FEFLOW 8.1: Introductory Tutorial

February 29, 2024

DHI A/S

Contents

Preface

0.1 Scope and structure of this manual

This training manual serves as a basis for FEFLOW training courses held by DHI instructors.

It is intended to help you to follow the training more easily, to understand the theoretical background of the program and to be able to repeat the examples at home on your own.

It should neither be considered as an alternative FEFLOW Help System nor as a step-by-step tutorial for personal training.

0.1.1 Terms and notations

This manual has been designed to be as self-explanatory as possible. Still, some notations should be clarified in advance.

At one side of each page you find some white space to take notes related to the contents of the page. The instructor will inform you about helpful details not contained in this manual, and you will probably wish to write down parts of these explanations.

In the manual, you will basically find four different types of content:

- A detailed description of the steps necessary to complete the examples.

- A basic overview of the theoretical or conceptual background of the steps applied.

- Pointers to the most frequent mistakes.

- References to the other parts of the FEFLOW documentation.

0.1.2 Description of the examples

In addition to verbal descriptions of the required screen actions we make 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:

	menu command
	toolbar
	panel
Bu	button
1^{23}	input box for text or numbers
	switch toggle
	radio button
	check box
	context-menu command

At some points in the text you will find file names printed in color. The respective files are located on the USB memory stick (or similar) that comes together with this training manual. The exact path of these files is given in the introduction to each chapter.

You may use these files as a starting point for your studies later (if you do not want to start your exercise at the beginning of the chapter again). Please note that DHI does not take any responsibility for the correctness and usability of these data.

Other files (e.g., GIS data or background images) are printed in the same color.

0.1.3 Theoretical / conceptual notes

> Notes concerning the theoretical background or the basic concepts in FEFLOW can be found in light blue boxes. They offer a short, practically-oriented overview of the theory related to the described steps. For more detailed information please have a look at the corresponding chapters in the FEFLOW documentation.

0.1.4 Pointers to the most frequent mistakes

> The most common mistakes are enclosed in light red boxes.

0.1.5 References

References to other parts of the FEFLOW documentation, such as the Help System or articles of the White Papers are provided in combination with a book sign.

The notation of references is as follows:

WP II 1.3.2

White Papers, Vol. II, Chapter 1.3.2

0.2 Exercise data

Your course material includes a training DVD, a USB memory stick or any other digital form, which contains this training manual and the presentation slides as well as the complete FEFLOW documentation (all PDF format).

All data that you need to create the models discussed in this manual can be found in the folder /model. At the beginning of each exercise, the location of the specific files necessary for the section is indicated.

0.3 Where to find additional information

To get further information, please also refer to other parts of the FEFLOW documentation. The following additional documentation material is available:

0.3.1 The FEFLOW documentation

- License Installation Guideline for Windows

- License Installation Guideline for Linux

- Introductory Tutorial

- Help System: The main core of the FEFLOW documentation is provided via the FEFLOW Help System. The help is available locally (requires user installation) and online. The help gives a detailed description of the user interface. It explains the

essential work steps of model set-up, simulation, post-processing, IFM Programming and Python Interface.

In each chapter, the respective topic is introduced and the relevant FEFLOW tools are presented. The underlying concepts and workflows are described, followed by a tutorial section.

- White Papers Vol. I - V:

A collection of articles about specific modeling approaches and methods used in FEFLOW.

The entire documentation can be found in PDF format in the /documentation folder of the FEFLOW DVD or USB.

0.3.2 The FEFLOW Help system

The Help system provides detailed background information at all stages of modeling. It can be opened from the FEFLOW menu ▭ **Help - Help (Online)** or ▭ **Help - Help (Online)** (requires previous installation).

0.4 Trademark legal notice

All product names, logos, and brands are property of their respective owners. All company, product and service names used in this training manual are for identification purposes only. Use of these names, logos, and brands does not imply endorsement.

1 Part I: Groundwater flow

All files related to this exercise can be found on the training USB memory stick (or similar) in the project folder /models/Friedrichshagen.

1.1 Objective

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 north- east of town. The other possible source is an abandoned waste-disposal site further east. In this exercise, we will focus on building the flow model. In a future section, we will set up the mass-transport part of this model.

1.2 Model Scenario

As mentioned before, a fictitious contaminant has reached the groundwater. As a result, 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 north-east 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 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.

1.3 Geometry

1.3.1 Maps and model bounds

Let us start by creating a new project in FEFLOW clicking on 🖼 *File - New*. From this window, Click on ◉ *2D or layered 3D mesh* and press ⬚Bu *Next*. Pick the option ◉ *Super-mesh import from maps* and click ⬚Bu *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
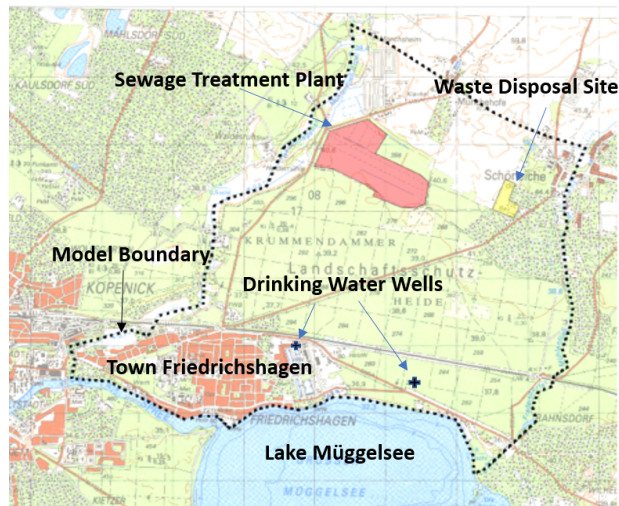
the keyboard) to ensure that FEFLOW uses the bounding box of all the maps to define the initial domain bounds. You may have to select All Maps in the Files of Type selector at the bottom of the dialog.

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 parametrization 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 Bu *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.
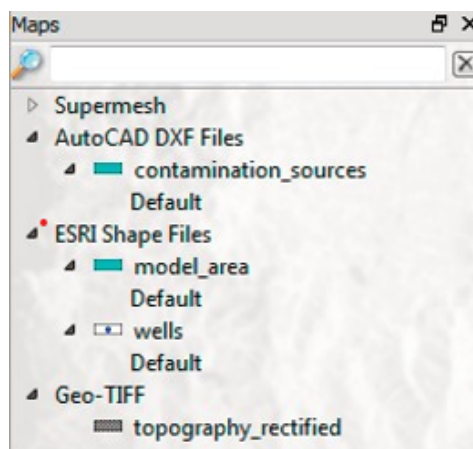

Except for the maps contamination_sources and topography_rectified the maps are ESRI shape files. These vector files occupy their own branch in the tree, each with a default

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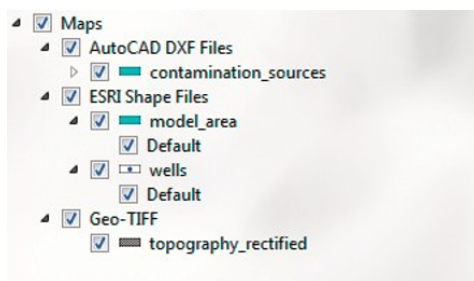
The drawing order of maps can be modified by dragging them with the mouse to another position in the tree (this might become necessary as the model area polygon may overlay the polygons of the mass sources). The topmost map is drawn on top.

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.

1.3.2 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 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.smhx

1.3.3 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.
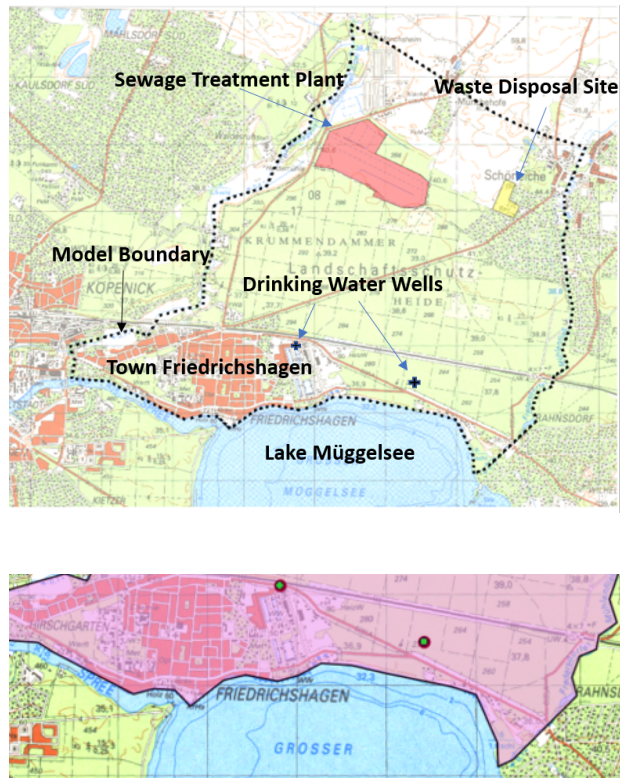
For this example, we will generate the 3D Finite element mesh directly from the super-mesh.

As mentioned before, for this example, we are going to set up a model with three different geological layers. The upper layer corresponds to the upper aquifer. This is limited by the ground surface on top and by an aquitard on bottom (second layer). The third layer corresponds to a second aquifer and it lays above a low permeable unit of unknown thickness. This underlying stratigraphic layer is assumed to be impervious and is not part of the simulation.
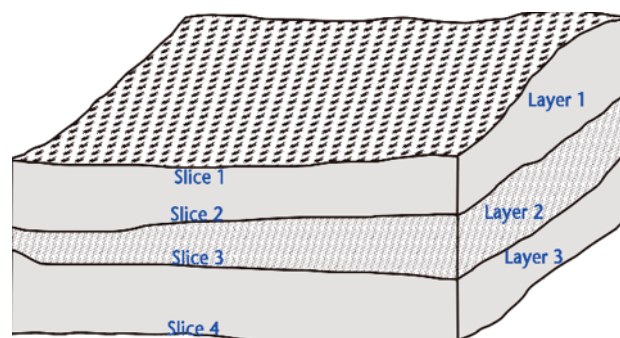
Figure 1.1: Model scenario - map overview

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. For this exercise, we are provided with a file containing information of the respective elevation of each slice a a given coordinates. Following, we are going to take a look at this file structure.
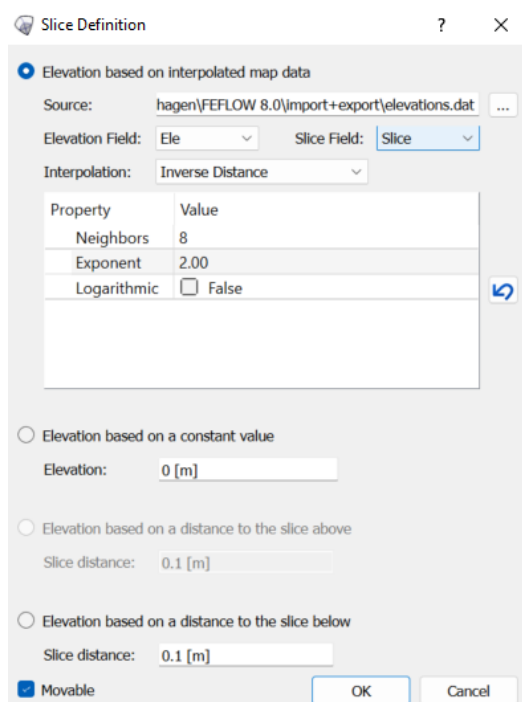
Elevation Data

This file was created from regionalizing elevation data contained in map files. The basic data have been derived from a DEM and from borehole logs, and was combined into an Excel file (*.xlsx and *.dat) 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 A.S.L.

|	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

To set-up the above mentioned layout, click the

⁂ **Insert below** button from the **Extension to 3D Mesh** section in the ▤ **Meshing** panel. A new window will pop up, giving us different options and fields to define the elevation of our model. In the Source field, import elevations.dat and select **Slice** from the Slice field drop down menu. We will use the **Inverse Distance interpolation**. Finally, enter 8 as number of neighbours used in the interpolation and click [Bu] **OK**, as shown below.

Once this step is finished, we can see that only one slice has been added to our supermesh. We now need to create 3 more slices to represent the 3 layered-geological structure. Since the input file elevations.dat contains the information of the elevation of all of our slices, we do now need to load different files every time we create a new slice. Therefore, we just need to click on the icon

2x **Duplicate** and add 3 more layers by typing ₁₂³ **3** as **No. of duplicates**. FEFLOW will automatically copy all the setting from the first slice to the new slices added.
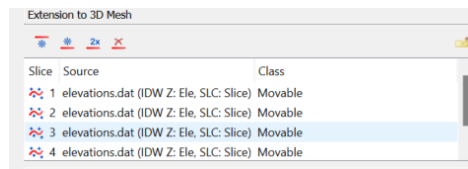
Refinement and mesh generation

Choose the mesh generation algorithm ▾ *Triangle* in the ▤ *Meshing* panel.

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, first check ☑ *Refine points* and set *Point Gradation* to 🔢 **3** and *Point Target size* to a value of 🔢 **20**. These two values define how refined should the point features be (in our case, our wells) and how far should this refinement propagate around the points.

Moreover, as our intention is to use this mesh to model transport of contaminants, we will need to increase the number of elements to avoid converging issues during the simulation as well as unexpected behaviour in our results. To do so, in the section *From Supermesh Elements* in the ▤ *Meshing* panel, click on Supermesh, and enter 🔢 **6000** as *Proposed Elements*. Finally, click on

⚙ *Generate Mesh* to finally create our 3D mesh.

📂 💾 exercise_fri2.smhx
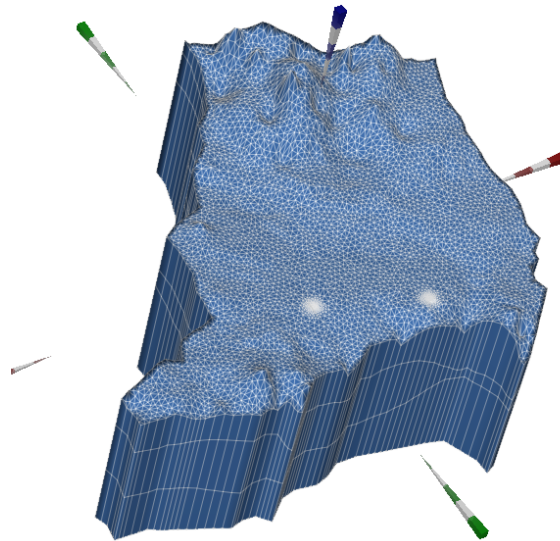
The main advantage of this workflow is that one can always go back to the mesh definition in the Supermesh view. From there, the user has the possibility to load new Shapefiles, change the mesh settings i.e. number of elements or minimum angle of the elements and, re-generate a 3D finite element mesh without the need to define all these properties again.

📂 💾 exercise_fri1.fem

1.4 Problem settings

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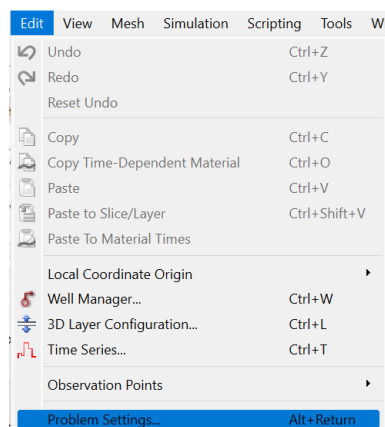
All these settings are organized in thematic pages controlled by a tree view on the left-hand side of the dialog.
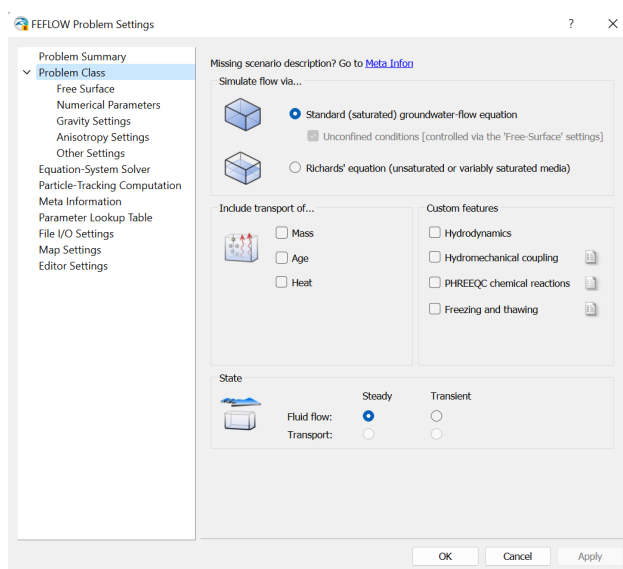

Problem class

The principal type of the FEFLOW model is defined in the **Problem Class** page. Below the Scenario description it's possible to choose from two different problem classes - saturated media and unsaturated/variably saturated media.

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

The second option - Richards' equation (unsaturated or variably saturated media) - would use Richards' equation, 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 defined. 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 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 `Bu` **Apply** to apply the changes

📤 💾 exercise_fri2.fem

1.5 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 a tree view:

- Geometry

- Process Variables

- Boundary Conditions

- Material Properties

- Auxiliary Data

- User Data

- Discrete Features

1.5.1 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**.
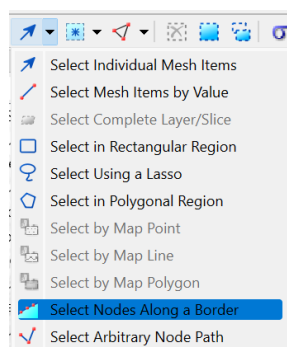
Having this tool selected, click on the westernmost node of the boundary and hold down the left mouse button, move the mouse cursor to the eastern most node and release the button. The nodes of the northern border are highlighted as yellow points. The selection is shown in the 3D view simultaneously.
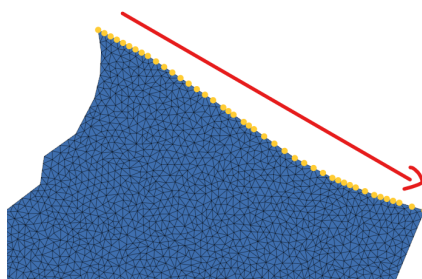

Next, the selection is extended to the other three slices of the model. A time-saving way to do this is the application of the
🔁 **Copy Selection to Slices/Layers** tool. Click on this icon and select all slices in the upcoming dialog (manually or by hitting <Strg>+<A>). Click on ▭Bu▭ **OK**. Bring up 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.
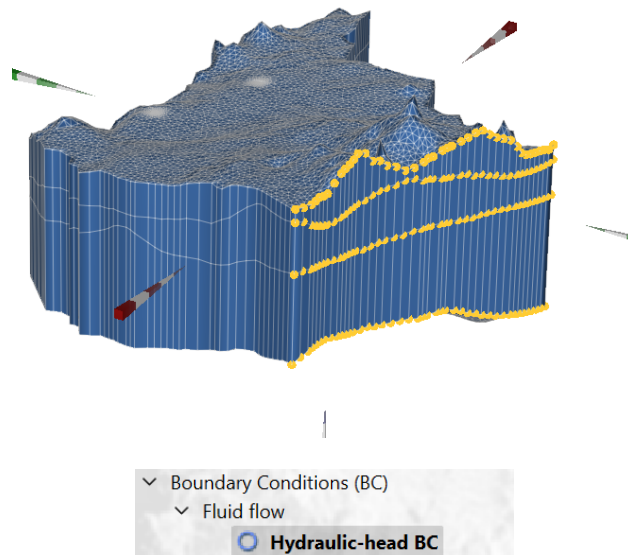
To manually assign a hydraulic-head boundary condition make sure that the **Assign Values** method is active in the ▣▣▣ **Editor** toolbar (see figure). If not, open the context menu by a right click on the symbol in the input box and choose **Assign Values**.
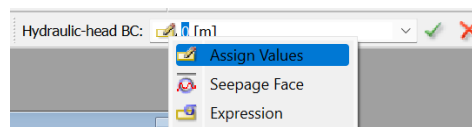

Enter ▣ **46** m in the input box of the ▣▣▣ **Editor** toolbar and click on the ✓ **Assign** button.

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 show the boundary conditions in 3D as well. 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.
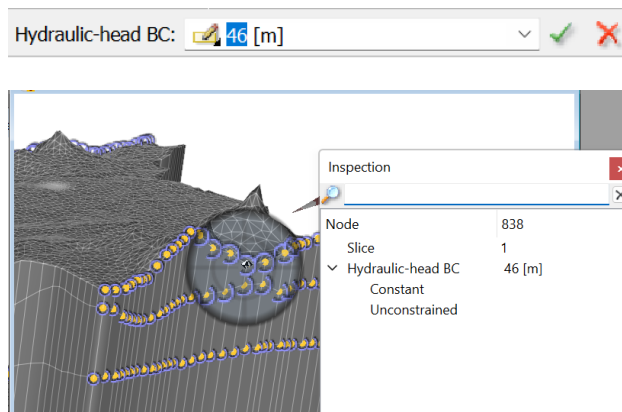
- Make sure that **Hydraulic-head BC** in the 📄 **Data** panel is still active.

- Type ₁₂³ **32.1 m** in the input box of the ▢▢▢ **Editor** toolbar and click on [Bu] **Assign**.

- Store the selection for later use, naming it **Southern Boundary**.

- Click on

  ⊠ **Clear Selection** to clear the selection.

📂 💾 exercise_fri3.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.

We can apply all these properties from the **Well Manager** available in FEFLOW. We can find this window from the 🗔 **Edit - Well Manager...** (or press <CTRL>+<W> on the keyboard). From this window, we can add wells or multilayer wells based on node selections, delete them with one click, or import a whole set of wells from an external file.
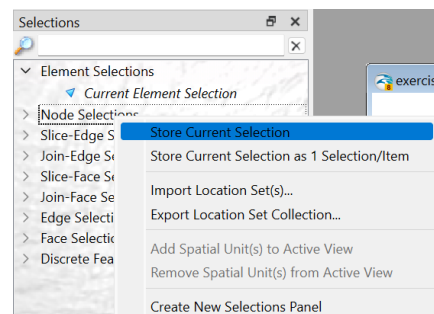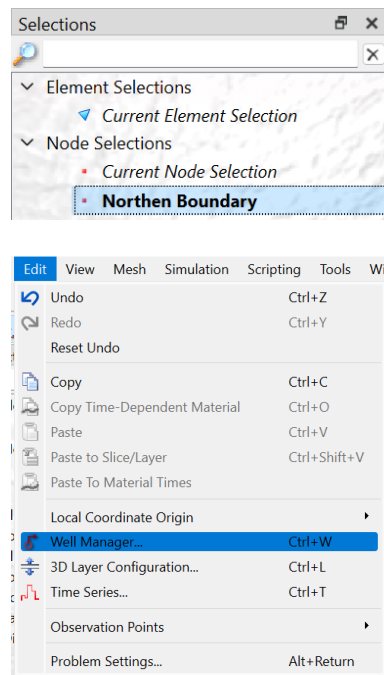
For this exercise, this external file has already been prepared for us. Therefore, we only need to import it from this window. To do so, we click on the [Bu] *Import*. Following that, we need to select the Well manager.dat located in the import+export folder. Once we click on [Bu] *OK*, we can see that all the fields for two different wells type MLW (Multi-Layer Wells) have been already filled for us. At this point, we only need to click on the

[?] *Verify*. This option checks if there is a node in our mesh located at the coordinates specified for each well in the imported file. Moreover, the screen of our well, defined by the Z[top] and Z[Bottom] fields, are also verified to check if there is a node at that specific elevation in our mesh. It is completely normal if those fields change their values slightly, they are just being approximated to the nearest node. This approximation is based on the snap distance which can be found in the menu *Tools - Global Settings - Tool Properties - Well Manager - Nearest Node Distance*, or it can be directly accessed by means of the icon

[≡] *GlobalSettings* in the upper right corner of the *Well Manager* window.

Finally, we can click [Bu] *OK* to close the window.

Now that we have defined all the boundary conditions, let us visualize them all at once. 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 [✓] *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. In the following section we will now concentrate in setting up the material properties of our geological setup.

exercise_fri4.fem
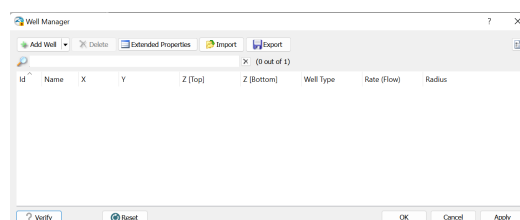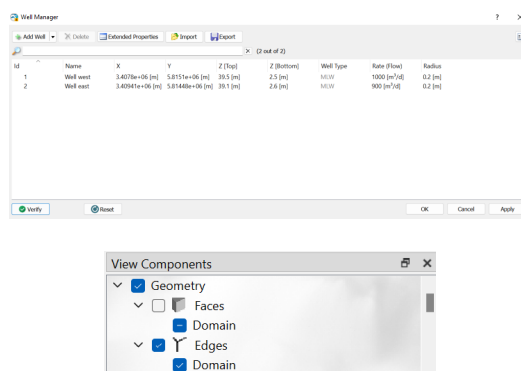
1.5.2 Material Properties

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 ¹²³ **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.
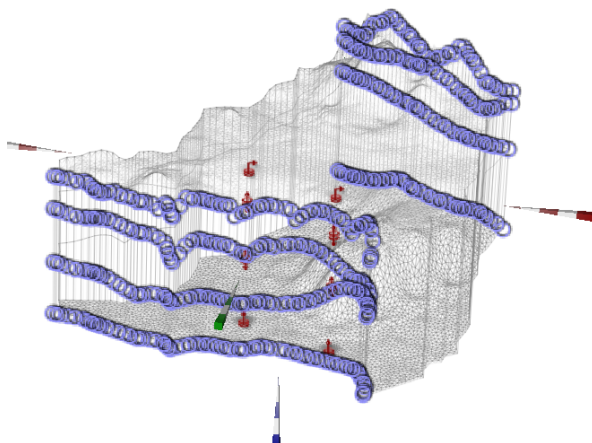
- Input a value of ⌨ **0.1** and hit **<Enter>**.

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

- Add the map conduc2d.trp to the 📑 **Maps** panel by right-clicking on the panel and choosing **Add map(s)**. This file contains point-based conductivity values for the top aquifer in the unit 10^{-4} m/s.

- Right-click on the map and choose **Link to Parameter. . .** .

- Associate the attribute column **Value** to the FEFLOW parameter **Kxx**. Kyy and Kzz will be calculated from Kxx and assigned later.

- Choose 10^{-4} m/s as **Source data unit** in the panel below the link.

- As a regionalization method choose **inverse distance** with 8 Neighbours. Close the dialog with Bu **OK**.

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

- In the 📑 **Maps** panel, double-click on **Linked Attributes - Value - Kxx**.

- To finally assign the conductivity values, click the Bu **Assign** in the ☐☐☐ **Editor** toolbar.

Conductivity: 🔳 conduc2d   ✓ ✗

📂 💾 exercise_fri5.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 using the selection tool
▱ **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 Kxx (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 Kxx.

- Enter 🔢 **0.1** for the Drain-/fillable porosity.

- Uncheck all other properties.

- Click the [Bu] *OK* button to finalize the assignment.

- Clear selection.

📂 💾 exercise_fri6.fem

Anisotropic hydraulic conductivity

While Kxx has been assigned already, Kyy and Kzz will be derived from Kxx. Kyy is equal to Kxx, it will be assigned using a simple copy-paste procedure:

- Select All elements in the 3D view by means of the icon

  🟦 *Select All*.

- Choose *Copy...* from the context menu of *Material Properties - Fluid Flow - Conductivity - Kxx*.

- Double-click on *Material Properties - Fluid Flow - Conductivity - Kyy*.

- Choose *Paste...* from the context menu of *Material Properties - Fluid Flow - Conductivity - Kyy*.

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

- Double-click on *Material Properties - Fluid Flow - Conductivity - Kzz*.

- 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.
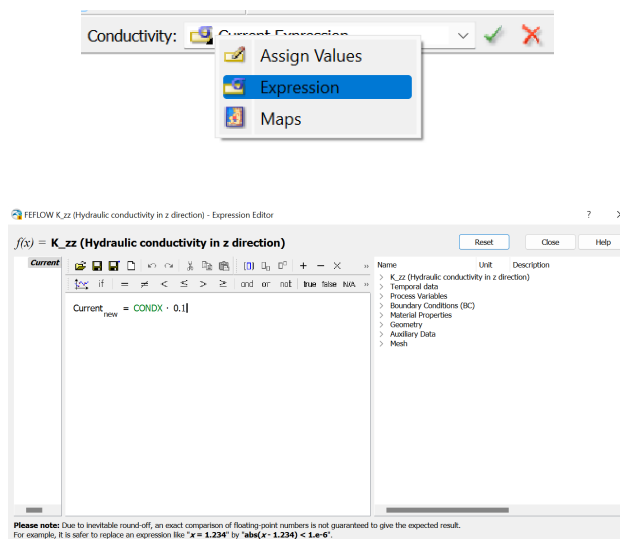

Figure 1.2: Expression defining Kzz

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 1^{23} **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 `Bu` *Assign* in the `□□□` *Editor* toolbar.

Clear selection.

📂 💾 exercise_fri7.fem

Reference Field Data

To be able to compare the computed groundwater levels to measurements, a few observation points will be loaded into the model. Go to the 📃 *Maps* panel, right-click on it and choose *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 `Bu` *OK* to proceed. The now imported observation wells can be shown by a double-click on *Observation Points* in the 📃 *Entities* panel.

📂 💾 exercise_fri8.fem

1.6 Simulation Run

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_fri8.fem in this case.

Starting the simulation

To run the model, click

▶ *Start* in the `□□□` *Simulation* toolbar.

As the model is unconfined, the resulting 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 **Hydraulic Head - 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.
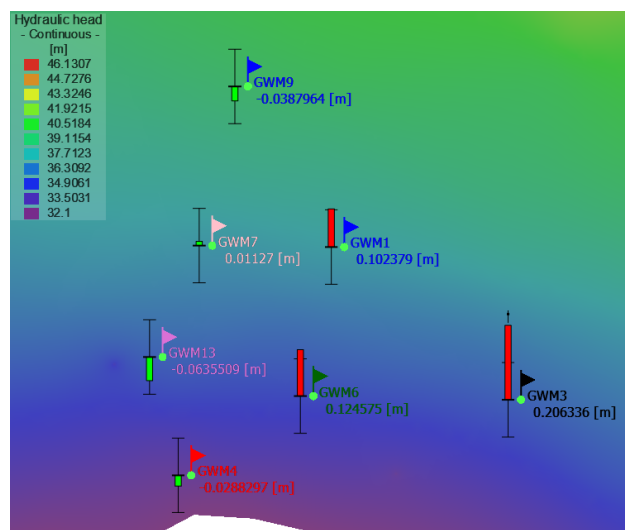
From the figure above, we can see some of the error bars in our observation points are displayed with a red color. That means, the simulated hydraulic head at those points are not within the confidence intervals of our measurements (0.1 m in for this exercise). This however, is expected as so far we have only run our model as a steady state.

1.7 Postprocessing

1.7.1 Scene Library

In the Slice View, right-click and choose **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. In case of loading the file, run the simulation from the file exercise_fri9.fem before proceeding.

1.7.2 Budget

To check whether the model indeed has reached the steady state, the over- all water balance is calculated. The 📄 **Rate-Budget** panel provides the means for this, as shown in Figure **??**.

If it has been closed accidentally, open the 📄 **Rate-Budget** panel via **View - Panels - Rate-Budget**. Check the ✅ **Active** 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. In this simulation, the imbalance is sufficiently small to accept the results.
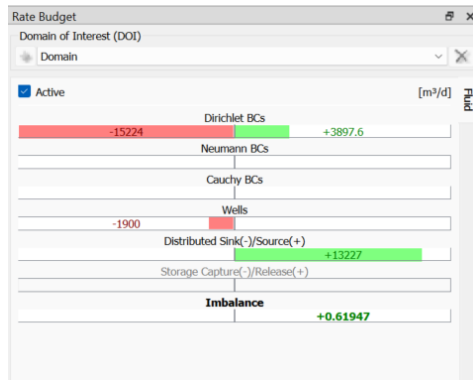
Figure 1.3: Rate Budget steady-state

2 Part : Mass transport

All files related to this exercise can be found on the training USB memory stick (or similar) in the project folder /models/Friedrichshagen.

2.1 Objective

In this exercise, the 3D steady-state flow model is extended to a 3D transient mass-transport model. Basic concepts of mass-transport modelling including how to assign boundary conditions and material properties are explained. Similarly, good practices on how to refine the 3D mesh to better set-up flow and mass models are mentioned.

2.2 Model Scenario

As mentioned in the previous chapter, we are building this model to evaluate the presence of a fictitious contaminant in groundwater. 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 north- east of town. The other possible source is an abandoned waste-disposal site further east.
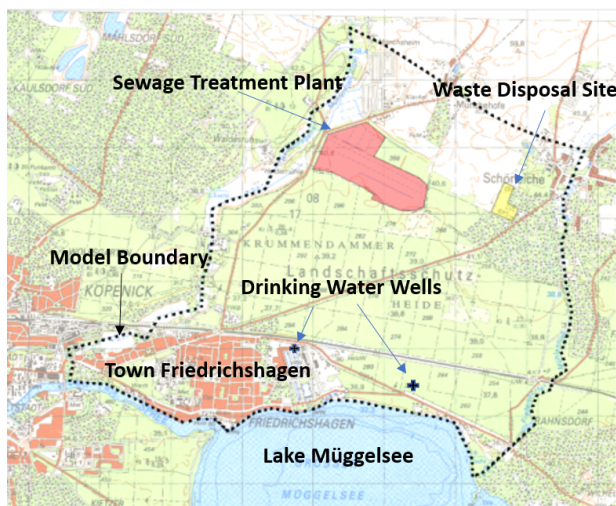
2.3 Problem Settings

Problem Class

Let us start by loading our previous 3D flow model run exercise_fri9.fem from the training material. We can see that the the process variable Hydraulic head has changed. After the run, it does not contain the initial conditions any more, but the final results from our

steady-state run. This is important, since we now have a head gradient to run our model as a mass transport class.

Open the **Problem Settings** dialog located in the 🔲 **Edit** menu. Choose **Problem Class** from the index and activate the option **Include transport of...** ☑ **Mass**. Finally, confirm with the 🔲 **Apply**.
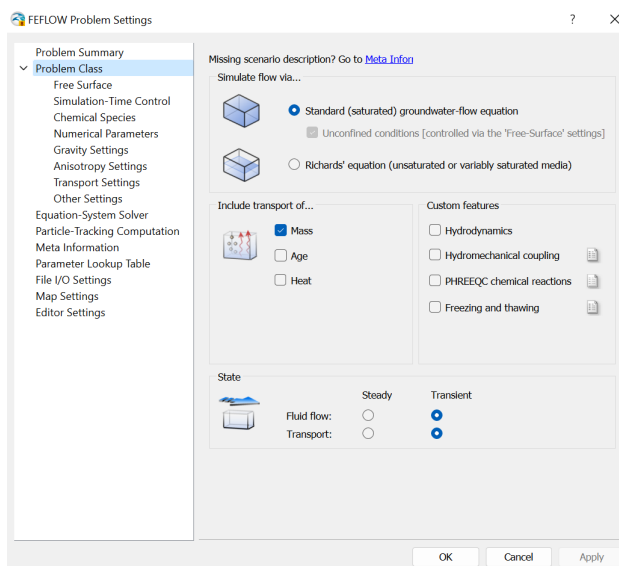

Figure 2.1: Problem-class settings for steady mass transport

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** to close the window.

🗂️ 💾 exercise_fri10.fem
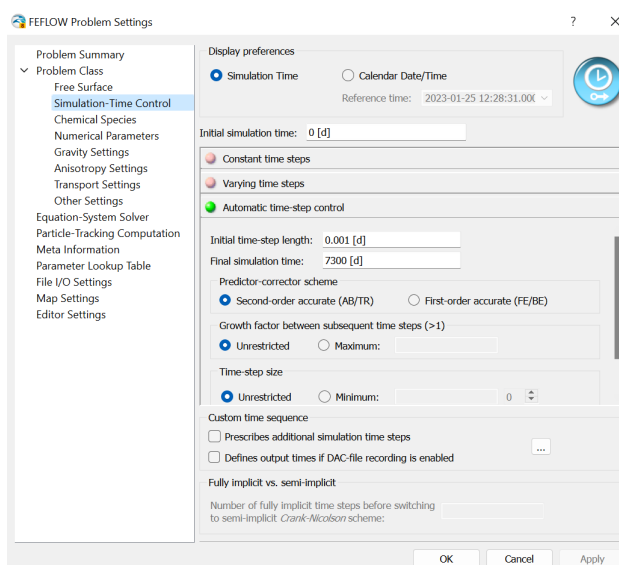
2.4 initial conditions

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.dxf. Choose the option

🔳 **Select by Map Polygon** from the dropdown selector of the ▫▫▫ **Selection** toolbar. Make sure the **Snap** distance is set to ₁₂₃ **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 (using **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 ₁₂₃ **4** Neighbors and Exponent of ₁₂₃ **2**. Click on ⌨ **OK**.

Double-click on the created link CONC ->Mass concentration in the Maps panel and assign the values by clicking on ⌨ **Assign**. Clear selection.

📂 💾 exercise_fri11.fem

2.5 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 ₁₂₃ **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.
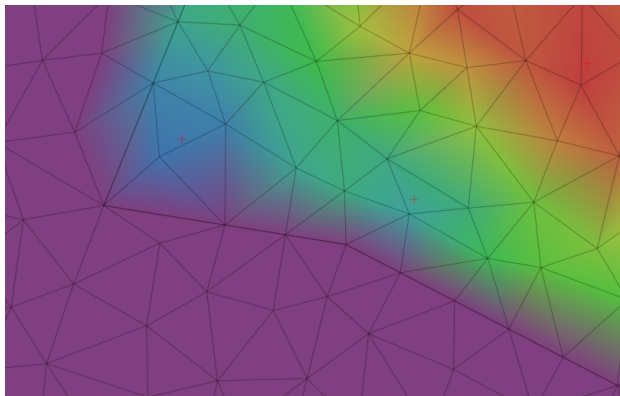
At the transition between the refined and non-refined area, the elements are now quite irregularly shaped (large angles). To improve the mesh quality after manual refinement, the mesh will be smoothed in this area. Make sure that the

⊞ *Add to Selection* option in the ▭▭▭ *Selection* toolbar is active. Define a Snap distance of ⌗ **500** m and click on

🔲 *Select by All Map Geometries* again in order to create a selection covering the refinement zone and its adjacent area. Press the

🔺 *2D Mesh Smoothing (with X-Y Inheritance in 3D)* button to perform the smoothing.
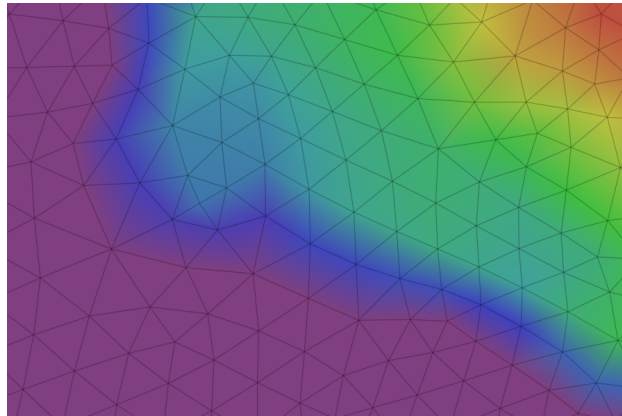
💾 exercise_fri12.fem

2.6 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 nodal 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.

> 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 (+).
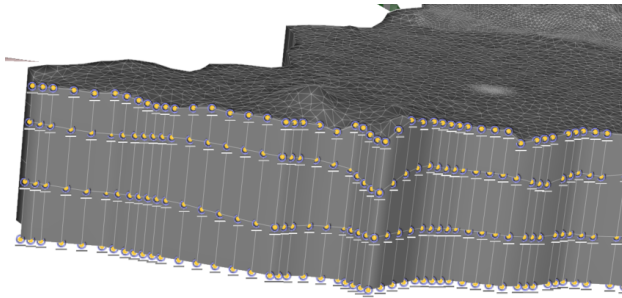
Wells

Both wells shall be operated by time-varying pumping rates. Go to the menu 📥 **Edit - Time Series...** and use the [Bu] **Import...** to load the Time Series from the pre-defined file well_rates.pow. Choose this file in the file selector dialog, confirm with the [Bu] **Open** button and [Bu] **OK** to all. Notice the gap in the time series 1: After 5200 days the western well will stop pumping. Hit [Bu] **OK** for applying the time-series and closing the dialog.
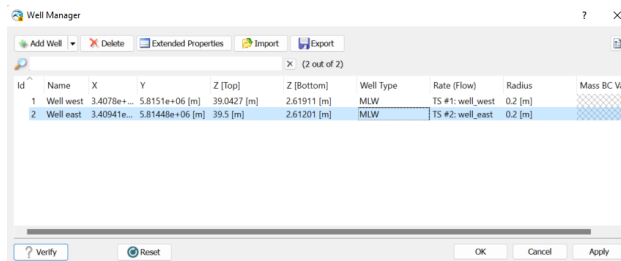
📂 💾 exercise_fri13.fem

2.7 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 parnameLink to Parameter... from its context menu. In the Parameter Association dialog on the right, browse to **Material Properties - Fluid Flow - In/Outflow on top/bottom** 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).
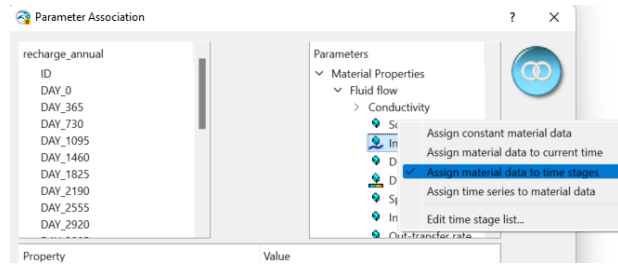
The list has to contain the same values as given in the attribute fields of the map (0;365;730;...). Instead of populating the list manually, the [Bu] **Import...** lets you get those time stages automatically from the pre-defined file recharge_time_stages.pow. Choose this file in the file selector dialog an confirm with the [Bu] **Open**.
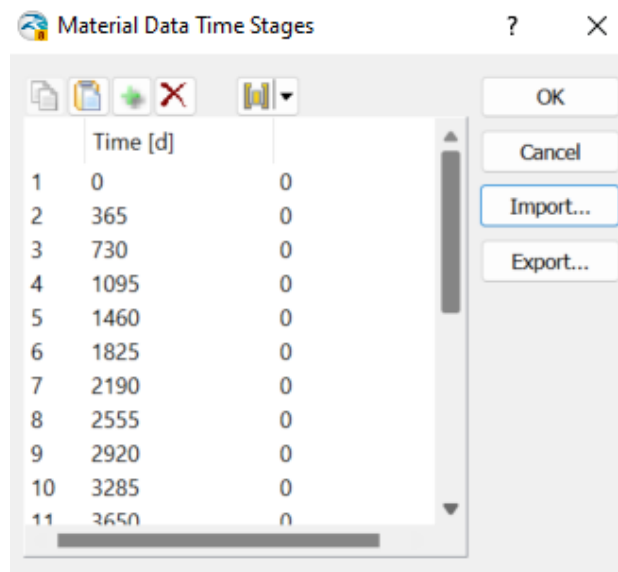
This will fill the list with annual time stage intervals up until 7300 days, as shown above. Click [Bu] **OK** to close the dialog. The links between the attribute fields and the time stages is done the same way as for constant model properties. The process, however, can be accelerated by creating a 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 newly created link to set **In/outflow on Top/Bottom** active and

🔲 **Select All** elements. Finally, set the **Snap** distance to [123] **0** m and assign the values by clicking on [Bu] **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_fri14.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.
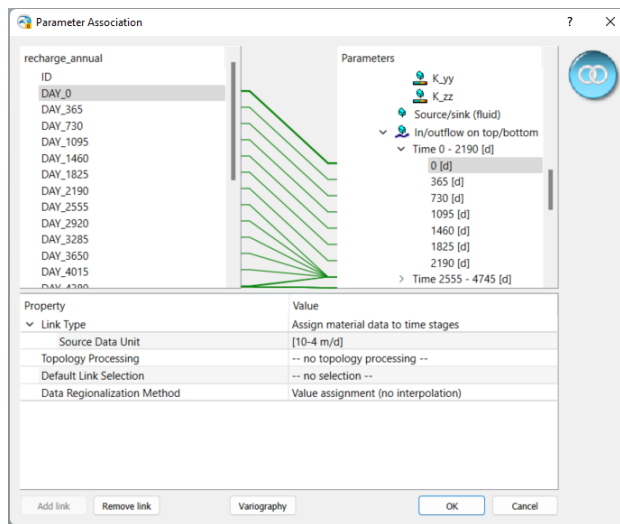
Right-click on **Material Properties - Mass Transport** in the 📄 **Data** panel and choose **Assign Multiple...** from the context menu. Afterwards:

- 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_fri15.fem

2.8 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, as shown in Figure **??**.
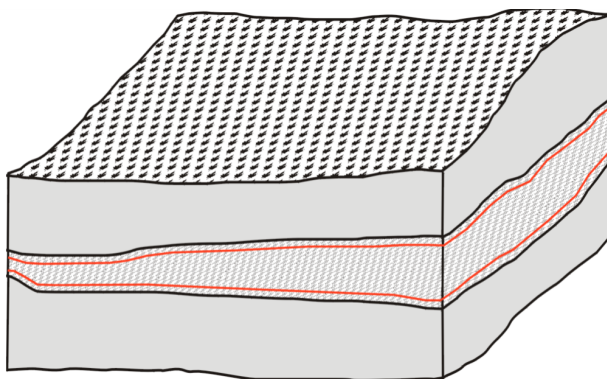
Figure 2.2: Adding extra layers in aquitard

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 [1²³] **0.1** m in the **Distance between slices** input box and click [Bu] **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 slices. The lower list, Data flow for layers describes the data flow for all material data and for elemental selections.

FEFLOW, in the **Data flow for slices**, suggests to transfer data 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, as shown in Figure **??**.
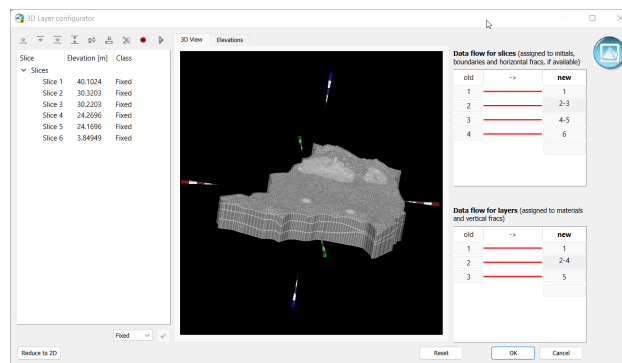
Figure 2.3: Adding Slices to model in the 3D Layer Configurator

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_fri16.fem

2.9 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 [▫▫▫] **Simulation** toolbar. Activate ☑ **Save complete results (DAC file)** as shown in Figure **??**. 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 [▫▫▫] **Simulation** 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 [▫▫▫] **Simulation** toolbar.
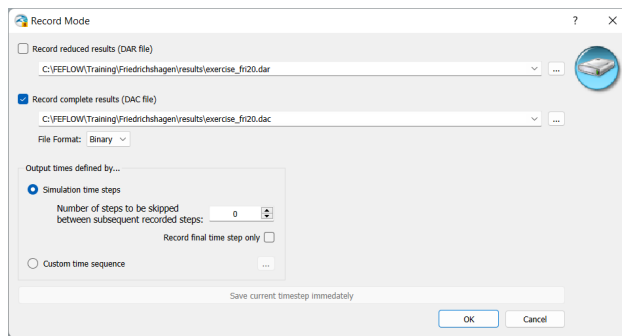
Figure 2.4: Record Mode window to define DAC file

In the Time-Steps chart (which can be opened via [icon] **View - Charts** if not already shown) the actual time step length versus the total simulation time is plotted. As shown in Figure **??**, the most constant conditions lead to a steadily increasing time step length, with a reduced time-step whenever a change of the groundwater recharge occurs.
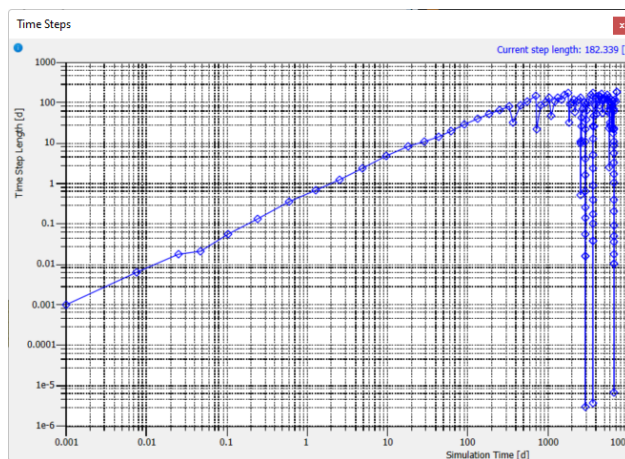

Figure 2.5: Time Steps chart

The simulation stops after 7300 days (the final time that has been set in the Problem Settings dialog before).

Click

[icon] **Stop** to exit simulation mode.

2.10 Postprocessing

[icon] **Open** the file exercise_fri16.dac.

2.10.1 Scene Library

Go to the [icon] **Scene Library** panel and and use the context menu of the previously stored exercise_fri1:1[*] item from **Create View from Scene**. Using the [icon] **Simulator** toolbar, browse through the saved time steps of the simulation file with the

[icon] **Previous Step** and

[icon] **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.

2.10.2 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).

2.10.3 Subdomain Budget

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

- Choose

  🔷 *Select Elements 3D* and

  🟦 *Select All*.

- From the context menu of the *Element Selection* in the 🗒 *Selection* panel, choose *Store Current Selection* and name this selection ₁²³ **Layer 5**.

- ❎ *Clear Selection*.

- Activate the stored selection Layer 5 in the 🗒 *Selection* 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 ⎡Bu⎤

  ∫...dt *Integral* to calculate the budgets for all time steps and accumulate them into the period budget.

The budget (see Figure **??**) 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*.

2.10.4 Imbalance

- Click on the stored selection Layer 5 in the 🗒 *Selection* 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 *Integral*

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.
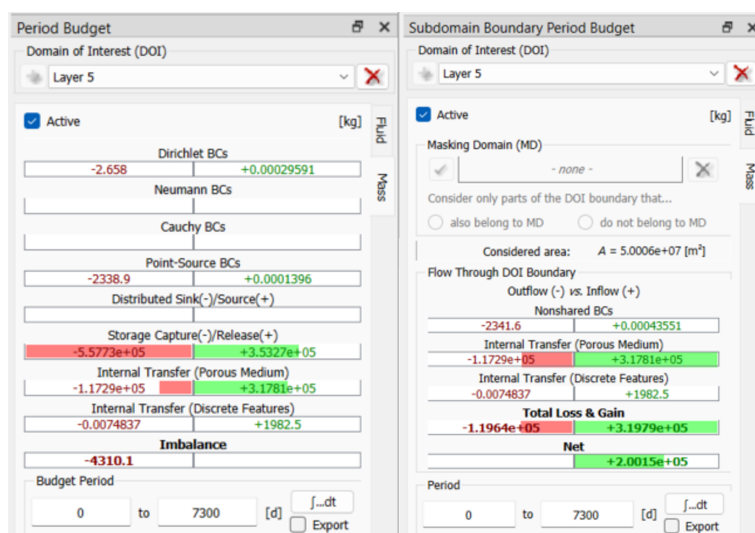
Figure 2.6: Period Budget and Subdomain Boundary Period Budget for Layer 5

2.10.5 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 Figure **??**.

In the ▤ **Entities** 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**, as shown in Figure **??**.

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 to have a View as shown in Figure **??**.

2.10.6 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, ₁²³ **10** mg/l and ₁²³ **20** mg/l. Close the dialog by clicking [Bu] **OK**, and click [Bu] **Apply** in the ▤ **Properties** panel. The isosurface visualization is changed to reflect the newly set concentrations.
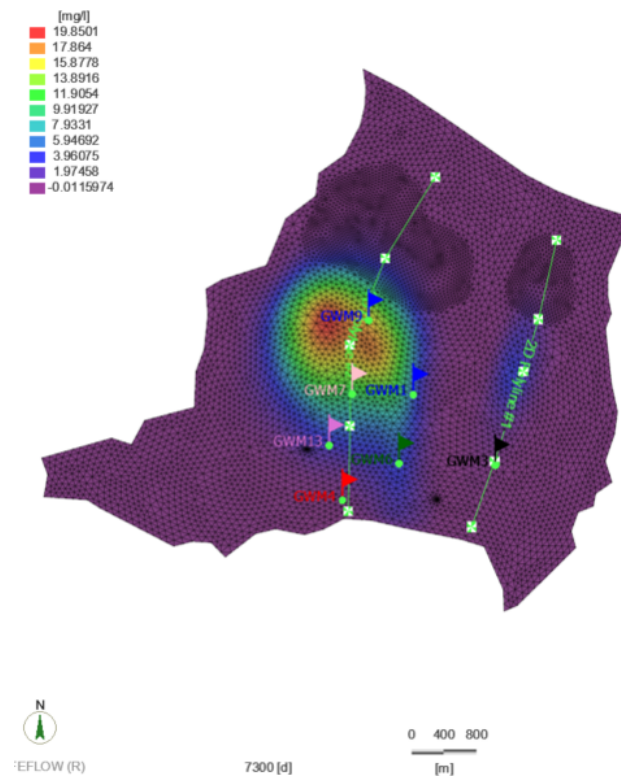
Figure 2.7: 2D Lines for Cross-Section plot

2.10.7 Breakthrough Curves

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

2.11 More Information

This completes 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 the website: `https://www.mikepoweredbydhi.com/products/feflow`
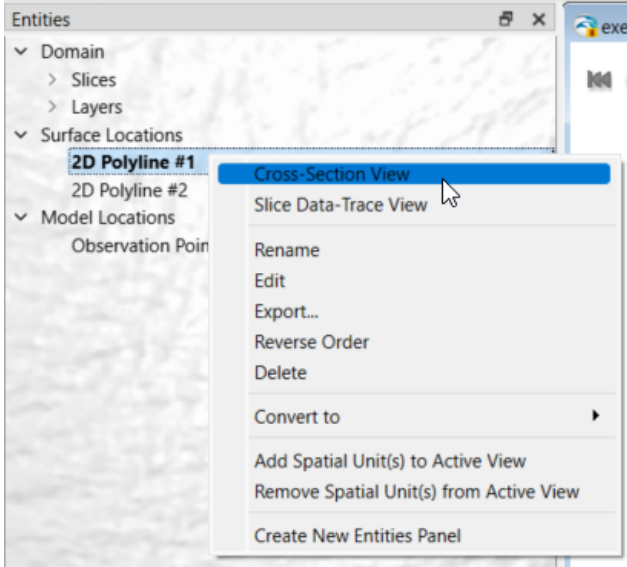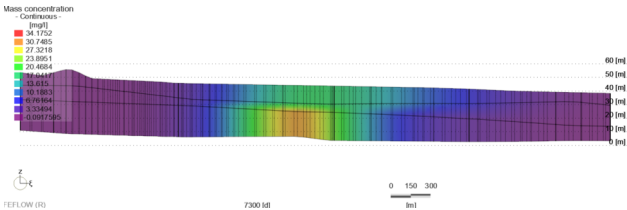
Figure 2.8: 2D Polyline #1 context menu

Figure 2.9: Cross-section view of the plume spreading