen, we can add a new aquifer at the base of the present aquifer.

@ [Alter-grid mode] / [F11] / 2 / [OK]

Four cells are added to the “Param” table, with the following codes: C2, T2, H2 and Q2

The water table in the top of the aquitard is assumed fixed at 12 m.

The resistances of the aquitards is 800 (top aquitard) and 200 days (between aquifers).

The transmissivities of the aquifers is 600 (top aquifer) and 300 m<sup>2</sup>/d (base).

@ [Input mode] / Select C1 / 800 / [F5] / T1 / 600 / [F5] / C2 / 200 / [F5] / T2 / 300 / [F5]

The water table in the top of the aquitard is assumed fixed at 12 m.

The model boundary condition is no-flow all around.

@ [Input mode] / Select H0 / 12 / [F5]

The well is located at x=3645 y=2600 and discharges 600 m<sup>3</sup>/d from the top aquifer.

We want a well node at the exact position.

@ [Drawing mode] / [F6] / spacing = 10000 / offset X = 3645 / offset Y = 2600 / [OK]

A cross hair appears showing the well location (why?). Zoom into that area.

@ [Alter-grid mode] / [Ins]

@ Point at the node closest to the cross hair, click and shift the node.

@ Use [+] a few times to further zoom in and bring the node further in position.

@ Now Use [-] a few times to see part of the grid again.

@ Relocate some neighboring nodes to make the elements more or less equilateral again.

@ When you are satisfied [Ins] to switch “relocate nodes” off.

@ [Input mode] / current node at cross hair node / Select Q1 / 600 / [Enter] / Select Label 1 / well / [Enter]

@ Right click in the map / Fit to scale

@ Menu bar: Calculate / Options ... / Set “Error of heads” to 1E-8 / [OK]

@ Menu bar: Options / Save settings

This stopping criterion will now automatically be chosen for future MicroFEM sessions. The settings are stored for the active user, and so depend on the user who has logged on to the system. The default settings can be restored at any time using: Options / Defaults

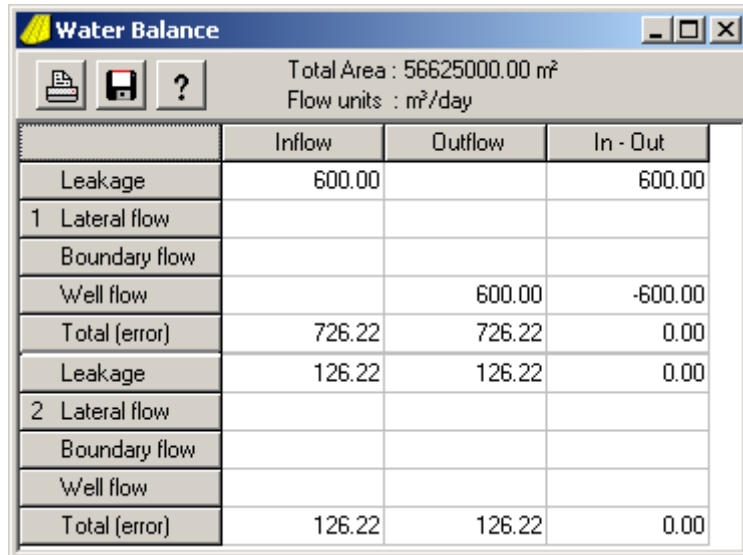
@ Menu bar: Calculate / Options ... / Calculate / [Stop] / [OK]

@ [Walking mode] / [F5]

You probably have to enlarge the water balance table to see both layers.

Notice there is both upward and downward flow in the C2 aquitard.

Why are they the same?



The screenshot shows a window titled "Water Balance" with a toolbar containing icons for print, save, and help. The window displays the following information:

Total Area : 56625000.00 m<sup>2</sup>  
Flow units : m<sup>3</sup>/day

	Inflow	Outflow	In - Out
Leakage	600.00		600.00
1 Lateral flow			
Boundary flow			
Well flow		600.00	-600.00
Total (error)	726.22	726.22	0.00
Leakage	126.22	126.22	0.00
2 Lateral flow			
Boundary flow			
Well flow			
Total (error)	126.22	126.22	0.00

Close the water balance table.

@ Draw the head contours of the 1st and 2nd aquifer in different colors.

What are the Minimum heads? And the largest drawdowns?

@ Check nodes at the boundary and note that drawdown in the 2nd aquifer  $\geq$  than in the top aquifer.

@ Draw head contours of the 2nd aquifer with a very small interval to find out what part of the model boundary is most effected by the well, and how much.

For the computation of heads it is sufficient when T-values (transmissivities) and c-values (vertical resistances) are known. Up to now we did not use K-values (hydraulic conductivities) and layer thicknesses. This latter information is only required when we want to compute 3D-flowlines or flow vectors, when we want to see sections of the model, and when we want the transmissivity of unconfined aquifers to be a function of the water table height.

For now, we will only add uniform thicknesses to all layers.

@ Menu bar: Project / Project manager ... / add unit to project [green +] / New / [OK] / Close project manager

In the table, next to the "Param" tab, a new tab has appeared "Thick". It contains 5 cells.

@ [Input mode] / top level = 12 / [F5]

Top level is the height of the top of the model (water table) above reference level.

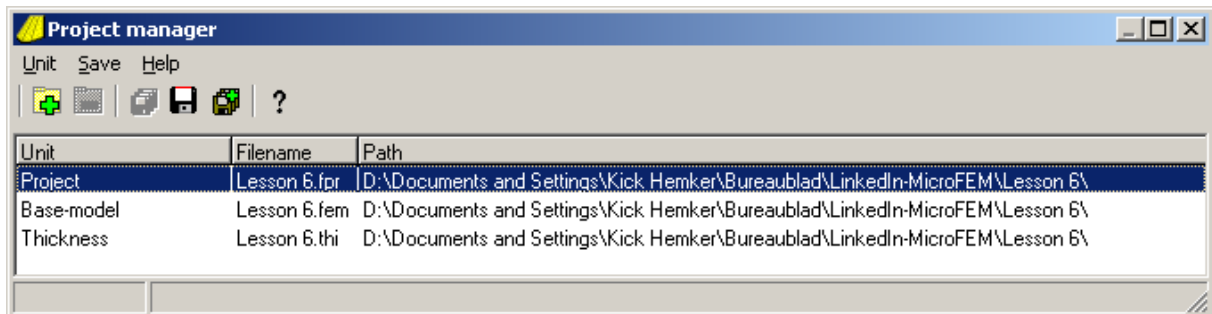
@ Similarly enter thicknesses for all (4) layers: from top to base = 10, 20, 2 and 12 m.

Save the model

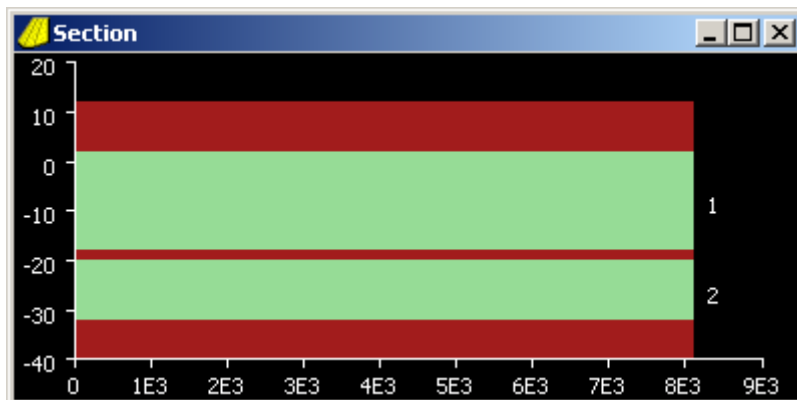
@ Menu bar: Files / Save all

Have a closer look at the Project manager.

Thickness data are stored in a separate thi-file



@ [Drawing mode] / [F12] / try left and try right mouse clicks / [F12]



The [F12] button is only available when the thickness unit is added to the model.



## ===== MicroFEM Lesson 7A: Marking nodes and Heterogeneity

When you are working with a word processor and you want to perform an action with part of the text, e.g. copy that text, you first have to select that part of text and then execute the action. MicroFEM works the same way. When you want to perform an action with only part of the model, e.g. assign values to some parameter or compute the water balance, you first have to mark that particular part of the model and then execute the action.

When you start with a model, no nodes are marked. Nodes can be marked either one by one, along a line, or in some area. In previous lessons we have been marking nodes a few times: the model boundary (lesson 1b), all nodes (lesson 3A) or only a single node using the Auto-marker [F10] (lesson 5B).

Another useful command in Walking mode is [Line marker (F11)]. It will mark all nodes on a line between nodes that are clicked. First switch [F11] on and then click two nodes in order to obtain a marked line. When there are many nodes between the two clicked nodes and the grid is irregular, it is not always obvious which path will be chosen. When you have a particular path in mind, make regular clicks. Also, don't forget to switch off the line-marker once you have finished marking the required line of nodes.

We will use the line-marker to mark the northern part of the model and assign a low transmissivity for the upper aquifer. We will make this "northern area" equal to the northern quadrangle we used when we created the grid.

@ Load the "Lesson 6" model

@ [Line marker (F11)] / Click on the most NE-boundary node / Click on a node somewhere halfway on the straight line that connects to the Inner area / + + / continue with marking the straight line until the Inner area is reached.

When zooming (keyboard + or - , or zoom buttons [+] or [-]) the model is also shifted to place the current node in the centre of the map (unless the current node is so close to the model boundary and shifting will only reduce the visible part of the model). A [+][-] or [-][+] sequence shifts the current node to the centre, keeping the scale unaltered.

@ Continue with the Line-marker and follow the Inner area northern boundary to the left until its NW node, then follow the straight line to the most NW boundary node / [F11]

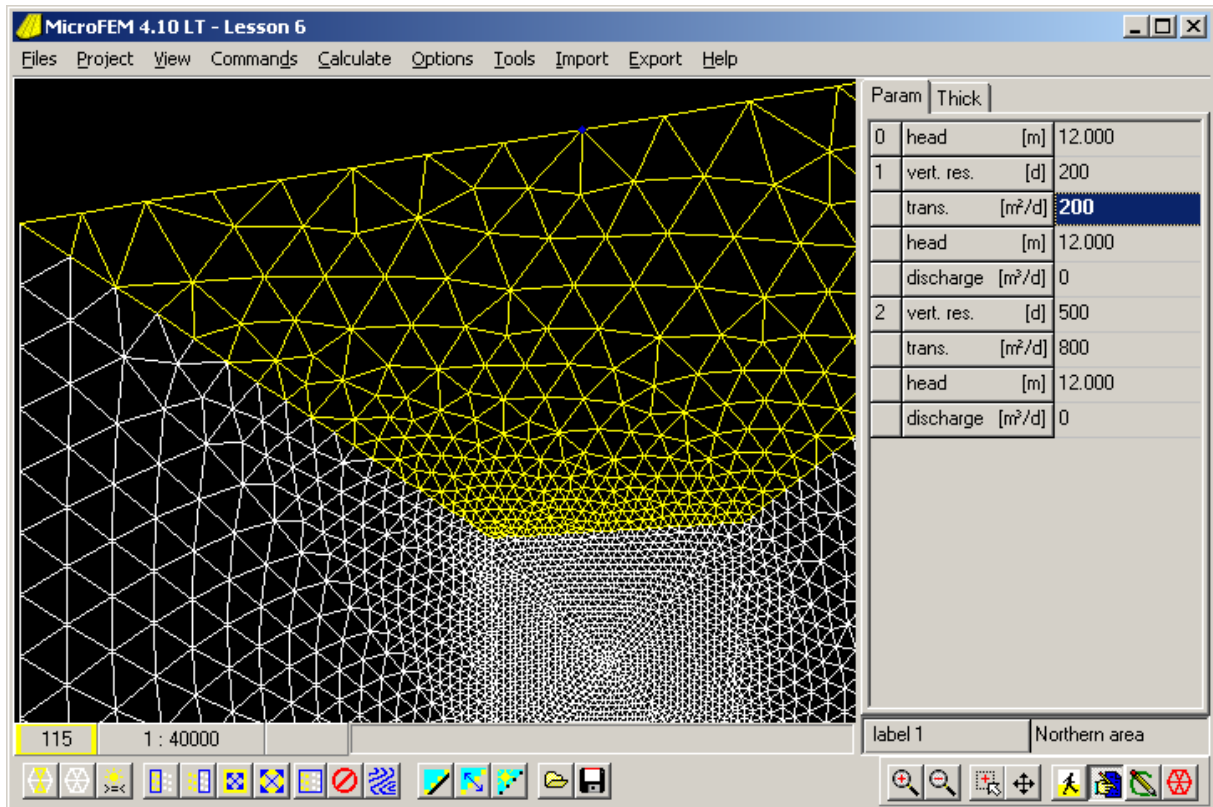
@ Click on one of the unmarked nodes in the northern quadrangle / [Mark up to marked (F8)]

All nodes of the northern quadrangle are now marked.

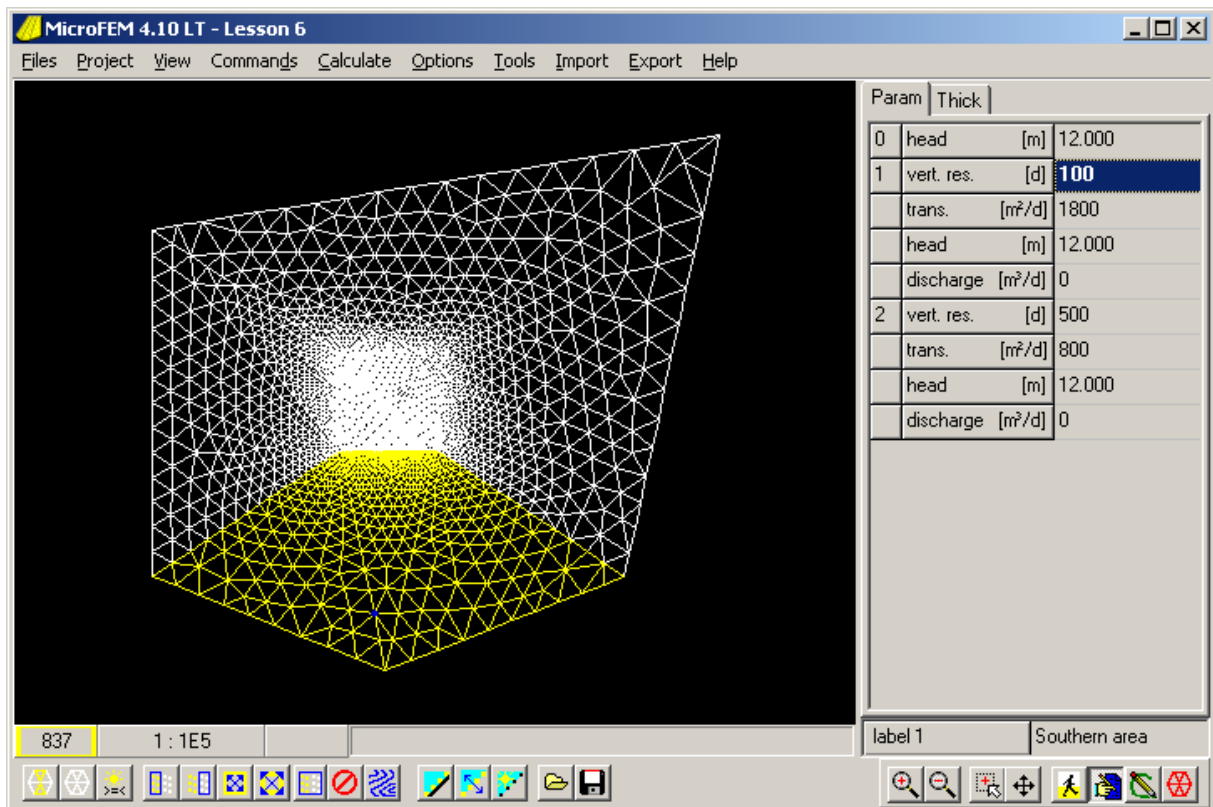
@ [Input mode] / Select Label 1 / Enter "Northern area" / [Marked nodes (F3)]

@ Select T1 / 200 / [F3]

We can mark any part of the model and then assign a value to the marked nodes for some parameter using [F3]. We can also assign to all non-marked nodes using [F4] or to all nodes with [F5]. The position of the current node is not important in those cases. When we assign only to the current node, we use the Enter-key.



@ First remove all marks, and then do the same marking procedure for the southern area. Assign the label "Southern area", a T1-value of 1800 m<sup>2</sup>/d and a C1-value of 100 days.

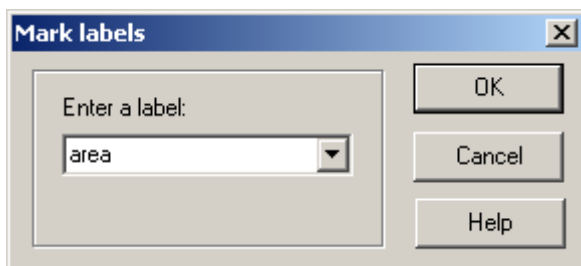


When you draw contours of Label 1, the resulting lines show 4 separate areas: “Northern area”, “Southern area”, “well“ (a single node only) and the area in between without a particular label.

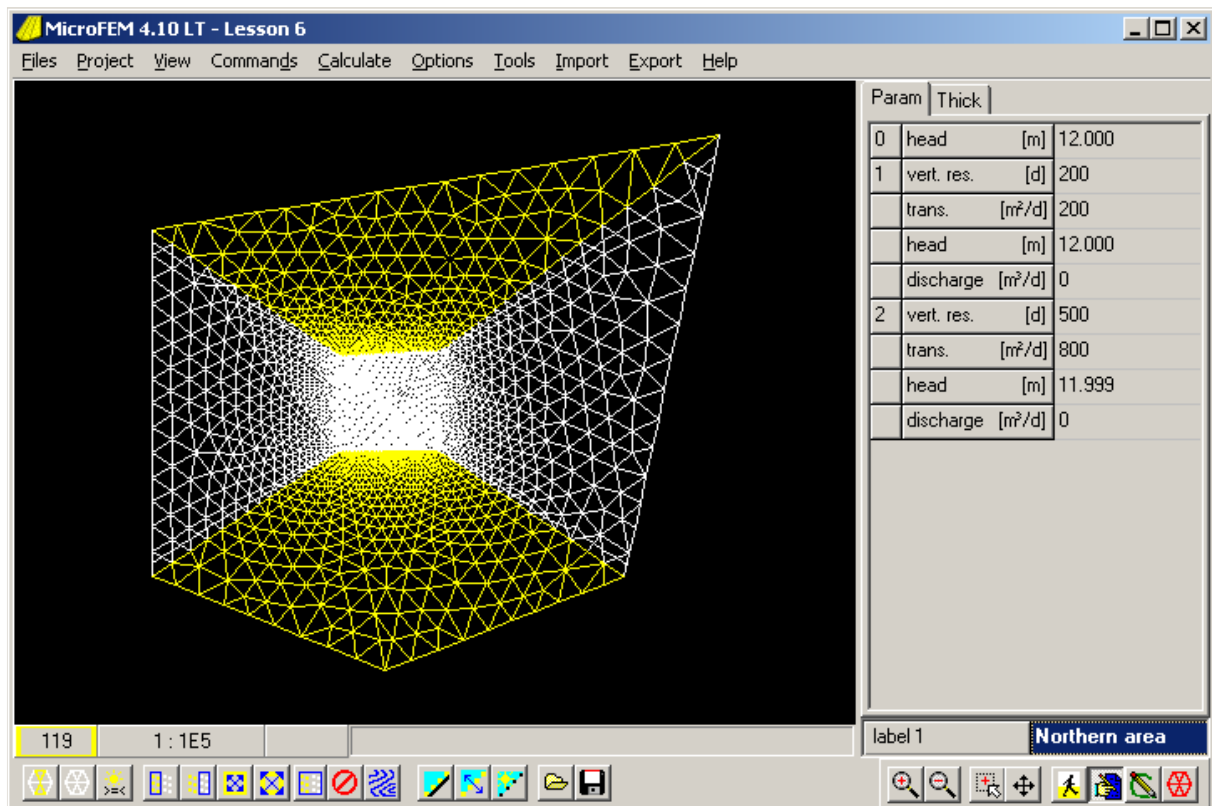
To mark the north area again, place the cursor on one of its nodes, select Label 1, [Input mode] / [Ins] / [Del] / [Mark data or labels (F2)] / [OK]

In this case “area” is part of both labels. That allows us to mark the north and the south area at the same time.

@ Select Label 1 / [Input mode] / [Ins] / [Del] / [Mark data or labels (F2)] / “area” / [OK].



This trick only works when there exists no label “area”. So, MicroFEM first looks at all nodes to see whether “area” exists, and when none is found it starts all over and marks all nodes that have “area” as part of their label. The same trick works in this case when we enter “them” or even “o”.



## ===== MicroFEM Lesson 7B: Using formulas and the Xtra worksheet

The same [Mark data or labels (F2)] button can be used for parameter values. When we want to mark the area where T1 is between 500 and 1500 m<sup>2</sup>/d, select T1 / [Input mode] / [Ins] / [Del] / [Mark data or labels (F2)] / select the middle radio button “values in between” and enter 500 and 1500 in the upper and lower box / [OK].

=== Formulas: constants, operators, functions, parameters and variables

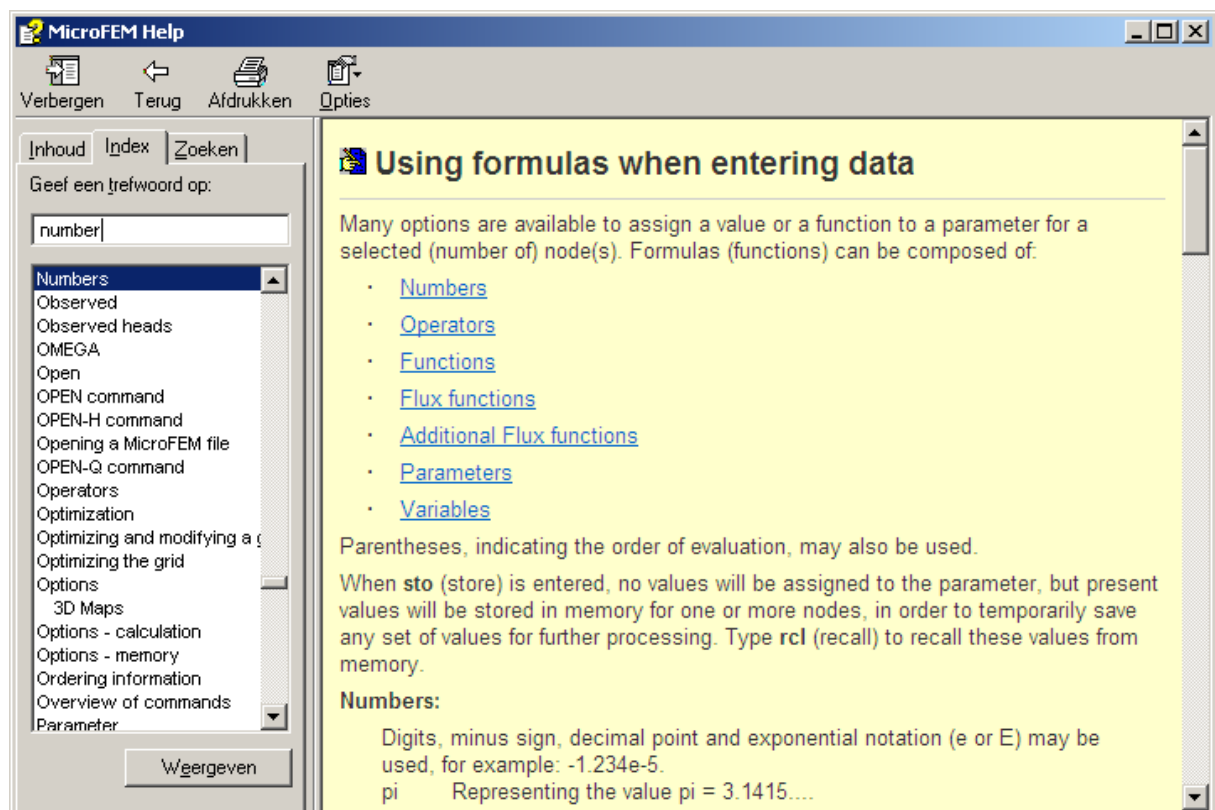
So far, we only assigned constant values in Input mode, for all or for a selection of nodes, e.g. 600 or 1800. However, similar to spreadsheets, it is allowed to also enter “formulas” and use brackets. Unlike Excel, you don’t start with an = .

Some examples:

When the well discharge is given as 57.5 m<sup>3</sup>/h, you can enter a discharge of:  $57.5*24$

When the hydraulic conductivity is given as 1.6E-4 and the aquifer thickness as 20 m, you can enter a transmissivity of  $1.6E-4*3600*24*20$

Open the Online Help (F1 key) and search the Index for “numbers” or “operators” to find the page “Using formulas when entering data” with all possible options. The same page can be found in Chapter: “Input of model data”, section: “Using formulas”.



Of these options many will only seldom be used. Nevertheless, expressions like:  $(1200-334)/12*1.2^3$  or  $\text{sqrt}(1000)+\text{exp}(3*\ln(1.2))$  are perfectly allowed.

In previous lessons it became clear that all parameter have codes, e.g. H0, C1, T1, H1, Q1, C2, etc. All such parameter codes may also be used in formulas. The value of a code is (or can be) different for each node.

For example, when the transmissivity of aquifer 1 is heterogeneous, varying from 200 m<sup>2</sup>/d in one area to 1800 m<sup>2</sup>/d in another, and you decide at a later stage that all these transmissivities are too low, and should be 100 m<sup>2</sup>/d higher, all you have to enter is: "T1+100" and [F5] to assign this to all nodes. Similarly, "T1\*1.1" will increase all values by 10%.

When we enter a formula like: (H0-H1)/C1\*1000 and assign that to all nodes, we compute the downward flux (mm/d) in the upper aquitard for each node of the entire model.

There are also a limited number of variables that you can use in formulas. A variable is also a code that represents a value that is (possibly) different for all nodes. Here are some examples:

- a – nodal area (1/3 of the sum of all neighbouring elements)
- d – distance to the nearest marked node
- r – distance to the current node
- v – value of to the nearest marked node
- x – X-coordinate of a node.

Formulas are not stored when you enter them in a cell. They are immediately evaluated and the result is assigned to the parameter of the appropriate nodes. Later on we will see how such formulas can be stored for repeated use (when we discuss batch files).

When we use formulas like (H0-H1)/C1\*1000 to compute results that should not be assigned to any model parameter, we need additional "working space" to store such results. This is where the "Xtra worksheet" comes in handy.

=== The Xtra worksheet

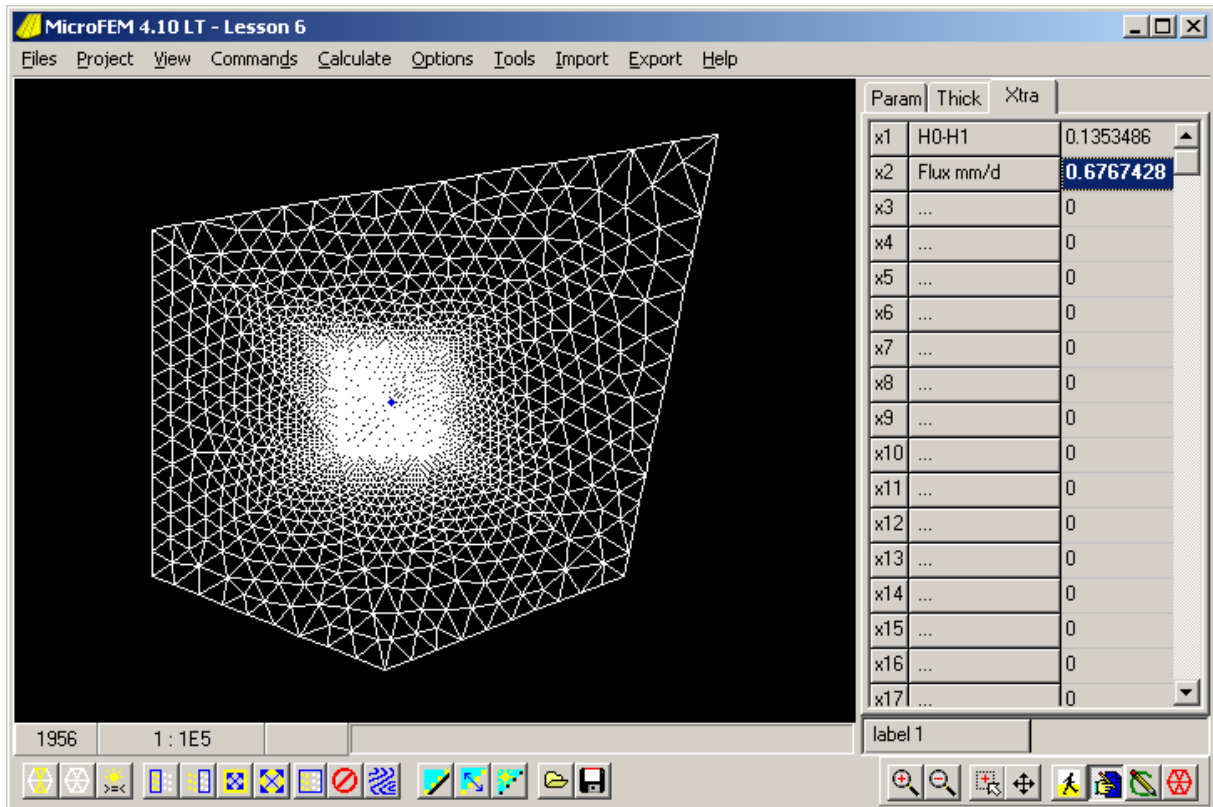
When assigning values in Input mode, the results of the formula are assigned to the parameter that corresponds with the cell where the formula is entered. For example, when we use the upper cell of the "Param" list, the resulting values are assigned to the H0 (fixed head at the top of the model) of each node. Each cell in the list has its own meaning, especially relevant when the model is run. This implies that we have a problem when we want to store temporal information that is no regular input or output data of the model. So we developed a special unit for that: the Xtra worksheet. It is sort of a spreadsheet, where each cell (row) contains data for all nodes.

@ Menu bar: Project / Project manager ... / add unit to project [green +] / Select Xtra worksheet / New / [OK] / Set the number of registers (cells, rows) to 20 / [OK] / Close project manager

A new tab is added to the Table "Xtra". Click on it to display X1, X2, ... X20. In the left part there are no texts (only three dots) and in the right part are only 0 values.

@ Right click on the three dots of X1 / Enter a new text: "H0-H1" / [OK] / Open the cell to the right / enter the formula "H0-H1" / [F5]

@ Right click on the three dots of X2 / Enter text "Flux mm/d" / [OK] / Open the cell to the right / X1/C1\*1000 / [F5]



This assignment will fail when the C1 value of one or more nodes is zero.

The parameter codes of the Xtra worksheet are: X1, X2, ..., etc.

The Xtra worksheet will be saved with the model, but its contents are not used for model computations. It is only used to store additional information. The list can be as long as 99 cells (registers).

The Xtra worksheet can be useful in many situations. For example, you can store the computed heads of different scenarios and compare results afterwards. To copy H1 into X8, all you have to do is open the cell X8, enter H1 and click the [F5] to copy the H1 value for all nodes.

The X-parameters can be contoured in Drawing mode and displayed in the 3D viewer.

## ===== MicroFEM Lesson 8A: FemGrid grid generation

In this lesson we will discuss the FemGrid generator and then build a simple unconfined model.

Originally MicroFEM was developed for regional models. Such models often have irregular shapes, while spacing contrast (ratio of largest to smallest elements) is not extremely high. In those days MicroFEM came with a single grid generator: FemGrid. Later on, when MicroFEM was also used for civil engineering models (construction dewatering, dumping-grounds, groundwater remediation) that often need very small elements locally (sheetpile walls), FemMesh was developed as an alternative generator.

We will start by inspecting the example grid

@Files / New grid / Create new grid / FemGrid / [OK] / “Lesson 8” / [OK] / [Example input (F4)]

There are three tabs in the Tables: Fixed nodes, Segments and Regions.  
The tables contain the input for the FemGrid example grid.  
Check that the number of Fixed nodes = 8, Segments = 6 and Regions = 3

@ [Fixed nodes (F8)] / [Nodes on segments (F8)]

There are 51 nodes in the map, in the shape of the MicroFEM logo.  
Note: You can press (Esc) anytime you want to start again.

Click a cell in the Fixed nodes table and then use up- & down-arrow keys, while watching the map.  
Do the same for the Segments table, and for the Regions table.

The Segments and Regions tables are wider than displayed on the screen: drag the bar between the Table and the Map to the left until all numbers in the tables are displayed.

Each Fixed node, Segment and Region has its own number.  
The order of the Fixed nodes in the table is irrelevant. This also applies to Segments and Regions.

Each Region is defined by the numbers of its bordering segments.  
The columns of the Region table show:  
1- The required spacing of nodes within the region  
2- The number of bordering segments  
3- The numbers of those segments.

Similarly, each Segment is defined by the numbers of connected Fixed nodes.  
The columns of the Segments table show:  
1- The required spacing of nodes on that Segment  
2- The number of connected Fixed nodes  
3- The numbers of those Fixed nodes.

Each Fixed node is defined by its (x, y) coordinates.

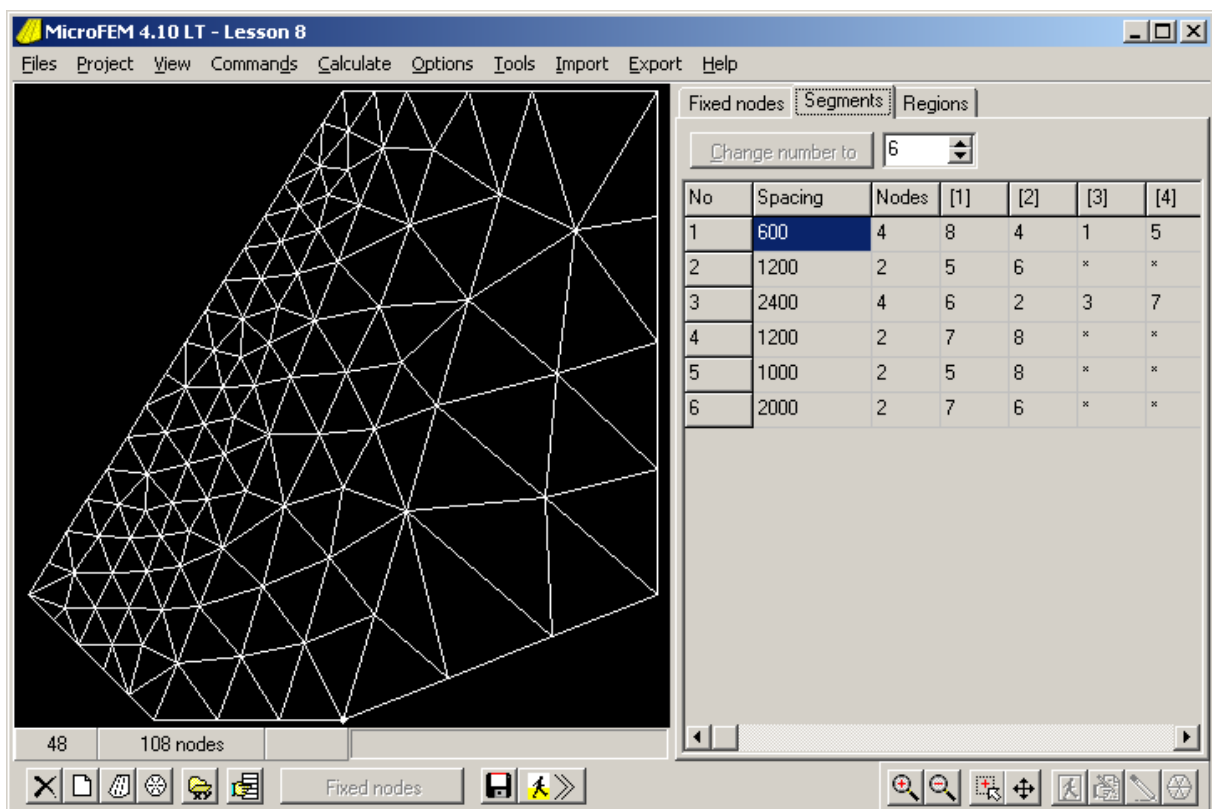
There is one IMPORTANT rule when defining your segments: A segment can be no longer than the boundary between TWO regions, or the outer boundary of ONE region.

Each region and each segment can have its own spacing, but it is recommended to use a spacing ratio of no more than TWO for adjacent regions. Also, choose a segment spacing between the spacings of the adjacent regions.

Check that these rules and recommendations apply to the example grid.  
All other steps to generate the grid are the same as for FemMesh.

@ [F8] / [F8] / [F8] / [F8]

The grid consists of 108 nodes and 178 elements.



@ [Clear input data (F3)] / No

@ [Start with a hexagon (F5)] / [OK] / [F8] / [F8] / [F8] / [F8] / [F8] / [F8]

Now there is only one Region, bounded by only one Segment

The Segment is a closed line. It connects 6 nodes, but to define this segment we have to enter 7 numbers, because the first and last Fixed node is the same (Fixed node 2).

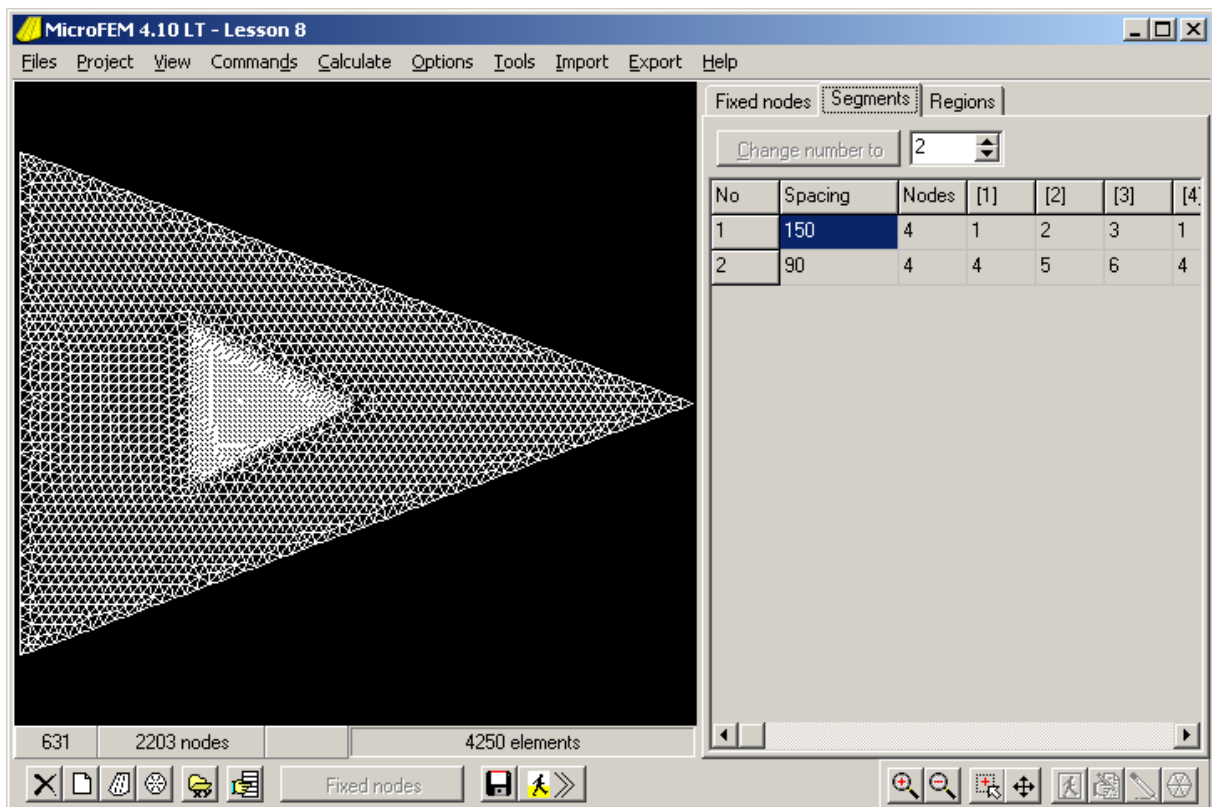
There are 7 Fixed nodes. Only 6 are used to define the segment. Node 4 is a “free Fixed node”.

There can be as many free Fixed nodes as you like.



We will now create the grid for our new model. It is a triangular island bounded by the following coordinates: (0, 0), (8000, 3000) and (0, 6000). Somewhere near the centre of the island is a well. Around the well we want a finer spacing. The finer spaced region is also a triangle: (2000, 2000), (4000, 3000) and (2000, 4000). The well is located at (2620, 3025).

- @ Enter the coordinates for all (7) Fixed nodes
- @ There are 2 segments: 1 = outer triangle, 2 = inner triangle
- @ There are 2 regions: 1 = bounded by the two triangles, 2 = within the inner triangle
- @ Spacing for inner triangle = 60 m and outer area = 120 m
- @ Choose your own spacing for the segments
- @ Create the grid.



===== MicroFEM Lesson 8B: Water table aquifer

When the spacing of the segments is chosen as 150 and 90, while keeping the region spacings at 60 and 120, there will be 2203 nodes in the grid.

The lake level is at 20 above reference level. The horizontal impervious base of the unconfined aquifer is 10 m below the lake level. We assume a hydraulic conductivity of 25 m/d. Average recharge by net precipitation is 0.001 m/d. The well is not yet producing.

@ [Save data file (F9)] / Save FEN-file / [Create aquifers (F10)] / accept 1 / [OK]

@ Walking mode / [F7]

@ Input mode / H1 / [F9] / 20 / [F5]

We don't know yet the saturated thickness of the aquifer. When the water table would be at lake level, thickness is 10 m and transmissivity is 250 m<sup>2</sup>/d.

@ Input mode / T1 / 250 / [F5]

There is no aquitard at the top of the aquifer, so there is no "head-dependent recharge" at the top: we keep the vertical resistance C1 at 0 (infinite resistance: see Lesson 5A, step 4).

The effective precipitation (precipitation minus evaporation) causes a constant recharge of 0.001 m/d at the top of the model.

@ Menu bar: Project / Project manager ... / add unit to project [green +] / Select Precipitation / New / [OK] / Close project manager

A new tab is added to the Table "Top". Click on it. The parameter code for Precipitation is PPN.

@ Input mode / PPN / 0.001 / [F5]

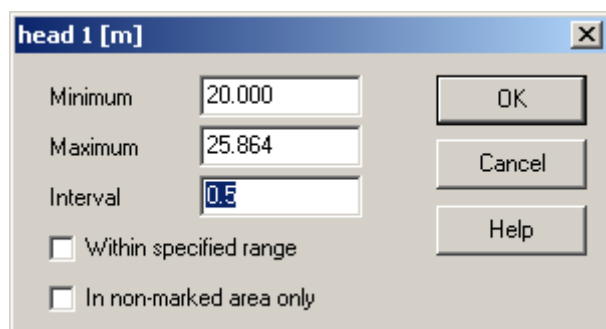
@ Menu bar: Files / Save all

@ Run the model

@ Check the model water balance.

When the water balance shows an error, adjust the stopping criterion and rerun the model (Lesson 4B).

@ Drawing mode / H1 / [F7] / Write the highest head on a piece of paper / [OK]



The displayed head shows the level of the groundwater table. The highest point is computed at 25.864 m, almost 6 m above lake level.

That implies that the saturated thickness is almost 16 m in the centre part of the island.

The transmissivity must be much larger than the assumed 250 m<sup>2</sup>/d.

The saturated thickness can be computed as:  $H1-10$

And the transmissivity as:  $(H1-10)*25$

@ Input mode / T1 /  $(H1-10)*25$  / [F5]

@ Run the model

@ Draw the contours of the water table.

The increased transmissivity makes that the highest water table level is now somewhat lower: 24.614 m.

When we repeat the above three steps (enter a new T1, run and draw contours) we find 24.752 m.

Repeated calculations show: 24.740 m, 24.741 m and again 24,741 m.

Our iterative procedure apparently requires 5 or 6 steps to find a solution.

Next lesson we will learn a more automatic way to account for head-dependent transmissivities.

We will now add the thickness unit (as in Lesson 6B)

@ Menu bar: Project / Project manager ... / add unit to project [green +] / New / [OK] / Close project manager

The “top level” is the water table height above reference level. This is different for all nodes and computed by our model as H1. The aquitard thickness is zero (not present).

The aquifer thickness is also different for all nodes, dependent on H1, and equal to  $(H1-10)$ .

@ [Input mode]

@ Table: Thick tab / top level / H1 / [F5] / aquifer / H1-10 / [F5]

Show the water table in a West-East profile

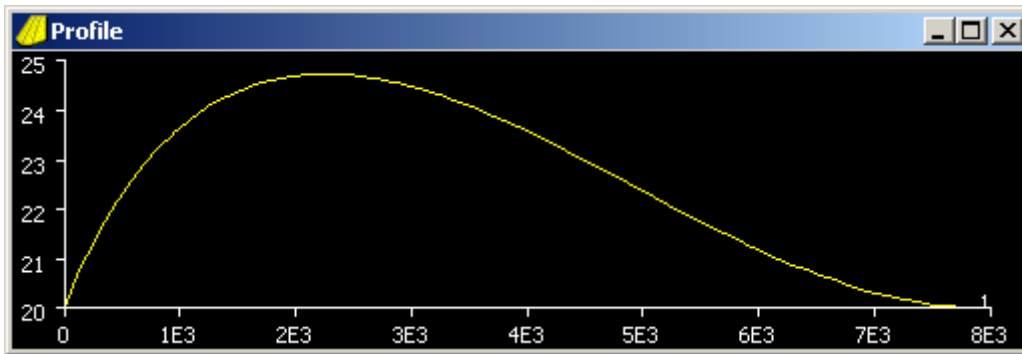
@ Drawing mode / [F10] / [F11]

The vertical scale ranges between 0 and 30, while the water table is at and above 20 m.

This is because not only H1 is displayed in the profile, but also H0, while all H0 values are still zero.

The H0 parameter plays no role in this model, since we have unconfined conditions (no top aquitard).

@ Left click in the Profile / Uncheck the H0 box / [OK]

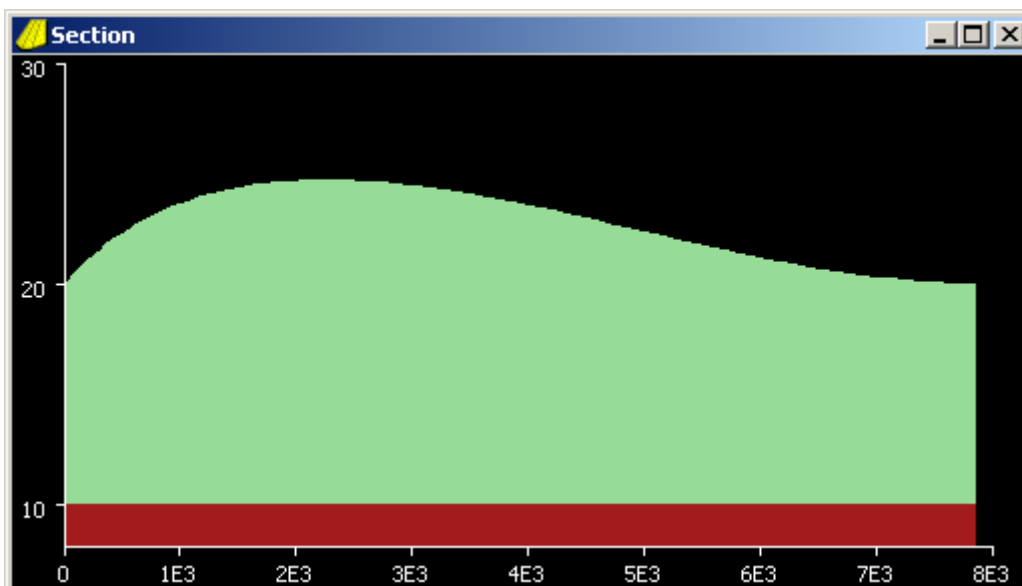


Now show the saturated aquifer in a West-East profile

@ Drawing mode / [F12]

The vertical scale ranges from 10 to 30 m.  
When we also want to see the impervious base

@ Left click in the Section / Scale / Uncheck Vertical automatic scale / Min = 8 / [OK]



@ Save your model.

We will use the same model in Lesson 9 to draw 3D flowlines and introduce the batch-file editor. In Lesson 10 we will continue with the Batch-file editor and start with transient modelling.

===== MicroFEM Lesson 9A: 3D-flowlines

@ Load the model of lesson 8 (click Lesson 8.fpr)

We will show how to draw 3D flowlines.

Before starting with this, we need:

- a - a steady-state model with its heads computed
- b - a thickness unit with the thicknesses of all aquifers and aquitards
- c - for each flowline a starting point in 3D space.

A starting point requires the x-, y- and z-coordinates of a point somewhere in the model.

MicroFEM uses the x- and y-coordinate of the current node.

The z-coordinate is given in a different way.

The flowline starts in the aquifer indicated by the active cell in the Param Table.

There is only one aquifer in our model. We select H1 (T1 or Q1 have the same effect).

The z-coordinate also depends on the depth within that aquifer.

This has to be specified as a “dimensionless depth”. A dimensionless depth of 0 means at the upper boundary of that aquifer, 1 means at the base of that aquifer and 0.5 means right in the middle.

A dimensionless depth can be any value on the interval [0 ... 1].

@ Drawing mode / [Del] / [F2] / Click at a node near (2000, 3000)

@ Select H1 in the Table / [Draw flowlines (F9)]

A window appears “Drawing 3-dimensional flowlines”.

Possible settings are:

- Direction: Forward or Backward. Forward is with the flow. Select “Forward”.
- Save file: Flowline file is the default. The default filename is given near the base: “Lesson 8.#01”
- Time step (y) = 10. There will be dots on the flowline indicating travel time steps of 10 years.
- Levels = 0.5. A level on the interval [0 ... 1] is the dimensionless depth of the starting point. Set level = 0.
- Porosity = 0.3. To compute travel times an (average) value for the porosity is required.
- Max time (y) = 0. A maximum travel time can be specified, but here 0 (zero) means: “no time limit”.

By selecting Forward and Level = 0 we indicate that we start at the water table and go with the flow.

(An alternative way to start at the water table is to select H0; the Level is not required then).

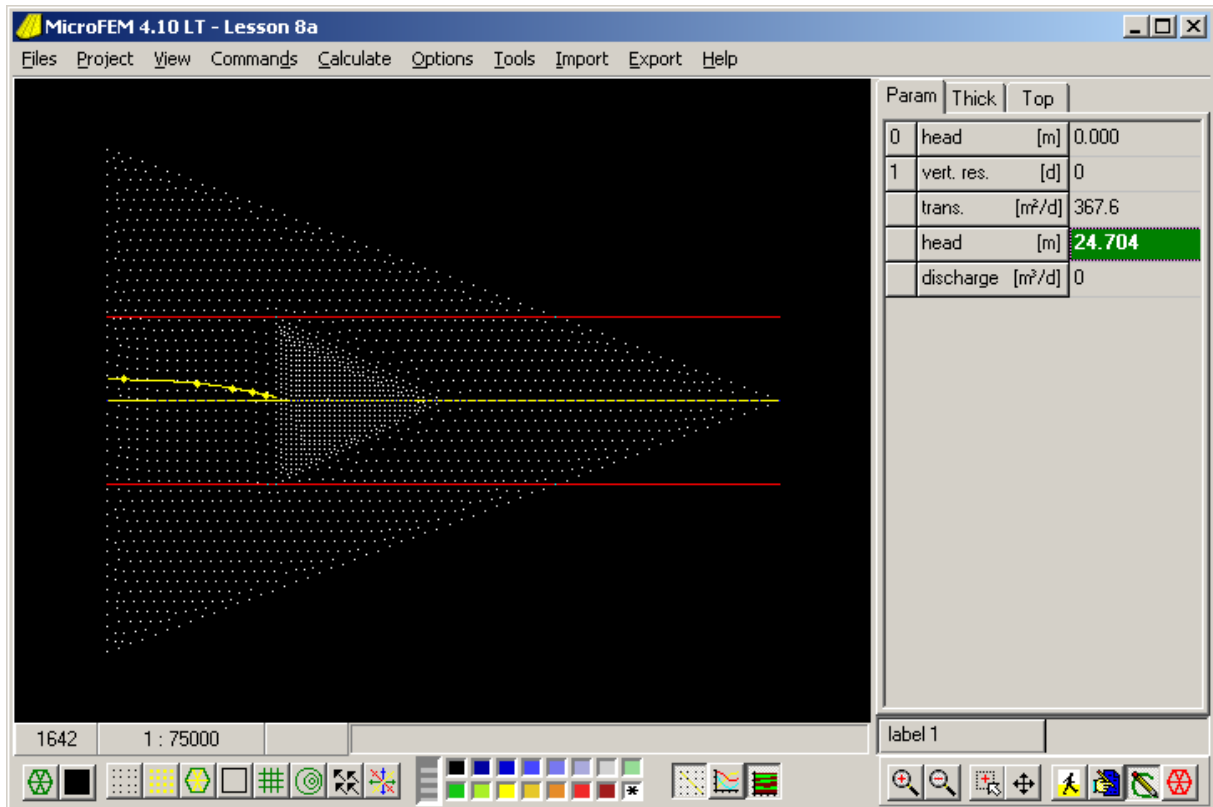
@ Check that “Forward” is selected and that Levels = 0 / [OK]

The 3D flowlines window disappears. A yellow flowline is drawn in the map.

When counting the dots, the travel time must be between 50 and 60 years.

Zooming is allowed.

@ [F10] / [F12]



In the map the yellow Profile line is bordered by two parallel red lines.

Any 3D flowline between the red lines will be projected on the vertical 2D plane of the Section.

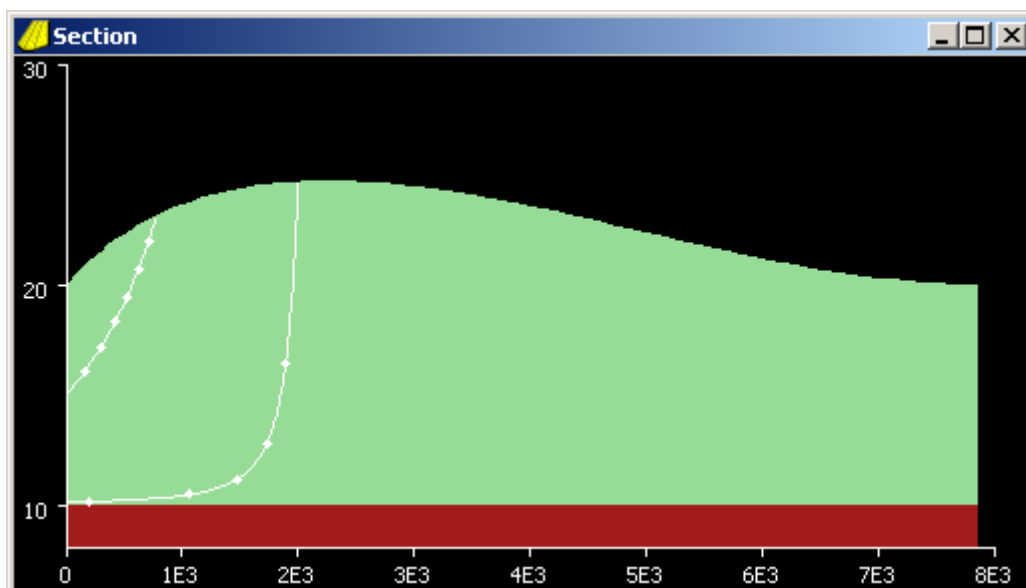
In the left (western) part of the Section the same flowline with time steps is displayed in white.

Zooming in the Section is also allowed (start with right-click).

@ [F10] / [F12] / Click a node at the model boundary, just north of the flowline / H1 / [F9]

@ Backward / Time step = 1 / Level = 0.5 / [OK]

@ [F10] / [F12]



The computation of the new flowline starts at a depth of 5 m. The line is computed Upstream. It represents a rain drop that infiltrated some 800 m east of the boundary. Travel time 6 or 7 years.

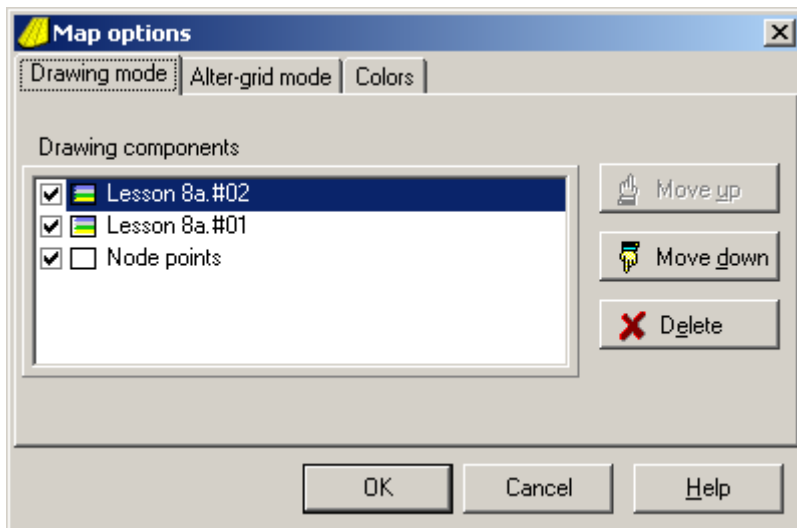
Here are some further details about 3D flowlines:

One or more flowlines:

- Usually there is only one flowline (forward and backward) through one point in 3D space. Wells are an exception. When you start in a well, you can specify any number of flowlines (Lesson 10)
- Flowlines will start in the current node and ALSO in all nodes with the same label.

Flowline files:

- Information of each flowline is saved in a #nn-file. Each new file has a follow-up number.
- You can use the [Browse ..] button to specify a different name or number
- The filenames of flowlines is added to the Drawing components list (Map options).



- The flowline files are saved. They can be read again with: Main menu / Import / Flowlines.

Colour of flowlines:

- The flowline colour in the map depends on the aquifer: Aquifer 1 = yellow, 2 = blue.
- These colours can be found and modified in: Options / Map / Colors
- The colour of a loaded flowline files depends on the Palette colour.
- The flowline colour in a Section also depends on the Palette colour.

@ Save the model.

## ===== MicroFEM Lesson 9B: Batch-file

Before we start with transient models, we have to discuss the batch-file editor. In MicroFEM these two are closely related. As soon as text is entered in the editor it is assumed that we want a transient flow solution.

@ Load the “Lesson 8” model / Calculate / Options

At this moment “Steady state” is selected and it is not possible to select “Transient”.

@ Click the tab: “Batch-file editor” and enter some nonsense “qqqqq” in the blue text window.

@ Return to the Options tab.

Because of text in the editor it is assumed that we are dealing with a Transient model.

When we calculate the model (click [Calculate]) the commands in the batch-file are processed one-by-one.

Only commands known to MicroFEM are processed; other texts are considered Comments and only copied to the black “Calculation of heads” screen.

The commands that MicroFEM understands are 14 “main commands” and several “secondary commands” (also called “specifications”) that always look like “something = something”.

The available main commands are given in a list to the right of the editor window. Clicking one of the commands in the list opens (if applicable) another list with possible secondary commands.

Double-clicking a command enters the command in the editor window.

@ Empty the editor using the Backspace-key / double-click “EVAL” / double-click “Q1” / enter “12345”

This produces the following text in the editor:

```
EVAL
  Q1=12345
```

@ [Calculate]

We have run the batch-file. The black screen repeats the commands and adds: “New values assigned to 2203 nodes”.

@ [Stop] / [OK]

As a result of our action, all nodes in the model show a discharge of 12345 m<sup>3</sup>/d. From this simple example we learn that in stead of manual assignments in Input mode, we can also assign values to parameters with the EVAL command in a batch-file.



When we want to run a steady-state model from the batch-file, we can use the main command RUN.

However, since this command is meant for transient models, MicroFEM checks whether a time period is specified with the TIME command and whether the Storativity unit is present. The trick now is that when all storativities are set to zero, running the model for any period will produce the steady-state solution.

@ Use the Project manager to add the “Storativity” unit to the model

@ Also add an “Xtra worksheet” (12 registers is OK)

Click the Stor tab in the Table and find that the storativities of all nodes are zero by default. (Try to draw contours!)

@ Edit the batch-file as follows:

```
EVAL
  Q1=0
  T1=250
  H1=20
TIME
  days=1
RUN
```

@ Run the batch-file (click [Calculate] / [Stop] / [OK])

@ Check the model water balance and draw the head contours.

This result is the same as after the first run of Lesson 8 (Maximum head = 25.864)

Since the EVAL command can also be used to assign formulas to parameters, we can extent the batch-file to include a number of iterations similar to Lesson 8.

```
EVAL
  T1=250
  H1=20
TIME days=1
RUN
EVAL
  T1=(H1-10)*25
TIME
  days=1
RUN
EVAL
  T1=(H1-10)*25
TIME
  days=1
RUN
EVAL
  T1=(H1-10)*25
TIME
  days=1
```

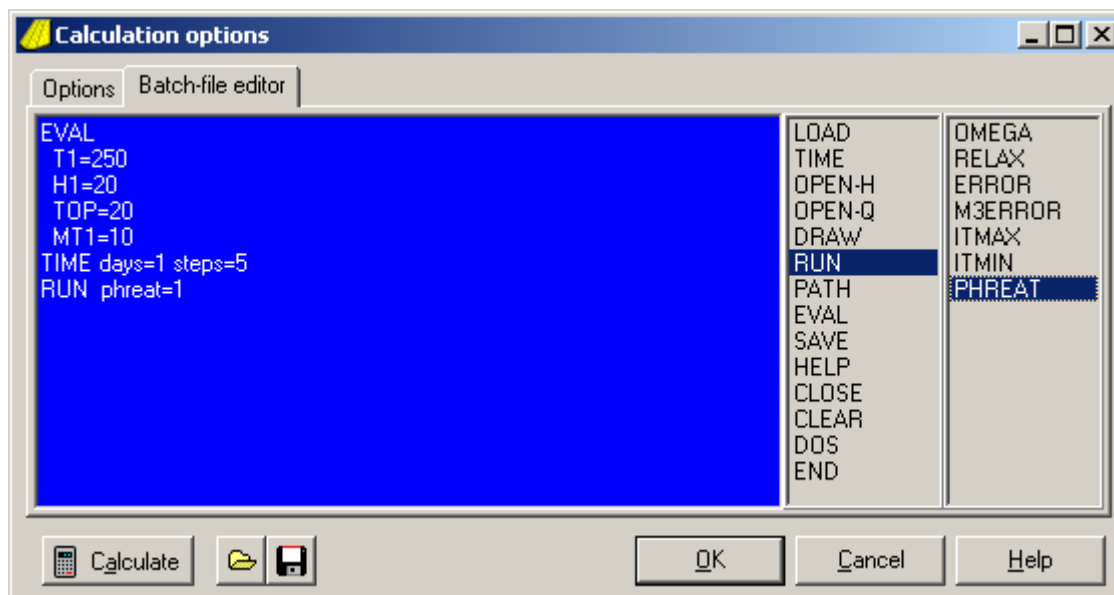
```
RUN
EVAL
  T1=(H1-10)*25
TIME
  days=1
RUN
```

```
EVAL
  TOP=H1
  MT1=H1-10
```

With the last 3 lines we assign values to the “top level” and the aquifer thickness.

Although somewhat easier than the procedure of Lesson 8, the iterative steps in the above batch-file are still rather awkward. There is a better way in MicroFEM to handle the condition that the transmissivity of an unconfined aquifer depends on the water table height. All you have to do is check the box “Phreatic top aquifer” on the Options tab (or add “phreat=1” to the RUN command) and make sure that you start with a consistent set of values for: H1, T1, TOP and MT1.

```
EVAL
  T1=250
  H1=20
  TOP=20
  MT1=10
TIME days=1 steps=5
RUN phreat=1
```



After each time step T1, MT1 and TOP are updated according to the new H1.

- @ Check results
- @ Save the model.

===== MicroFEM Lesson 10A: Water divide and Capture area

When we created the grid in Lesson 8, we added a Fixed node at the location of a well at (2620, 3025).

The well discharges 120 m<sup>3</sup>/hour somewhere in the centre of our unconfined triangular island.

@ Click “Lesson 8.fpr” to load the model we saved at the end of Lesson 9

@ Start the Project manager. Note that the model consists of 7 files, including the batch-file

@ Close the Project Manager, save the model as “Lesson 10”

@ Drawing mode / Label 1 / [F7] / [OK]

The small square in the centre is drawn around the node with Label “Fixed node 7”.

@ Zoom in and click on this node / Input mode / enter the text “well” / Enter / Q1 / 120\*24 / Enter

The batch-file editor shows the following text

EVAL

T1=250

H1=20

TOP=20

MT1=10

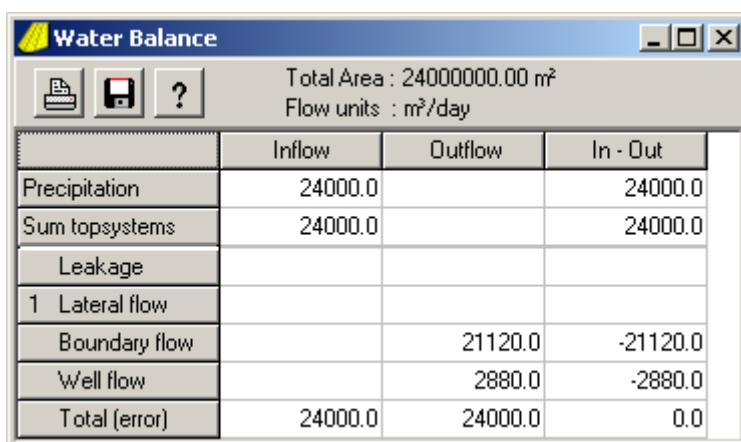
TIME days=1 steps=5

RUN

@ Check the box “Phreatic top aquifer” or (and) change the Run command: RUN phreat=1

@ Run the model ([Calculate] / [Stop] / [OK])

@ Check the water balance.



The screenshot shows a window titled "Water Balance" with a toolbar containing icons for print, save, and help. The window displays the following information:

Total Area : 24000000.00 m<sup>2</sup>  
Flow units : m<sup>3</sup>/day

	Inflow	Outflow	In - Out
Precipitation	24000.0		24000.0
Sum topsystems	24000.0		24000.0
Leakage			
1 Lateral flow			
Boundary flow		21120.0	-21120.0
Well flow		2880.0	-2880.0
Total (error)	24000.0	24000.0	0.0

Note that 88% of all precipitation (21120 m<sup>3</sup>/d) is boundary outflow.

Since the well is now labelled, we can specify any well discharge in the batch-file:

EVAL

T1=250

H1=20

Q1=120\*24 label=well

TOP=20

MT1=10

TIME days=1 steps=5

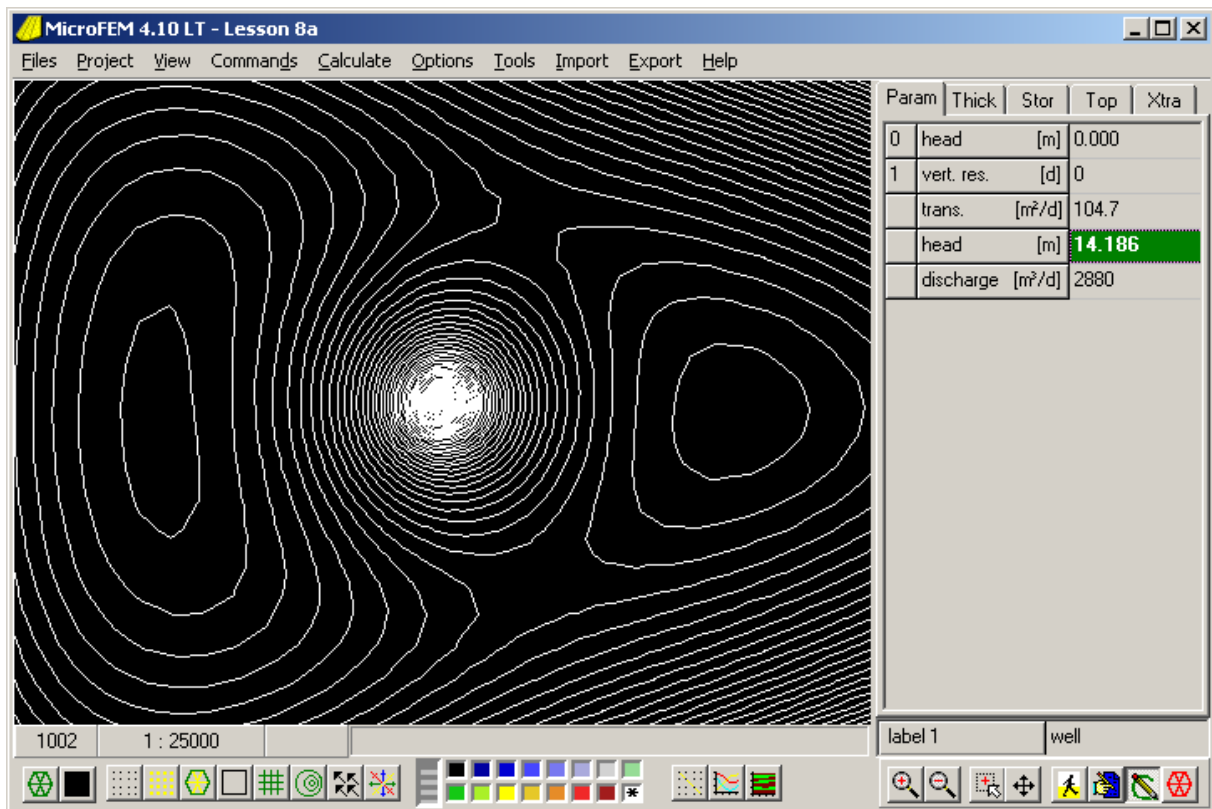
RUN phreat=1

The discharge of 120\*24 m<sup>3</sup>/d is assigned to all nodes with label “well”, which is only one node in this case.

@ Run the model

There is a well-defined capture zone where all infiltrated precipitation flows toward the well. All other precipitation in this model flows to the model boundary. To find this capture zone we can draw the contours of the water table and see the shape of the drawdown cone.

@ Right-click / Fit to scale / Drawing mode / H1 / [F7] / Choose a small interval = 0.1 / [OK]



The hydraulic head at the well is found to be 14.186 m.

W, NNE, E and SSE of the well are locations where the contour lines are relatively far apart, indicating areas of low head gradient. They are on the water divide at the boundary of the drawdown cone. The four points give an impression of the shape of the drawdown cone. A more accurate view can be obtained with 3D flowlines.

@ Walking mode / [F12] / Next label / select “well” / [Jump] / [Close]

Zoom somewhat in on the well (+ + +)

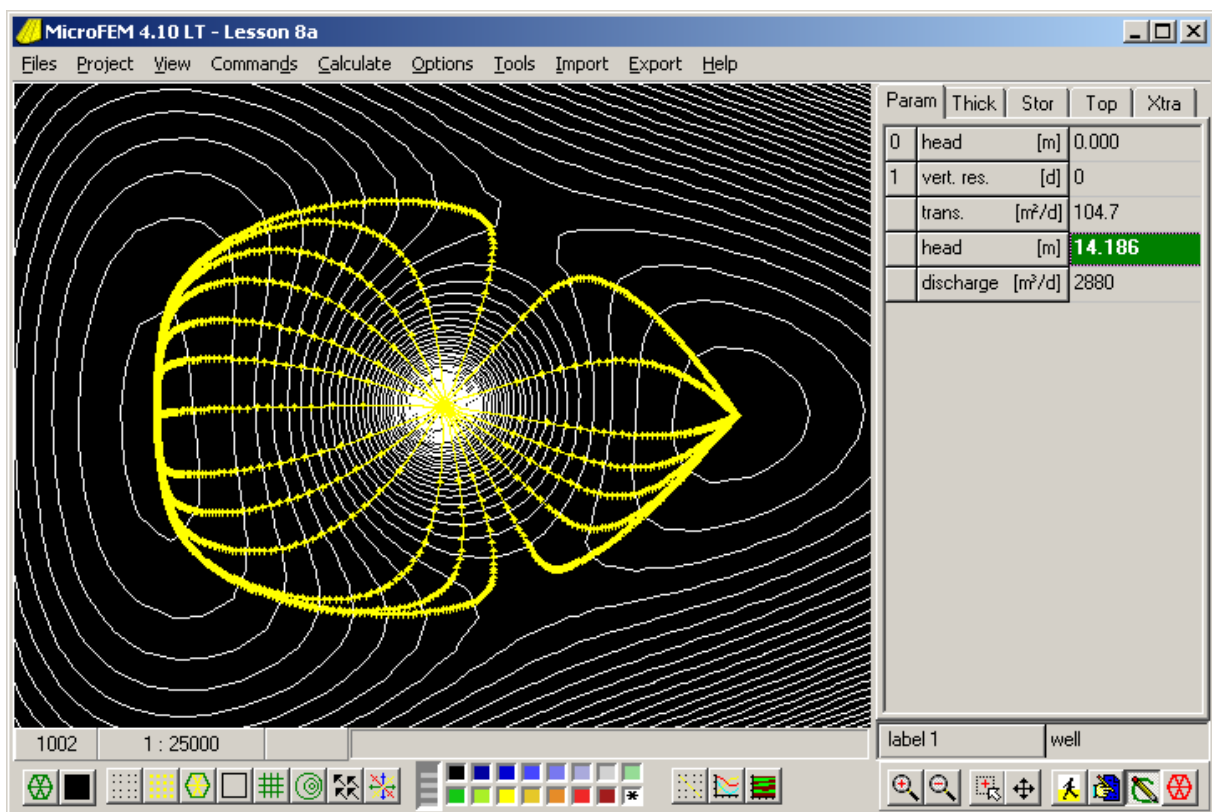
@ Drawing mode / [F9] / Backward / n lines = 20 / [OK]

When you cannot specify the number of lines, you are not in the well node,

The yellow flowlines do not reach up to the groundwater divide.

@ [F9] / Backward / n lines = 20 / levels = 1 / [OK]

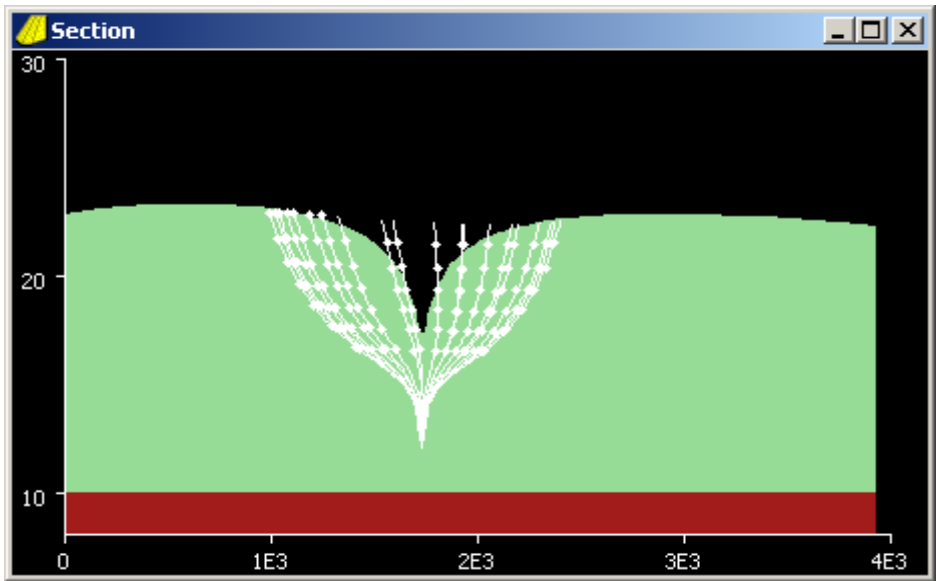
This second set of 20 flowlines starts at the base of the well screen and flowlines are computed backwards until an area with no gradient is reached, the 2 highest points of the water divide. The shape of the capture area becomes clear.



When you have doubts about the open parts in the NNE and SSE direction, increase the number of lines (e.g. 200) and try again.

When you draw a section ([F10] / [F12]) you will only see the first set of 20 flowlines starting at a height of about 12 m. When levels was set at 1, flowlines start at the base of the fully penetrating well, which in this case is also the impervious base of the aquifer. Since the vertical flow component is zero here, these flowlines will remain at the base. To display these flowlines set the Minimum of the vertical scale lower than 10 m (click in the Section / Scale / Min = 9 / [OK]). For a better view of the capture zone in the Section, try levels = 0.999.

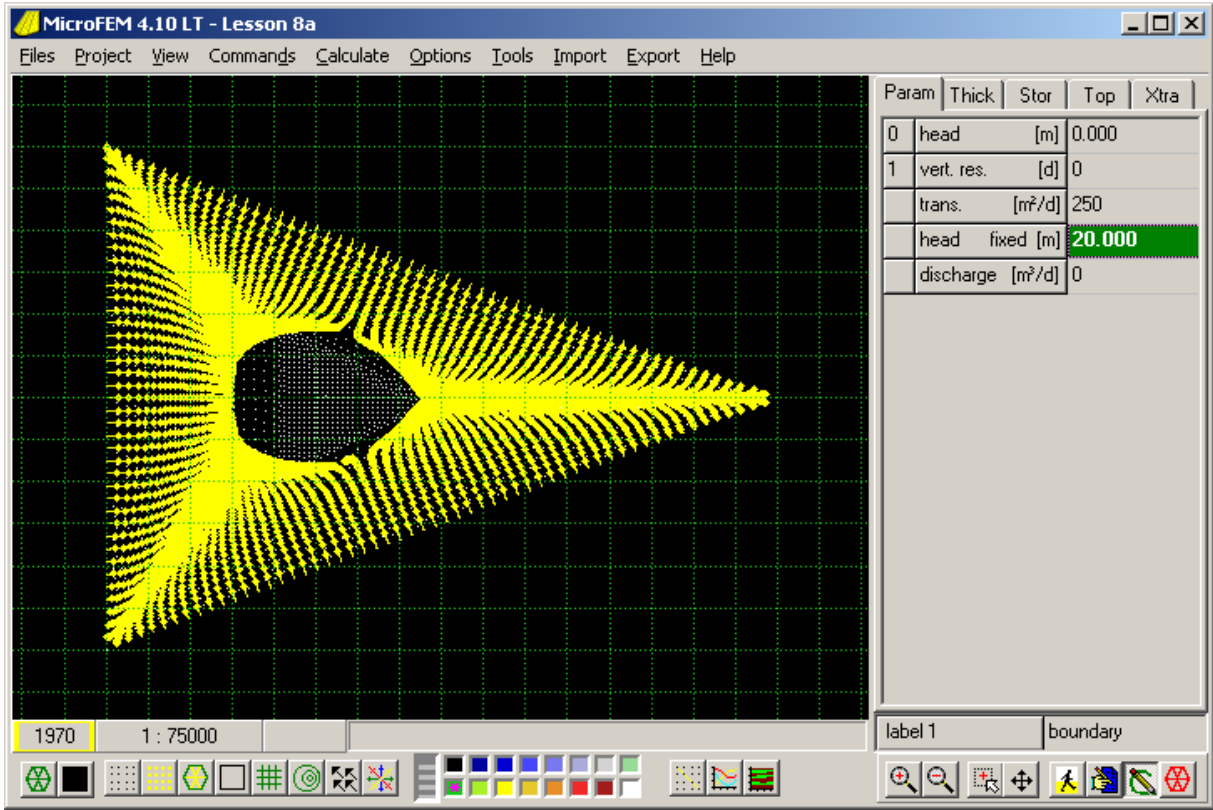
In the section right through the drawdown cone, several flowlines starting at Level = 0.5 appear above the water table. Can you explain that?



We can also approach the water divide around the capture zone from the outside. First label all boundary nodes.

- @ Walking mode / [Del] / [F7]
- @ Input mode / Label 1 / "boundary" / [F3]
- @ Drawing mode / [Del] / [F2] / H1 / Click a boundary node / [F9] / Levels = 0.999 / [OK]

Draw a 500 m map grid [F6] to estimate the capture zone area.



Make your own water balance of the capture zone and find that its surface area must be 2.88 km<sup>2</sup> (24\*120 m<sup>3</sup>/d / 0.001 m/d).

===== MicroFEM Lesson 10B: Transient modelling

When head changes with time (as in all natural situations) we are dealing with transient (time dependent, unsteady, nonsteady state) flow. We can simulate such flow for a given period. It requires an additional aquifer parameter: (elastic) storativity (for confined aquifers) or specific yield (unconfined aquifers).

The total simulation period must be subdivided into a number of (one or more) stress periods. Within a stress period the boundary conditions do not change. A stress period is subdivided into a number of time steps. For each time step the heads are computed for all nodes in all aquifers. The results of a transient model also depend on the initial condition, the heads at the start of the first stress period. We usually start with a steady-state model.

EVAL

T1=250

H1=20

Q1=0 no well discharge

S1 = 0 storativity (specific yield) is zero

PPN = 0.001 precipitation is 1 mm/d

TOP=20

MT1=10

TIME days=1 steps=5

RUN phreat=1

When you run this we are back to our original model with precipitation and no well.

Maximum height of water table = 24.741

This will be our initial condition. Save the model.

When running a transient model, MicroFEM doesn't save all computed heads and fluxes for each time step, but only the data that you ask for. When we are only interested in the simulated transient heads at the nodes that represent observation wells around the pumping well, then we have to assign labels to those nodes. The smart way is to use labels that have some text in common, such as: obs.1, obs.2, obs.a, and obs.b3, or we can use whatever names we like and end for example with the number sign # (e.g. observ# , old\_well# and forest-lake#).

@ Select six nodes close to and further from the well and assign a different label to each of them, all ending with the # sign.

Add the following text to the batch-file

EVAL

S1= 0.1

Q1=120\*24 label=well

OPEN-H #=fth

TIME days=50 steps=50

RUN

The OPEN-H command opens a file “Lesson 10.fth” and writes for the start (t=0) and at the end of each time step a line with the time (in days) and the computed heads for each node that has a label with a #-sign.

@Run the model.

When ready the total running time is displayed.

@Close the blue and black windows.

@Menu bar: Tools / Time curves ... / [Open file] (second button upper left) / Click FTH file

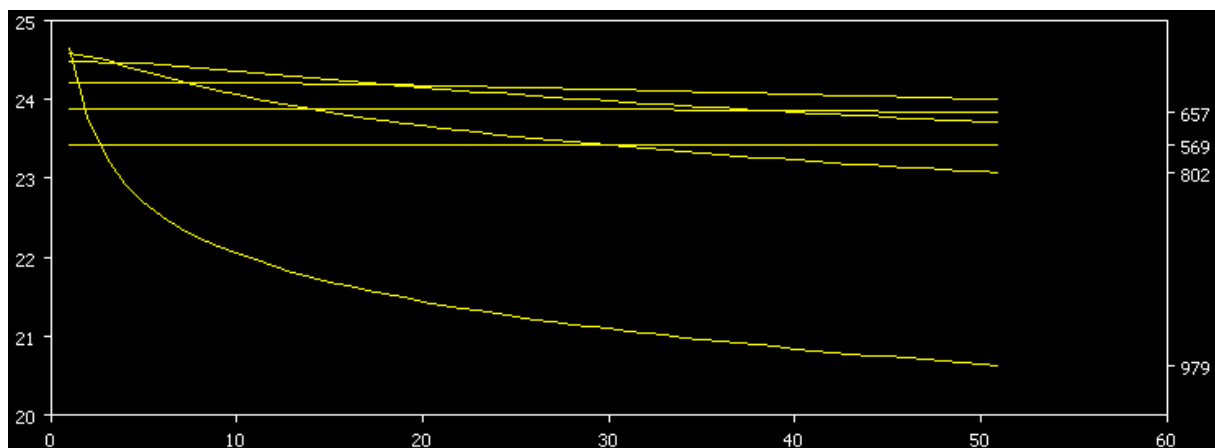
The Selection Tab can be used to make a section of nodes (observation wells) and times

The Option Tab can be used for a few settings of the Time graph

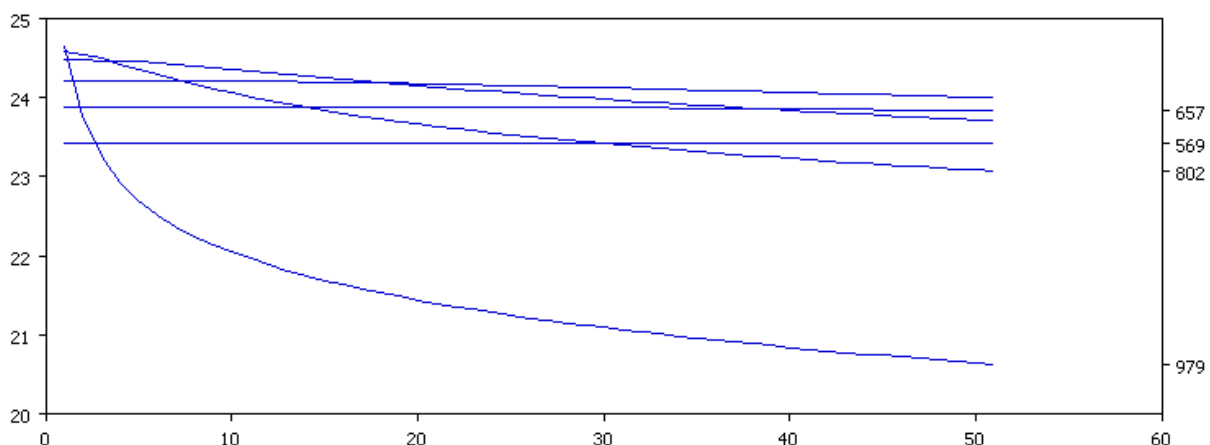
The Time-Head curve displays the resulting time-graph.

@ Menu bar: Export / Copy to clipboard / Time-curve

A copy of the time-graph can now be pasted into Word or other windows software for presentation purposes.



Although a black background, white axes and yellow curves work fine on the screen, a white background, black axes and darker curves are better for presentations. The same applies to maps, profiles and sections.





When it is not the well discharge but the recharge by precipitation that changes in time, then the batch-file could look like:

```
EVAL
  S1= 0.1
  PPN=0.003
OPEN-H #=fth
TIME days=185 steps=30
RUN
EVAL
  PPN=-0.001
TIME days=180 steps=30
RUN
EVAL
  PPN=0.003
TIME days=185 steps=30
RUN
```

At the end or in between stress periods you can also save files or copy results, e.g.:

```
SAVE fem="lesson 10a"
```

to save the fem-file: lesson 10.fem

```
EVAL x4=h1
```

to copy the computed heads (water table heights) to X4 of the Xtra worksheet.

Another interesting command is DRAW. When you start the batch-file with DRAW and draw a contour map before you run the model, the map will be updated each time step. You have to shift the blue and black screen to the right for a better view of the map. Any section (or profile) on the screen will also be updated.

This lesson is only a simple introduction to the batch-file commands.

See the online Help texts and the microfem site [www.microfem.nl](http://www.microfem.nl) for further information.

---

---

Above Lessons were first published in the LinkedIn group:  
'MicroFEM groundwater flow modeling'

---

---