About ✕

MicroFEM
Version 4.10.77

© Copyright 1997..2020 by C.J. Hemker and R.G. de Boer

OK   Help

MicroFEM for Windows

This text is a short guide for the WIN-version of MicroFEM. New users of MicroFEM will get a first impression, but many details are only explained in the MicroFEM Help text boxes.

The WIN-version of MicroFEM uses a standard Windows type of GUI (graphical user interface), with its mouse, menus, hot keys, dialog boxes, file handling, help system, etc.

Summary of the most important features of MicroFEM, as discussed in this guide:

1) Overview
2) Projects and Files
3) Grid Generation (FemGrid & FemMesh)
4) Marking Nodes (Walking Mode 1)
   + Computing Water Balances
5) Data Entry (Input Mode 2)
6) Presentation of Results (Drawing Mode 3)
   + Flow Vectors and Flowlines
   + Profiles and Sections.
7) Grid Manipulation (Alter-grid Mode 4)
8) Top systems
9) Steady-State and Transient Calculations
10) Time-Head Curves
11) Import and Export data
12) Water Table Calculations
13) The 3D viewer
14) Model optimization
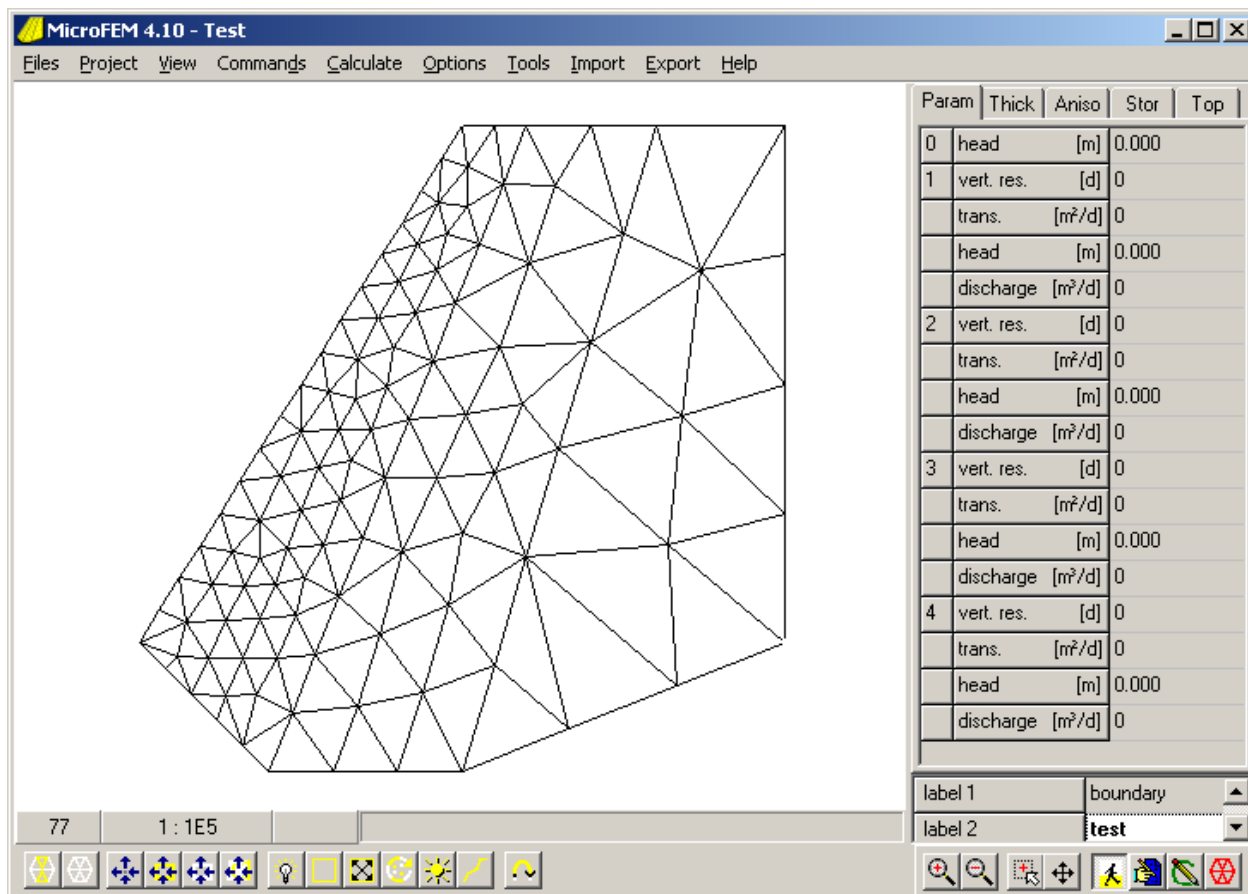
Example of a MicroFEM screen display, showing the default grid on a white background, a four-aquifer model with thickness, anisotropy, storativity and topsystem data and with two labels for each node.

1)   Overview

a)   MicroFEM has all the general advantages of a Window application, such as multitasking and high resolution video. At this stage MicroFEM performs best when using a screen resolution of 800 by 600 (or higher) and small characters.

b)   Layout and components of the screen

- Title bar with MicroFEM version number and model name
- Menu bar: Files, Project, View, etc.
- Graphical window (showing grids and other maps)
- One or more parameter lists and labels
- Status bar
- Toolbar

c)   Menu bar:

- Files   Starting new projects/grids
                Opening files: FEN, FEM, FEB or FPR
                Saving results
                Exiting MicroFEM
- Project   Listing of all current datafiles and batchfile that comprise the project
                Adding or removing files
                Renaming and saving files
                Project information (user defined)
- View   Selecting mode 1, 2, 3 or 4 (Walking, Input, Drawing, Alter-grid)
- Commands List of commands for the current mode
- Calculate   Setting calculation options
                Running steady-state and transient models
                Creating/editing/saving a batch file
- Options   Map = list of current drawing objects
                Profile = graph of heads and water table
                Section = cross section showing aquifers and aquitards
                Colors, Units, etc.
- Tools   Drawing time-head and distance-head curves
                Parameter optimization (automated calibration)
                Three-dimensional model viewer
- Import   Opening DXF-files, flowline files and reading xyz-data files
- Export   Saving graphical window maps and drawings to the clipboard
                Saving special ASCII-files
- Help   MicroFEM Help
                MicroFEM copyright and version number

d)   Graphical window (map of the model):

The vertical separation line between the map window and the parameter lists can be shifted. Background/foreground colors can be selected in Drawing mode.
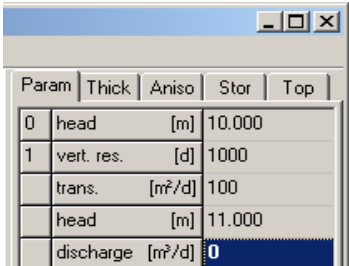
e)   Parameter lists:

All parameters can be shown in one of six parameter lists. The standard list contains heads, vertical resistances, transmissivities and discharges. Other lists are optional: layer thickness, anisotropy, storativities, various top-system parameters and a worksheet for additional data.

Param	Thick	Aniso	Stor	Top
0	head	[m]	10.000	
1	vert. res.	[d]	1000	
	trans.	[m²/d]	100	
	head	[m]	11.000	
	discharge	[m²/d]	0	

The current (active) parameter cell is colored white, blue, green or red, depending on the current mode 1, 2, 3 or 4. A different current parameter can be chosen by left-clicking in the parameter list. The value can be changed in Input mode by double-clicking, typing the new value (or equation) and assigning this value to one or more nodes.

All parameters in each list (except top system) are saved together in a unit file – base model standard parameters (.fem), thickness (.thi), anisotropy (.ani), storativity (.sto) and worksheet (.xtr).

All parameters have their own name (code), which can be used when assigning values as a function of other parameters in Input mode or when loading parameter files during batch processing. The same codes are used as defaults for filename extensions, when parameter files are saved in mode 2.

Parameter list	Parameters	Codes
- Param (.fem)	standard parameters:	h0; c1, t1, h1, q1, c2, t2, h2, ...etc.
- Thick (.thi)	top reference elevation of the uppermost aquitard or water table:	top
	thickness of aquitards and aquifers:	mc1, mt1, mc2, ...etc.
- Stor (.sto)	storativities:	s1, s2, ...etc.
- Aniso (.ani)	principal horizontal anisotropy direction and factor:	ad1, af1, ad2, af2, ...etc.
- Top	topsystem parameters:	
	- precipitation	ppn
	- drains	dh1, dc1, ...etc. (max 3)
	- rivers	rh1, rc1, ri1, ...etc. (max 2)
	- wadi	wh1, wl1, wc1
	- evaporation	el1, eh1, em1
- Xtra (.xtr)	extra worksheet:	x1, x2, x3, ...etc.

| 59 | 1 : 75000 | 105 | x = 5087.43 y = 9357.68 |

f) The status bar shows information about the current (blinking) node including its number, and if marked, a yellow (or the selected marking color) line around the node number box; the drawing scale; and information about the node closest to the mouse cursor including its number, marking, and coordinates.

g) The toolbar is a selection of buttons, dependent on the current mode. The eight buttons at the right end of the bar remain the same for each mode: the first four buttons control zooming and panning and the other four buttons change the mode.

Walking mode 1, Input mode 2, Drawing mode 3 and Alter-grid mode 4 toolbars:

4

h)   Operations available in the four modes:

Mode 1 = Walking mode (white)
-- drawing the grid and deleting the marks (selected nodes)
-- calculating water balances
-- marking (= selecting) a part of the model
-- jumping to a particular node or label

Mode 2 = Input mode (blue)
-- drawing the grid and deleting the marks
-- assigning parameter values to the nodes
-- setting boundary conditions (fixed head or no-flow boundary)
-- loading and saving parameter files

Mode 3 = Drawing mode (green)
-- drawing the grid and clearing the map screen
-- drawing nodes, boundary and map grid
-- drawing contours
-- drawing flow vectors or circles
-- drawing (2D or 3D) flow lines
-- selecting drawing color
-- drawing (head) profiles and cross sections

Mode 4 = Alter-grid mode (red)
-- modifying the finite-element grid
-- increasing or decreasing the number of model layers or worksheet layers

i)   One of the most obvious features of the MicroFEM interface is the use of the mouse to move about the grid, to select the current (blinking) node, and to mark lines between nodes. The mouse can be used to select parameters in the parameter lists or to access commands (menu bar, toolbar) or to perform many other GUI interactions. You do not have to use a mouse. Practically all the operations have keyboard equivalents.

j)   Right-clicking is only required in a limited number of occasions:
    -   To unmark (de-select) marked nodes if the Automarker is active (Walking mode)
    -   To open the file/edit menu during data entry (Input mode)
    -   To select the right end node of the yellow profile line (Drawing mode).
    -   To open the file menu to save a log-file (= file created of the on-screen window when a batch file is processed during transient calculations)
    -   To enter or change the name of a worksheet register or label (Input mode)

Right-clicking can also be useful in other occasions:
    -    To open a zoom menu (All modes, Profiles and Sections)
    -   To change the background color by clicking the color-palette (Drawing mode)
    -   To open the file/edit menu in the batch-file editor

k)   All active on-screen windows (e.g. Profiles, Sections, Water balances, Legend, etc.) can be copied to the clipboard by pressing Alt + Print screen. Maps and drawings in the graphical window can be copied to the clipboard with Export / Copy to clipboard.

2) Projects and Files

a) All MicroFEM files are ASCII-files and can be edited with any text editor, including MicroFEM's own batch-file editor.

b) MicroFEM project files (with the extension FPR) contain a list of all the files (and their path locations) that makes up a model. A model (or project) is a set of files that includes at least a Base model FEM-file and often a number of other units (or parameter files), such as, thickness-files, storativity-files, anisotropy-files, or various topsystem files. Transient calculation batch-files are also listed in the project file.
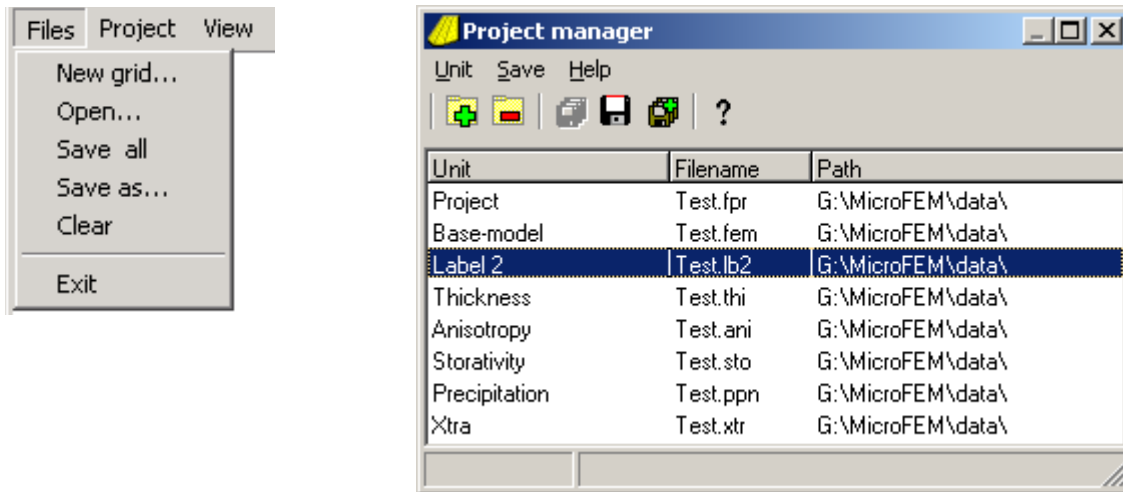
c) To save or change the name of any file in the Project manager, select the file and use the Save file as… button. If a unit/parameter file name is changed the FPR-file must be resaved. To save all project files at a new location or with a new name, use the Save all files as .. button.

d) To save the current set of unit/parameter-files under their current names, use the Save all modified files button. If one or more of the files is untitled (i.e., added to the project since the last save) the user will be prompted to save the file(s) with the current project FPR file name as a default. The untitled file(s) can also be saved under any file name; however, do not change the extension of FPR, FEM or FEN-files.

e) If you try to exit before saving untitled unit/parameter files or if you have changed a unit/parameter-file name and did not resave the project FPR-file, you will be prompted to save the files.

f) To change a project by adding or removing one or more units, such as thickness, storativity, anisotropy, or topsystems, use the Project / Project manager / + Add unit or / -Remove unit buttons.


h) Parameter files can also be opened and saved in the Input mode (Mode 2).

3)   Grid Generation

a)   A new element grid can be generated by selecting Files / New grid.

b)   An existing FEN-file can be used or a completely new grid can be created using the FemGrid or FemMesh grid generator. FemGrid creates grids based on one or more irregular regions, each with its own nodal spacing. FemMesh creates grids based on one or more triangles and quadrangles with a gradually changing node-spacing.
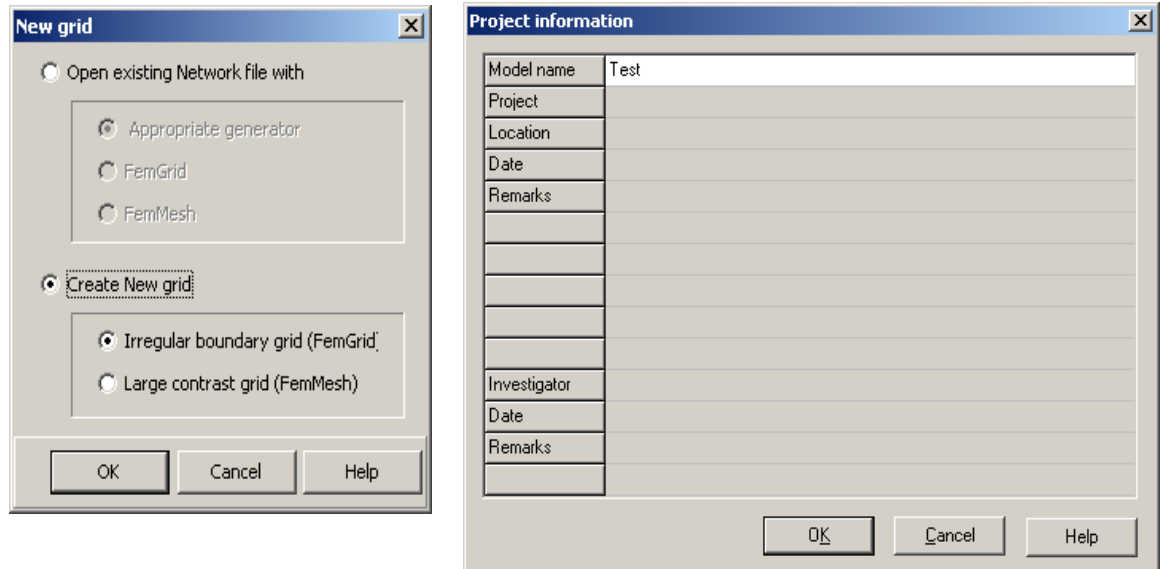
c)   Building a new grid starts with a box to enter project information. The Model Name is used as the default file name for the FPR-file. The project information can be accessed and changed using Project / Information… on the Menu bar.

d)   The FemGrid and the FemMesh toolbars are virtually the same:

-
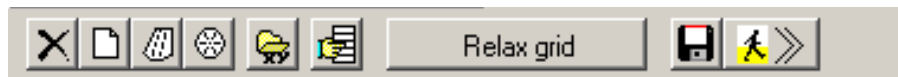
-    cancel grid generation
-    start with a new grid
-    start with FemGrid or FemMesh example grid
-    start with a hexagonal grid

-    load a file of x- and y-coordinates (only FemGrid)
-    insert or delete data

-    successive steps of grid generation

-    save a FEN-file (or 'incomplete' FEN-file)
-    use the grid to create a model and enter the number of aquifers (max 20).

FemGrid data entry list

Fixed nodes | Segments | Regions

Change number to 6

No	Spacing	Nodes	[1]	[2]	[3]	[4]
1	600	4	8	4	1	5
2	1200	2	5	6	x	x
3	2400	4	6	2	3	7
4	1200	2	7	8	x	x
5	1000	2	5	8	x	x
6	2000	2	7	6	x	x

FemMesh data entry list

Fixed nodes | Triangles | Quadrangles

Change number to 8

No	x-coordinate	y-coordinate	spacing
1	0	0	400
2	10000	0	1000
3	10000	10000	1000
4	0	10000	400
5	2000	3000	100
6	2050	2925	100
7	2050	6050	100
8	2000	6000	100

e)   The size of the lists for segments and regions is often too small to enter all the data. If a wider list is required, you can drag the left margin of the list (the boundary between the data list and the graphical window) to the left to temporarily resize the list area.

f)   The size of the entry lists is limited.
Rows:
- maximum number of fixed nodes = 50.000
- maximum number of segments = 250
- maximum number of regions = 50
- maximum number of triangles = 100
- maximum number of quadrangles = 100
Columns:
- maximum number of nodes per segment = 100
- maximum number of segments per region = 30

g)   By highlighting a cell in the list of fixed nodes, the blinking cursor is moved to the corresponding node. This works both ways: if you left-click on a fixed node, the corresponding node in the list is highlighted.

Pointing at a cell in the list of segments or regions colors the corresponding segment(s) yellow (or any other selected marking color).

h)   The order of steps to generate a grid is:
- draw fixed nodes
- draw boundary nodes
- draw internal nodes
- draw elements
- relax grid
- renumber nodes
The procedure restarts if the input data are changed, the Esc-key is pressed, or when a FEN-file has been read.

i)   Please refer to the Help system "Building networks / Grid generation" for a description of the procedures and the details to create grids according to the FemGrid and the FemMesh techniques.

4)   Marking Nodes & Computing Water Balances

Ins	=	draw grid
Del	=	remove all marks
F2	=	water balance current node on/off
F3	=	water balance marked nodes on/off
F4	=	water balance non-marked nodes on/off
F5	=	water balance entire model on/off.
F6	=	blink marks
F7	=	mark model boundary
F8	=	mark all the nodes within a marked boundary
F9	=	change marks of all the nodes
F10	=	automarker on/off
		- left-click = mark; right-click = remove mark
F11	=	line-marker on/off
		- left-click the starting node and one or more subsequent
		nodes to mark a line
F12	=	jump to a node:
		- the node with a given number
		- the next (higher number) node with a given label
		- the next node with a discharge unequal to 0 in the current layer
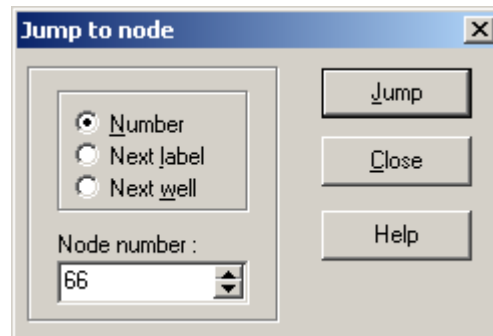
Water Balance

a) The three columns show the computed Inflow, Outflow, and Inflow minus Outflow

b) Leakage indicates vertical flow through the top of an aquifer. Inflow means downward flow to the underlying aquifer, while outflow is upwards.

c) The lateral flow component only indicates horizontal flow from or to adjacent parts of the model, while the boundary flow component crosses the model boundary.

d) The table can be printed or saved to file (two buttons), the text can be copied to the clipboard (right mouse click) and used in e.g. Excel, or the screen can be copied to the clipboard (Alt + PrintScreen).

e) The water balance error is the difference between total inflow and total outflow. This error should be relatively small, compared to the total flow components, when the heads are computed for a steady-state model. Errors can be reduced by increasing the number of iterations. If the water balance table is computed for transient flow models, the presented error is affected by changes in the amount of storage

Water Balance

Total Area : 685598757.81 m²
Flow units : m³/day

	Inflow	Outflow	In - Out
Precipitation	342799.4		342799.4
Sum topsystems	342799.4		342799.4
Leakage	263247.4	411977.7	-148730.3
1 Lateral flow			
Boundary flow	30754.4	126121.9	-95367.5
Well flow		39367.7	-39367.7
Total (error)	641420.7	641417.1	3.5
Leakage	63949.8	4619.4	59330.3
2 Lateral flow			
Boundary flow	3491.9	12328.4	-8836.5
Well flow		33360.0	-33360.0
Total (error)	71664.9	71656.3	8.6
Leakage	21348.5	4223.2	17125.3
3 Lateral flow			
Boundary flow	3899.7	4880.6	-980.9
Well flow		12898.6	-12898.6
Total (error)	29967.5	29963.2	4.3
Leakage	7960.7	4719.3	3241.4
4 Lateral flow			
Boundary flow	5060.6	7374.3	-2313.8
Well flow		919.0	-919.0
Total (error)	13021.3	13012.7	8.6

5) Data Input

Ins   =   draw grid
Del   =   remove all marks (make all nodes unselected)
F2   =   mark nodes if current parameter values are > , < ,or between limits
               or mark nodes with a specified label

F3   =   assign to all marked nodes
F4   =   assign to all non-marked nodes
-          =   assign to all nodes up to a line of marked nodes
-          =   assign to all nodes up to and including a line of marked nodes
F5   =   assign to all nodes
F6   =   undo last assignment (restore old values)
F7   =   interpolate zero values

F8   =   set the marked nodes in the current aquifer to a no-flow boundary
               condition
F9   =   set the marked nodes in the current aquifer to a fixed-head boundary
               condition
F10   =   mark all the nodes with a fixed-head boundary condition in the current
               layer

F11   =   open parameter file for current parameter
F12   =   save parameter file of current parameter

a)   The number of labels per node can be increased up to five. Labels can be added (and removed) in the Project manager: Project / Project manager / + / Labels / New.

b)   Clicking the active parameter (blue cell) with the mouse (left-button), or pressing [Enter], enables data entry. Once the cell in the parameter list has been activated numerical values can be entered directly. Values can be assigned to one node only (Enter-key), to all nodes or to a sub-set of nodes. It is also possible to enter a formula (function) or to assign names (labels) to nodes.

c)   Right-clicking in the active cell opens a pop-up window with some edit-functions. In this way text (formulas) can be copied to the clipboard and pasted at another time or location, e.g. the batch-file editor. (A batch file can be saved to disk.)

d)   Please refer to the Help system "Input of model data" for a more detailed description of all available functions and variables for assigning values to nodes.

6)   Presentation of Results

Ins	=	draw element grid
Del	=	clear graphics screen
F2	=	draw the nodes
F3	=	draw only marked nodes
F4	=	draw lines between marked nodes
F5	=	draw model boundary
F6	=	draw coordinate grid
F7	=	draw contours (isolines) of the current parameter or label
		(options: only within a specified range, or only in non-marked areas)
F8	=	draw flow vectors (specific discharges) of the current aquifer or draw circles around all nodes to indicate the sign and value of the parameter.
F9	=	draw 2D or 3D flowlines
		- 3D flowlines can only be drawn if aquifer thicknesses are specified in parameter list 'Thick'

Colors are chosen from the palette:

            -- left-click (*)   : foreground color
            -- right-click   : background color

The multi-color button toggles color fill on/off.
New palette colors can be chosen from the Menu bar: Options / Colors / Custom colors.

F10	=	profile line on/off
		-- left-click = left end of line
		-- right-click = right end of line
F11	=	draw profile on/off
F12	=	draw cross section on/off

a)   DXF-files and flow line files can be added to a map with Import on the menu bar.

b)   Maps and drawings in the graphical window can be copied to the clipboard with Export on the menu bar.

c)   The order of drawing objects can be changed: Options / Map / Drawing mode / Move up or Move down. The objects in the list are drawn starting from the bottom of the list.

d)   A further description of the functions for drawing maps is given in the Help system "Display of results".

Flow Vectors and Flow lines

e)   Drawing mode / Draw vectors or circles (F8) draws flow vectors of the aquifer discharge. It is possible to draw vectors for various aquifers and with different colors in the same map. Circles or discs can be drawn of any parameter.
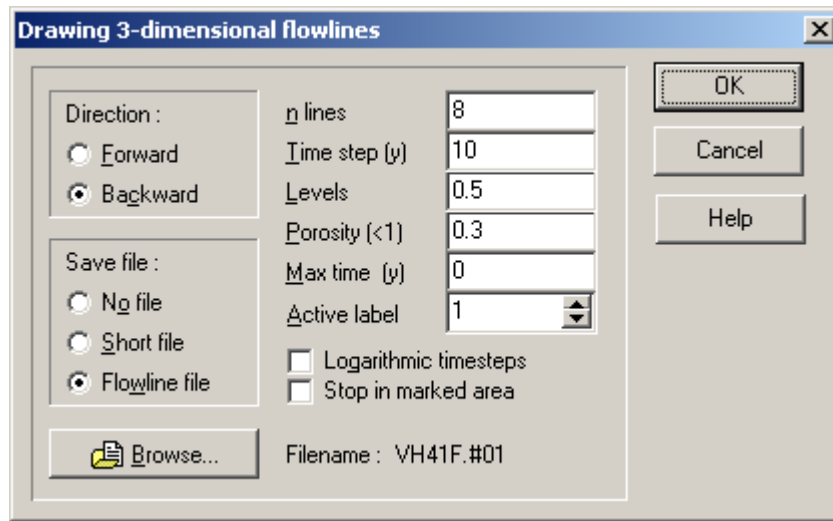

f)   Drawing mode / Draw flowlines (F9) opens a dialog box to enter information for drawing 2D- or 3D-flowlines. 2D-flowlines will be created if no layer thickness unit is part of the model. If the thickness is included (use the Project manager) 3D-flowlines are selected automatically. Additionally there is an option to stop flowlines in marked areas. 3D-flowlines are color coded with regard to the involved aquifer as specified in Options / Map / Colors.

g)   When zooming, flowlines are not automatically redrawn. However, flowlines will be redrawn if the flowlines are saved in a flow line file.

h)   Existing flow line files can be added to a map by using Import / Flowlines. Depth (aquifer) color coding of 3-D flowlines is only applied if the full-color button is enabled when the file is read.

j)   Set the active label register in the Draw flow lines dialog box if flow lines are started from equally labeled nodes in other than label register 1.

k)   Refer to the Help system for further information: "Display of results / Drawing flow lines and travel times".

Profiles and Sections

l)   Use Options / Profile or left-click on the graph to set the scale, colors, and other drawing options for profiles

m)   In a similar way Options / Section can be used to set the drawing options of a section. Use Options / Section / Flowline to specify the width of the flowline area in which the flow lines will be projected on to the section; this area is indicated in the map by two parallel red lines.

n) The size and position of a profile and section window will be saved when Options / Save options on the Menu bar is used.
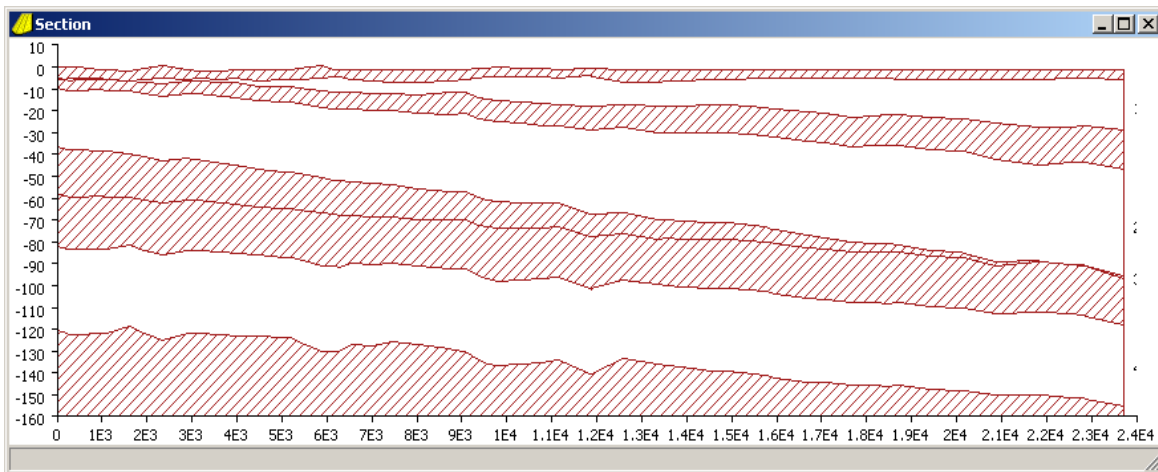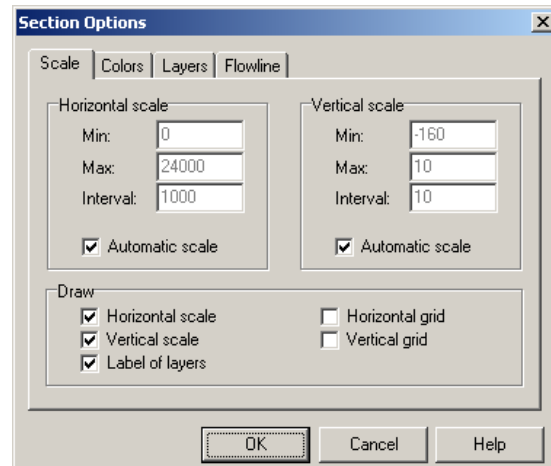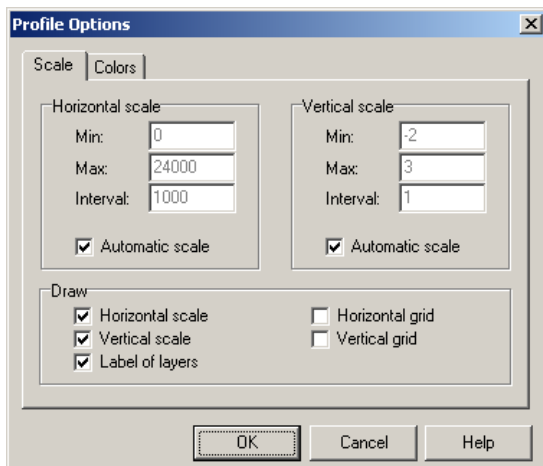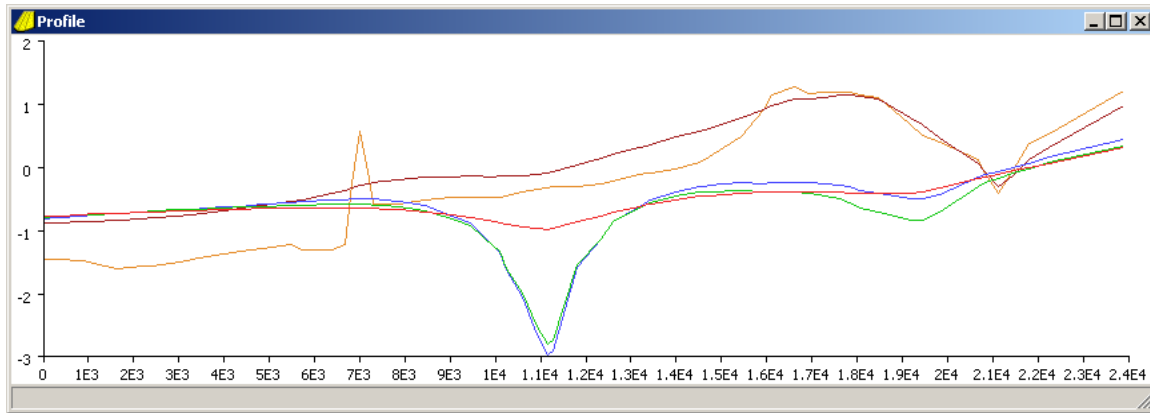
o) 3-D flow lines can only be drawn on the section if a current flow line file has been saved or a saved flowline file has been loaded.

p) The top level (top reference elevation) of the uppermost aquitard or water table controls the depths of the hydrostratigraphic units shown on the section. If the uppermost unit is an unconfined aquifer, then the final water table (h1) heads must be assigned as top level before drawing the section.

q) Please refer to the Help system: "Drawing profiles of hydraulic heads" and "Drawing sections of model layers and flow lines" for a more detailed description.

14

7) Grid Manipulation

a) The functions to swap node connections (F2) and to add a node (F3) require the selection of a red line, which can be obtained by clicking two neighboring nodes.

Ins	=	relocate nodes by dragging on/off
F2	=	swap the red line connection (if a red line is created by clicking two neighboring nodes)
F3	=	add a node on the red line
F4	=	delete the current node
F5	=	add nodes on all the marked connections
F6	=	delete nodes on all the marked connections
F7	=	delete a boundary node and its neighboring elements
F8	=	delete all the marked boundary nodes and elements
F9	=	mark nodes with large angles or many neighboring nodes
F10	=	relocate (optimize) all interior marked nodes
F11	=	Add a model layer at the base of the model or in between layers If Xtra is the current parameter list: Add a worksheet layer
F12	=	Delete a model layer at the base of the model or in between layers If Xtra is the current parameter list: Delete a worksheet layer

b) Use Options / Map / Alter-grid mode on the Menu bar to specify a value for rounding off the coordinates of nodes relocated by dragging.

c) The maximum number of aquifers is 20, while the number of nodes in each layer is preferably not much larger than 50,000 (no hard limit).

d) Please refer to the Help system: "Modifying and optimizing an existing grid" for a more detailed description.
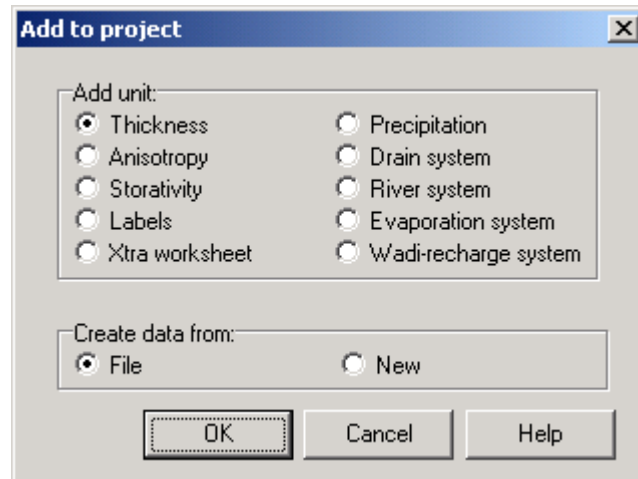
8)   Top systems

a)   Top systems can be added to a model by selecting: Project / Project manager / + Add unit from the Menu bar.

Available top systems are:
  Precipitation
  Drain system (maximum = 3)
  River system (maximum = 2)
  Wadi-recharge system
  Evaporation system

The topsystem can be created from either an existing file or by entering new values in the parameter list 'Top'.

b)   All topsystem parameters have codes that can be used for different purposes:
1) as a variable in functions in data input mode
2) as a code for loading parameter files in a batch file
3) as a default extension of parameter files.

  - precipitation: ppn
  - drain: dh1, dc1; dh2, dc2; dh3, dc3
  - river: rh1, rc1, ri1; rh2, rc2, ri2
 - wadi: wh1, wl1, wc1
  - evaporation: el1, eh1, em1

c)   Top system components can be removed by selecting: Project / Project manager / Remove unit from project.

d)   A further description can be found in the Help system: "Modeling top-boundary systems".

9) Steady-State and Transient Calculations

a) Start steady-state calculations by selecting: Calculate / Go calculate on the Menu bar.

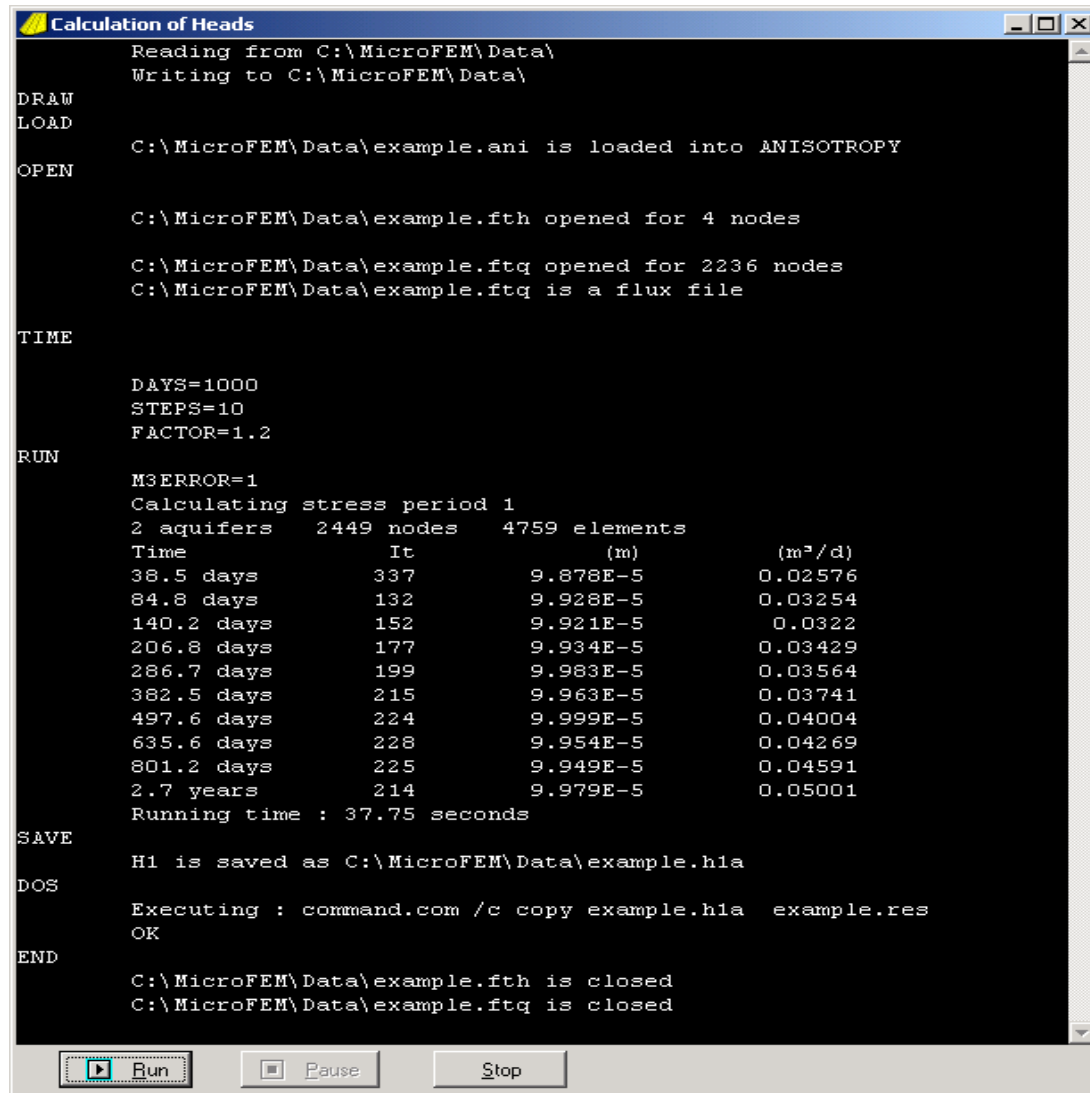b) Use Calculate / Options to set the stop-criteria.
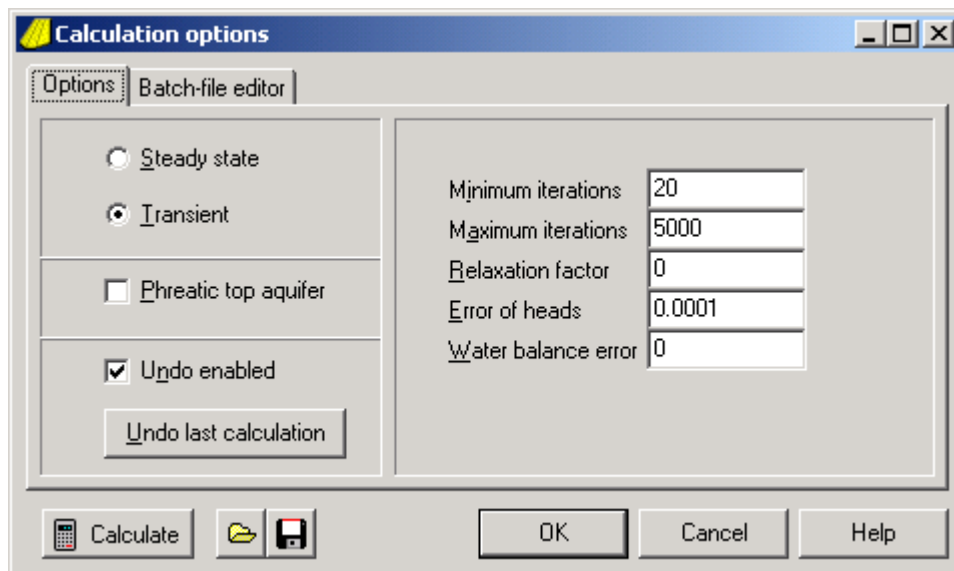
Relaxation factor = 0 means: automatic adjustment of the relaxation factor.
Water balance error = 0 means: water balance not used as a stop-criterion.
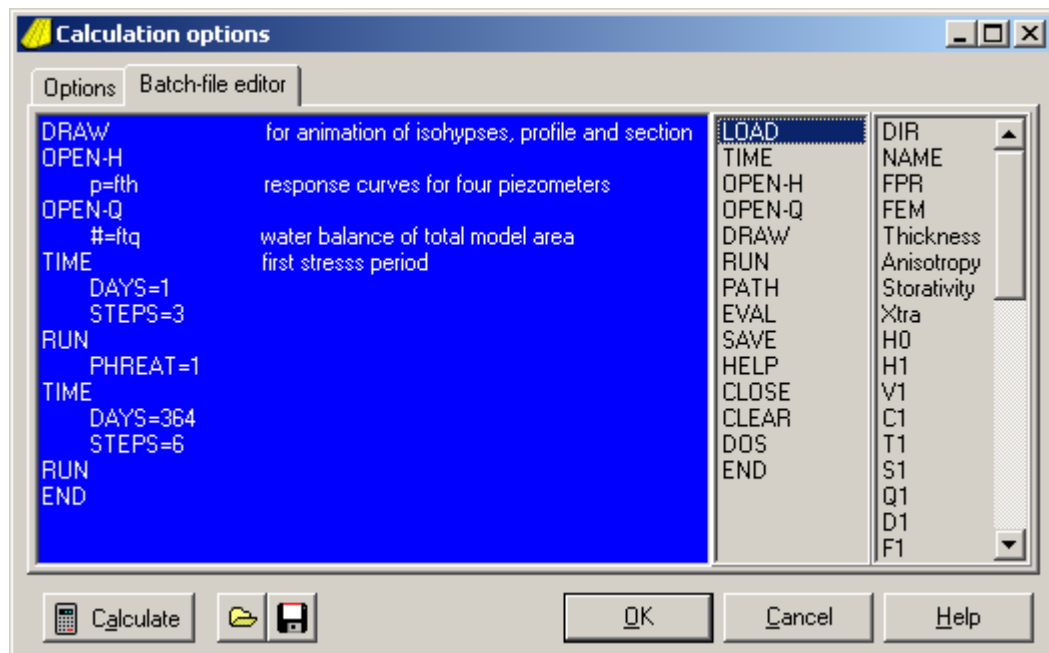
c) All the topsystems are available for steady-state and transient models.

d) Start transient calculations by selecting Calculate / Options / Transient on the Menu bar. This option is only available if the storativity parameter list has been entered into the model and if text is entered in the batch-file editor.

e) Use Calculate / Edit batch file to enter the editor and specify the commands for the transient calculations. On the right-hand side the list of main commands is displayed. Left-clicking produces a second list of possible specifications. All commands in the lists are copied to the editor by left-clicking. Batch-files are saved with the .feb extension. A batch-file is part of a project and its location is saved in the project FPR file.

f) Units (thickness, anisotropy, storativity, extra worksheet) and individual parameter files (including top systens) can be loaded into memory by using the LOAD command. Loading an FPR- or FEM-file is only allowed if no model is in memory at starting up MicroFEM, or after clearing all files from memory by selecting Files / Clear from the Menu bar, or with the CLEAR command in the batch file.

```
Calculation of Heads                                              _ □ ×
          Reading from C:\MicroFEM\Data\
          Writing to C:\MicroFEM\Data\
DRAW
LOAD
          C:\MicroFEM\Data\example.ani is loaded into ANISOTROPY
OPEN

          C:\MicroFEM\Data\example.fth opened for 4 nodes

          C:\MicroFEM\Data\example.ftq opened for 2236 nodes
          C:\MicroFEM\Data\example.ftq is a flux file

TIME

          DAYS=1000
          STEPS=10
          FACTOR=1.2
RUN
          M3ERROR=1
          Calculating stress period 1
          2 aquifers   2449 nodes   4759 elements
          Time             It             (m)          (m³/d)
          38.5 days        337        9.878E-5        0.02576
          84.8 days        132        9.928E-5        0.03254
          140.2 days       152        9.921E-5         0.0322
          206.8 days       177        9.934E-5        0.03429
          286.7 days       199        9.983E-5        0.03564
          382.5 days       215        9.963E-5        0.03741
          497.6 days       224        9.999E-5        0.04004
          635.6 days       228        9.954E-5        0.04269
          801.2 days       225        9.949E-5        0.04591
          2.7 years        214        9.979E-5        0.05001
          Running time : 37.75 seconds
SAVE
          H1 is saved as C:\MicroFEM\Data\example.h1a
DOS
          Executing : command.com /c copy example.h1a  example.res
          OK
END
          C:\MicroFEM\Data\example.fth is closed
          C:\MicroFEM\Data\example.ftq is closed

   ▶ Run          ■ Pause          Stop
```

g) During processing the batch commands, a text window 'Calculation of Heads' is opened summarizing the calculation process. An option can be set to save the log file or, afterwards, by right-clicking in the window a file/edit menu appears to save the results.

h) Please refer to the Help system: "Transient flow modeling" for a detailed description of all the available commands to create a batch file for the calculation of transient models.
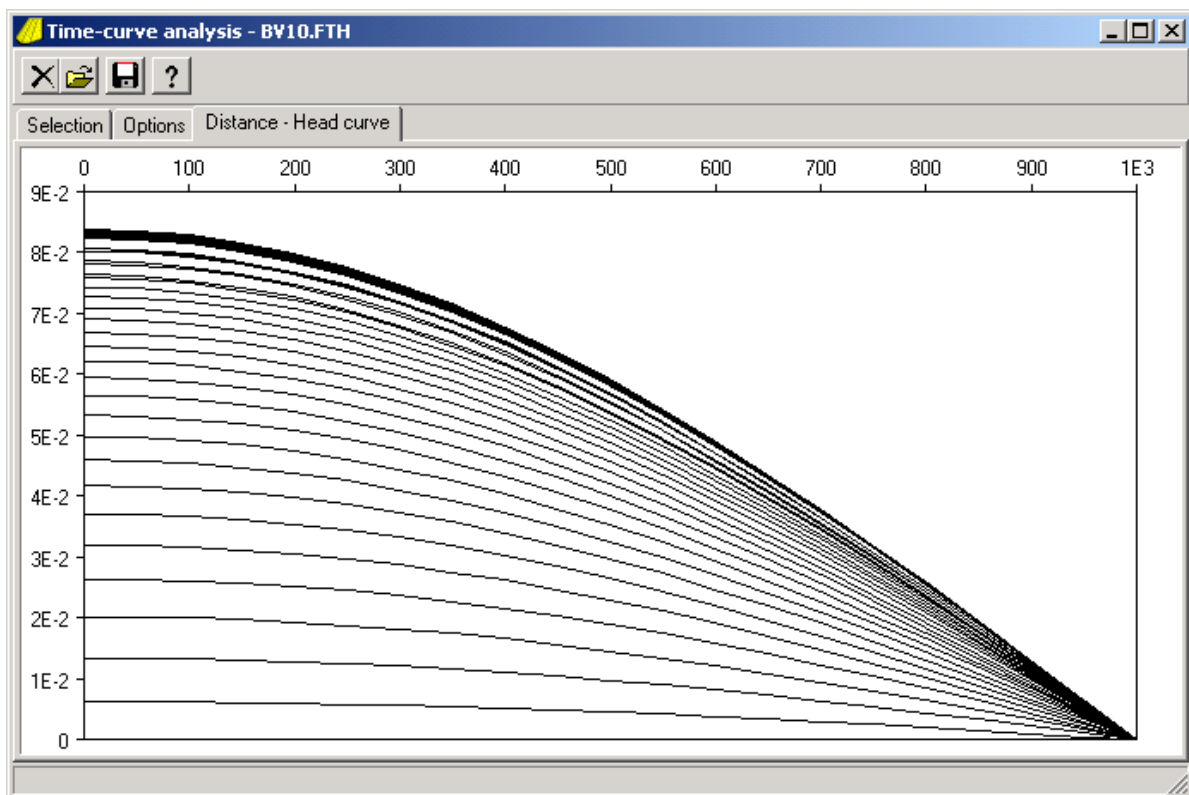
10) Time-Head Curves

a)   A utility for drawing time-head curves and distance-head curves is available by selecting: Tools / Time-curves on the Menu bar. After the FTH-type file is read, the times and nodes to be graphed can be selected. Use Tools / Time-curves / Options to select the drawing options of the graph to be displayed.
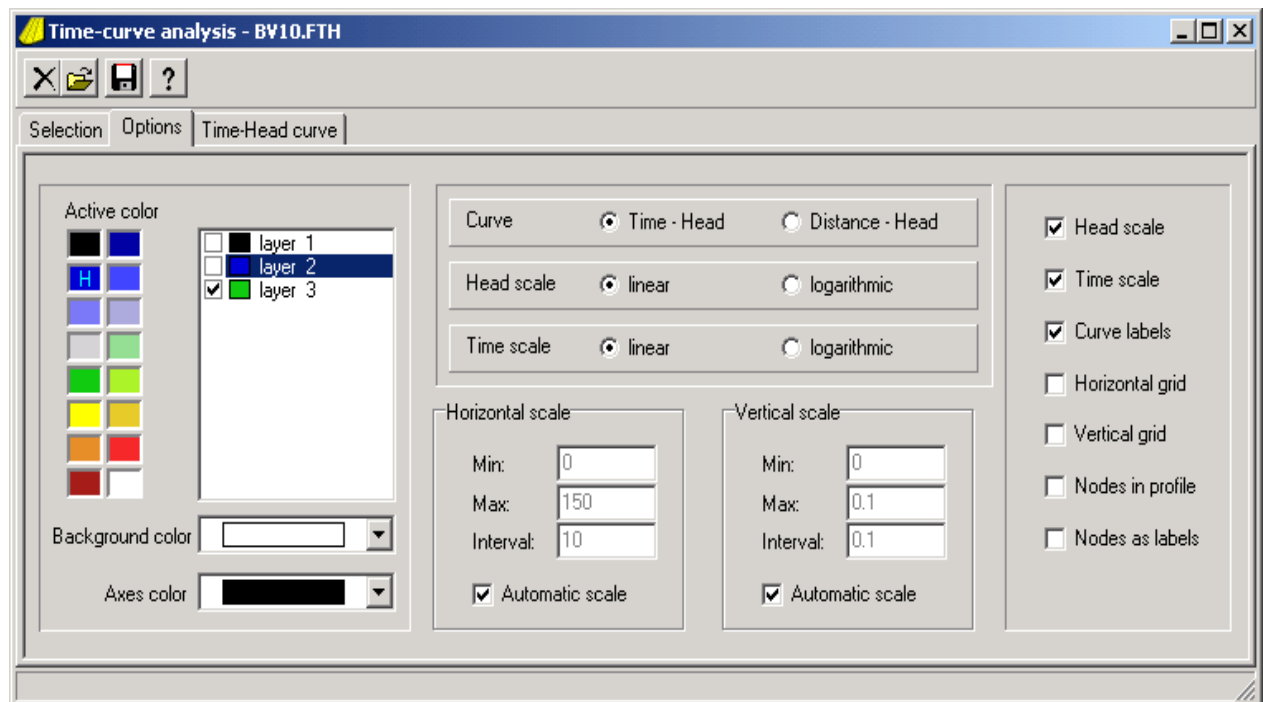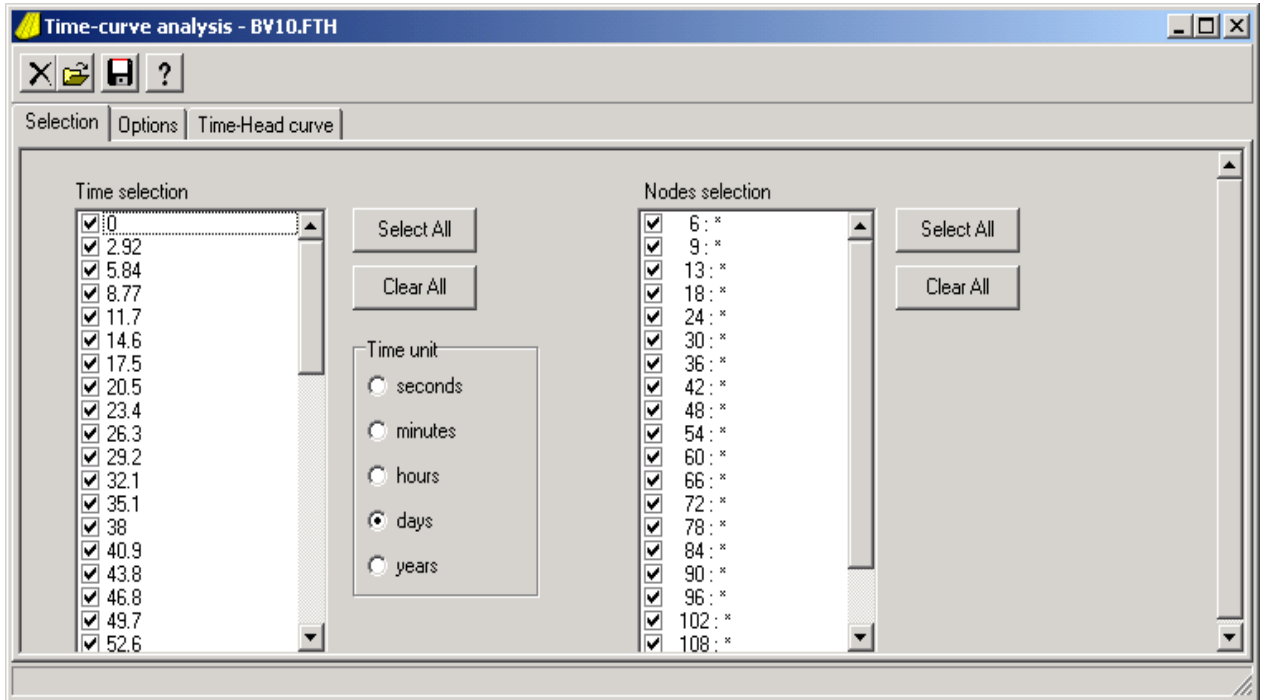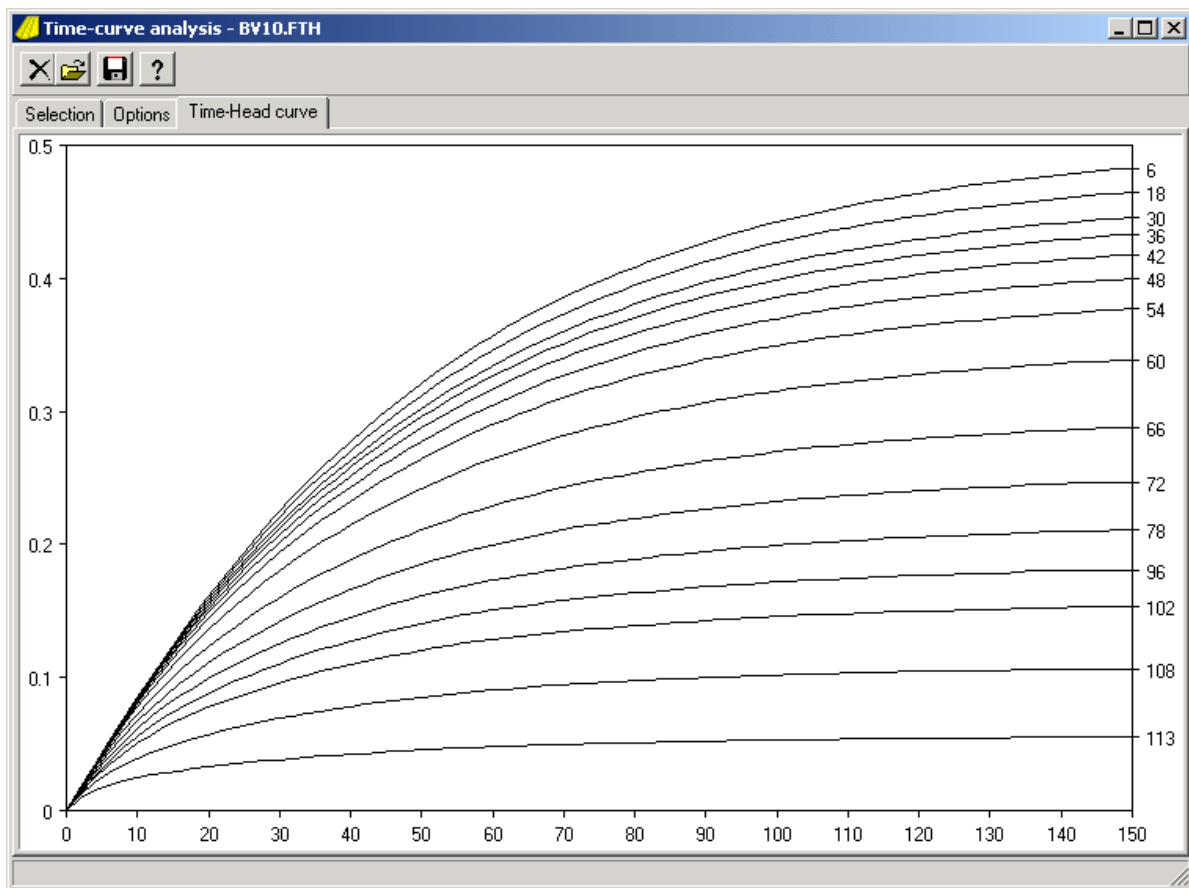
Time-head curves

Distance-Head curves

## 11) Import and Export data

a) Import options:

- DXF-files   to read and display files in AutoCAD format, also available in Grid-generation mode
- Flowline files   to read and display 2D and 3D-flowline files
- Xyz-files   to read a list of xyz-values and assign results to a parameter
- Special ASCII-files   to read nodal data into one model parameter (.&.. file) or to 'fill' a whole model with data (.&&n and .CSV files).
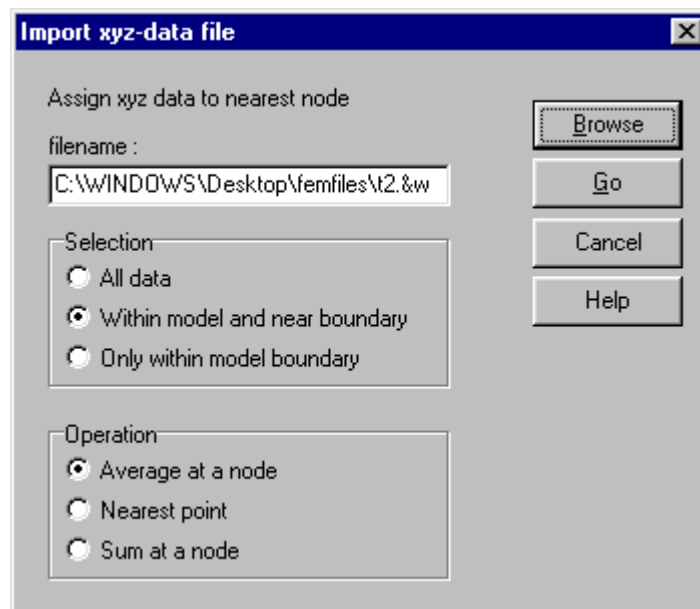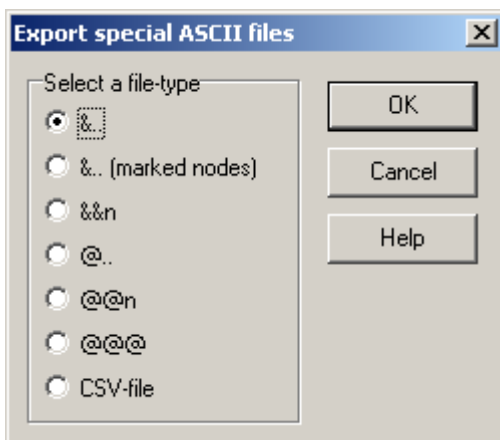
b)   Export options:

  - Clipboard   to copy the map, profile, section or time-curve to the clipboard
  - DXF-files   to save drawing objects in AutoCAD-format
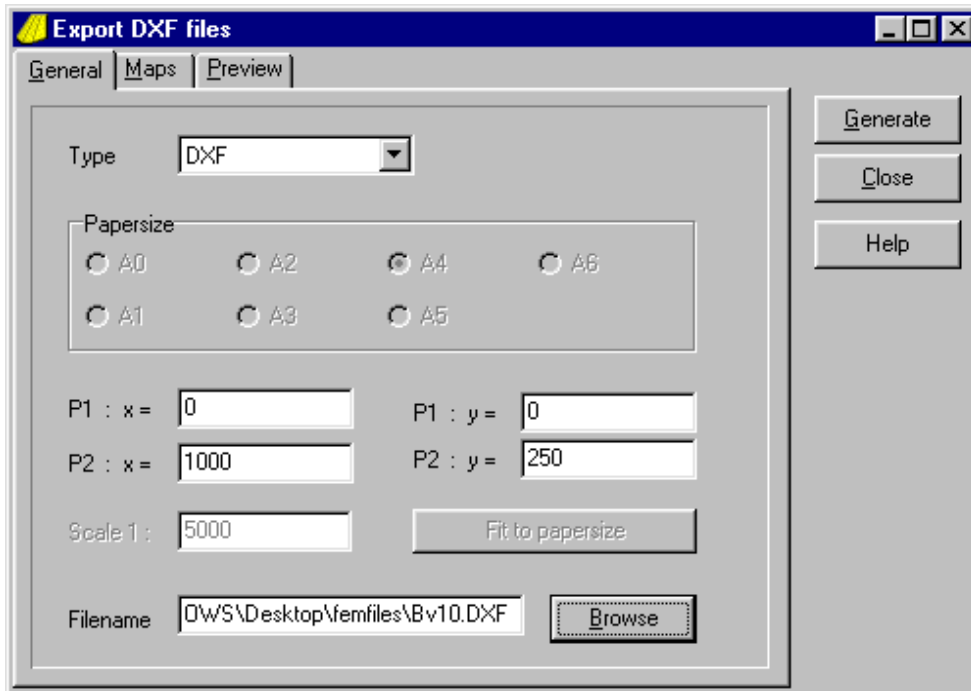
  - HPGL-files   to save drawing objects in HP-plotter format
  - Special ASCII-files to save model data in various formats, for import in
   GIS-programs and Excel

  - Connectivity list   to save a list of element node numbers.

12) Water Table Calculations

a) Water table conditions can be modeled in the upper aquifer (or parts of the aquifer) by setting $c_1 = 0$. Recharge is modeled with the precipitation top system. Time-variant transmissivities (transmissivities of homogeneous aquifers, dependent on a changing water table level) can be modeled for transient models by adding a thickness unit specifying the (positive) initial saturated thickness of the upper aquifer and the initial heads and transmissivities in the h_1 and t_1 registers. If the Calculation / Options / "Phreatic top aquifer" box is ticked, then, for all nodes with $c_1 = 0$, t_1 will be adjusted depending on h_1 changes. Confined conditions in the other parts of the upper aquifer can be modeled by setting c_1 to very high (fully confined) or lower (leaky) values. (It is important to specify h_0 for leaky areas.) For steady-state problems, set storativities $= 0$.

b) If sections are drawn the top reference elevation/level of the uppermost aquitard or water table controls the depths of the hydrostratigraphic units shown on the section. If the uppermost unit is an unconfined aquifer, then the top reference elevation (thickness unit) should be set equal to the initial water table heads before starting water table calculations, or the final water table heads must be entered as the top reference elevation before drawing the section.

13) The 3D Viewer

a) A utility for drawing three-dimensional pictures of the model is available by selecting: Tools / 3D Viewer on the Menu bar.
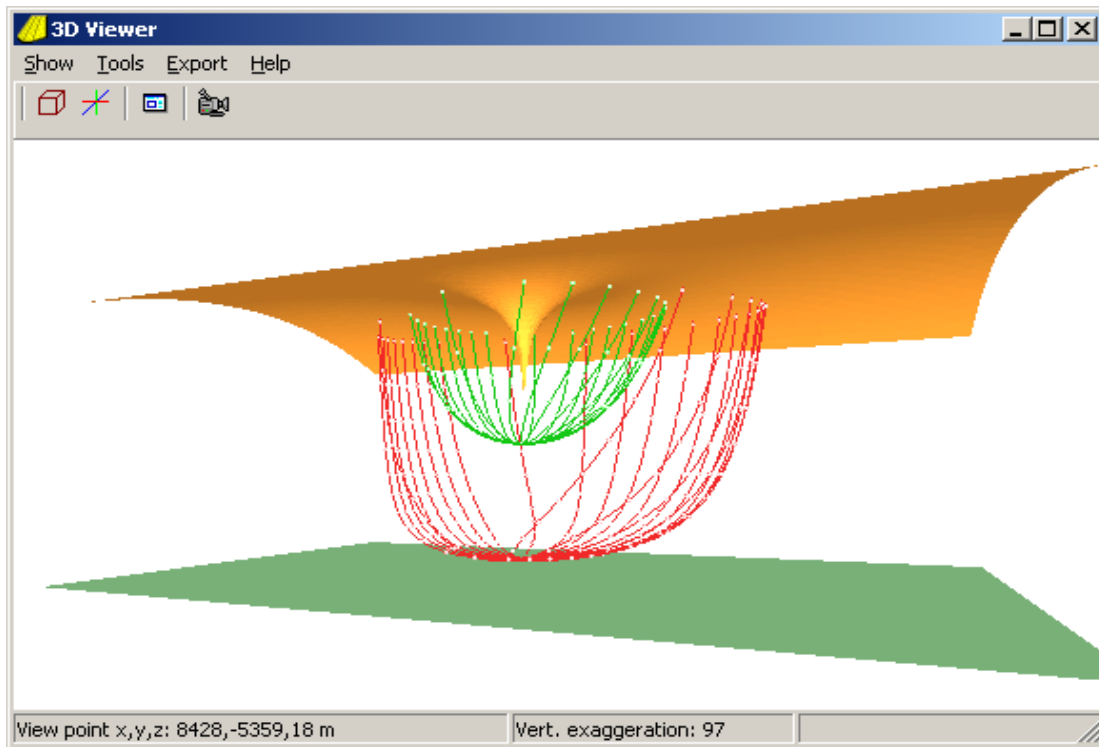
b) The 3D Viewer drawing objects include the heads, aquifer and aquitard top and base, contents of Xtra registers, wells and flow lines.
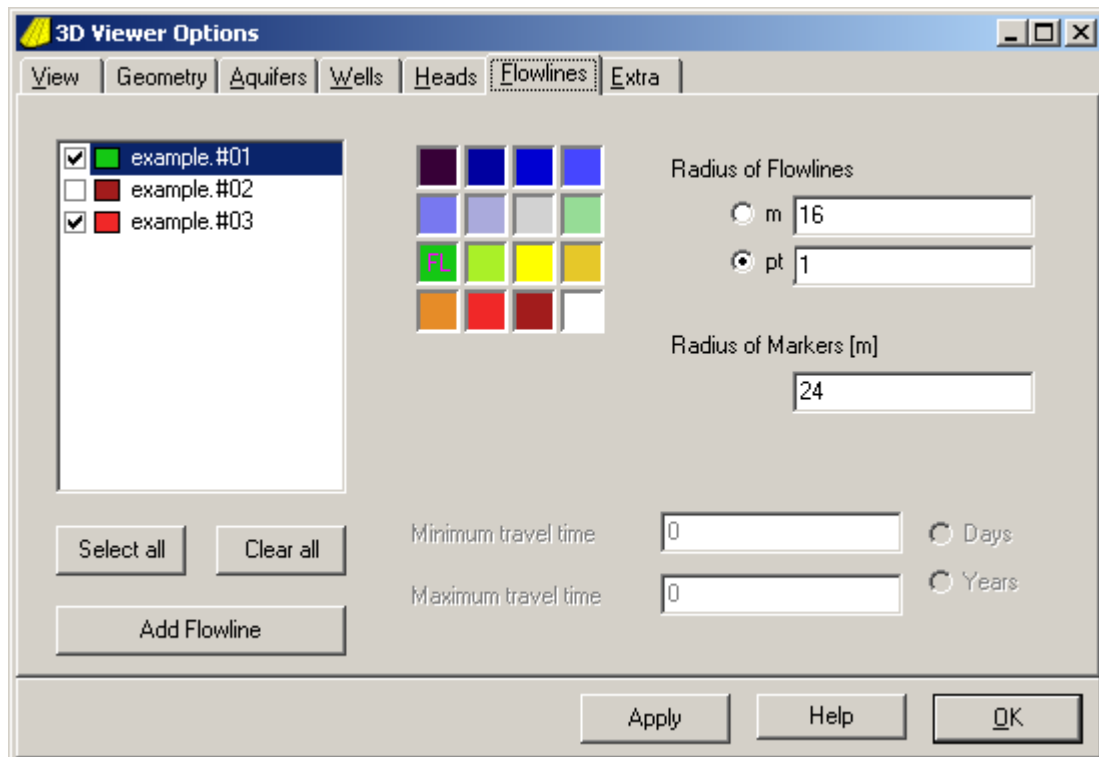
14) Model optimization

a) Steady state model can be calibrated with the parameter optimization tool: Select Tools / Optimization.

b) The Xtra registers are used to store the field data (measured heads) for each aquifer.

c) Up to 40 parameters, aquifer properties and boundary conditions for any part of the model, can be optimized in one run.

d) Residuals can be saved to the Xtra registers to be inspected for systematic errors.

e) Results include the standard error of each optimized parameter and a correlation matrix.

Parameter Optimization

Field data | Parameters | Optimize | Results | Save residuals | Accept model

Number of parameters to be optimized: 6   Clear Parameters

No	Parameter	Label or fragment	Number of nodes	Relative value	Unit	Optimize
1	T1	z1	16	100	%	☑
2	T1	z2	20	100	%	☑
3	C1	z3	16	100	%	☑
4	C1	z4	20	100	%	☑
5	H1	All Nodes	13	0.00	m	☑
6	Q1	All Nodes	73	100	%	☑

Close   Help