

**MIKE Powered by DHI Software**

# **FEFLOW<sup>®</sup> 7.2**

Finite Element Subsurface Flow  
& Transport Simulation System

## **Introductory Tutorial**



Copyright notice:

No part of this manual may be photocopied, reproduced, or translated without written permission of the developer and distributor DHI.

Copyright © 2018 DHI – all rights reserved.

## **DHI**

Agern Alle 5, 2970 Horsholm, Denmark

Phone: +45-45-16 92 00,

E-Mail: [mike@dhigroup.com](mailto:mike@dhigroup.com)

Internet: [www.mikepoweredbydhi.com](http://www.mikepoweredbydhi.com)



# Contents

<b>1</b>	<b>Introduction</b>	<b>4</b>	<b>7</b>	<b>Flow and Transport Model</b>	<b>33</b>
1.1	About FEFLOW	4	7.1	Problem settings	33
1.2	Scope and Structure	5	7.2	Initial conditions	35
1.3	Terms and Notations	5	7.3	Horizontal Refinement	36
1.4	Requirements	6	7.4	Boundary Conditions	36
1.5	Model Scenario	6	7.5	Material properties	38
			7.6	Vertical resolution	40
<b>2</b>	<b>Getting Started</b>	<b>7</b>	7.7	Simulation Run	41
2.1	Starting FEFLOW	7	7.8	Postprocessing	42
2.2	FEFLOW 7.2 User Interface	7		<b>More Information</b>	<b>46</b>
<b>3</b>	<b>Geometry</b>	<b>9</b>			
3.1	Maps and Model Bounds	9			
3.2	Supermesh	10			
3.3	Finite element mesh	11			
3.4	Expansion to 3D	13			
<b>4</b>	<b>Problem settings</b>	<b>18</b>			
<b>5</b>	<b>Model Parameters</b>	<b>20</b>			
5.2	Boundary conditions	20			
5.3	Material properties	26			
<b>6</b>	<b>Simulation</b>	<b>30</b>			

# 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 plugins 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, government 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>.

## Introductory Tutorial

### 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 philosophy 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



 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



*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.*



*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](http://www.feflow.com).*

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.

## 1.4 Requirements

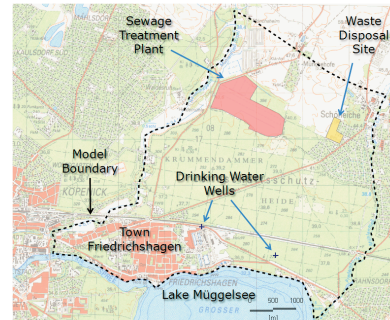
If not already done, please install the FEFLOW software including the demo data package. A license is not necessary to run this tutorial (FEFLOW can be run in demo mode).

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

## 1.5 Model Scenario

A fictitious contaminant has been detected near the small town of Friedrichshagen, in the south-east 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.

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 a northwest-southeast groundwater level contour line north of the two potential contamination sources.



## Introductory Tutorial

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 7.2 via the corresponding desktop icon or the startup menu entry.



#### On Linux Systems

- Type `feflow72q` 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 7.2 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  **View** > **Toolbars** and  **View** > **Panels**, respectively. Please keep in



1\_maps.mp4



The exercise workflows are also available as video screen-casts 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.

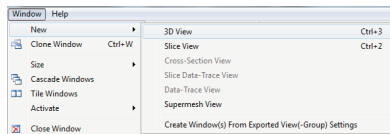



In this exercise, different file types are used as data source at the different stages of modelling to show the number of options. In practical projects, it may be preferred to store basic data in one file type, e.g., \*.shp when using GIS.

mind that not all panels and toolbars are displayed by default. Thus this exercise 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.

## Introductory Tutorial

### 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 7.2.
```


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:

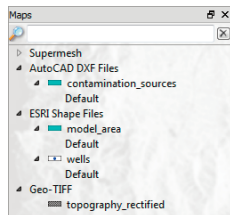
- **topography\_rectified.tif** (a georeferenced raster image of the model area for better orientation)
- **model\_area.shp** (a polygon map that contains several polygons denoting the outer model boundary and embedded contamination areas)
- **contamination\_sources.dxf** (the footprint of the sewage 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.




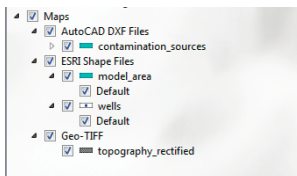
2\_supermesh.mp4



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:



## Introductory Tutorial

- 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 in the 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.

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.



*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.*

## 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.

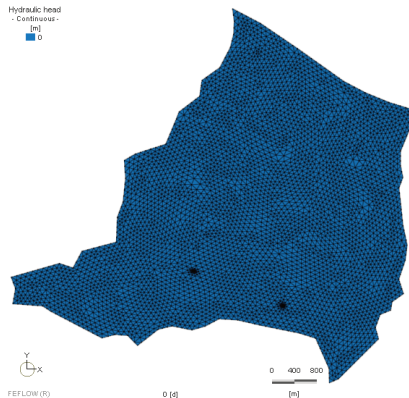
Click on  **Generator Properties** in the  **Meshing** panel if the generator properties are not yet shown.

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**



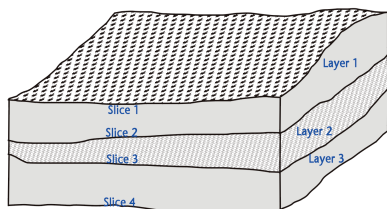
## Introductory Tutorial

### 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.

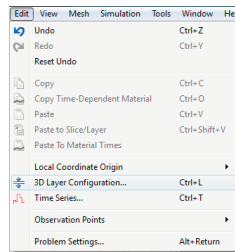


When using a layered approach, FEFLOW distinguishes between layers and slices in 3D. Layers are three-dimensional bodies that typically represent geological formations like aquifers and aquitards. The interfaces between layers, as well as the top and bottom model boundaries, are called slices.

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**.



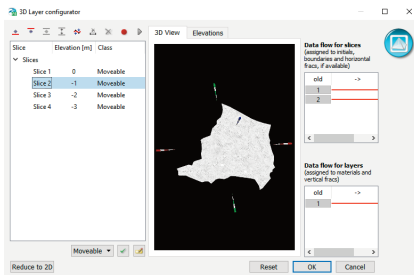
On the left side of the dialog, the current slices are shown (*Slice 1* and *Slice 2*), corresponding to one single layer with the default top and bottom elevations of 0 m, 1 m resp. We add two more slices



3\_3D\_geometry.mp4

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.



Click on **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.

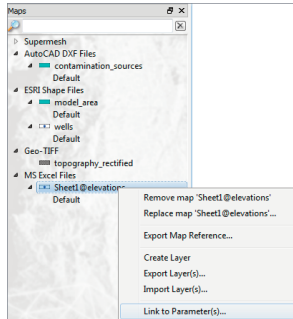
	A	B	C	D
1	<b>X</b>	<b>Y</b>	<b>Ele</b>	<b>Slice</b>
2	3403999.50	5818015.00	39.50	1
3	3403999.50	5818405.50	39.50	1
4	3403999.50	5819254.00	39.50	1
5	3403999.75	5814206.00	34.50	1
6	3403999.75	5813779.50	37.00	1
7	3403999.75	5813957.50	37.00	1

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.


## Introductory Tutorial

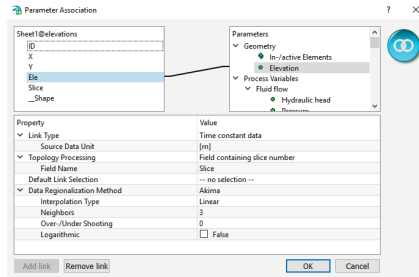
As a next step, the attribute values of the data file need to be associated with (linked to) their respective FEFLOW parameter, in this case with the elevation. In order to do this, open the context menu of the map *elevations* with a right click and choose  **Link to Parameter(s)...**



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  **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.

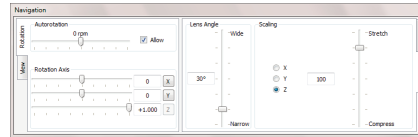
Click on **OK** to apply these settings and to close the dialog.

## 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:

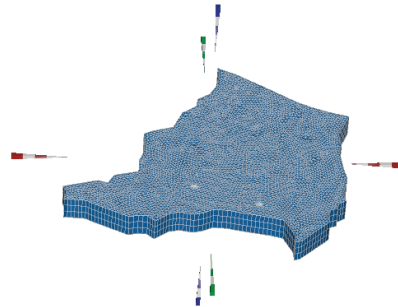
- Left mouse button: rotate the model around its center of gravity
- Center mouse button (mouse wheel): pan the model
- Right mouse button: zoom (in/out)

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**).

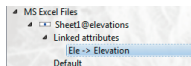
Click on the tab **Projection** and move the **Scaling** slider bar upwards until you have achieved a convenient view on the 3D model. Alternatively, the **<Shift>** key in combination with the mouse wheel can be used to change the stretch factor for the active **3D** view.





## Introductory Tutorial





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.

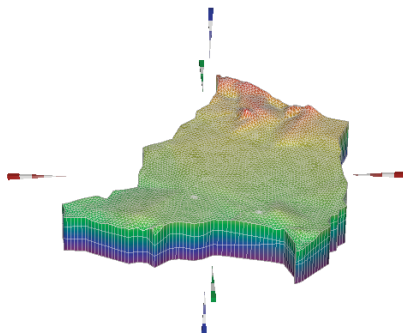


- Click on  **Select All** in the  **Selection** toolbar.
- Click  **Assign** in the  **Editor** toolbar to apply the new elevation data to all nodes.

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

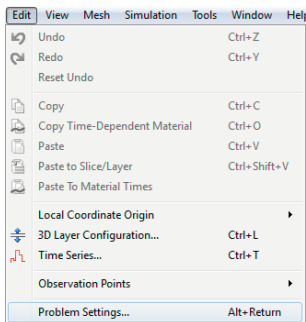
FEFLOW provides the means to simulate a number of different physical processes in different spatial and temporal dimensions, ranging from simple 2D steady-state flow models to transient, unsaturated, density-coupled reactive transport models. As the input parameters depend on the model



4\_problem\_settings.mp4

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.

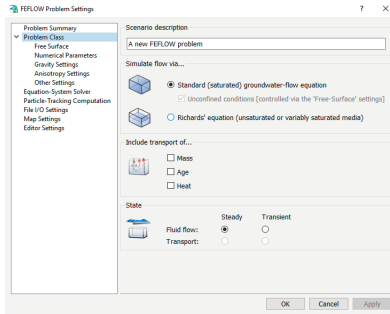
Keep the default setting (**Standard groundwater-flow equation**) for this exercise.

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.



# Introductory Tutorial



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

Click on **Bu 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\_flow\_bcs.mp4

## 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*
- *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 1<sup>st</sup> 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 (hydraulic 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<sup>3</sup>/d and 1,000 m<sup>3</sup>/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.



## Introductory Tutorial

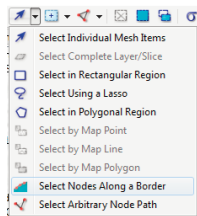
The recommended tool for navigation in the *Slice* view is the  **Pan** tool in the  **View** toolbar. The mouse buttons are associated with the following functions:

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

### Northern Boundary

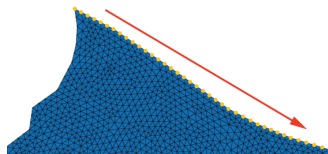
Zoom to the northern boundary.


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



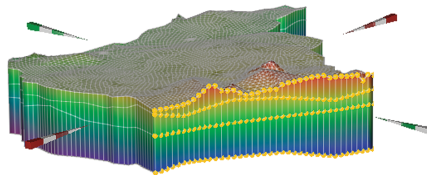
Having this tool selected, click on the westernmost node of the boundary and hold down the left mouse button, move the mouse cursor to the easternmost node and release the button. The nodes

of the northern border are highlighted as yellow points. The selection is shown in the *3D* view simultaneously.

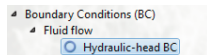


Next, the selection is extended to the other three slices of the model. A time-saving way to do this is the application of the  **Copy Selection to Layers/Slices** tool. Start the tool and select all slices in the upcoming dialog (manually or by hitting `<Strg>-<A>`). Click on .

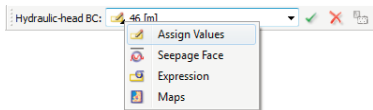
Bring up the *3D* view and ensure that indeed all nodes at the northern boundary are selected.



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**.

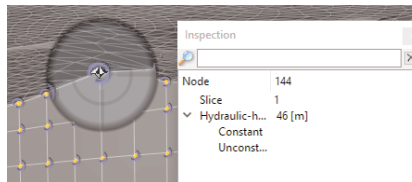


Enter **46 m** in the input box of the **Editor** toolbar and click on the **Assign** button.



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.

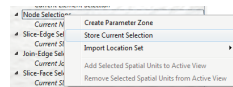
The values of the boundary condition can be checked using the inspection tool, which is activated by clicking on **Inspect nodal/elemental values** in the **Inspection** toolbar. Move the hair-cross to a node with a boundary condition. The values of all properties currently visible in the active view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

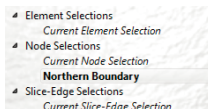
After defining the boundary condition at the northern boundary, we store the current node selection for later use:

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





## Introductory Tutorial

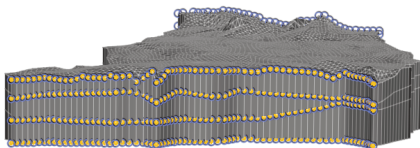
In the *Node Selection* dialog, change the name to  **Northern Boundary** and click  OK. The stored selection appears in the  *Selections* panel. Click on  *Clear selection*.










### 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.



6\_multilayer\_wells.mp4

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.

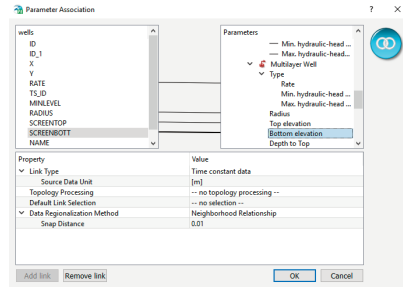
The *wells* map contains attribute data that need to be associated with (linked to) their respective FEFLOW parameter. In order to do this, open the context menu of the map *wells* (in the *Maps* panel) with a right click and choose *Link to Parameter...* to open the *Parameter Association* dialog.

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 **Bu** *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

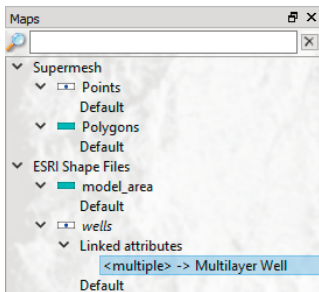
- **RADIUS** (link to *Radius*)
- **SCREENTOP** (link to *Top elevation*)
- **SCREENBOTT** (link to *Bottom Elevation*)
- **NAME** (link to *Name*).






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  $10^{-2}$  **0.01** meters in the input field. Click **Bu** *OK* to close the dialog.

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

## Introductory Tutorial



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





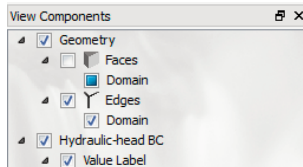
- Click on  **Select All** in the  **Selection** toolbar.
- Click on  **Assign** in the  **Editor** toolbar. Even though all edges in the model have been selected, the multilayer wells will be assigned each to the closest edges only.  **Clear** the selection.

  **exercise\_fri6.fem**

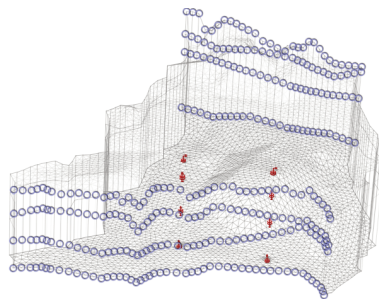
Finally check all boundary conditions you have set.

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**



7\_material\_properties\_1.mp4

## 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** toolbar and 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<sub>xx</sub>**. **K<sub>yy</sub>** and **K<sub>zz</sub>** will be calculated from **K<sub>xx</sub>** 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<sub>xx</sub>** in the **Data** panel.

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<sub>xx</sub>**.



## Introductory Tutorial



- 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  **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  **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** >  $K_{yy}$ .

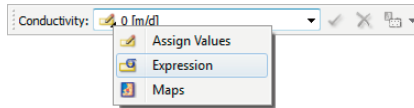


8\_material\_properties\_1l.mp4


- Choose  **Paste...** from the context menu of **Material Properties > Fluid Flow > Conductivity > K<sub>yy</sub>**.

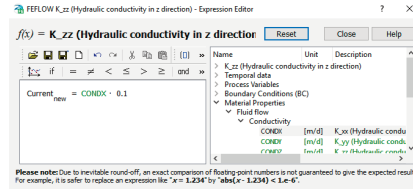
Besides manual assignment and data import, FEFLOW allows to calculate model properties from mathematical expressions. This will be used for the vertical conductivity  $K_{zz}$ , which is assumed to be 10% of the lateral conductivity  $K_{xx}$ .

- Double-click on **Material Properties > Fluid Flow > Conductivity > K<sub>zz</sub>**.
- 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 expression 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  **Close**.



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



 **Clear selection**.

  **exercise\_fri10.fem**




## Introductory Tutorial


### 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 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  **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.



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.

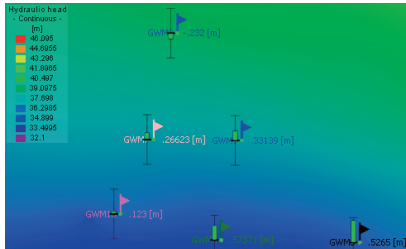
During and after the simulation, all visualization tools in FEFLOW can be used to monitor and post-process the simulation results.

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 > Continuous**. 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

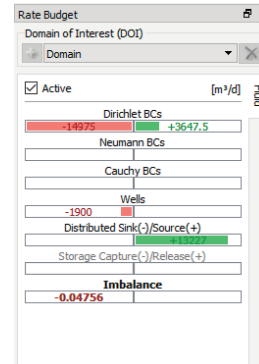
In the **Slice View**, chose **Create Scene from Window**. This stores a copy of the current view and all its settings in the **Scene Library** panel, in order to use it later for restoring the view.

**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.



## Introductory Tutorial

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.




### 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.

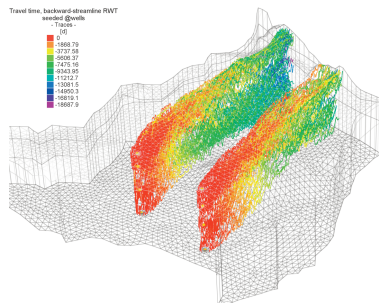
In the now opened  **Properties** panel, enter **100** m as the **Radius**, press **<Enter>** and click  **Apply**. Still in the same panel, right-click on the color scale on the left and choose  **Presets... > Rainbow**.



11\_mass\_transport\_1.mp4


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*.

### 7.2 Problem settings

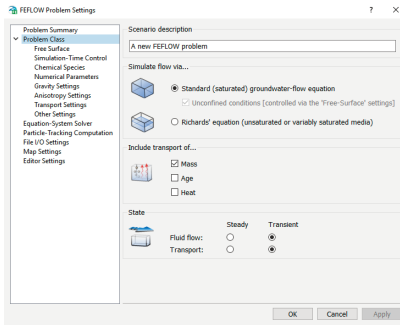
#### Problem class

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.

# Introductory Tutorial

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**.



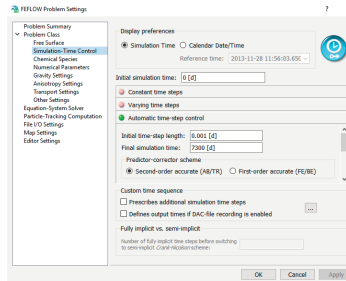
## Time stepping

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 **123 7300** days in the **Final Time** input box.


Click on **Bu Apply** and **Bu 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 aquifer. 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**

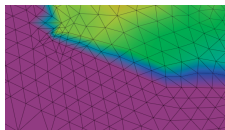


## Introductory Tutorial

### 7.3 Horizontal Refinement

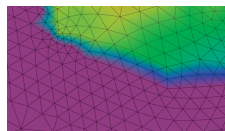
Transport models typically require a finer discretization than flow models. For this reason, the mesh will be horizontally refined around the contamination sources.

First, the area to be refined is selected as a nodal selection. Increase the **Snap** distance to  $10^3$  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  $10^3$  500 m and click **Select by All Map Geometries** again in order

to create a selection covering the refinement zone and its adjacent area. Press the **Smooth Mesh** button to perform the smoothing.



**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**.



[12\\_mass\\_transport\\_II.mp4](#)

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 predefined 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 pump-



*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 (+).*

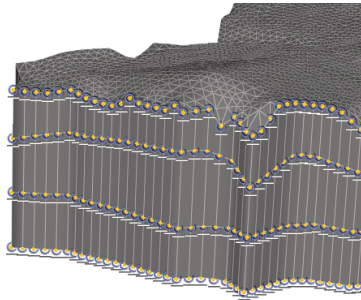
## Introductory Tutorial

ing. Hit **Bu** **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 clicking **Edit Link...** Click on the link for the **Rate** and choose **Bu** **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.**



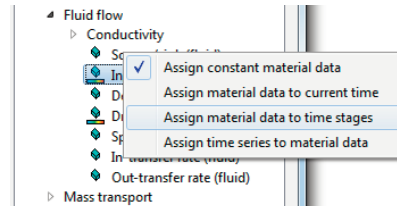
  **exercise\_fri17.fem**

## 7.5 Material properties

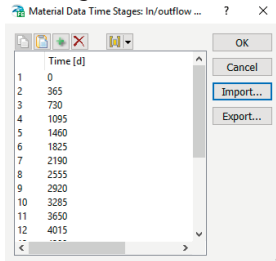
As annual rainfall data show a significant variability during the simulated period, groundwater recharge is assumed to be time-varying in the model. The file **recharge\_annual.shp** contains the spatial distribution of the approximated recharge for annual periods each in a separate attribute field.

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).



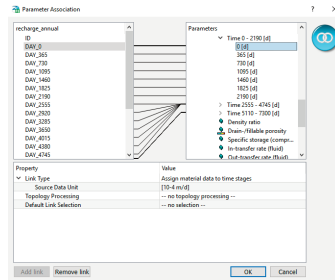
The list has to contain the same values as given in the attribute fields of the map (0;365;730;...). Instead of populating the list manually, the **Import...** button lets you get those time stages automatically from the pre-defined file **recharge\_time\_stages.pow**. Choose this file in the file selector dialog and confirm with the **Open**



This will fill the list with annual time stage intervals up until 7300 days. Click **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 **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 **12<sup>3</sup> 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**

## Introductory Tutorial

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,

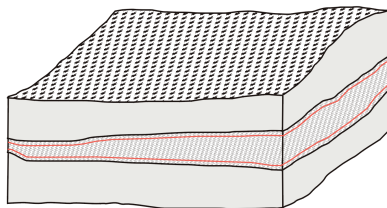
- enter  **0.2** for **Porosity**
- enter  **70** m for **Longitudinal dispersivity (mass)** and
- enter  **7** m for **Transverse dispersivity (mass)**
- Deactivate all other entries and click the  **OK** button to finalize the assignment

  **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.

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**.

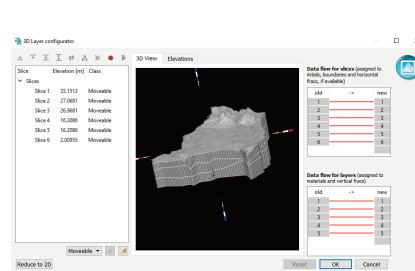
Select **Slice 2**. Click on **Insert Slice(s) Below...**, type a value of  **0.1** m in the **Distance between slices** input box and click  **OK**.

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**.



13\_mass\_transport\_III.mp4

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 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  $1^{23}$  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.

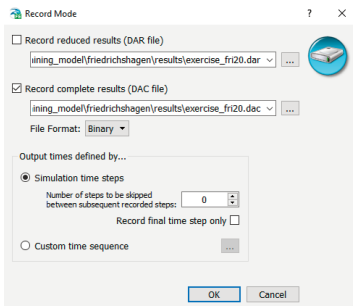
Click on **Bu** **OK** to exit the *3D Layer Configurator* and to apply the changes to the model.

[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**.

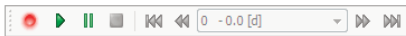
## Introductory Tutorial



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.

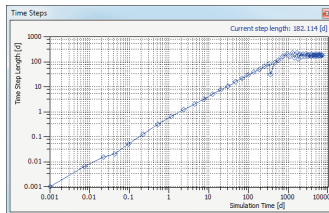
To run the model, click  **Start** in the  **Simulator** toolbar. The simulation takes approximately 5 minutes on a system with an Intel i7 processor.

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



14\_mass\_transport\_simulation.mp4

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).

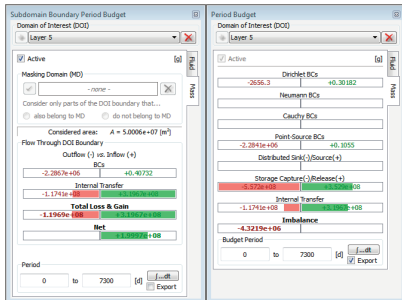
## Subdomain Budget

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  *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.



## Introductory Tutorial



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 *...dt*.

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

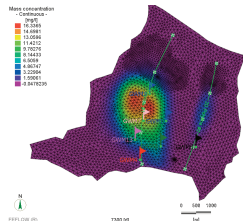
## Cross Sections


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.

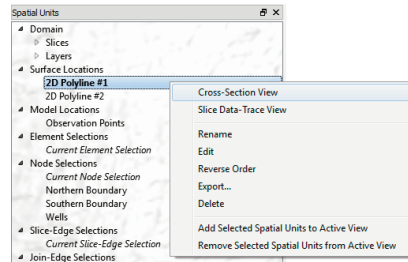
Click on the sewage fields (the contamination source in the north-west) to define the starting point of the line here. The line is extended by adding points with a single mouse click. Follow the flow path to the western well and further to the lake Müggelsee. Finish the line with a double click.

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




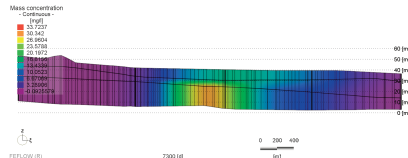
In the  **Spatial Units** panel, two new entries **Surface Locations** > **2D Polyline #1** / **#2** have been added.

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



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.



## Introductory Tutorial

### 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 introductory 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](http://www.feflow.com)