MIKE Powered by DHI Software

FEFLOW® 8.0

Finite Element Subsurface Flow

& Transport Simulation System

Introductory Tutorial

DHI

Agern Alle 5, 2970 Horsholm, Denmark

Phone: +45-45-16 92 00,

E-Mail: mike@dhigroup.com

Internet: www.mikepoweredbydhi.com

Contents

1 Introduction

1.1 About FEFLOW

FEFLOW (Finite Element subsurface FLOW and transport system) is an interactive groundwater modeling system for

- three-dimensional and two-dimensional
- regional and cross-sectional (horizontal, vertical or axisymmetric)
- fluid density-coupled, also thermohaline, or uncoupled
- variably saturated
- transient or steady state
- flow, groundwater age, mass and heat transport
- reactive multi-species transport

in subsurface water environments with or without one or multiple free surfaces.

FEFLOW can be efficiently used to describe the spatial and temporal distribution and reactions of groundwater contaminants, to model geothermal processes, to estimate the duration and travel times of chemical species in aquifers, to plan and design remediation strategies and interception techniques, and to assist in designing alternatives and effective monitoring schemes.

Sophisticated interfaces to GIS and CAD data as well as simple text formats are provided.

The option to use and develop user-specific plug-ins via the programming interface (Interface Manager IFM) allows the addition of external code or even external programs to FEFLOW.

FEFLOW is available for WINDOWS systems as well as for different Linux distributions.

Since its birth in 1979 FEFLOW has been continuously extended and improved. It is consistently maintained and further developed by a team of experts at DHI WASY. FEFLOW is used worldwide as a high-end groundwater modeling tool at universities, research institutes, govern-ment agencies and consulting companies.

In 2013, the FEFLOW book - written by FEFLOW founder Prof. Hans-Jörg Diersch - has been published by Springer. It represents a theoretical textbook and covers a wide range of physical and computational issues in the field of porous/fractured-media modeling.

For additional information about FEFLOW please do not hesitate to contact your local DHI office, or have a look at the FEFLOW web site http://www.feflow.com.

1.2 Scope and Structure

This exercise provides a step-by-step description of the setup, simulation, and post processing of a three-dimensional flow and mass transport model based on (simplified) real-world data, showing the philosopy and handling of the FEFLOW user interface.

The introductory tutorial is not intended as an introduction to groundwater modeling itself. Therefore, some background knowledge of groundwater hydrology and modeling is required, otherwise respective literature should be consulted in parallel.

The exercise covers the following work steps:

- Definition of the basic model geometry

- Generation of a 3D finite-element mesh
- Setup of a transient transport model, including initial conditions, boundary conditions and material properties
- Import of GIS data and regionalization
- Simulation run
- Results visualization and post processing

For additional information and other physical processes covered by FEFLOW, please refer to the FEFLOW help system in the main menu or by pressing the F1 key.

1.3 Terms and Notations

In addition to the verbal description of the required screen actions this exercise makes use of some icons. They are intended to assist in relating the written description to the graphical information provided by FEFLOW. The icons refer to the kind of setting to be done:

main menu

context menu

toolbar

panel

Bu button

input box for text or numbers

switch toggle

radio button

checkbox

All file names are printed in bold red, map names are printed in red italic and numbers or text to be entered by the user in bold green. Keyboard keys are referenced in <italic> style. All required files are available in the FEFLOW demo data. The symbols indicates an intermediary stage where either a prepared file can be loaded to resume this exercise or - if working with a licensed copy of FEFLOW - it is recommended to save the model. Thus the exercise does not have to be done in one step even in demo mode.

You can skip any of the steps in this exercise by loading already prepared files at certain stages. These model stages are not necessarily ready to run.

1.4 Requirements

If not already done, please install the FEFLOW software including the demo data package. A license is not necessary to run this tutorial (FEFLOW can be run in demo mode).

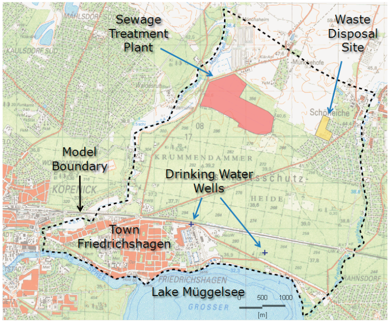
The latest version of FEFLOW can be downloaded from the website www.feflow.com. In case of any problems or additional questions please do not hesitate to contact the FEFLOW technical support (mike.de@dhigroup.com).

1.5 Model Scenario

A fictitious contaminant has been detected near the small town of Friedrichshagen, in the southeast of Berlin, Germany. An increasing concentration can be observed in two water supply wells. There are two potential sources of the contamination: The first are abandoned sewage fields close to a waste-water treatment plant located in an industrial area northeast of town. The other possible source is an abandoned waste-disposal site further east.

For following the exercise, the demo data files for FEFLOW have to be installed. The demo data installation package is available for download from www.feflow.com.

A three-dimensional groundwater flow and contaminant transport model is set up to evaluate the overall threat to groundwater quality, and to quantify the potential pollution. First, the model domain needs to be defined. The town is surrounded by many natural flow boundaries, such as rivers and lakes. There are two small rivers that run north-south on either side of Friedrichshagen that can act as the eastern and western boundaries. The lake Müggelsee can limit the model domain to the south. The northern boundary is chosen along an northwest-southeast groundwater level contour line north of the two potential contamination sources.



The geology of the study area is comprised of Quaternary sediments. The hydrogeologic system consists of two main aquifers separated by an aquitard. The top hydrostratigraphic unit is considered to be a sandy unconfined aquifer up to 7 meters thick. The second aquifer located below the clayey aquitard has an average thickness of approximately 30 meters.

The northern part of the model area is primarily used for agriculture, whereas the southern portion is dominated by forest. In both parts, significant urbanized areas exist.

2 Getting Started

2.1 Starting FEFLOW

On Windows Systems

- Start FEFLOW 7.5 via the corresponding desktop icon or the startup menu entry.

On Linux Systems

- Type feflow75q in a console window and press <Enter> .

If no FEFLOW license is available, FEFLOW can be set to demo mode via Tools > License... In demo mode, loading and saving of files is limited to 2500 nodes. Specially prepared demo files provided with FEFLOW are an exception. Such files are available for all relevant steps of this example so that the model setup can be interrupted and picked up again.

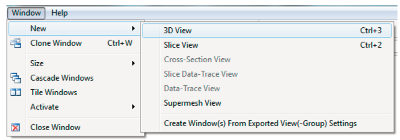
2.2 FEFLOW 7.5 User Interface

The user interface components are organized in a main menu, toolbars, panels, view windows, and dialogs.

While the main menu is always visible, the other parts of the interface can be customized, adding or hiding particular toolbars and panels by using the menu command w > Toolbars and w > Panels, respectively. Please keep in mind that not all panels and toolbars are displayed by default. Thus this exersice may require to access a function in a toolbar or panel that is not visible at that moment. The toolbar or panel has to be added then.

During the work with FEFLOW models, view windows display a certain type of view on the model and its properties. There are different types of view windows: Supermesh view, Slice view, 3D view, Cross-Section view and Data-Trace view. The availability of different functionality like toolbars depends on the currently active view type.

View windows can be closed via the corresponding button in the view frame. New view windows can be opened by selecting Window > New and choosing the respective view window type.



The last type of user interface component relevant for the exercise are charts. Looking very similar to panels, they contain plots of time curves. Missing chart windows can be added to the user interface by opening View > Charts from the menu and choosing the required chart type from the list.

Last, but not least it might be worth to mention that all steps done in FEFLOW can be undone and redone via the corresponding toolbar buttons. There is no limit on the number of undo steps.

1_maps.mp4

The exercise workflows are also available as video screencasts on the FEFLOW installation USB drive and in the FEFLOW channel on YouTube.
The video symbol and the file name indicate the respective video for the following workflow. The first video starts here.

*In this exercise, different file types are used as data source at the different stages of modelling to show the number of options. In practical projects, it may be preferred to store basic data in one file type, e.g., *.shp when using GIS.*

3 Geometry

3.1 Maps and Model Bounds

After opening FEFLOW, start a new model by using the menu command File > New or the New button in the Standard toolbar.

All necessary files for this exercise are provided with the FEFLOW Demo Data package and are located in the project folder demo/exercise (by default in

C:\Users\Public\Documents\DHI FEFLOW 7.5.

The map files are found in the subdirectory import+export.

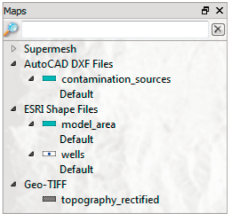
Click on 2D or layered 3D mesh and press Next. Pick the option Supermesh import from maps and click Next. On the upcoming page of the wizard, press the button for adding maps and load all the following maps at once (by holding <Ctrl> on the keyboard) to ensure that FEFLOW uses the bounding box of all the maps to define the initial domain bounds. You may have to select All Maps in the Files of Type selector at the bottom of the dialog.

The particular map files that are needed now are:

- topography_rectified.tif (a georeferenced raster image of the model area for better orientation)
- model_area.shp (a polygon map that contains several polygons denoting the outer model boundary and embedded contamination areas)
- contamination_sources.dxf (the footprint of the sewage fields and the waste disposal as polygons)
- wells.shp (the positions of the wells)

Some of these maps will also be used for model parameterization later on. After import, the maps are shown in a list. The contamination source areas do not need to be converted into supermesh polygons, thus we uncheck Convert for the map contamination_sources and click Finish.

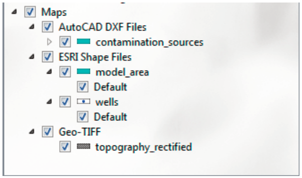
In the Maps panel, all loaded maps are shown, by default sorted by their file type (see figure). A double click on the Geo-TIFF topography_rectified adds the georeferenced topographic map to the active Supermesh view window.



Except for the maps contamination_sources and topography_rectified the maps are ESRI shape files. These vector files occupy their own branch in the tree, each with a default map layer. Double-click on all the Default layer entries to add the visualization layers of the shape files to the Supermesh view.

Now have a closer look at a second panel, the View Components panel. This panel lists the components that are currently plotted in the active view window.

When loading the map layers to the view, the maps have also been added to the tree in the View Components panel.



The drawing order of maps can be modified by dragging them with the mouse to another position in the tree (this might become necessary as the model area polygon may overlay the polygons of the mass sources). The topmost map is drawn on top.

To switch a map on and off, the checkbox in front of the map name can be checked/unchecked. If checking/unchecking the checkbox of an entire branch all the maps in this branch become visible/invisible at the same time.

The topographic map has mainly been loaded for providing a regional context. For more clarity, it can be switched off before starting with the following operations. Make sure that the other maps are visible.

3.2 Supermesh

In the simplest case, the supermesh contains a definition of the outer model boundary. In addition, geometrical features such as the position of pumping wells, the limits of areas with different properties or the courses of rivers can be included to be considered for the generation of the finite-element mesh. Additionally, the polygons, lines and points specified in the supermesh can be used later on to assign boundary conditions or material properties.

As mentioned above, a supermesh may contain three types of features:

- polygons
- lines
- points

At least one polygon has to be created to define the model area boundaries.

The editing tools for the supermesh are found in the Mesh Editor toolbar:

By having imported the maps with the checkbox for conversion set, the geometries inthe corresponding maps have been converted into supermesh polygons and points automatically. Thus we do not have to manually edit the supermesh here.

Both polygons and points are shown in the Supermesh view.

exercise_fri1.smh

3.3 Finite element mesh

Once the outer boundary and other geometrical constraints have been defined in the supermesh, the finite-element mesh can be generated.

All necessary tools can be found in the Meshing panel.

2_supermesh.mp4

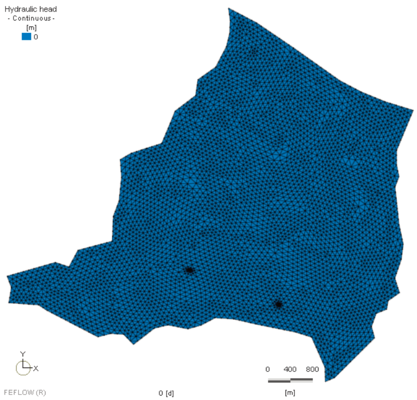
For this example, choose the mesh generation algorithm Gridbuilder. Click Generate Mesh to start mesh generation.

A new Slice view is automatically opened, depicting the resulting finite-element mesh.

For our purpose, especially for the simulation of contaminant transport, this initially generated mesh does not seem to be appropriate. A finer spatial resolution is required.

Activate the Supermesh view again and note that the Supermesh toolbar become visible again.

In the section From Supermesh Elements in the Meshing panel, click on Supermesh, and enter 6000 as Proposed Elements. Click on Generate Mesh again. The finite-element mesh in the Slice view is updated, showing a finer discretization now.



Local refinement

At the pumping wells steep hydraulic gradients are expected at the center of the well cone. To better represent these, a locally fine discretization is desirable.

Click on Generator Properties in the Meshing panel if the generator properties are not yet shown.

To obtain a refinement around the well locations, set the Point refinement value to 10.

Click Generate Mesh one last time and check the mesh for changes.

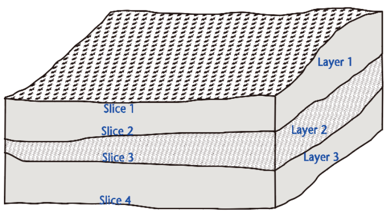
exercise_fri1.fem

3.4 Expansion to 3D

Up to this point we have worked on the model seen in top view, not considering the vertical direction. Starting from this 2D geometry, a 3D model consisting of several layers is set up.

The actual elevation of the layer tops and bottoms is derived by an interpolation based on map data (point-based data).

For this example, three geological layers are considered for the model. An upper aquifer is limited by the ground surface on top and by an aquitard on bottom. A second aquifer is situated below the aquitard, underlain by a low permeable unit of unknown thickness. This underlying stratigraphic layer is assumed to be impervious and is not part of the simulation.

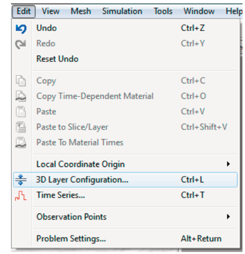


When using a layered approach, FEFLOW distinguishes between layers and slices in 3D. Layers are three-dimensional bodies that typically represent geological formations like aquifers and aquitards. The interfaces between layers, as well as the top and bottom model boundaries, are called slices.

In a first step, the numbers of layers and slices are defined. The actual stratigraphic data are applied in a separate step afterwards.

Initial 3D Setup

Open Edit > 3D Layer Configuration.

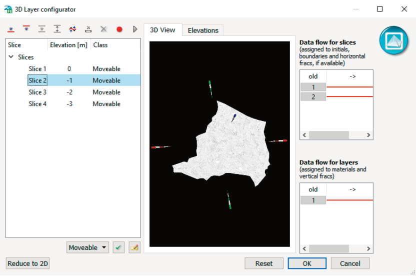


On the left side of the dialog, the current slices are shown (Slice 1 and Slice 2), corresponding to one single layer with the default top and bottom elevations of 0 m, 1 m resp. We add two more slices by clicking on Slice 1 and then on ICON Insert Slice(s) above. In the upcoming dialog, increase the Number of slices to 2 and click OK.

The elevations of the slices are not relevant at this stage.

Zooming functions can be used at any time. Press and hold the right mouse button, move the mouse up/down). Pan by pressing and holding the mouse wheel and moving the mouse to any direction. Also the mouse wheel may be used for

Besides refinement at points or polygon borders, FEFLOW also provides the means to edit the desired relative mesh density on a polygon-by-polygon basis.



Click on Bu OK to apply the settings and to exit the dialog. After finishing the basic layer configuration of the 3D model, a 3D view automatically opens. This view shows the actual 3D geometry of the model, now containing 4 planar slices with a distance of 1 m each. The 3D view background by default is set to black. To achieve better visibility in print, for all images in this exercise a white view background has been applied.

exercise_fri2.fem

Elevation Data

This raw geometry will be formed into its real shape by regionalizing elevation data contained in map files.

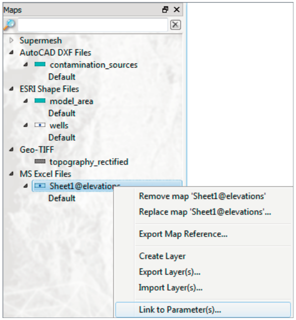
The basic data have been derived from a DEM and from borehole logs, and have been combined into an Excel file (*.xlsx) with four columns: X, Y, Ele, and Slice. Such a file can only be edited in spreadsheet software such as Microsoft Excel or Open Office / LibreOffice. Containing the target slice number as a point attribute, the file can be used as the basis for regionalization of elevations for all slices at once. The elevations are given in meters ASL.

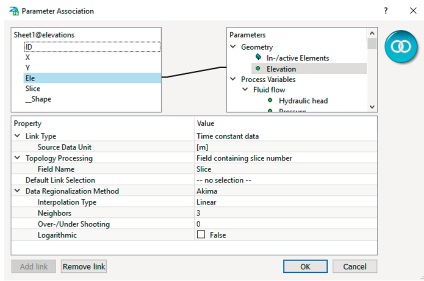
	A	B	C	D
1	X	Y	Ele	Slice
2	3403999.50	5818015.00	39.50	1
3	3403999.50	5818405.50	39.50	1
4	3403999.50	5819254.00	39.50	1
5	3403999.75	5814206.00	34.50	1
6	3403999.75	5813779.50	37.00	1
7	3403999.75	5813957.50	37.00	1

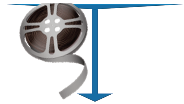
The file has to be loaded as a map before its attribute data can be used as the basis for interpolation. In to the Maps panel click on Filter and use Add Map(s)... to load elevations.xlsx. It is not necessary to visualize the map in the view.

The usage of Excel files requires the installation of a licensed copy of Microsoft Excel. On Linux systems or where Excel is not installed, please load elevations.dat instead of the Excel file.

As a next step, the attribute values of the data file need to be associated with (linked to) their respective FEFLOW parameter, in this case with the elevation. In order to do this, open the context menu of the map elevations with a right click and choose Link to Parameter(s)….







3_3D_geometry.mp4

On the left-hand side of the dialog, the available attributes of the map are listed. Select the entry Ele with a mouse click.

On the right-hand side, a tree view contains all available FEFLOW parameters that can be associated with the data. In this tree, open the Process Variables > Elevation branch and click on Elevation.

Click on Bu Add Link to establish a connection between the values in the map and the elevation data, or - alternatively - double click on Elevation to set the link.

Besides linking the attribute field to the model properties, a number of settings has to be done to ensure appropriate regionalization when importing the map data to the nodal values.

By default, FEFLOW expects elevation data to be in the unit meters, which is correct in this case.

Data are regionalized by applying two-dimensional interpolation. To separate data for the different slices, select Field containing slice number in Node/Slice Selection. In the next line, choose the attribute Slice as Field containing slice number (see image).

From the dropdown menu for Data Regionalization Method in the lower part of the dialog, choose the Akima method. As the properties, set:

- Interpolation type: Linear

- Neighbors: 3. Only the three map points that are closest to a mesh node are used for the interpolation.
- Over-/Under Shooting: 0. Thus the resulting values may not exceed the range of input values.

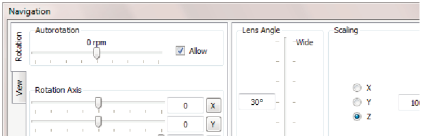
Click on OK to apply these settings and to close the dialog.

Elevation data assignment

Click into the 3D view to bring it to front. Make sure to have the Rotate tool in the View toolbar activated. Rotation, panning, and zooming can be easily performed using the mouse:

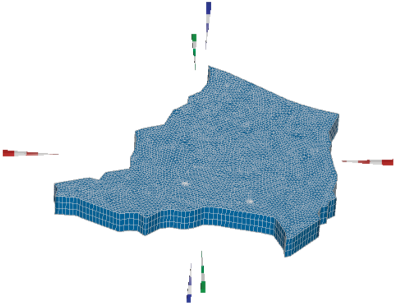
- Left mouse button: rotate the model around its center of gravity
- Center mouse button (mouse wheel): pan the model
- Right mouse button: zoom (in/out)

As the model has a rather small vertical extent compared to its horizontal dimensions, the (vertical) z-axis should be exaggerated.



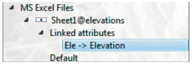
This can be done in the Navigation panel (View > Panels).

Click on the tab Projection and move the Scaling slider bar upwards until you have achieved a convenient view on the 3D model. Alternatively, the <Shift> key in combination with the mouse wheel can be used to change the stretch factor for the active 3D view.



To finally assign the elevation data by regionalization from map data to the selected nodes, two more steps are required:

- In the Maps panel, open the branch Maps > ASCII Table Files > elevation. Under Linked Attributes, double-click on Ele -> Elevation.



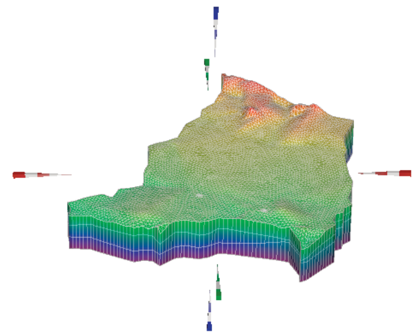
- Hereby, in the Editor toolbar, the map elevation is automatically set as data source in the input box and the model property Elevation is activated as parameter. Note that also Elevation has been chosen in the Data panel and is now shown in bold letters.

- Click on Select All in the Selection toolbar.
- Click Assign in the Editor toolbar to apply the new elevation data to all nodes.

In the 3D view the node elevations are immediately updated.

Click on Clear Selection in the Selection toolbar.

The result looks as shown in the figure below. Probably the Scaling has to be adjusted again (Navigation panel > Projection tab or <Shift> - mouse wheel) to account for the changed vertical extent.

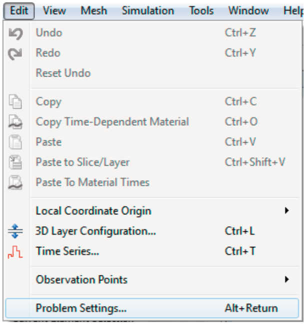


exercise_fri3.fem

4 Problem settings

FEFLOW provides the means to simulate a number of different physical processes in different spatial and temporal dimensions, ranging from simple 2D steady-state flow models to transient, unsaturated, density-coupled reactive transport models. As the input parameters depend on the model type, the general problem settings are typically done in the beginning.

Go to Edit > Problem Settings to open the Problem Settings dialog, where all general settings related to the current model are done.



All these settings are organized in thematic pages controlled by a tree view on the left-hand side of the dialog.

Problem class

The principal type of the FEFLOW model is defined on the Problem Class page.

Below the Scenario description (which is not mandatory to be modified) one of two general types of problems - saturated media and unsaturated/variably saturated media - is chosen.

By default Standard (saturated) groundwater flow equation is selected, applying Darcy's equation. Though this option is selected, the model is able to account for phreatic conditions.

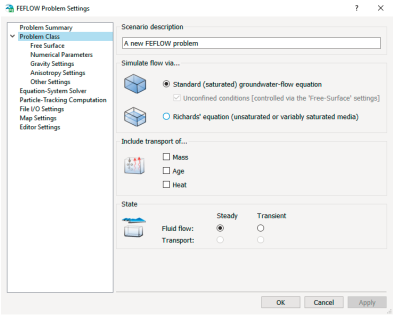
The second option - Richards' equation (unsaturated or variably saturated media) - would lead to Richards' equation being applied, accounting for both saturated and unsaturated conditions within one model.

In this particular case, it is not expected that considering the unsaturated/variably saturated zone would change the model result to an extent that would justify the additional effort for the solution.

Keep the default setting (Standard groundwater-flow equation) for this exercise.

On the same page, also other Problem class settings are done. Besides choosing between transient and steady-state conditions, it is also possible to add mass, groundwater age or heat transport processes to the flow simulation.

However, activating the transport option now would increase the complexity to an extent that is not necessary at this stage. Thus we first focus on the flow model and add mass transport at a later stage.



The flow model is to be run for steady-state conditions, so switch to Steady.

Click on Apply to apply the changes.

Free Surface

The first aquifer in the simulation area is known to be unconfined, so a phreatic water table is to be simulated. For this example, an approximation of the phreatic level by applying a pseudo-unsaturated technology is chosen. Hereby, a reduced conductivity is applied during the simulation to model layers if they are located above the water table. For more information on the handling of free surfaces in 3D models, please refer to the help system .

The settings for unconfined conditions are located on the Free Surface page. First of all, switch to Unconfined aquifer(s). In the Status column, open the drop down list of Slice 1 and choose the option Phreatic.

For the slices 2 and 3 keep the option Dependent (the status of the bottom slice 4 is fixed and cannot be changed).

Finally, set the Residual water depth for unconfined layers to a value of 0.05 m, hereby increasing the residual conductivity of dry elements for adding additional stability.

Close the dialog by clicking Bu Apply and Bu OK.

exercise_fri4.fem

5 Model Parameters

In the following sections, the physical properties of the study area are applied to the finite-element model.

The respective parameters are found in the Data panel. The parameters are organized in the following main branches in the tree view:

- Geometry

4_problem_settings.mp4

- Process Variables
- Boundary Conditions
- Material Properties
- Auxiliary Data
- User Data
- Discrete Features

5.1 Boundary conditions

To calculate the hydraulic head distribution between the upstream and downstream boundary, appropriate boundary conditions are applied. For the sake of simplicity, they will be kept in a rather simple way:

- Southern border: The lake Müggelsee completely controls the head along the southern boundary. The lake water level of 32.1 m is used as the value for a 1st kind (Dirichlet) hydraulic-head boundary condition.
- Northern border: As there is no natural boundary condition like a water divide close to the boundary, a head contour line will be used instead (hydraulich head = 46 m).
- Western and eastern border: Two small rivers (the Fredersdorfer Mühlenfließ and the Neuenhagener Mühlenfließ) form the boundaries at the western and eastern limits of the model. As they roughly follow the groundwater flow direction, we assume these heavily clogged creeks to represent boundary streamlines. No exchange of water is expected over this boundary and therefore a no-flow boundary condition is assumed.
- Finally, two wells, with a pumping rate of 900 m^3/d and 1,000 m^3/d, respectively, are located in the southern part of the model. These represent a number of large well fields in reality.

The hydraulic head boundary conditions are entered manually, while the wells are derived from a map.

Manual editing is often easier if being done in a 2D view. Thus switch to the Slice view. If you have accidentally closed it, a new view can be opened via Window > Slice view.

Slice view

This view type always shows a single slice or layer. Browsing between the slices is easiest by hitting the <Pg Up> and <Pg Down> keys, respectively. Alternatively, the layer/slice to be seen in the view can be directly selected in the Spatial Units panel.

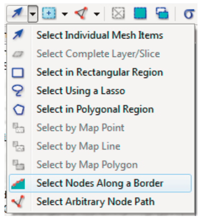
The recommended tool for navigation in the Slice view is the Pan tool in the View toolbar. The mouse buttons are associated with the following functions:

- Left and center mouse button: pan
- Right mouse button: zoom (in/out)
- Mouse wheel: zoom (in/out) in steps

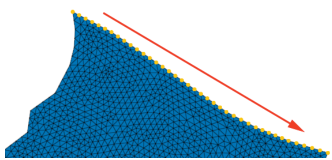
Northern Boundary

Zoom to the northern boundary.

For the Slice view, the Selection toolbar provides additional tools for selecting nodes compared to the 3D view. Choose Select Nodes Along a Border.

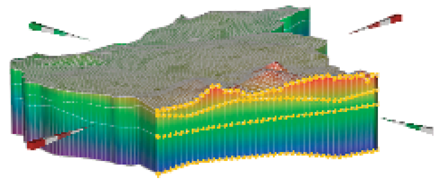


Having this tool selected, click on the westernmost node of the boundary and hold down the left mouse button, move the mouse cursor to the east-ernmost node and release the button. The nodes of the northern border are highlighted as yellow points. The selection is shown in the 3D view simultaneously.

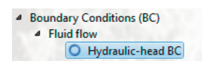


Next, the selection is extended to the other three slices of the model. A time-saving way to do this is the application of the Copy Selection to Layers/Slices tool. Start the tool and select all slices in the upcoming dialog (manually or by hitting <Strg>-<A>). Click on Bu OK.

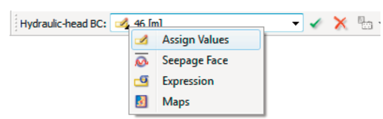
Bring up the 3D view and ensure that indeed all nodes at the northern boundary are selected.

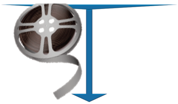


Go to the Data panel and double-click on Boundary Conditions (BC) > Fluid flow > Hydraulic-head BC, the type of boundary condition to be applied.



To manually assign a hydraulic-head boundary condition make sure that the Assign Values method is active in the Editor toolbar (see figure). If not, open the context menu by a right click on the symbol in the input box and choose Assign Values.



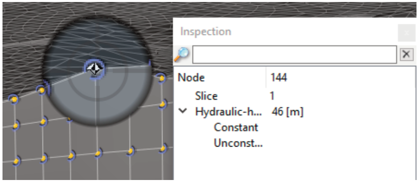


5_flow_bcs.mp4

Enter 46 m in the input box of the Editor toolbar and click on the Assign button.

Blue circles appear around the selected nodes to indicate the Hydraulic-head BC. While having the 3D view active, double-click on Hydraulic-head BC in the Data panel to also show the boundary conditions in 3D.

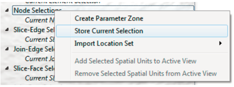
The values of the boundary condition can be checked using the inspection tool, which is activated by clicking on Inspect nodal/elemental values in the Inspection toolbar. Move the hair-cross to a node with a boundary condition. The values of all properties currently visible in the active view are shown in the Inspection panel.



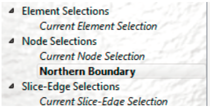
The inspection tool can be closed by hitting <Esc> or by activating another tool (e.g., the default tool Rotate) in the View toolbar.

After defining the boundary condition at the northern boundary, we store the current node selection for later use:

Open the context menu of the Spatial Units panel and choose Store Current Selection.



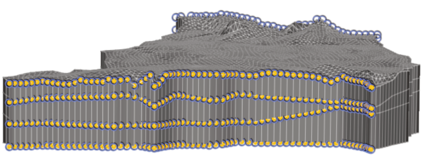
In the Node Selection dialog, change the name to Northern Boundary and click OK. The stored selection appears in the Selections panel. Click on Clear selection.



Southern Boundary

The assignment of boundary conditions along the southern boundary is done in the same way:

- In the Slice view, zoom/pan to the southern border.
- Select nodes along a border.
- Copy selection to all slices.
- Switch to the 3D view and check that the selection is set correctly.



- Make sure that Hydraulic-head BC in the Data panel is still active.
- Type 32.1 m in the input box of the Editor toolbar and click Assign.
- Store the selection for later use, naming it Southern Boundary.
- Clear selection .

exercise_fri5.fem

Remaining outer boundaries

At nodes without an explicit boundary condition set, FEFLOW automatically applies a no-flow condition. Therefore, no further action is required for the western, eastern, top and bottom model boundaries, which are assumed to be impervious except for groundwater recharge to be added later.

Pumping wells

The wells are to be set in the southern part of the study area based on the map wells. They are assumed to be screened throughout the whole depth of the model.

This kind of well, which stretches along a number of layers, is called a multilayer well. Multilayer wells are assigned along vertical element edges. Nodes along these edge selections are connected automatically by a high-conductive finite element that mimics the borehole.

Several parameters are necessary to assign a multi-layer well, including the pumping rate, the radius of the well and elevation of the top and bottom end of the screen.

While it is also possible to manually enter these values, it is more convenient to import them from a map.

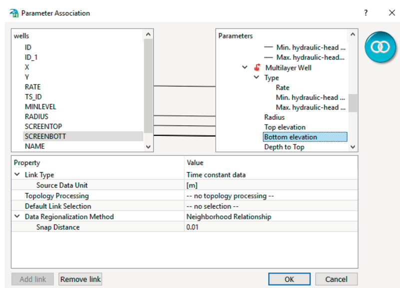
The wells map contains attribute data that need to be associated with (linked to) their respective FEFLOW parameter. In order to do this, open the context menu of the map wells (in the Maps panel) with a right click and choose Link to Parameter… to open the Parameter Association dialog.

On the left-hand side of the dialog, you see the available data of the map.

The attribute RATE relates to the abstraction rate of the well, select it with a mouse click.

On the right-hand side, open the Boundary Conditions > Fluid Flow > Multilayer Well branch and click on Type > Rate.

Click on Add Link to establish a connection between the values in the map and the Multilayer well or double click on Rate to set the link.



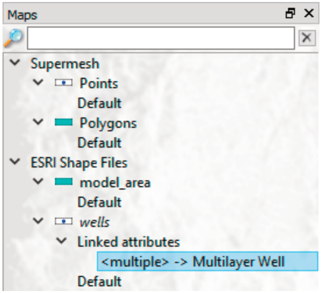
Repeat these steps for the attribute fields

- RADIUS (link to Radius)
- SCREENTOP (link to Top elevation)
- SCREENBOTT (link to Bottom Elevation)
- NAME (link to Name).

Each well will be created along that edge that is closest to its corresponding data point, but still within a user-defined search radius (snap distance). The Snap Distance should be small but greater than zero. Enter 0.01 meters in the input field. Click OK to close the dialog.

The actual assignment is done in a similar way already performed while importing the elevation data.

- In the Maps panel, open the branch Maps > ESRI Shape Files > wells. In Linked Attributes, double-click on <multiple>->Multilayer Well.

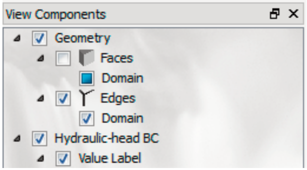


- Click on Select All in the Selection toolbar.
- Click on Assign in the Editor toolbar. Even though all edges in the model have been selected, the multilayer wells will be assigned each to the closest edges only. Clear the selection.

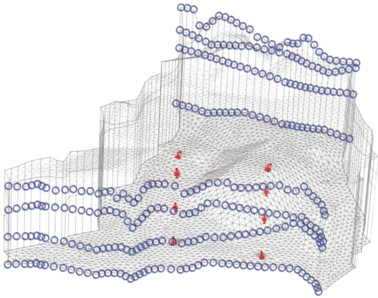
exercise_fri6.fem

Finally check all boundary conditions you have set.

Go to the 3D view. Make sure that Domain is selected in the Spatial Units panel. Double-click on Boundary Conditions (BC) > Fluid flow in the Data panel. All boundary conditions are shown in the view. Uncheck the checkbox of Geometry > Faces in the View Components panel to see into the domain.



Blue circles are shown on the northern and southern border, and four red symbols for each well.



exercise_fri7.fem

5.2 Material properties

Top Aquifer (Layer 1)

In the top aquifer, the hydraulic conductivity, the porosity as well as the groundwater recharge are to be set.

Although groundwater recharge from a mathematical point of view is rather a boundary condition, it is handled as a material property in FEFLOW. In a 3D model, the respective parameter to be set is In/Outflow on top/bottom.

The input procedures for material properties are completely analogous to the ones for the boundary conditions; material properties however are assigned to elements instead of nodes:

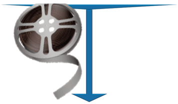
- Go to the 3D view.
- Reactivate the checkbox of Geometry > Faces in the View Components panel.
- Activate Material Properties > Fluid Flow > In/Outflow on top/bottom in the Data panel with a double-click.
- Choose the Select Complete Layer/Slice tool (in the dropdown selector on the left of the toolbar) in the Selection toolbar and select all elements in the top layer by clicking on it.
- In the Data panel, right-click on In/Outflow on top/bottom and choose Set Unit > mm/a from the context menu.
- Input a value of 195 mm/a into the box in the Editor toolbarand hit <Enter>.

The porosity is applied to the same selection:

- Activate Drain/Fillable Porosity in the Data panel with a double-click.
- Input a value of 0.1 and hit <Enter>.

The hydraulic conductivity will be assigned by interpolating data from field samples.

- Add the map conduc2d.trp to the Maps panel (Add map(s)). This file contains point-based conductivity values for the top aquifer in the unit 10^{-4} m/s.
- Associate (Link to Parameter…) the attribute column Value to the FEFLOW parameter K_xx. K_yy and K_zz will be calculated from K_xx and assigned later).
- Choose 10-4 m/s as Source data unit
- As a regionalization method, choose Akima, Linear, with Neighbors 3 and 0 % Over-/Under Shooting. Activate the checkbox Logarithmic. Close the dialog with OK.
- Double-click on Conductivity > K_xx in the Data panel.



6_multilayer_wells.mp4

If the top layer is no longer selected, choose the Select Complete Layer/Slice tool in the Selection toolbar and select it.

- In the Maps panel, double-click on Linked Attributes > Value -> K_xx.
- To finally assign the conductivity values, click the Assign button in the Editor toolbar.

exercise_fri8.fem

Aquitard (Layer 2)

In the second layer, we only need to assign constant values for the conductivity and drain-/fillable porosity.

- Clear selection and select the elements in the second layer applying Select Complete Layer/Slice again.

A very efficient way to assign multiple model properties is by right-clicking on Material Properties > Fluid Flow in the Data panel and choosing Assign Multiple... from the context menu.

In the following dialog,

- Enter 1e-6 m/s for K_xx (including the unit!).
- Enter 0.15 for the Drain-/fillable porosity
- Uncheck all other properties.
- Click Bu OK to finalize the assignment.

Lower aquifer (Layer 3)

Repeat the same steps as for layer 2:

- Clear selection and select the elements in the third layer applying Select Complete Layer/Slice again.
- Choose Assign Multiple... from the context menu of Material Properties > Fluid Flow.
- Enter 2e-4 m/s for K_xx.
- Enter 0.1 for the Drain-/fillable porosity.
- Uncheck all other properties.
- Click the Bu OK button to finalize the assignment.
- Clear selection.

exercise_fri9.fem

Anisotropic hydraulic conductivity

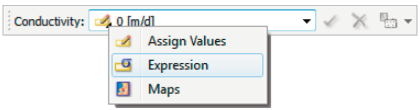
While K_xx has been assigned already, K_yy and K_zz will be derived from K_xx.

K_yy is equal to K_xx, it will be assigned using a simple copy&paste procedure:

- Select All elements in the 3D view.
- Choose Copy... from the context menu of Material Properties > Fluid Flow > Conductivity > K_xx.
- Double-click Material Properties > Fluid Flow > Conductivity > K_yy.
- Choose Paste... from the context menu of Material Properties > Fluid Flow > Conductivity > K_yy.

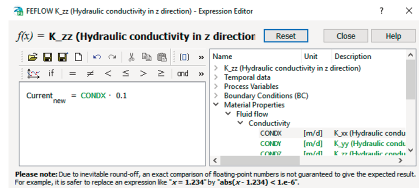
Besides manual assignment and data import, FEFLOW allows to calculate model properties from mathematical expressions. This will be used for the vertical conductivity K_zz, which is assumed to be 10% of the lateral conductivity K_xx.

- Double-click on Material Properties > Fluid Flow > Conductivity > K_zz.
- Switch to the Expression input method by clicking on the icon in the input box of the Editor toolbar or by right-clicking into the box and selecting from the drop-down list. Then double click on Current Expression. The Expression Editor opens.



The Expression Editor is a tool to create arbitrary mathematical expressions for various purposes. At the top of the dialog several toolbars provide basic mathematical operations. On the right hand side there is a list of all model parameters that can be used within the expression. All operations and parameters can be added to the expession by a double click, or by directly typing them into the expression using the keyboard.

For this particular operation, select and delete the word current on the right hand side of the equality sign. Afterwards, double-click on Material Properties > Fluid Flow > Conductivity > CONDX in the list on the right to insert it into the expression. Afterwards, click on the multiplication symbol in the toolbar on the top and finally type 0.1 on your keyboard. The resulting expression (see figure) calculates the vertical conductivity as 10% of the horizontal conductivity. Afterwards, press Bu Close.



To finally assign the new values, click the Assign button in the Editor toolbar.

Clear selection.

exercise_fri10.fem

Reference Field Data

To be able to compare the computed groundwater levels to measurements, a couple of observation points will be loaded into the model.

Go to the Maps panel and use the button (Add Map(s)...) to add observation_wells.dat to the list of loaded maps. It is not necessary to visualize the map in the view.

Open the Slice view again if it has been closed.

Right-click on the map entry in the Maps panel and choose Convert to ... > Observation Points from the context menu. The map file contains information about the location, slice number and measured

7_material_properties_I.mp4

hydraulic head in the observation well. As default headers are used in this example file, the association of attributes to the properties of the observation points works automatically and no changes need to be done in the upcoming dialog. Click the Bu OK button to proceed.

The now imported observation wells can be shown by a double-click on Observation Points in the Entities panel.

exercise_fri11.fem

6 Simulation

The flow part of the flow and transport model is complete. By running the steady-state model, a hydraulic head distribution will be computed that will also act as initial condition for the following transient simulation.

In case that FEFLOW is run in licensed mode, save the model to be able to return to the initial properties later! If running FEFLOW in demo mode, this model cannot be saved as the number of nodes per slice exceeds the allowed maximum of 500. Please use the prepared file exercise_fri11.fem in this case.

Starting the simulation

To run the simulation, click Start in the Simulator toolbar.

As the model is unconfined, the resulting nonlinear equation system needs solved iteratively, taking into account that the saturated thickness of unconfined layers depends on the actual solution for hydraulic head. The Error Norm History chart provides information about the remaining error in each simulation iteration. The simulation stops after eight iterations, the error reaching values below the defined error criterion.

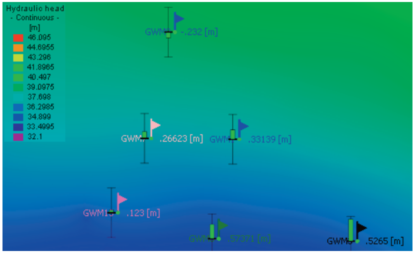
During and after the simulation, all visualization tools in FEFLOW can be used to monitor and postprocess the simulation results.

Make sure to select Domain in the Spatial Units panel. Go to the Data panel and double-click on Process Variables > Fluid Flow > Hydraulic head for visualization of the resulting hydraulic head distribution.

In addition, go to the Spatial Units panel and select (single click) Model Locations > Observation Points. Afterwards, double click Process Variables > Fluid Flow > Hydraulic head in the Data panel.

In the View Components panel, check both boxes for Hydraulic Head > Fringes and Isolines and uncheck Hydraulic Head > Continous. To achieve better visibility in print, the element edges have been removed in the figure.

Make sure to select Domain in the Spatial Units panel afterwards.



Scene Library

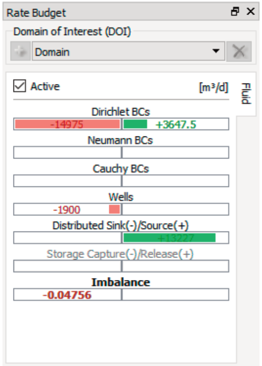
In the Slice View, chose Create Scene from Window. This stores a copy of the current view and all its settings in the Scene Library panel, in order to use it later for restoring the view.

exercise_fri12.fem

In case of loading the file, run the simulation before proceeding.

Budget

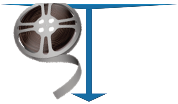
To check whether the model indeed has reached the steady state, the overall water balance is calculated. The Rate-Budget panel provides the means for this.



If it has been closed accidentally, open the Rate-Budget panel via View > Panels > Rate-Budget Panel.

Check the Active checkbox to activate the budget calculation. The budgeting is turned off by default as it can cause significant computational effort, especially when being done at each time step during a transient simulation run.

The budget shows inflows in green, outflows in red for the different boundary condition types, the areal sources and sinks (groundwater recharge) and - for transient models only - the storage capture or release. The Imbalance value shows the numerical error. it is sufficiently small to accept the solution as steady state.



8_material_properties_II.mp4

Streamlines

One way to visualize the flow field is the plotting of streamlines.

Streamlines are calculated by tracking the path of virtual particles that are released („seeded“) at certain starting points. In our case, multiple streamlines are released from around the nodes along the well screens. The random-walk method is used to add a diffusive/dispersive component to the particle tracking, hereby accounting for uncertainty.

First, a selection is created containing all nodes along the well screens.

Go to the 3D view.

In the Data panel, right-click on Boundary Conditions > Fluid Flow > Multilayer Well and choose Convert Parameter to > 3D Nodal Selection from the context menu. To save this selection, go to the Spatial Units panel and right-click to open the context menu.

Choose Store Current Selection and give the saved selection the name Wells.

Now, uncheck Geometry > Faces in the View Components panel to be able to see inside the model domain.

In the Selections panel, click on Node Selections > Wells.

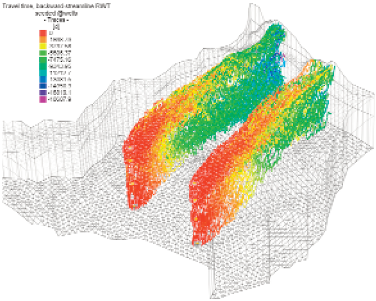
In the Data panel, double-click on Process Variables > Fluid Flow > Random-Walks > Backward.

In the View Components panel, right-click on Travel time, backward streamlines (RWT) seeded@wells and choose Properties from the context menu.

In the now opened Properties panel, enter 100 m as the Radius, press <Enter> and click Apply. Still in the same panel, right-click on the color scale on the left and choose Presets... > Rainbow.

Finally, activate the checkmark Travel time, backward streamline RWT seeded@wells > Legs in the View Components panel.

As a result, the random-walk tracks are shown in the 3D view. The color scale displays the travel time along these pathways.



From the result it can be seen that the western well is certainly influenced by the sewage fields. The eastern well seems to be reached by contamination from neither source. .

For more detailed analysis, a transport model seems useful.

Reactivate the checkbox of Geometry > Faces in the View Components panel before proceeding.

7 Flow and Transport Model

To exit simulation mode in order to apply changes to the model, click Stop in the Simulator toolbar.

When the flow model has been run, the process variable Hydraulic head has changed. After the run, it does not contain the initial conditions any more, but the final results.

In our case these will be used as the initial condition for the transient flow model.

exercise_fri13.fem

To obtain the correct status, run the simulation before proceeding and click on Stop.

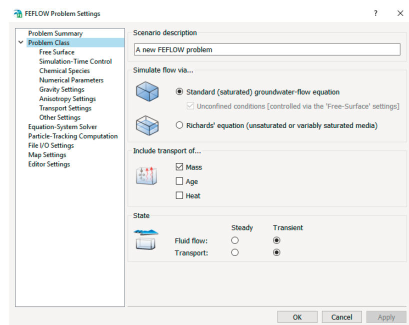
7.1 Problem settings

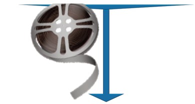
Problem class

From the preliminary streamline analysis based on the results of the flow model it cannot be excluded that the contamination sources are located within the capture zone of the production wells. To be able to provide more quantitative estimations, the model is extended to a flow and mass transport model.

To change the problem class, go to Edit > Problem Settings to open the Problem Settings dialog. In Problem Class, Include transport of... Mass and choose the Transient option for both the Fluid Flow and the Transport simulation.

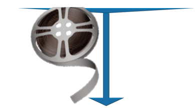
Confirm with Bu Apply.





9_observation_points.mp4

It is recommended to save the file before starting the simulation (if working with a license). During the simulation, the process variables will change and you would lose the initial conditions of the model.



10_flow_simulation.mp4

On computers with multi-core CPUs or multiple CPUs the simulation result and budget result may be slightly different in each run - even with identical input parameters. This is due to possibly different summation order when using parallelization.

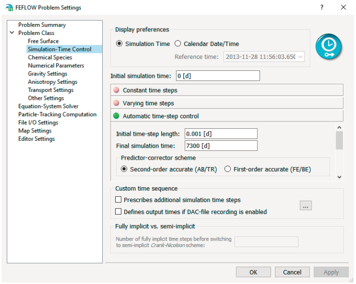
Time stepping

In a transient model temporal discretization has to be defined. The corresponding settings can be found on the Simulation-Time Control page.

By default, FEFLOW uses an automatic time-step control scheme. Hereby, an appropriate time-step length is determined internally by monitoring the changes in the primary variables (hydraulic head and concentration).

Enter a value of 7300 days in the Final Time input box.

Click on Apply and OK.



exercise_fri14.fem

7.2 Initial conditions

Click on Domain in the Entities panel.

Switch to the 3D view. Double-click on Process Variables > Mass transport > Mass Concentration in the Data panel. The 3D view shows the default initial concentration of 0 mg/l, representing fresh water. No changes are necessary for the background concentration.

Contamination sources

The contamination sources are represented by a higher initial concentration in the areas of the sewage fields and the landfill within the first aquifer.

First, a selection of the nodes belonging to the contamination source is needed.

Go to the Slice view and browse to Slice 1. Double-click on Process Variables > Mass transport > Mass Concentration in the Data panel. In the Maps panel, activate (double-click) the map contamination_sources. Choose the option Select by Map Polygon from the dropdown selector of the Selection toolbar. Make sure the Snap distance is set to 0 m in the Snap-Distance toolbar. Click Select by All Map Geometries in the Selection toolbar.

The contamination in both areas is found to reach down to the top of the aquitard. Use Copy Selection to Slices/Layers to copy the selection to slice 2.

The initial concentrations in these areas are to be interpolated from observed data.

Go to the Maps panel and add the map file conc_init.shp.

Associate (Link to Parameter…) the attribute column CONC to the FEFLOW parameter Process Variables > Mass transport > Mass concentration by defining the link. Choose Inverse Distance as the Data Regionalization method and set 4 Neighbors and Exponent of 2.

Click OK.

Double-click on CONC -> Mass concentration in the Maps panel and assign the values by clicking on Assign.

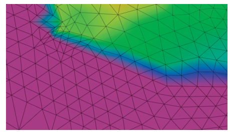
Clear selection.

exercise_fri15.fem

7.3 Horizontal Refinement

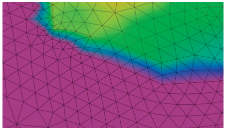
Transport models typically require a finer discretization than flow models. For this reason, the mesh will be horizontally refined around the contamination sources.

First, the area to be refined is selected as a nodal selection. Increase the Snap distance to 200 m in the Snap-Distance toolbar. Choose Select by Map Polygon in the Selection toolbar and click on Select by All Map Geometries. This will select all nodes within and up to a distance of 200 m around the contamination sources. Hit the Refine button in the Mesh-Geometry toolbar once.



At the transition between the refined and non-refined area, the elements are now quite irregularly shaped (large angles). To improve the mesh quality after manual refinement, the mesh will be smoothed in this area. Make sure that the Add to Selection option in the Selection toolbar is active. Define a Snap distance of 500 m and click Select by All Map Geometries again in order to create a selection covering the refinement zone and its adjancent area. Press the Smooth Mesh button to perform the smoothing.

11_mass_transport_I.mp4



Clear selection.

exercise_fri16.fem

7.4 Boundary Conditions

Northern and southern boundaries

Any water entering the domain through the northern or southern boundary is fresh water with a concentration of 0 mg/l. Therefore a fixed concentration of 0 mg/l is assigned as a boundary condition at these locations.

Go to the Selections panel and open the context menu of the previously stored node selection Northern Boundary. Choose Add to current selection.

Repeat this step with the node selection Southern Boundary.

In the 3D view all nodes along both borders are shown as selected. Click on Domain in the Entities panel.

Double-click on Boundary Conditions > Mass Transport > Mass-Concentration BC in the Data panel. In the input box of the Editor toolbar, ensure that the Assign Values mode is active, input a value of 0 mg/l and click Assign.

Blue circles indicate that first kind boundary conditions are set, similar to the flow boundary conditions.

Constraints

As stated before, water entering the model at the northern or southern border is fresh water. Depending on the hydraulic head distribution, however, at these boundaries also outflow is possible. In this case, a free outflow of contaminated water is to be preferred over applying a fixed concentration.

This requires a dynamic change of the mass transport boundary condition depending on the flow direction, which can be implemented by applying a constraint.

A constraint in our case is used to limit the mass flow at a 1st kind boundary condition (fixed concentration) to a minimum or maximum value. For this exercise, the constraint is set to limit the mass flow to a minimum value of 0 g/d, applying the concentration boundary condition only for inflowing water.

The constraints are technically applied in the same way as boundary conditions. However, for the sake of clarity, the constraints are not shown by default in the Data panel and have to be added first:

In the Data panel, open the context menu of Boundary Conditions > Mass transport > Mass-Concentration BC and choose Add Constraint > Min. mass-flow constraint.

Expand the tree view and activate (double-click) Min. mass-flow constraint and assign 0 g/d to all the selected nodes with <Enter>. The minimum constraint is indicated by a bar below the associated boundary condition symbol.

Clear selection.

Wells

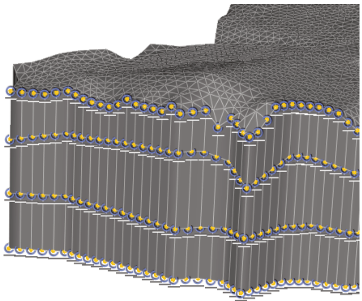
Both wells shall be operated by time-varying pumping rates.

Go to the menu Edit > Time Series... and use the Import... button to load from the pre-defined file well_rates.pow. Choose this file in the file selector dialog, confirm with the Open button and OK to all. Notice the gap in the time series 1: After 5200 days the western well stops pumping. Hit OK for applying the time-series and closing the dialog.

Go to the Maps panel and edit the parameter link <multiple> -> Multilayer Well for the map wells.shp. by Edit Link... Click on the link for the Rate and choose Remove Link in the Parameter Association dialog. Establish a new link between TS_ID and Rate and set the Link Type to Time-varying (time series id). Activate the link by double-clicking on <multiple> -> Multilayer Well. Then click on Select All in the Selection toolbar.

and Assign in the Editor toolbar.

Clear selection.



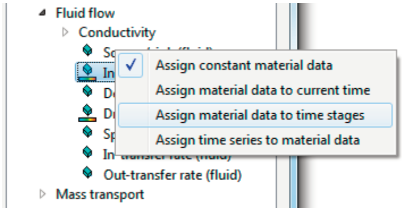
exercise_fri17.fem

7.5 Material properties

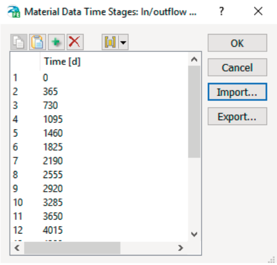
As annual rainfall data show a significant variability during the simulated period, groundwater recharge is assumed to be time-varying in the model. The file recharge_annual.shp contains the spatial distribution of the approximated recharge for annual periods each in a separate attribute field.

Go to the Maps panel, load the map recharge_annual.shp and choose Link to Parameter... from its context menu.

In the Parameter Association dialog, browse to Material Properties > Fluid Flow > In/Outflow on top/bottom on the right and open the context menu with a right-click. Choose the option Assign Material Data to Time Stages. The upcoming dialog lets you define the time stages for which time-varying recharge data shall be assigned (for time steps between these time stages, the recharge is temporally interpolated during the simulation).



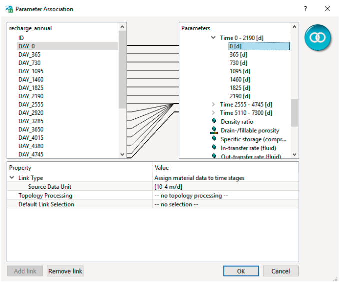
The list has to contain the same values as given in the attribute fields of the map (0;365;730;...). Instead of populating the list manually, the Import...



This will fill the list with annual time stage intervals up until 7300 days. Click Bu OK to close the dialog.

The links between the attribute fields and the time stages is done the same way as for constant model properties. The process, however, can be accelerated by creating a multiple selection of all attribute fields (DAY_0 ... DAY_7300) before creating a link with the 0 [d] time stage. In this case, FEFLOW will automatically create links between all subsequent attribute fields and time stages (see figure).

Click Bu OK to close the dialog.



Go to the Slice view and browse to Slice 1.

In the Maps panel, double-click on the link entry recharge_annual > Linked Attributes > <multiple> -> In/Outflow on top/bottom and Select All elements. Finally, set the Snap distance to 0 m and assign the values by clicking on Assign.

The values for the time stages have now been imported. When looking at In/outflow on Top/Bottom in the Data panel, notice that a tilde symbol marks the material property as time varying.

Clear Selection.

Visualize the different recharge values for time stages by right-clicking on In/Outflow on Top/Bottom in the View Components panel and choosing one of the entries in Material Time in the context menu.

exercise_fri18.fem

To simplify the data input of the remaining parameters, the material parameters effective for mass transport processes (porosity as well as longitudinal and transversal dispersivity) are assumed to be homogeneous throughout the model.

Go to the 3D view and activate (double click) Material Properties > Mass transport > Porosity in the Data panel.

Right-click on Material Properties > Mass Transport in the Data panel and choose Assign Multiple... from the context menu. Afterwards,

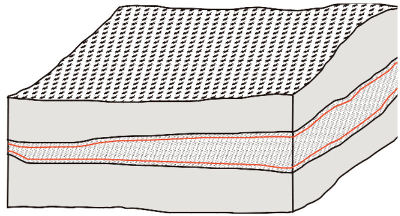
- enter 0.2 for Porosity
- enter 70 m for Longitudinal dispersivity (mass) and
- enter 7 m for Transverse dispersivity (mass)
- Deactivate all other entries and click the OK button to finalize the assignment

exercise_fri19.fem

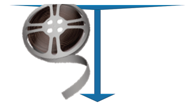
7.6 Vertical resolution

To ensure a correct representation of low flow velocities in the aquitard as a basis for transport simulation, this model layer has to be further subdivided.

The best choice to minimize errors due to the nodal nature of the calculated velocity field is to apply thin layers on top and bottom of the aquitard.



The two additional slices are placed within the aquitard with a distance of 10 cm to the aquitard top and bottom.



12_mass_transport_II.mp4

Go to Edit > 3D Layer Configuration.

Select Slice 3. Click on Insert Slice(s) Above..., type a value of 0.1 m in the Distance between slices input box and click OK.

Select Slice 2. Click on Insert Slice(s) Below..., type a value of 0.1 m in the Distance between slices input box and click OK.

The aquitard has now been divided into three layers. To ensure that the data are transferred correctly from the old to the new slices and layers, have a look at the Data Flow lists on the right of the 3D Layer Configurator.

There are two lists that provide control over the data flow between the previous and the new slices and layers. The upper control called Data flow for slices describes the data flow of the process variables, boundary conditions, and nodal selections from the old slices to the new ones. The old slices are shown as number buttons in the left column, the new ones in the right column. The data flow is symbolized by lines connecting the old with the new

Algebraic signs are handled differently for Constraint and Boundary Conditions: For Constraints inflows are positive (+), outflows are negative (-). For Boundary Conditions inflows are negative (-), outflows are positive (+).

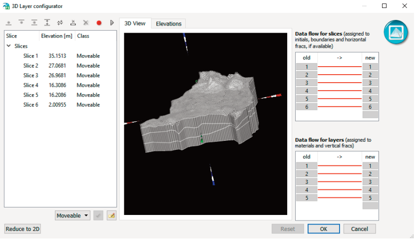
slices. The lower list, Data flow for layers describes the data flow for all material data and for elemental selections.

FEFLOW suggests to transfer the model properties from old slice 2 to slices 2–4. In order to change this, double click on the number box showing 5 in the new column. Change this entry to 123 4–5.

As a result, the link from old slice 2 points to new slices 2-3 and from old slice 3 to new slices 4-5.

The data flow in the lower list for the material properties describes the same data characteristics from the old center layer (aquitard) to the new layers 2, 3 and 4. No changes are necessary.

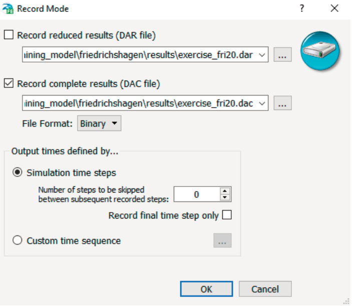
Click on Bu OK to exit the 3D Layer Configurator and to apply the changes to the model.



exercise_fri20.fem

7.7 Simulation Run

If working with a licensed version of FEFLOW, the results can be saved to a file during the simulation run. Click on Record in the Simulator toolbar. Activate Save complete results (DAC file). By default, the results file (*.dac) is saved with the same name as the current model in a subdirectory results. To avoid overwriting the prepared file, define another name. Confirm by clicking Bu OK.

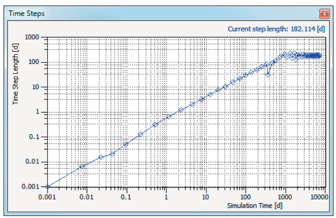


If running FEFLOW in demo mode, it is not possible to save the results to a file, and only the prepared file (exercise_fri20.fem) can be run.

To run the model, click Start in the Simulator toolbar. The simulation takes approximately 5 minutes on a system with an Intel i7 processor.

The current simulation time is displayed in the dropdown box of the Simulator toolbar.

In the Time-Steps chart (which can be opened via View > Charts if not already shown) the actual time step length versus the total simulation time is plotted. The mostly constant conditions lead to a steadily increasing time step length, with a reduced time-step whenever a change of the groundwater recharge occurs.



The simulation stops after 7,300 days (the final time that has been set in the Problem Settings dialog before).

Click Stop to exit simulation mode.

7.8 Postprocessing

Load the recorded results file.

exercise_fri20.dac

Scene Library

Go to the Scene Library panel and use the context menu of the previously stored exercise_fri11:1[*] -Slice 1 item. Create View from Scene. Using the Simulator toolbar, browse through the saved time steps of the simulation file with the Previous Step and Next Step buttons, either step by step by single click or quickly by pressing, holding and releasing the left mouse button. Due to the significant deviation of the transient results from the steady-state flow solution, many of the Error Bars are shown in red, indicating that the differential between calculated and observed values exceeds the defined confidence intervals. Finally, move to the last time step at 7300 [d] by using the Last Step button and close the active Slice View.

Slice View

Go to the remaining Slice View and double click on Process Variables > Mass Transport > Mass Concentration in the Data panel.

Clear selection if necessary.

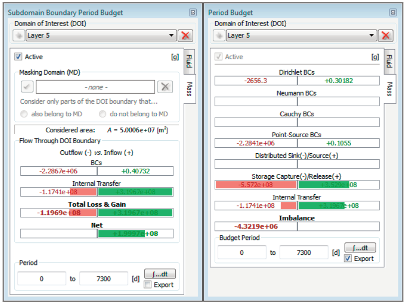
The spatial distribution of the concentration at the final simulation time is shown.

Use the <Pg Down> key to check the result for the lower aquifer (Slice 5 and Slice 6).

Subdomain Budget

To obtain the time-integrated contaminant mass budget for the lower aquifer, some additional steps are necessary:

- Choose Select Elements and Select All.
- From the context menu of the Slice view, choose Store Current Selection and name the selection 1²³ Layer 5.
- Clear selection.
- Activate the stored selection Layer 5 in the Selections panel by a single click.
- Go to the Subdomain Boundary Period Budget panel.
- Click on Add... for adding Layer 5 as Domain of Interest and switch to the tab Mass.
- Check Active and click on ∫...dt to calculate the budgets for all time steps and accumulate them into the period budget.



The budget shows inflows in green, outflows in red. It separates into boundary condition types and the internal transfer, which is in this case all contaminant mass crossing the border to the overlying aquitard. The Net value sums the positive and negative values of Total loss and gain.

Imbalance

- Click on the stored selection Layer 5 in the Selections panel.
- Go to the Subdomain Boundary Period Budget panel.
- Click on Add... for adding Layer 5 as Domain of Interest and switch to the tab Mass.
- Check both the Active and the Export box and click on Bu ...dt.

Accept the proposed name of the ASCII file output and proceed by hitting Bu Save.

Summing the mass amount of all boundaries, sources and sinks, storage losses and gains and internal transfers; the Imbalance shows the numerical error of the mass transport for the specified subdomain over the entire simulation period.

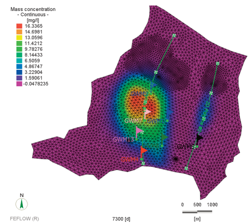
Cross Sections

Process variables and material properties can be visualized in cross-section views. In our case, we are interested in the vertical distribution of the contamination along the plumes.

Cross-section display is based on lines as entities. Switch to the Slice view and choose Draw a Surface (2D) Line from the Drawing toolbar.

Click on the sewage fields (the contamination source in the north-west) to define the starting point of the line here. The line is extended by adding points with a single mouse click. Follow the flow path to the western well and further to the lake Müggelsee. Finish the line with a double click.

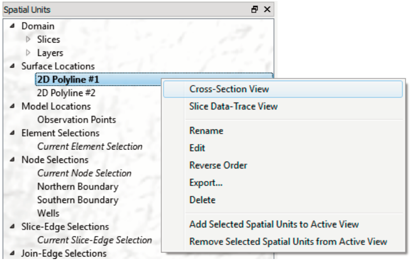
Repeat these steps for the waste dump (the eastern contamination source). The final result should look similar to the image.



In the Spatial Units panel, two new entries Surface Locations > 2D Polyline #1 / #2 have been added.

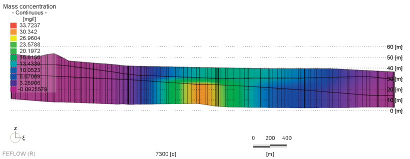
Open the context menu of 2D Polyline #1 and choose Cross-Section View.

13_mass_transport_III.mp4



A cross-section view showing the depth-related concentration along the cross-section is opened.

In the Navigation panel, go to the Projection tab and push up the lever to exaggerate the z-axis.



Isosurfaces

Go to the 3D view. Make sure that Domain is selected in the Entities panel. In the Data panel, double-click on Mass concentration to show this parameter in the view. In the View Components panel, uncheck Faces and Mass concentration > Continuous. Check Mass concentration > Isosurfaces > Domain. Instead, one isosurface is shown.

To edit the isosurface visualization properties, double-click on Isosurfaces. The Properties panel comes to front. Switch to the Custom mode and click on Edit. Specify two values, 10 mg/l and 20 mg/l. Close the dialog by clicking OK, and click Apply in the Properties panel. The isosurface visualization is changed to reflect the newly set concentrations.

Breakthrough Curves

Open from the menu View > Charts > Local Concentration History. The diagram shown contains the concentrations calculated at the different observation points during the simulation time.

More Information

This completes the dintroductory tutorial, that gives an overview of the basic functionality and workflows of FEFLOW.

Additional tutorials, application examples and more detailed descriptions of the program features are provided by the FEFLOW help .

For more information, including extensions, tutorial videos, user forum and more, please visit

www.feflow.com