

WASY Software **FEFLOW[®]**

Finite Element Subsurface Flow
& Transport Simulation System

Demonstration Exercise



Copyright notice:

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

Copyright (c) 2004 WASY GmbH Berlin - all rights reserved.

WASY and FEFLOW are registered trademarks of WASY GmbH.

WASY Institute for Water Resources Planning and System Research Ltd.,
Waltersdorfer Straße 105, D-12526 Berlin, Germany
Phone: +49-30-67 99 98-0, Fax: +49-30-67 99 98-99
E-Mail: mail@wasy.de

Contents

1. Introduction	5
1.1 About FEFLOW	5
1.2 Scope and Structure	6
1.3 Terms and Notations	6
1.4 Model Scenario	6
2. Getting Started	9
2.1 Starting FEFLOW	9
2.2 The FEFLOW Graphical User Interface	9
3. Setting up the Model	11
3.1 Creating the Finite Element Mesh	11
3.1.1 Loading Background Maps	11
3.1.2 Designing the Superelement Mesh	13
3.1.3 Generating the Finite Element Mesh	15
3.1.4 Mesh Geometry	15
3.2 The 3rd Dimension	17
3.2.1 Designing Slices and Layers	17
3.3 Problem Class	19
3.4 Temporal & Control Data	20
3.5 3D-Slice Elevation	21
3.6 Flow Data	23
3.6.1 Flow Initials	23
3.6.2 Flow Boundaries	25
3.6.3 Flow Materials	28
3.7 Transport Data	32
3.7.1 Mass Transport Boundaries	32
3.7.2 Mass Transport Materials	35
3.8 Reference Data	36
3.9 Reconfigure 3D Task	37
4. The Simulator	39
5. The Postprocessor	41
5.1 2D Visualization	41
5.2 3D-Options Menu	45
5.2.1 3D-Pathline Analyzer	46
5.2.2 3D-Isosurfaces	47
5.3 Final Remarks	48

Contents

Introduction

1.1 About FEFLOW

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

- **three-dimensional** and **two-dimensional**
- areal and cross-sectional (horizontal, vertical or axis-symmetric),
- fluid **density-coupled**, also **thermohaline**, or uncoupled,
- **variably saturated**,
- **transient** or steady state
- **flow, mass and heat transport**

in subsurface water resources with or without one or **multiple free surfaces**.

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

Through a sophisticated interface communication

between FEFLOW and GIS applications such as ArcInfo, ArcView and ArcGIS for ASCII and binary vector and grid formats is available.

The integrated Interface Manager (IFM) provides a comfortable interface for the coupling of external code or even external programs to FEFLOW.

It has been used to implement the parameter estimator PEST[‡]) in FEFLOW.

FEFLOW is available for WINDOWS systems as well as for different UNIX platforms.

Since its creation in 1979 FEFLOW has been continuously improved. The FEFLOW source code is written in ANSI C/C++ and contains more than 1,100,000 lines. FEFLOW is used worldwide as a high-end groundwater modeling tool at universities, research institutes, government offices and engineering companies.

For additional information about FEFLOW contact your local distributor or have a look at the FEFLOW web site www.feflow.info.

[‡]) based on the PEST version 2.04 (1995) by John Doherty, Watermark Computing, Corinda, Australia.

1. Introduction



You can skip any of the steps in this exercise by loading the problem files that are already prepared. These files are not ready to run in the simulator, you have to complete them first. For complete files install the sample data tutorial and benchmarks from the CD-ROM.

1.2 Scope and Structure

The scope of this exercise is to introduce the novice user to the philosophy of modeling three-dimensional flow and transport problems based on real world data with the help of FEFLOW. It also shows some of the capabilities of FEFLOW to users testing the code in demo mode. It is not intended as an introduction to groundwater modeling itself. Therefore some background knowledge of groundwater hydraulics and groundwater modeling is required.

Before starting the exercise FEFLOW should be installed on a suitable computer. For a detailed description of the installation process please refer to the booklet of the FEFLOW CD-Rom.

1.3 Terms and Notations

In addition to verbal description 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
	button
	input field for text or numbers
	switch toggle
	radio button or checkbox

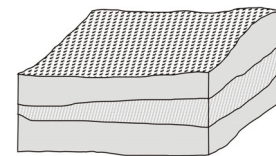
Please notice that the color of the corresponding element in FEFLOW may be different, depending on the window in which the element occurs. You will find for example green menus or yellow switch toggles as well as the blue ones shown above.

All file names are printed **in color**.

1.4 Model Scenario

A fictitious contaminant plume has been detected near the small town of Friedrichshagen, southeast of Berlin, Germany. An increased concentration of a contamination has been found in the town's two drinking water wells. There are two potential sources of the contamination; the first is the treatment plant located in an industrial area situated to the northeast of the town. The other option is a waste disposal site found to the northwest of Friedrichshagen.

For studying the groundwater threat and potential pollution, we need to design, run, and calibrate a three-dimensional groundwater flow and contaminant transport model of the area. First we need to define the model domain. The town is surrounded by many natural flow boundaries, such as rivers and lakes. There are two rivers that run north-south on either side of Friedrichshagen that can act as the eastern and western boundaries. The lake Mueggelsee will limit the model domain to the south. The northern boundary runs along an east-west flowline north of the two potential sources of the contaminant.



The geology of the model domain is comprised of Quaternary sediments. The hydrogeologic system concerns two aquifers separated by a clay aquitard.

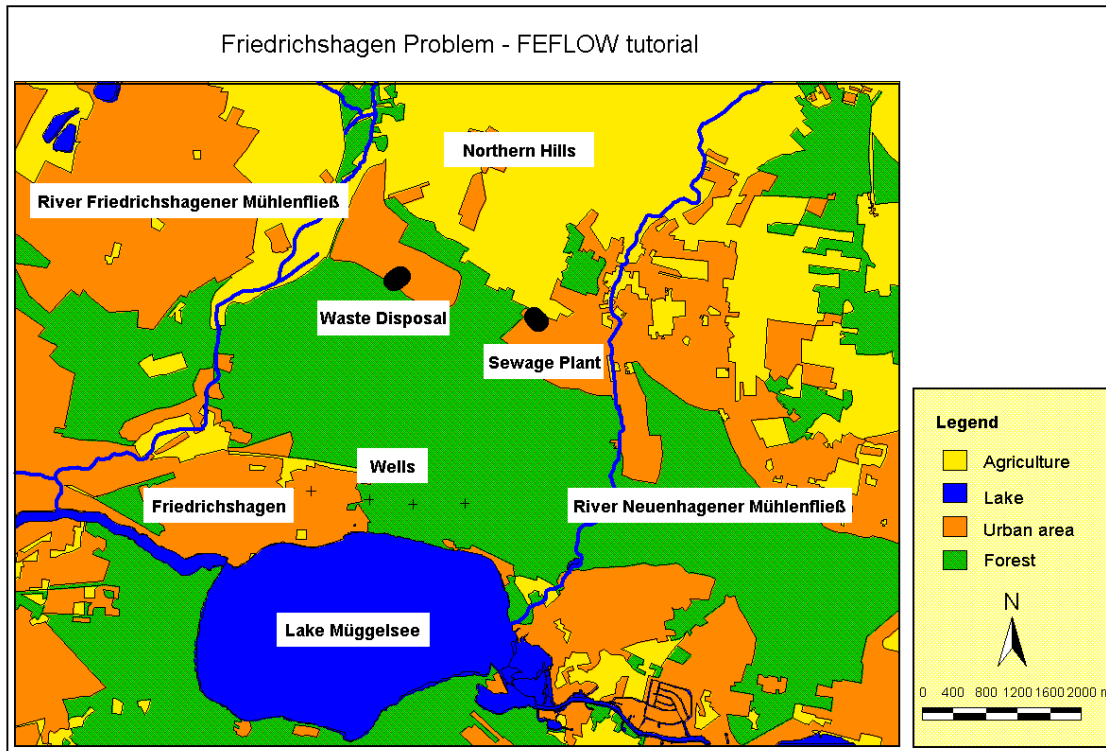
The top hydrostratigraphic unit is considered to be a sandy unconfined aquifer up to 7 meters thick. The second aquifer located below the clayey aquitard has a thickness of 30 meters.

The northern part of the model area is primarily used for agriculture, whereas the southern portion is dominated by forest.

1.4 Model Scenario



This map was created by importing GIS data for the model into FEPLLOT, a plotting & desktop mapping tool made by WASY, which is included with FEFLOW. FEFLOW supports the ESRI shape file format, ArcInfo compatible ASCII formats, DXF and FEFLOW specific plot formats for displaying background maps and exporting data.



1. Introduction

Getting Started

2

2.1 Starting FEFLOW

We assume that FEFLOW has been successfully installed on your system. For details of the installation process please refer to the booklet of your FEFLOW CD.

FEFLOW is started as follows:

On Windows Systems

- Start FEFLOW via the *WASY* entry in the *Programs* folder of the Windows *Start* menu.
- "FEFLOW 5.1"

On Unix Systems

- Type 'feflow' and hit the <Enter> key.

If no FEFLOW license has been installed, you are asked if you want to start FEFLOW in demo mode. The demo mode does not allow you to save any files or to open unregistered files, i.e., files not delivered with FEFLOW.

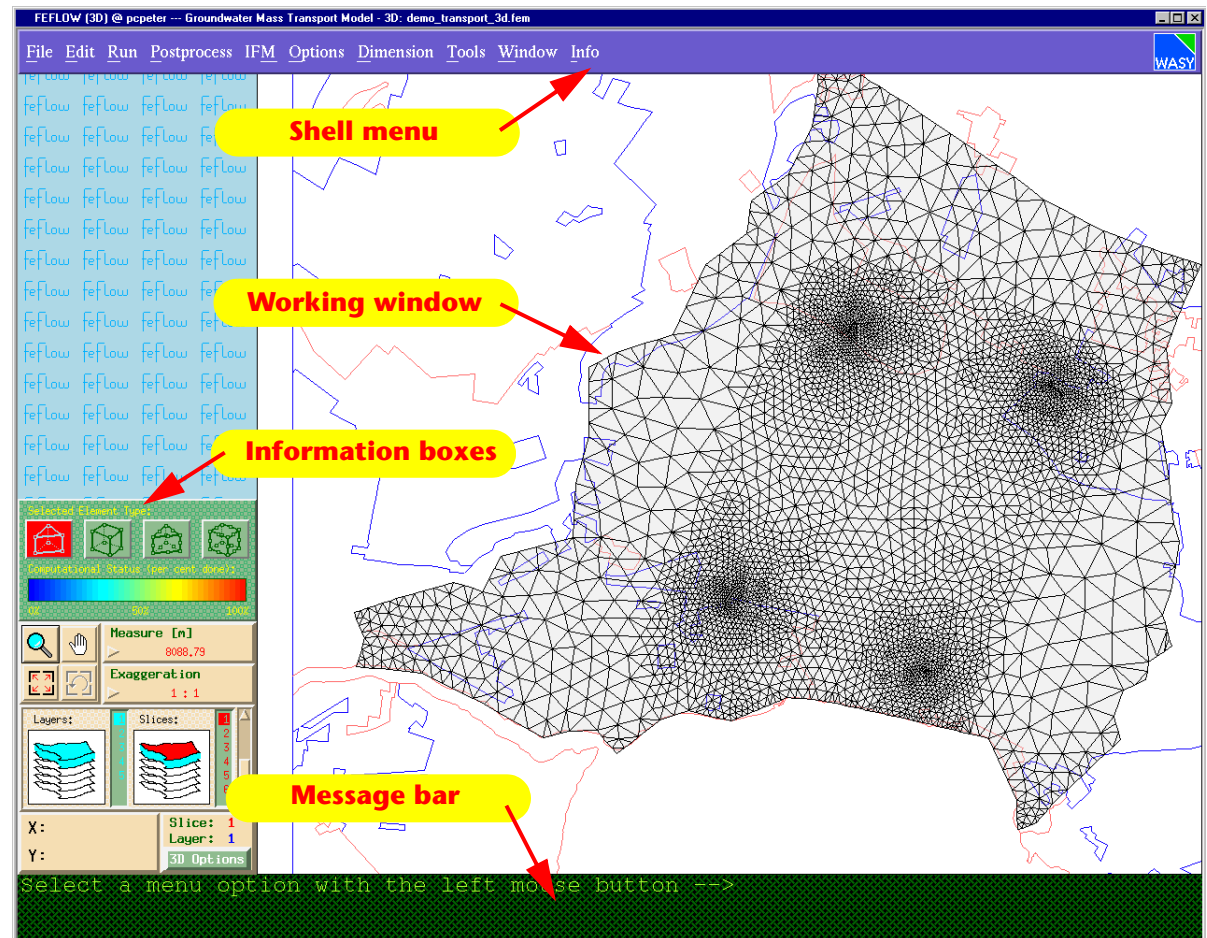
The Main Window of FEFLOW is displayed on your screen.

2.2 The FEFLOW Graphical User Interface

The FEFLOW window on your screen is divided as shown in the figure on the next page.

FEFLOW commands are grouped in several menu levels, i.e., the system is hierarchically structured. The **shell menu** forms the top level. All subordinate levels and menus are accessible from the *Shell menu entries* at the top of the window. All editing processes are executed interactively in the **Working window**. The **Information boxes** at the lower left side of the shell are visible in every menu level. They display information about the model, offer tools for zooming and switching between different slices/layers and provide the entry to the **3D-Options menu** for 3D view and analyses. The **Message bar** at bottom edge of the shell displays information about the current processes or available functions. For detailed online help hit the <F1> key or click on the "Help" buttons, which you can find in most of the menus and windows. The help is context-sensitive so that you always get support on the currently active functions.

2. Getting Started



Setting up the Model

3

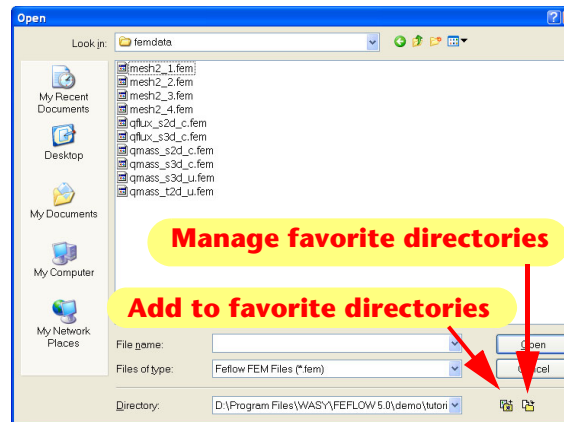
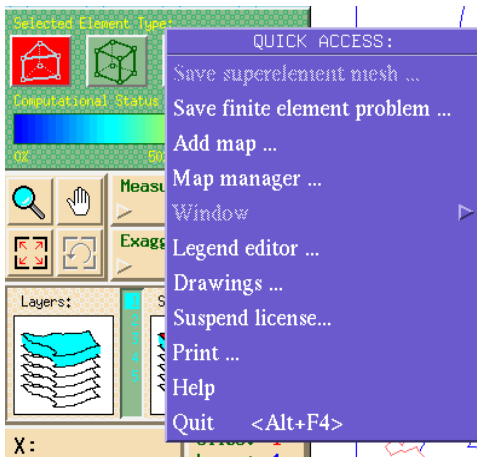
In this step the FEFLOW model is built from scratch. We begin creating the finite element mesh, extend the mesh to the third dimension, and assign all parameters needed for simulating a flow and mass transport problem.

3.1 Creating the Finite Element Mesh

3.1.1 Loading Background Maps

To define the model area and to construct the superelement mesh, we need to load background maps. This can be accomplished by using the Quick Access menu. Click anywhere on the green colored part at the left side of the screen. The **Quick Access** menu appears.

Holding the left mouse button, select  **Add map ...** from the menu.



3. Setting up the Model

The FEFLOW **File Selector** appears. The uppermost field, called the **Filter**, displays the current directory path. FEFLOW automatically searches for map data in the directory called **import+export**. The **Map type** field allows you to choose between different file formats.

The **Files** field displays all available files of the selected map type in the current directory. To navigate between directories use the 'Directories' field. Click a directory for opening it; navigate to the parent directory clicking '../'. You can find the files for this exercise in the project directory '.../WASY/FEFLOW/exercise/'. The maps are stored in the subdirectory 'import+export'.

- select **model_area.lin** in the **Files** list.
- **OK**

The **Map Measure Menu** appears. The **Map Measure Menu** allows you to define the extent of the background maps and the coordinates of the Working Window.

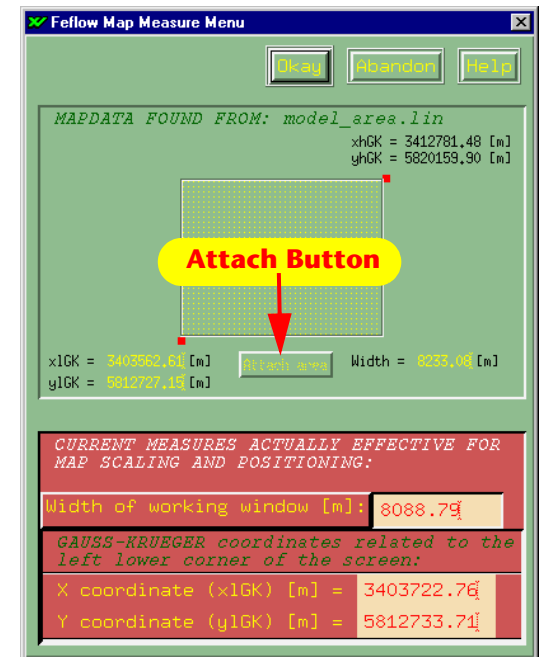
Attach area in the center of the window. The attach area button references all additional maps to the attached map coordinates.

- **Okay** to import the map.

FEFLOW automatically georeferences and scales the working window to the co-ordinates of the background map and displays the map in the working window.

Next we will import a map showing the landuse of the area.

- Click anywhere on the green colored part on the left of the screen. The **Quick Access** menu appears.
- Holding the left mouse button, select **Add map ...** from the menu. The FEFLOW **File Selector** pops up.
- select **landuse.lin** from the **Files** list.
- **Okay**



- The **Map Measure** window pops up.
- **Okay** to import the map. **Do not** attach the area a second time as the extent of the landuse map differs from the extent of our model area.

We have now imported the required maps for our investigation and proceed to the generation of the superelement mesh.^{‡)}

^{‡)} FEFLOW is also capable of importing other file formats as background maps including GIS (*.shp) and CAD (*.dxf) data as well as raster images (*.tif). If necessary, images can be referenced via the FEMAP assistant that is included with FEFLOW.

3.1 Creating the Finite Element Mesh

3.1.2 Designing the Superelement Mesh

To define outer and inner borders of the finite element model, a so called superelement mesh is constructed. The superelement mesh will provide the basic structure of the model. Designing the superelement mesh is accomplished via the **Mesh Editor** located in the **Edit** menu of the Shell.

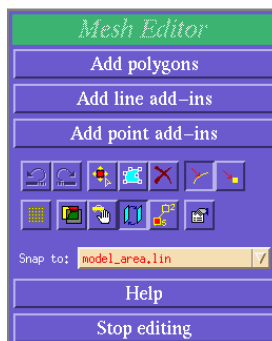
-  **Edit** in the top bar of the Shell menu
-  **Design superelement mesh**

The **Mesh Editor** menu appears along the left hand side of the window. For this exercise we use the so called 'New mesh editor' which will replace the former one in the future. In the moment both editors are implemented in FEFLOW as the new one does not contain all necessary features until now.

-  **New mesh editor**
-  to **model_area.lin** in the 'Snap to:' line
-  : activates the line-snapping mode

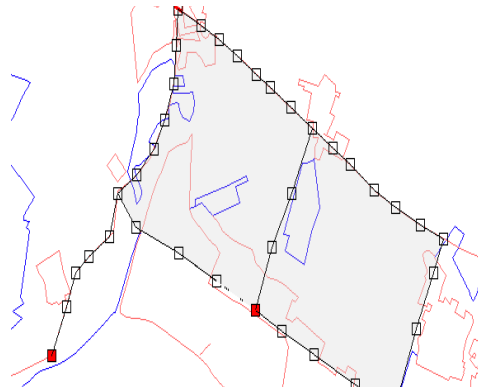
We must now define the outside boundary of our model.

-  **Add polygons** (The cursor becomes a crosshair.)



Move the cursor to the outline of the model area background map in the working window. If the cursor is within the snap distance to the outline, the corresponding part of the background map becomes red.

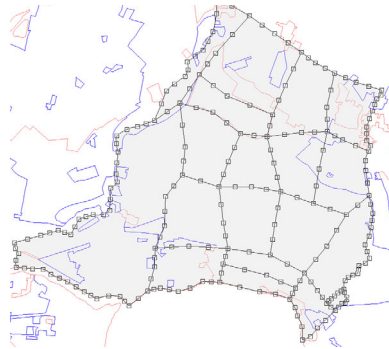
- Nodes should be set at fairly equal intervals around the perimeter of the model area. Define the nodes by clicking the left mouse button along the model outline. Where you set a node of a superelement FEFLOW will also create a finite element node. This is important for the exact assigning of boundary conditions.
- When you return to the first node, close the polygon by clicking the first node (marked by a red arrow) a second time. The enclosed polygon area is displayed in a shaded gray color.



You can undo the last digitized nodes by returning to a recently drawn node and hitting the left mouse button. This action is indicated by the cursor symbol .

The superelement mesh can be saved separately using the **Quick Access** menu - **Save superelement mesh ...**. This allows to keep the basic data for generating several finite element meshes for the same area. Furthermore a superelement mesh can be reloaded as template for assigning problem attributes on the corresponding model as described below.

3. Setting up the Model



By creating the polygon, you have defined the outer border of the model. Leave the the New mesh editor via **Bu Stop editing.**

Next we will import the so called **Add-ins**. Add-ins are lines or points which FEFLOW uses as focal points to create finite element nodes during the mesh generation. Add-ins are very useful to position boundary conditions like contaminant mass sources or wells in exact locations. First we have to load background maps containing the location of the Add-ins:

- Click anywhere on the green colored part of the screen. The **Quick Access** menu should appear.
- Holding the left mouse button, select **Add map ...** from the menu. The FEFLOW **File Selector** appears. The uppermost field, called the **Filter**, displays the current directory path. The **Files** field displays all the available files of the selected map type.
- select **mass_src.lin** in the **Files** list, the background map which describes the locations of the contaminant sources at the sewage treatment plant and the waste disposal.
- **Bu OK** - The **Map Measure Menu** appears.
- **Bu Okay** to load the map. Two small lines are displayed on the model.

The well locations are imported from a *.pnt ASCII point file format:

- **Bu Add map ...** from the Quick Access Menu
- Select **ASCII Point (*.pnt)** from the map type list.
- Repeat the steps described above for the background map **demo_wells.pnt**.

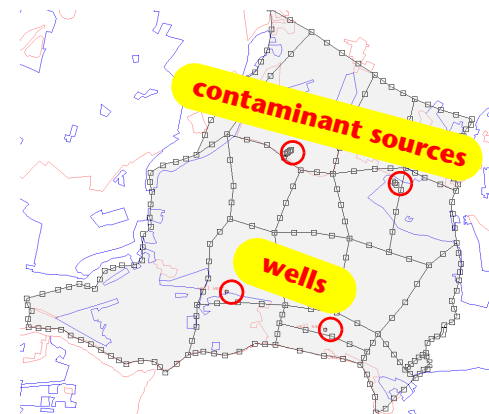
Now we activate the two background maps as Add-ins:

- **Bu Add-in lines/points**
- **mass_src.lin** as the background map.
- **Active** - The map is now activated.
- **Bu Add lines from map** - The lines are imported as Add-ins automatically.

Repeat these steps to include two pumping wells located in the southern portion of the map.

Use **demo_wells.pnt** for this purpose.

- **Inactive**
- **Active**
- **Bu Add point from map**
The supermesh should look like the figure below.
- **Bu Continue mesh design**



3.1 Creating the Finite Element Mesh

3.1.3 Generating the Finite Element Mesh

The finite element mesh is generated in the **Mesh Generator Menu**.

- **Bu** Start mesh generator

- **Bu** Generator options

Choose a **high refinement** for the areas around the Add-ins.

- **Bu** Okay

- **Bu** Generate automatically (The **Mesh density input** menu pops up.)

- Enter an element number of **500**. The element number is an educated estimate based on the size and type of model.

- **Bu** Start

The TMesh generator is an extremely accurate tool that

creates precise meshes based on the Add-ins and the boundary design. As an alternative generator for triangular meshes the Advancing Front algorithm is also available in FEFLOW. While able to create more regular meshes it cannot honor prescribed (Add-in) locations.

3.1.4 Mesh Geometry

For our mass transport simulation, the mesh is too coarse in the area where the contaminant will be distributed. Therefore the mesh has to be partially refined.

- **Bu** Mesh geometry

You will enter the **Mesh Geometry Editor**. Before starting the mesh refinement, please note some fundamental rules for using the FEFLOW editors (see box below).

To refine the mesh based on a background map, you

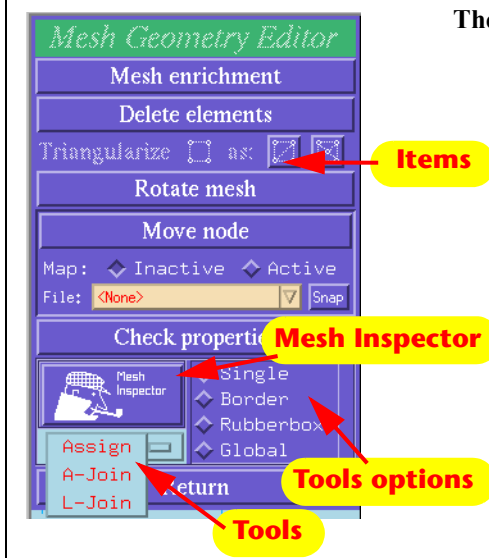


A mesh should always be created containing as few elements as possible. It can be refined later, but it never can be derefined to a coarser state than after mesh generation...

The FEFLOW editors:

The upper buttons define the items, e.g., *Mesh Enrichment*, *Delete elements* and *Check properties*. The icon showing a man's face activates the Mesh Inspector which gives you information about the parameters assigned to each node/element. The tools can be selected by clicking on the light blue button below the mesh inspector. Having selected a tool, different options are offered in the field right of the tools button. If you try to change the global value settings, you are warned and asked for confirmation. The use of the editor is as follows:

- Choose a **tool**.
- Select one of the **options** offered.
- Choose the **Bu** item/parameter you want to edit.
- Start editing.
- Exit the function by clicking the right mouse button or hitting <Esc>.



3. Setting up the Model



*You can also refine the mesh more accurately using the polygon background map. Select 'A-Join' instead of 'Assign'. Click the 'polygons' toggle and load **demo_refine.ply**. 'Cancel' the file selector box. Afterwards choose 'Mesh enrichment - refine' and click the polygon with the left mouse button. All following images are based on this refine-mesh.*

need to import a polygon map containing a polygon for the area that you want to refine. To do this,

- Click anywhere in the green colored part on the left of the screen. The **Quick Access** menu appears.
- Holding the left mouse button, select  **Add map ...** The **FEFLOW File Selector** appears.
-  **ASCII Polys (*.ply)** at the *Map Type* list.
- **demo_refine.ply** in the *Files* list.
-  **Okay**. The **Map Measure Menu** appears.
-  **Okay** to import the map. The area that we want to refine is shown as a colored polygon.

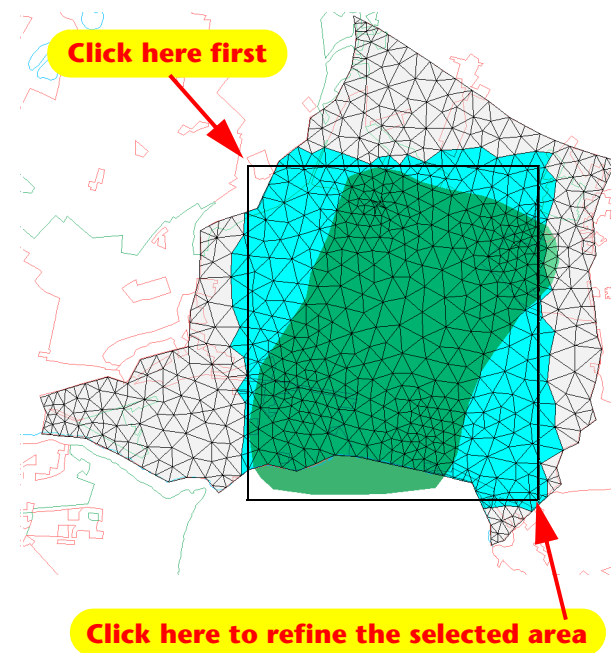
There are several different possibilities for refining the mesh. The simplest one is the **Rubberbox** option.

The tools button is set on  **Assign** by default.

-  **Rubberbox**
-  **Mesh Enrichment** and hold the left mouse button.
- Select  **Refine** from the small pop up menu and release the left mouse button.
- Click to set the upper left corner of the rubberbox.
- Move the cursor over the model until the light blue rubberbox covers the area of the polygon background map.
- Click again to set the lower right corner of the rubberbox. The mesh refinement is executed. To undo the mesh refinement, select  **Derefine** instead of 'Refine'. You can switch the background maps off by selecting the  **Map Manager** from the **Quick Access** menu.

We leave the Mesh Geometry Editor:

-  **Return**
-  **Exit to master menu**



3.2 The 3rd Dimension

Up to this point you have designed the geometry of a two-dimensional model. The following steps describe how to introduce the third dimension into your model using FEFLOW. A three-dimensional finite element model consists of a number of nodal planes, called **slices**. These slices can generally be regarded as the top or bottom planes of the (geological) layers.

3.2.1 Designing Slices and Layers

-  **Dimension.**

-  **Three-dimensional (3D)** - The 3D Layer Configurator pops up. See figure below.

The 3D-Layer Configurator controls the basic settings for the 3D model:

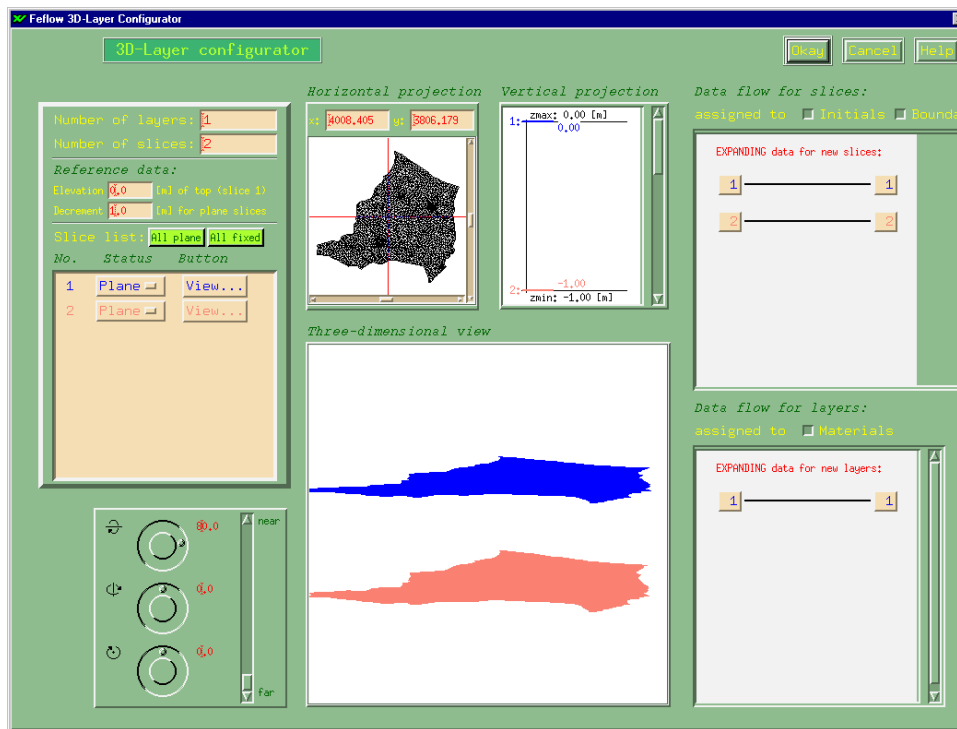
- the number of layers and slices,
- the data inheritance between slices or layers and
- the relative position between the slices.

The assignment of the real z coordinates is done later in the 3D slice elevation editor.

You will now define the number of layers/slices you need for this model. Of course the number may be changed afterwards, if necessary.



*If you have skipped the previous steps, please load the file **demo_2d.fem** via the Load finite element problem option from the File menu before beginning this exercise.*



When modeling one layer you need two slices, the top and bottom slice. When modeling two layers you need three slices, the top slice, the dividing slice between the two layers and the bottom slice. Initial conditions and boundary conditions are assigned to the nodes on slices, while material parameters are assigned to elements in layers.

3. Setting up the Model

In our case the upper aquifer is limited by the ground surface and by an aquitard at the bottom. The second aquifer is situated below the aquitard, underlain by a clay layer of unknown vertical extension. We will create the slices necessary for the stratigraphy of the existing area first:

In the *Reference data* box, enter an

- elevation for the top slice at **1000 m** in the **Elevation field** and a
- decrement of **100 m** in the **Decrement field**.

This will set the top slice of the model at an elevation of 1000 m, with all remaining slices set with a vertical distance of 100 m apart. These settings will prevent slices from intersecting when assigning real z-elevations from borehole data.

We will now specify the 3D layers.

- In the **Number of layers** input box, insert a value of **3**.
- Press **Return** to add the layers. The number of slices automatically changes to 4 slices.

As you can see, the 3D-Layer Configurator offers many other tools, some of which we will need later in this exercise. The online **Help** supplies you with more detailed information about the functionalities.

- **Okay** to exit the menu.

The screenshot shows the 3D-Layer Configurator interface. At the top, there are two input fields: "Number of layers:" with a value of 3, and "Number of slices:" with a value of 4. Below these is the "Reference data:" section, which includes "Elevation" set to 1000 (m) of top (slice 1) and "Decrement" set to 100 (m) for plane slices. The "Slice list:" section has two buttons: "All plane" and "All fixed". Below this is a table with columns "No.", "Status", and "Button". The table contains four rows, each representing a slice. Each row has a "Plane" button in the "Status" column and a "View..." button in the "Button" column.

No.	Status	Button
1	Plane	View...
2	Plane	View...
3	Plane	View...
4	Plane	View...

3.3 Problem Class

3.3 Problem Class

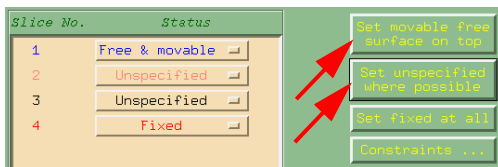
To define the parameters of the model we enter the **Problem Editor Menu**. All existing parameters are set to default values. We will only modify the most important of these parameters. To enter the Problem Editor,

-  **Edit**
-  **Edit problem attributes ...** - the **Problem Editor** appears.

According to the FEFLOW philosophy, the most efficient procedure to build a model is to work from the uppermost menu entry down to the lower ones.

In the Problem Class window we will define the principal type of the FEFLOW model. For our purposes we need transient flow with mass transport for an unconfined aquifer using the **BASD** moving-grid technique. For information about BASD click on the **Help** button in the **Free surface editor** and follow the link to the **Theory** section.

-  **Problem class** to enter the Problem Classifier.
- Since we are conducting a transient flow/transient mass transport model, perform the following settings:
-  **Flow and Mass Transport**.
-  **Transient flow/transient transport**
-  **Unconfined (phreatic) aquifer(s)**.
-  **Edit free surface(s)** - The **Free surfaces editor** appears. This will allow us to define the hydrogeologic properties of the slices.

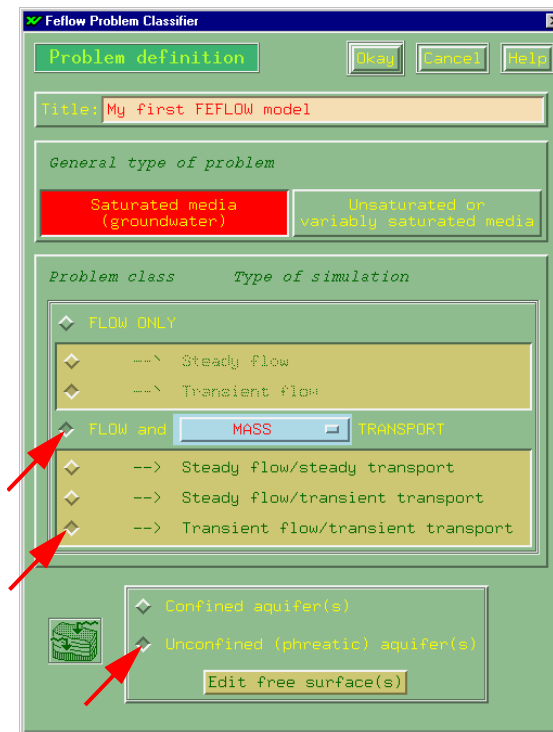


-  **Set movable free surface on top** - Now the top slice will follow the groundwater surface..
-  **Set unspecified where possible** - These slices will be distributed according to the moving groundwater surface and the stratigraphy
-  **Apply**
-  **Okay** to exit the Problem Classifier.

The model now describes an unconfined flow & mass transport problem with moving grid properties, i.e the mesh will follow the moving groundwater table avoiding that mesh elements get partially saturated or even dry.



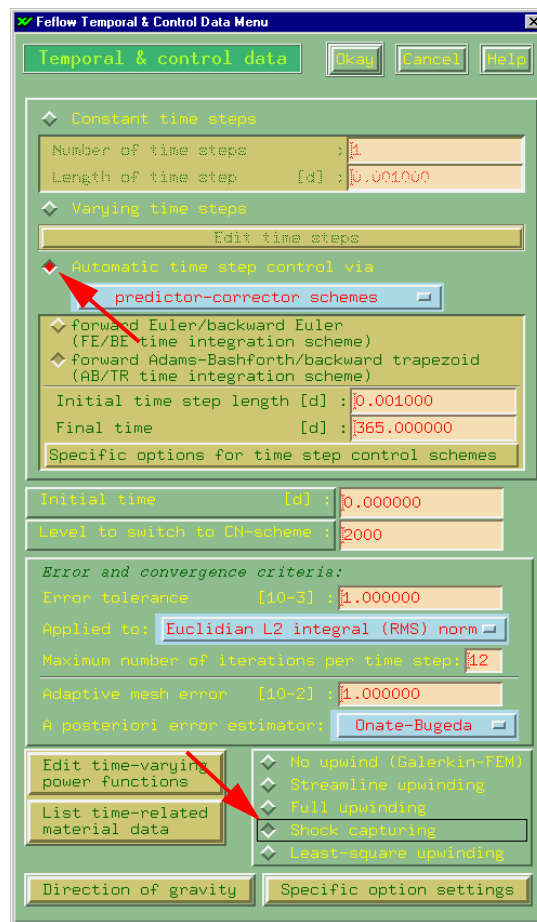
*If you have skipped the previous steps, please load the file **demo_3d.fem** via the **Load finite element problem** option from the **File** menu before beginning this exercise.*



3. Setting up the Model

3.4 Temporal & Control Data

In the Temporal & Control Data Menu, all parameters related to time and time-varying functions are defined or imported. Also, the error norms and the upwinding technique for suppressing numerical oscillations are specified here.



For our purpose, we will set the parameters for the time stepping during the simulation run. The simulation should run over 20 years by using the automatic time-step control option and the shock-capturing upwinding method for the mass transport simulation:

- **Bu** **Temporal & Control Data** to open the corresponding window
- **Automatic time-step control via predictor-corrector schemes.**
- Input an **initial time-step length** of **100** 0.001 days.
- Input a **final time** of **100** 7300 days.

We will now select the **Shock capturing** upwinding method which adds a nonlinear anisotropic damping factor to stabilize convection-dominant transport processes. It provides a powerful method to remove numerical oscillations around abrupt fronts of the contamination adding a minimal amount of numerical dispersion. Numerical damping is only performed in the vicinity of sharp gradients. Outside the fronts numerical dispersion vanishes or becomes significantly reduced compared to other upwinding methods, e.g., full upwinding.

- **Shock capturing** upwinding method at the lower right corner of the window.
- **Bu** **Okay** to exit **Temporal & Control Data**

3.5 3D-Slice Elevation

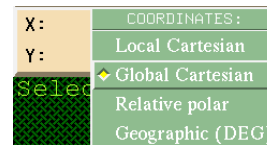
The 3D-Slice Elevation Menu allows you to define the slices based on real-world z-elevations by regionalization of irregularly distributed data points, i.e., from borehole logging. Database regionalization of all initial values, boundary conditions and material parameters can be assigned on the model as described for the z-elevations of the slices.

- **Bu 3D- Slice elevation** to enter the Slice Elevations Menu.

In the **Layer configurator** we had set the top slice to 1000 m and the lower slices with a vertical spacing of 100 m each. Now we will “pull down” the slices to their real-world elevations by assigning the corresponding z-elevations. To avoid intersection we begin at the lowest slice:

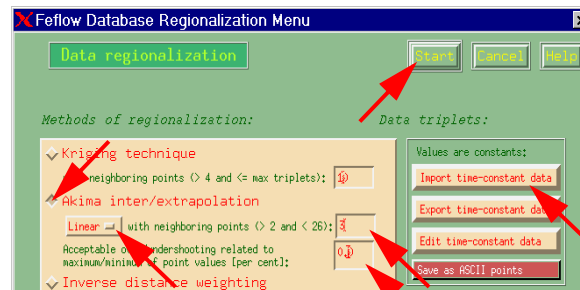


- Select slice **4** by clicking on the corresponding number in the Layers & Slices browser at the lower left side of the screen. In the browser, the left column lists the numbers of layers while the right column lists the number of slices.
- As the database for the z-elevations is georeferenced in global cartesian coordinates, we have to make sure the global system is set. Click the coordinates box in the lower left corner of the screen and choose **Global cartesian**.



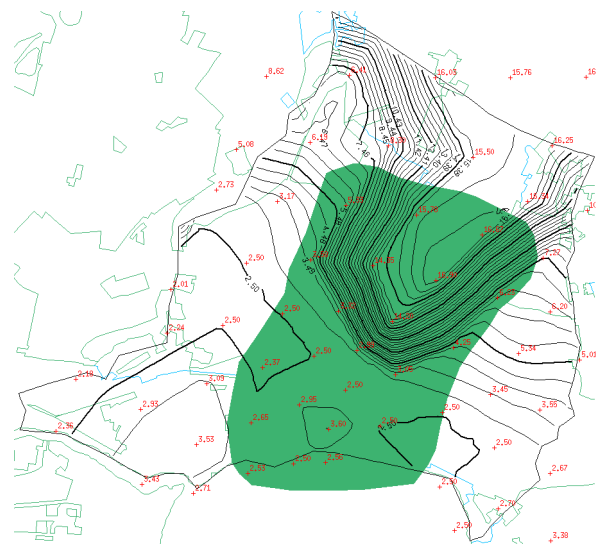
- To import z-elevations for a slice based on borehole data, we have to enter the **Database Regionalization Menu**.
- **Database** option (at the right side of the mesh inspector).
- **Bu z-Coordinates** - An alert box pops up asking if you want to overwrite the existing values.
- **Bu Yes**

In the Data Regionalization Menu different interpolation methods for sample data are offered.



- **Akima inter/extrapolation**
- **Linear**
- In the **neighboring points** field, type **3**. Only the nearest three neighboring data points will be used for the interpolation.

3. Setting up the Model



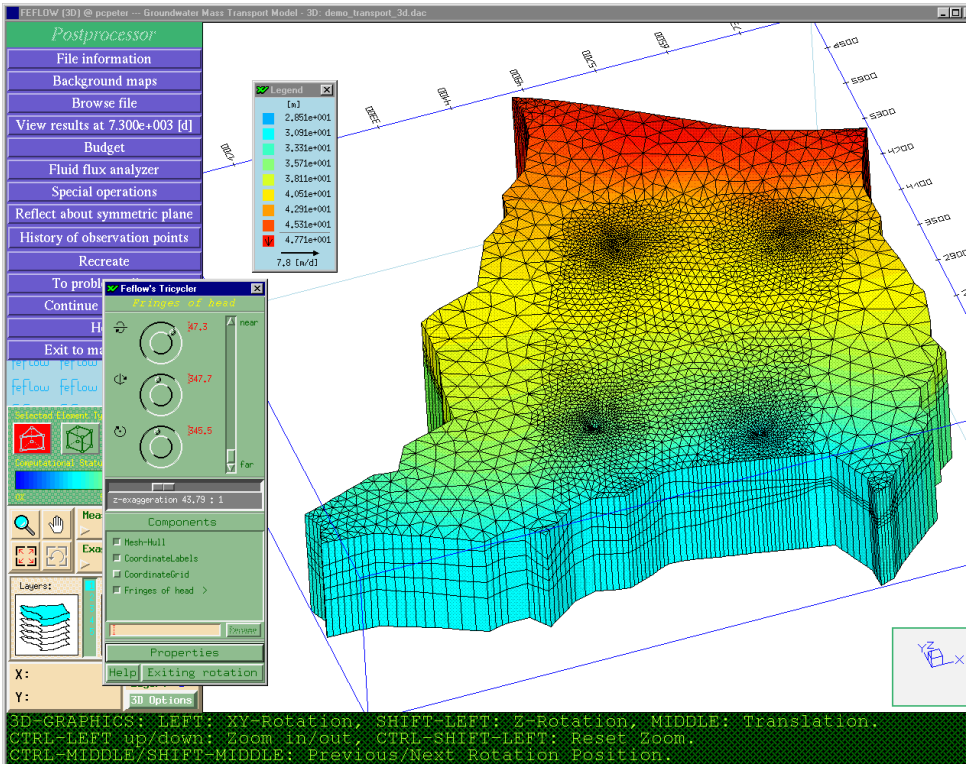
- In the *Acceptable over/undershooting* field, type **1000**. This assures that the interpolation will be executed without smoothing the resulting surface.
- **Import time-constant data** - The file selector appears and allows you to select the database for the interpolation
- Choose **bot_san2.trp** from the *Files* list. This ASCII file (so called 'triplets file') has an *x-coordinate*, *y-coordinate*, *function-value* format.
- **Okay**
- **Start** in the Database Regionalization Window and FEFLOW creates contours for slice 4, the bottom of layer 3 (see figure on the next page).
- Exit the Assign database function by clicking the right mouse button in the working window.

- Browse to the next upper slice using the Layers & Slices browser and repeat the steps above to assign the elevation data according to the list below.

slice	database
4 (bottom of lower aquifer)	bot_san2.trp
3 (top of lower aquifer/ bottom of aquitard)	bot_clay.trp
2 (top of aquitard/ bottom of upper aquifer)	bot_san1.trp
1 (relief)	demo_relief.trp

You can display the resulting model domain in a 3D view:

- **3D-Options** on the lower left side of the screen, hold the mouse button. A small menu pops up
- **Visualize** - A second menu pops up.
- **Body**, release the mouse button. The Tricycler window pops up and the model is displayed in 3-dimensional view (see figure).
- Move, rotate and zoom the model as described in the message bar below the working window.
- **Exiting rotation** in the Tricycler window
- **Return** to exit the 3D slice elevation menu



3.6 Flow Data

The Flow Data section controls all the necessary input parameters for the groundwater flow model. The Flow data menu consists of three submenus: the **Flow initials**, the **Flow boundaries** and the **Flow materials** menus. Click on the corresponding buttons to enter the submenus.

- **Bu Flow Data** to enter the first level of the **Flow Data Menu**

3.6.1 Flow Initials

The Flow initials menu allows you to assign the groundwater surface at the beginning of the transient simulation run.

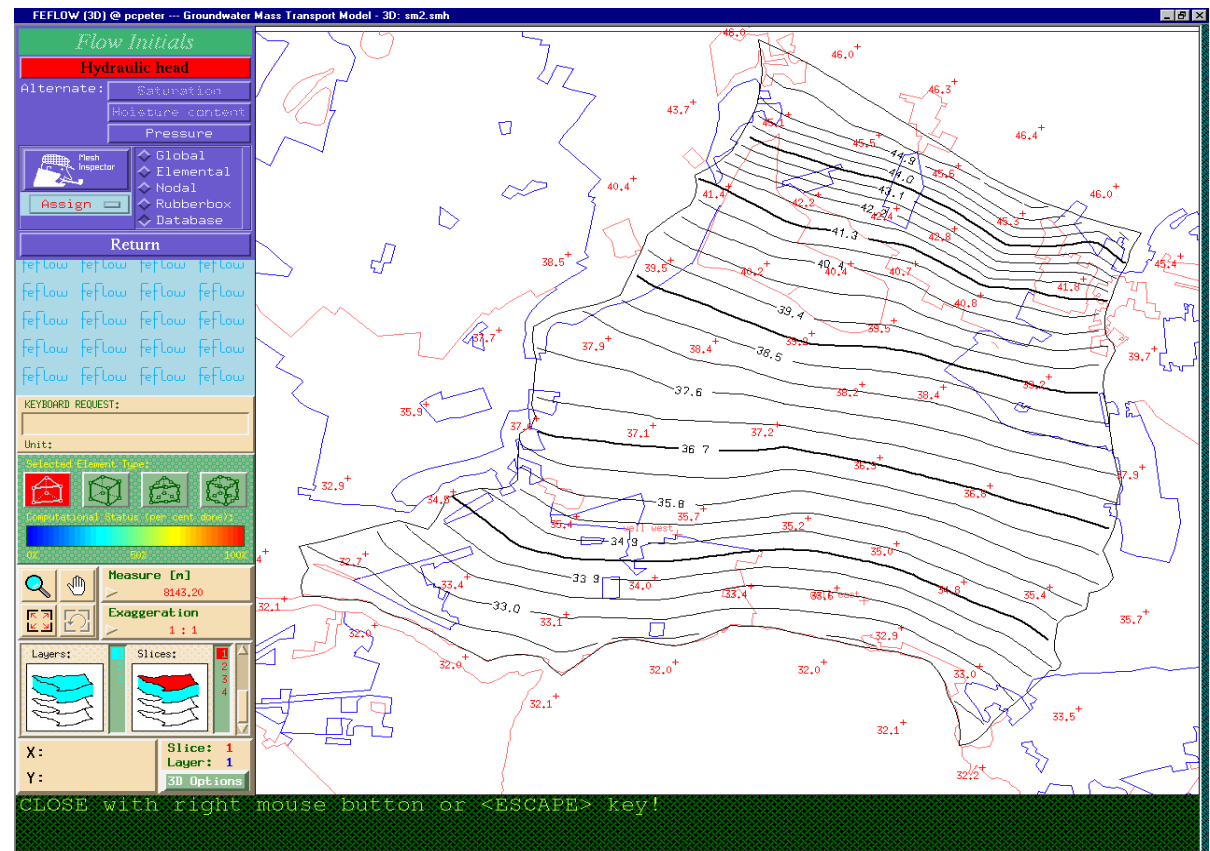
-  **Flow initials**
-  **Database** to import previously prepared point data
-  **Hydraulic head** - The **Database regionalization** window pops up.

3. Setting up the Model

In the *Methods of regionalization*, we select the Akima interpolation technique again.

-  **Akima** inter/extrapolation
- Switch to  **Linear**.
- In the **neighboring points** field, type  **3**.
- In the **Acceptable over/undershooting** field, type  **0**.

-  **Import database** - The file selector appears.
- Choose **demo_head_ini.trp** from the *Files* list.
-  **Okay**
-  **Start** - The regionalization will execute the interpolation of the data just imported (see resulting contours in the figure below).
- Exit the function by clicking the right mouse button in the model domain.

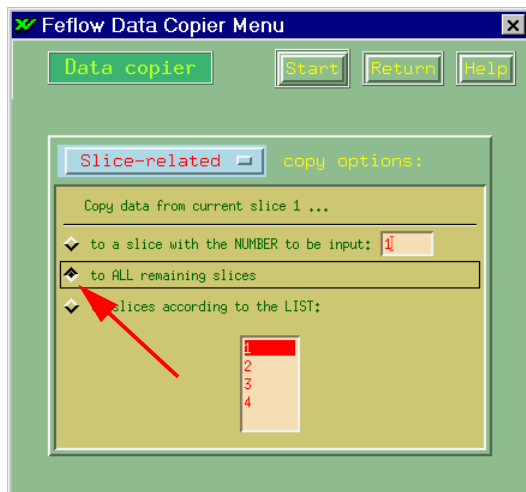


You can visualize the results.

- Switch from  **Assign** to  **Show**.
-  **Hydraulic head** - FEFLOW shows the hydraulic head distribution as colored fringes.
- Switch from  **Show** to  **Vanish**.
-  **Hydraulic head** - FEFLOW resets to normal view.

We will now copy the initial values from slice 1 to all the remaining slices.

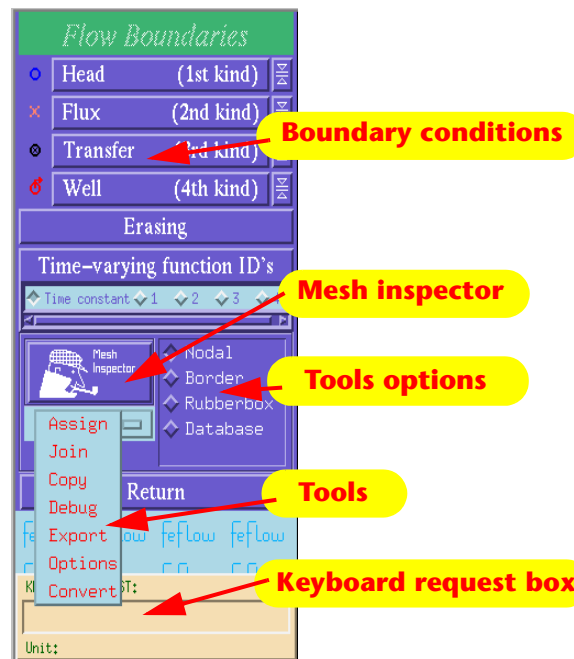
Go to the **Mesh Inspector** icon and switch the light blue Tools button to  **Copy**.



-  **Hydraulic head** - The **Data Copier** appears.
-  **to ALL remaining slices**
-  **Start** - You will be asked by an alert box if you really want to overwrite the current values.
-  **Yes**
-  **Return** to close the Data Copier
-  **Return** to exit the Flow Initials Menu.

3.6.2 Flow Boundaries

We should enter the Flow boundaries menu now and set the boundary conditions for our model. The picture below shows all parts of the menu.



We set the northern boundary condition first, which describes the northern hills. In this exercise, for the sake of simplicity, we will assume a reasonable hydraulic potential along this border. Therefore we will set a constant hydraulic head of 46 m.

-  **Assign**
-  **Border**
-  **Head (1st kind)**

3. Setting up the Model



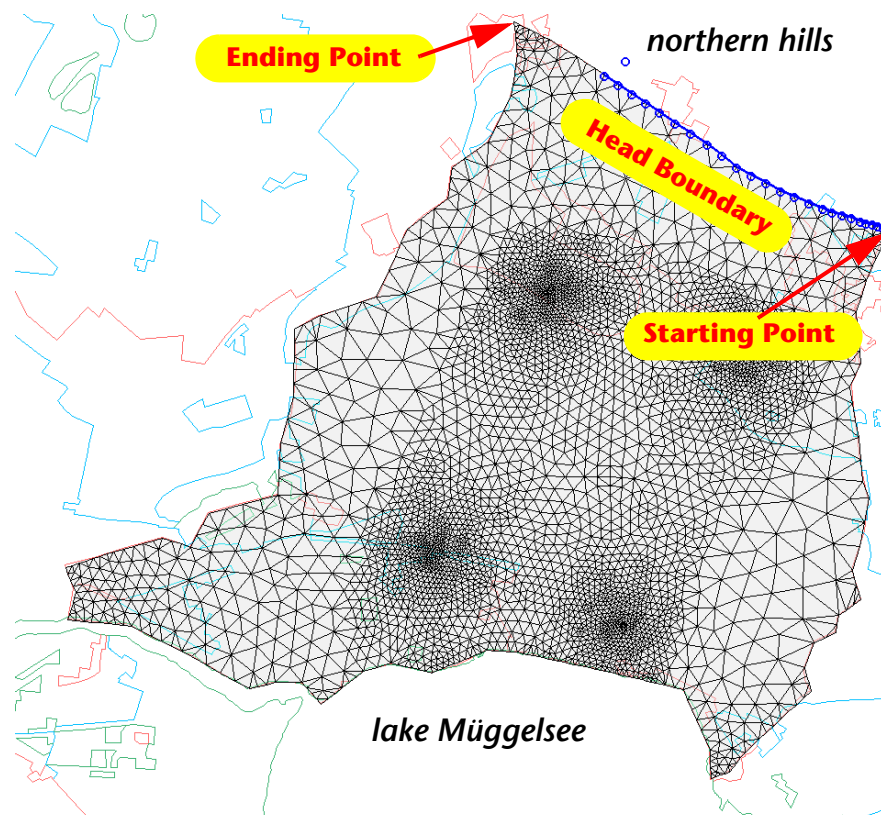
If you have assigned too many nodes, simply select the "Nodal" option in the "Assign" tool, click on the "Hydraulic Head" button, go to the head boundary conditions you want to delete and click on each unwanted node. Click the right mouse button to exit.

- To be able to write the value into the **Keyboard request box**, click into the box or hit the <TAB key>. Type **46** in the Keyboard Request box and hit the Return key.

KEYBOARD REQUEST:
46.000000
Unit: [m]

Move the cursor to the working window. Click the left mouse button over the northeastern corner of the model

domain and hold the button pressed. Move the cursor a little bit to the northwest along the boundary and release the mouse button. Move on along the border and click the last node in the northwest with the left mouse button. Notice that all nodes in between have been assigned a head value of 46 m. The head boundary condition is indicated by blue circles.



Now define the southern boundary conditions describing the shoreline of Lake Müggelsee and the river Müggelspree. We recommend to use the  **Assign - border tool** as described previously for the northern boundary. Set a  **Head** boundary condition with a hydraulic head of  **32.1** meters.

Afterwards switch the light blue Tools button to  **Copy**.

-  **Head (1st kind)** - The **Data Copier** appears.
-  **to ALL remaining slices**
-  **Start** - You will be asked by an alert box if you really want to overwrite the current values.
-  **Yes**
- Click  **Return** to close the Copier menu.

The western and eastern borders of the model have not been assigned any values so far. Usually the rivers would be described by the third kind boundary condition, **Transfer**. For this exercise let us assume the boundaries to be impervious. That means, we do not have to prescribe any boundary condition at these borders.

Once you have assigned the head boundary conditions for the northern and southern borders of our model, assign the wells with their specific extraction rate.

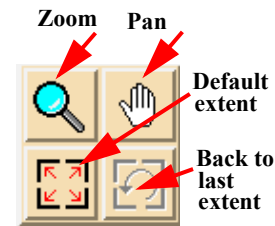
- Switch to the  **Join** tool.
-  **Supermesh**
-  **Load** below the *Supermesh* option. The file selector appears.

The selected supermesh will act as a frame for the finite element mesh. It contains the polygon describing the outer boundaries of the model and the Add-ins for

the exact positioning of the pumping wells and the contaminant sources (waste disposal site, sewage treatment plant).

- Choose **demo.smh** in the *Files* list.
-  **Okay**
- Now the imported supermesh including the Add-ins is displayed in the working window.

The wells (boundary conditions) should be set on the southern part of the model where the two Add-in points are positioned. For a more accurate setting of the wells, you can zoom into this area using the zoomer icons found among the information boxes.

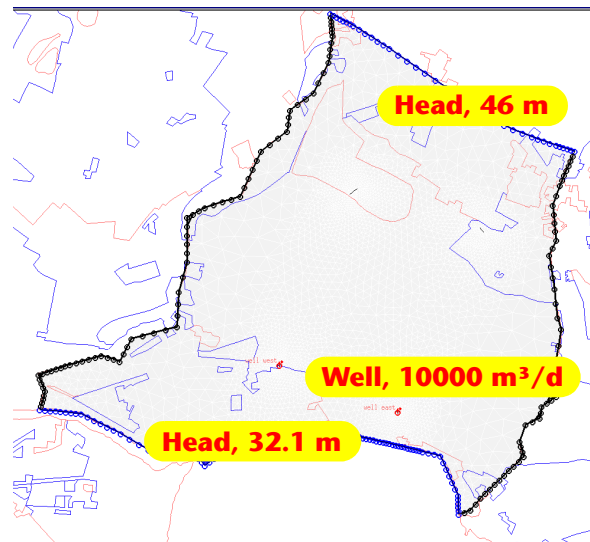


We will now set the wells as boundary conditions of 4th kind with a time-constant discharge rate. The procedure is the same as before:

-  **Join**
-  **Well (4th kind)**
- type a value of  **1.000** m³/d into the Keyboard request box
- Move the cursor over one of the Add-ins. The underlying mesh node should be highlighted by a red square.
- Click the left mouse button to set the boundary condition with the defined discharge rate exactly on the Add-in.
- Repeat this step for the second Add-in.
- Once completed, click the right mouse button to exit this function.

3. Setting up the Model

- We will now assign a value of zero (0) to the wells on all other slices. That causes the discharge rate specified on the first slice to be distributed automatically on the different slices.
- Select the next slice in the Layers & Slices browser located below the Zoom option by clicking on the **Slices** number.
- Repeat the previous steps for each slice using a value of 0 for the wells.
- Click **Bu** **Return** to exit the **Flow boundaries** menu.



3.6.3 Flow Materials

The **Flow Materials Menu** allows you to edit all material parameters which have to be set for modeling a groundwater flow problem.

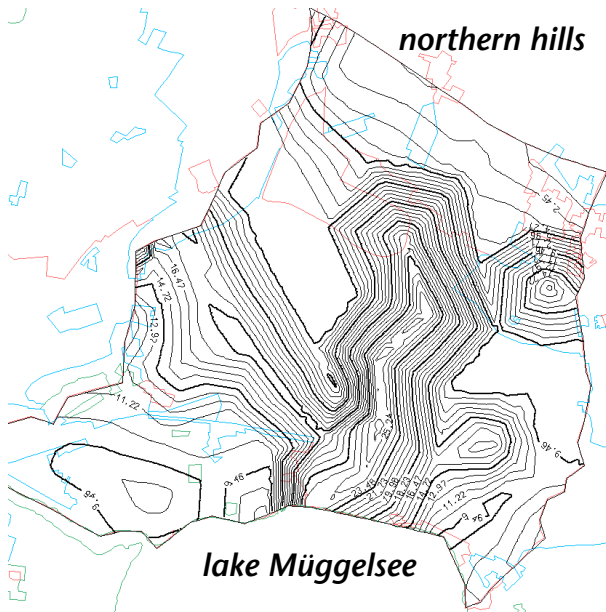
- **Bu** **Flow materials**

The menu structure is similar to the **Flow Boundaries** menu.

Upper Aquifer

The conductivity of the upper aquifer will be imported from borehole samples stored in an ASCII database (syntax: X-coordinate, Y-coordinate, Conductivity). The method used is similar to the assignment of z-elevations in the 3D Slice elevation menu.

- Ensure that the **Assign** tool is set.
 - **Database**
 - **Bu** **Conductivity [Kxx]** - The Database regionalization Window appears.
 - **Akima inter/extrapolation**
 - Set the number of neighbors to **100** **3** and the over/undershooting to **100** **0** %.
 - **Bu** **Import time-constant data**
 - Choose **conduc2d.trp** from the "Files" list.
 - **Bu** **Okay**
 - **Bu** **Start** in the Data Regionalization window.
- FEFLOW will now inter/extrapolate from the borehole data and will display the resulting distribution as a contour map of conductivities (see figure to the right). The contours of our example show low conductivities near the northern hills and a high conductivity flow channel going down from the north to the south dividing the model domain into two equal halves.
- Click the right mouse button after the interpolation is finished.



You can visualize the results of the interpolation as colored fringes.

-  **Show**
-  **Conductivity Kxx** - The value distribution is displayed.
-  **Vanish**
-  **Conductivity Kxx** - FEFLOW resets to normal view.

We will now specify the storativity (drainable porosity) for the layer.

-  **Assign**
-  **Global** - This will assign the same value to the entire layer.
-  **Storativity (drain/fillable)** - Here you assign the drain/fillable porosity.

- An alert box pops up asking if you are sure to overwrite the data. Select  **Yes**.
- Input a value of  **0.1** in the Keyboard request box and press the **Return** key.
- Exit the function by clicking the right mouse button.

Aquitard

Our next step is to prescribe the material properties of the Aquitard. Select **Layer 2** in the Layers & Slices browser.

-  **Assign**
 -  **Global**
 -  **Conductivity [Kxx]** - Again an alert box pops up asking if you really want to overwrite the current values.
 -  **Yes**
 - Input a value of  **1e-6** in the Keyboard request box corresponding to the unit 1e-4 m/s and press the **Return** key.
 - Exit the function by clicking the right mouse button.
- We will assign a new storativity value due to the low conductivity we have set.

-  **Assign**
-  **Global**
-  **Storativity (drain/fillable)**
-  **Yes**
-  **0.01**, press **Return**.
- Quit the function by clicking the right mouse button.

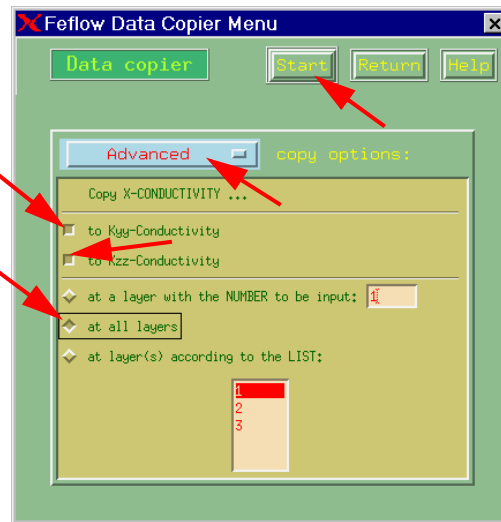
Bottom Aquifer

To the bottom aquifer we set a constant conductivity of 10^{-3} m/s.

- **Layers: 3** in the Layers & Slices browser
-  **Assign**
-  **Global**

3. Setting up the Model

- **Bu** Conductivity [Kxx]
- **Bu** Yes
- **100** 10, press **Return**.
- Exit the function by clicking the right mouse button. Now we copy the K[xx] values to the K[yy] and K[zz] parameter to create isotropic conductivities in all layers.
- **Copy**
- **Bu** Conductivity Kxx - The FEFLOW Data Copier appears.



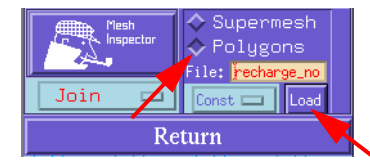
- Switch the light blue button from **Layer-Related** to **Advanced**
- Ensure that the **Copy to Kyy-Conductivity** and **Copy to Kzz-Conductivity** toggles are selected in the upper part of the menu
- **at all layers**
- **Bu** Start



- You will be asked twice by an alert box if you really want to overwrite the current values.
- Select **Bu** Yes both times.
- **Bu** Return to close the Data Copier

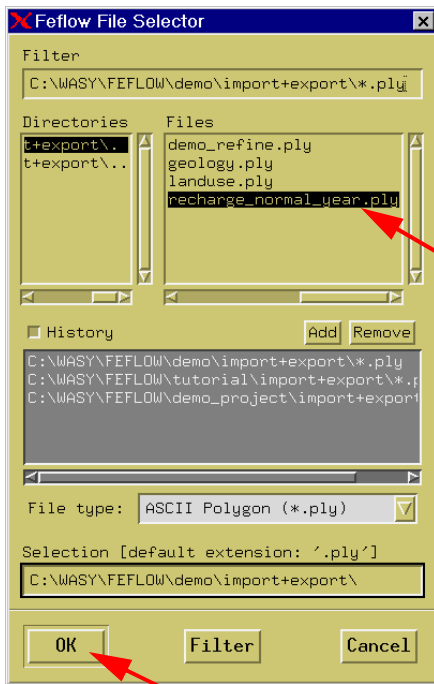
Groundwater Recharge

The assignment of the groundwater recharge will be executed using a template showing the areas of different landuses and an ASCII database containing the attribute data. The polygon file is linked with the database.



- **Layers:** 1 in the Layers & Slices browser
- **Join**
- **Polygon**
- **Bu** Load below the "Polygon" option. The File Selector pops up.
- Choose **recharge_normal_year.ply** in the **Files** list.

3.6 Flow Data



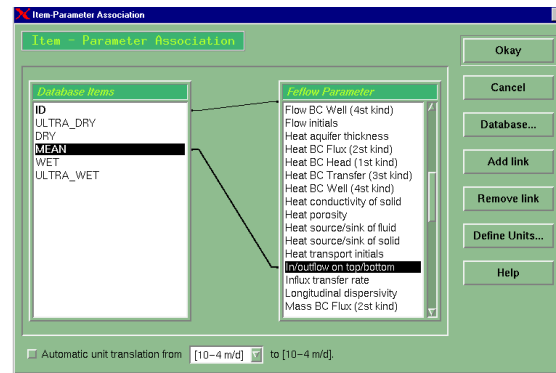
- **Bu** **Okay** to leave the file selector.

Now the **Parameter Association** window becomes visible.

On the left hand side the field names of the database are listed. On the right hand side the FEFLOW parameters are shown. Two pipelines connect the right hand list with the left hand side. The polygon IDs are linked with the field “ID” of the database. The data of the “MEAN” field is linked to the FEFLOW parameter “In/outflow on top/bottom”.

You can add and remove links. However, for this demo exercise just click

- **Bu** **Okay**



- The polygon file is displayed as template in the working window
- **Bu** **In(+)/out(-) flow on top** - An alert box appears telling you the different database link options (see next page).
- **Bu** **Overlay** in the alert box. The database values will automatically be assigned to the model. For detailed information about this database link see the FEFLOW online help.
- click the right mouse button.

Now leave the Flow materials and Flow data menus by hitting **Bu** **Return** two times.

You have created an executable transient flow problem now. Changing the problem class to “flow only” and eventually to „steady flow“ allows you to make a first trial on the simulation and to skip the next sections regarding the transport parameter settings.

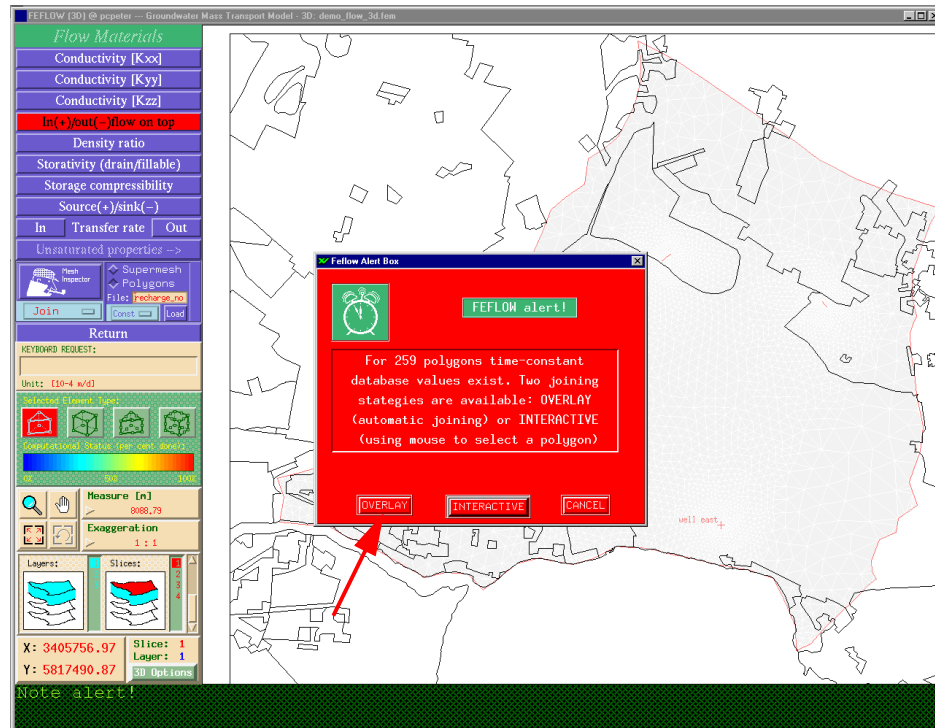


The links have been set before during the preparation of the exercise data. For using your own data they have to be set manually. If using ESRI shape files, the ID link is not necessary!



If FEFLOW does not display the alert box, please check your file access permissions. Writing permission in the WASY/FEFLOW/exercise/import+export/ directory is necessary!

3. Setting up the Model



3.7 Transport Data

Enter the **Bu** **Transport Data** menu from the Problem Editor. This menu contains all editors for defining mass and heat transport parameters. Its structure is similar to the Flow Data menu, i.e., you can set initial values, boundary conditions and material parameters.

The **Mass transport initials**, which describe the initial concentration of the model, remain on the default value of 0 mg/l.

3.7.1 Mass Transport Boundaries

In the **Mass transport boundaries** menu, "fresh water" conditions - at very low concentrations - will be assigned to the outer borders where water can enter the model. The contaminant sources are situated on the top slice.

Click on **Bu** **Mass transport boundaries** to enter the menu.

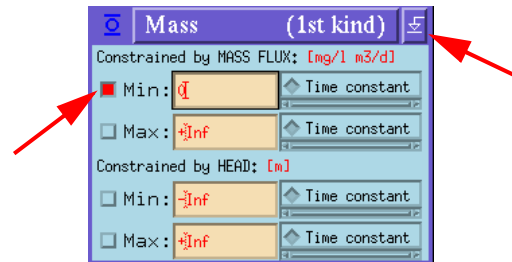
- Switch to the **Assign** tool .

3.7 Transport Data

- **Border**
- **Mass (1st kind)**
- Type the value of **1e-12** mg/l into the Keyboard request box.
- Move the cursor in the working window to the north eastern corner of the model.
- Click and hold the left mouse button on the first node at the northeastern border. Move the cursor along the model boundary still holding the left mouse button. Free the left mouse button. All border nodes you have passed should be marked by a blue circle. Go on until you reach the last node at the northwestern edge. Click the node with the left mouse button.
- Repeat the same procedure for the southern border.
- Exit the function clicking the right mouse button in the working window.

The model boundary is now set for freshwater conditions. These fresh water conditions have the disadvantage that outflowing water is set to this concentration, too, if passing the border. A contaminant plume cannot leave the model freely. Therefore we will limit the activity of the fresh water conditions by so-called **Constraints**. This guarantees that the first-kind boundary condition of fresh water is only set when water enters the model (inflow). On the other hand, if an outflow situation occurs at such a constrained boundary, the first-kind boundary condition of fresh water is automatically switched off and the contaminant mass can freely outflow (if boundary is open for convection). For detailed information see the **Reference Manual**. We assign for northern and southern borders a complementary minimum constraint of 0 m³/d mg/l.

Click on the **arrow-sign** right of the **Mass (1st kind)** button. The corresponding **Constraint Conditions** menu becomes visible.



Input the Min-constraint of **0** mg/l m³/d in the first row.

Click on the **Min** toggle left of the input field for activating the setting.

Click on **Mass (1st kind)**.

Assign the constraint along the northern and southern border. As the border option is not available here keep the left mouse button pressed and try to move the mouse over all border nodes.

Leave the **Constraints** menu via the **arrow-signed button**.

Next we will assign the boundary conditions for the two contamination sources, the sewage water treatment plant and the waste disposal site, which are placed in the northern part of the model.

- **Join**
- **Supermesh**
- **Load** below the Supermesh option. The file selector pops up.
- Choose **demo.smh** from the *Files* list
- **Okay** button. Now the supermesh including the Add-ins is displayed.
- **Mass (1st kind)**
- Enter a value of **500** mg/l in the Keyboard request box in order to represent contaminant release



Constraint and Boundary Conditions are handled differently for fluxes: For Constraint Conditions Inflows are positive (+), Outflows are negative (-). For Boundary Conditions Inflows are negative (-), Outflows are positive (+).

3. Setting up the Model

at the sewage treatment plant to the west and the waste disposal site to the east.

- Press the **Return** key.
- First, move the mouse to the sewage facility (west side) directly over the Add-in line. Notice that the line becomes highlighted.
- Click the left mouse button to assign the contaminant concentration.
- Repeat the step for the waste disposal facility (east side).
- Click the right mouse button to exit the function.

We will now copy these boundary conditions to all remaining slices.

-  **Copy**
-  **Mass (1st kind)** - The **Data Copier** appears.

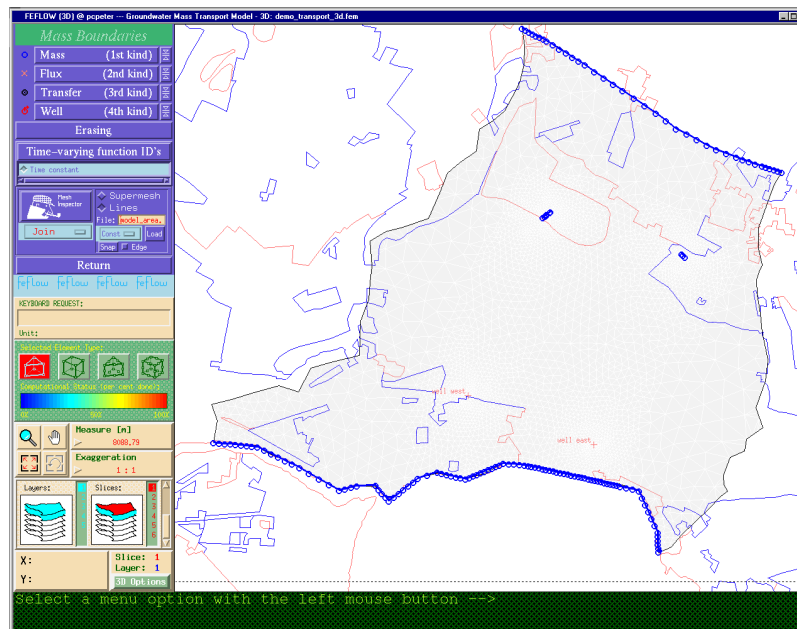
- Notice the possibility to copy boundary conditions with or without the related constraint conditions.

Choose the option  **Boundary conditions with related constraints if exist.**

-  **to ALL remaining slices**
-  **Start**
- A FEFLOW alert box pops up. Select  **Yes** to overwrite the values.
-  **Return"** to exit the Data Copier.

We will now leave the **Boundaries** menu and enter the **Mass transport materials** menu.

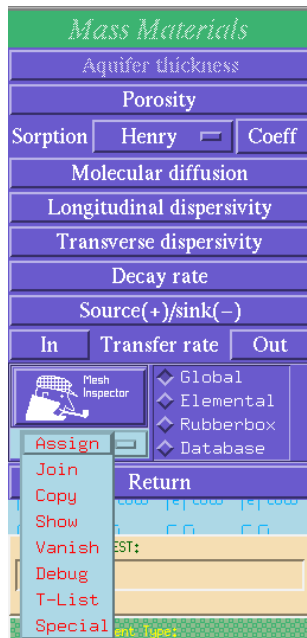
-  **Return**
-  **Mass transport materials**



3.7.2 Mass Transport Materials

All material parameters concerning mass transport are set in the Mass Transport Materials Menu. We will first assign the values for the total porosity of the aquifer for the top layer:

-  **Assign**
-  **Global**
-  **Porosity**
- A FEFLOW alert box appears. Select  **Yes** to overwrite the default value globally.
- Input a value of  **0.2** in the Keyboard request box.
- Press the **Return** key and leave the function hitting the right mouse button.



As the next step we specify the contaminant mass transport dispersivity for our model.

-  **Assign**
-  **Global**
-  **Longitudinal dispersivity**
-  **Yes**
- Input a longitudinal dispersivity value of  **70** m in the Keyboard request box.
- Press the **Return** key and leave the function hitting the right mouse button.
- Repeat the steps for  **Transverse dispersivity**, assigning a value of  **2.5** m
- Copy the dispersivity values to all layers using the  **Copy** tool.
- Exit the **Transport data** menu by selecting  **Return** and go up to the **Problem Editor Menu**.



„Porosity“ as a mass transport material property means the total porosity in contrast to the „storativity“ value of the flow materials, which only takes into account the drainable porosity.

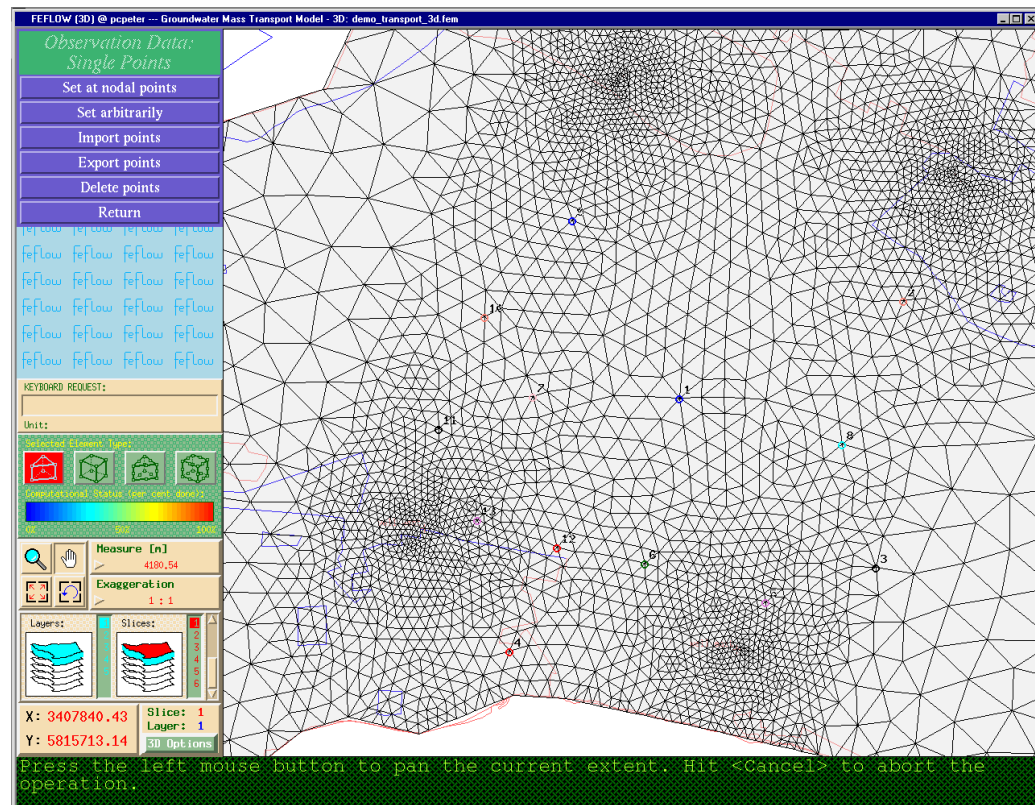
3. Setting up the Model

3.8 Reference Data

At last you should set some observation points to the top slice. For the points, all resulting data such as hydraulic heads or contaminant concentrations are visualized online in diagrams during the simulation run.

- **Bu** Reference data
- **Bu** Observation single points

- **Bu** **Import points** - The file selector pops up. Now you will import the observation points from an ASCII database.
- Choose **demo_obs_pnts.pnt**
- **Bu** **OK** - The points are loaded and visualized as colored circles.
- Leave the reference data editor for the **Problem editor** by clicking on the two **Bu** **Return** buttons.



3.9 Reconfigure 3D Task

The basic model parameters are now assigned, but we will add two additional layers to increase the accuracy of modeling the aquitard. Therefore we have to re-enter the **Layer configurator** via the **3D-Slice elevations menu**.

No.	Status	Button
1	Fixed	View...
2	Fixed	View...
3	Fixed	View...
4	Fixed	View...

- **Bu** 3D-Slice elevation in the **Problem Editor**
- **Bu** Reconfigure 3D Task

- Type a decrement of **1000** m in the **Decrement** box. This unrealistic high decrement makes FEFLOW divide the aquitard equally spaced in three new layers.
- Change the **Number of layers** in the input box at the upper left corner of the menu to **5**.
- Press Return.
- The **Slice Partitioner** pops up
- **Partitioning according to the list**
- Move the cursor to the **Partitioning list**.
- Scroll the **Partitioning list** using the vertical slider bar at the right of the menu and replace the **2** automatically set by FEFLOW beneath 4. *slice fixed* by **0**.
- Between the 2. *slice is fixed* and 3. *slice is fixed* replace the **0** by **2**.

The z-elevations of the new slices are interpolated from the nodal values of the upper and lower slice. Therefore the new slice will divide the aquitard evenly.

- **Bu** OK

There are already 4 fixed slices and 2 new (plane) slices should be created!

How to partition these new slices? **Partitioning according to the LIST**

Partitioning list (editable if set):

2. slice is fixed	add: 2
3. slice is fixed	

Additional option for partitioning between TWO fixed slices:
Spacing from the **UPPER** fixed slice(s)

3. Setting up the Model

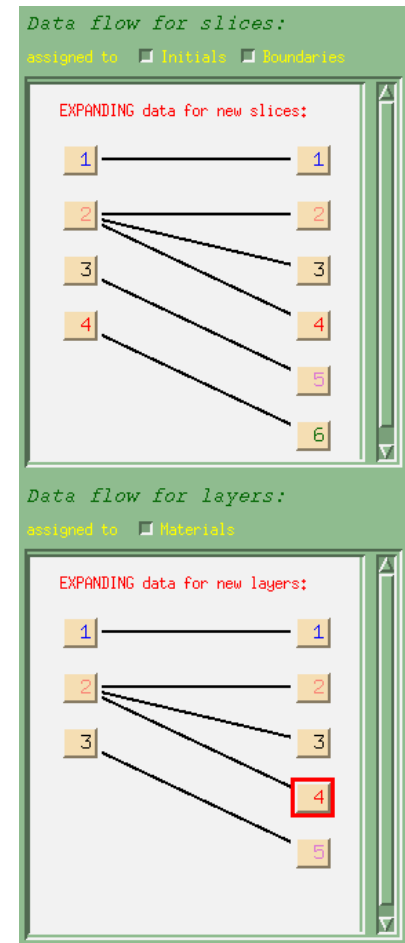
Now the aquitard is divided in three layers. To ensure that the data copy is executed correctly from the old to the new slices, we will utilize the Data Flow lists on the right of the **3D-Layer configurator** menu. As shown in the next picture, there are two lists that provide control over the data flow between the previous and the new layers. The upper control called **Data flow for slices** describes the data flow of the initial conditions and boundary settings 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.

The data flow in the upper list for the initial and boundary conditions is set automatically by FEFLOW. The information of the old slice 2 will be inherited by the new slices 2, 3 and 4. The data flow in the lower list for the materials data describes the same data characteristics from the old bottom layer (lower aquifer) to the new layers 2, 3 and 4. For our example, we will change the links.

Go to the lower **Data flow for layers** list.

- Click  in the left column. (represents the aquitard)
- Click  in the right column. A line will now connect the old layer 2 with the new layer 3. The data are inherited.
- Click  in the right column. A line will now connect the old layer 2 with the new layer 4.
- The resulting data flow lists should look like the figure to the right.

-  **OK** to exit the Layer configurator.
- Exit the **Problem editor** for the top **Shell** menu by clicking on the various  **Return** buttons.



The Simulator

4

In this step the simulation run is performed.

Enter the simulator via the  **Simulator** shell menu by selecting  **Run ...**

Start the simulation by clicking on  **(Re-)Run simulator**. Notice that FEFLOW automatically generates various windows that detail the ongoing results for the wells, observation points, hydraulic head and contaminant concentrations. These windows display the results as diagrams. For the possibilities of editing the diagram properties please see the online help.

The transient simulation will require approximately 10 minutes on a Pentium III 1.1 GHz. If you don't want to wait, we have already prepared a results file, which you can view in the **Postprocessor Menu**.

Having interrupted or completed the simulation, you can analyze the results of the current time step with the options offered in the Halt & View Results menu (isoline maps, velocities, particle tracking, data export), the Budget analyzer, the Fluid flux analyzer or in the spatial operations window. The same possibilities are available in the Postprocessor, where analyzing is enabled for all of the saved time steps.

-  **Return** to exit the **Simulator**.



*If you have skipped the previous steps, please load the file **demo_transport_3d.fem** via the Load finite element problem option from the File menu before beginning this exercise.*

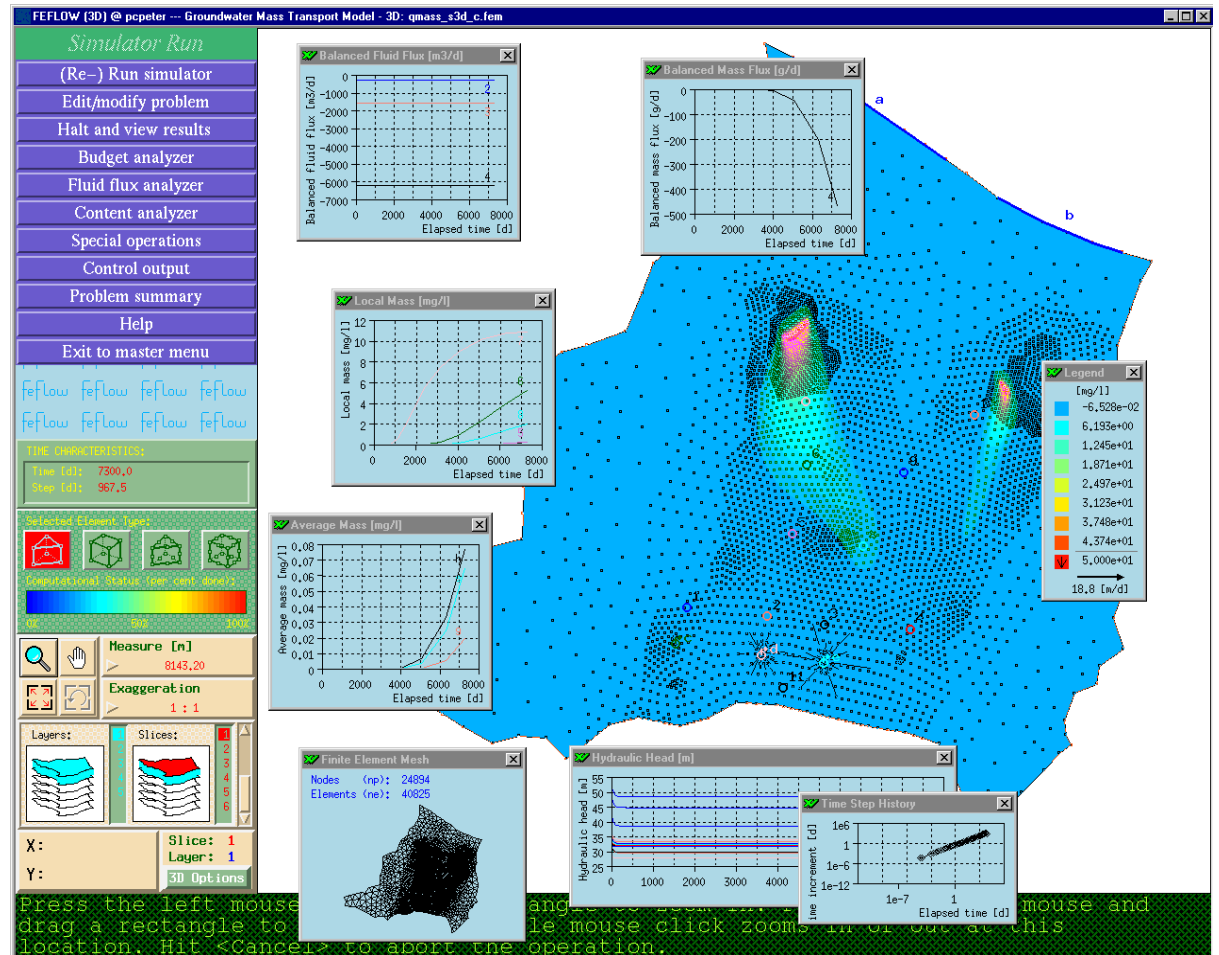


You can stop the simulation by pressing <Esc>. Restart by clicking on "(Re-)Run simulator".



You can perform a steady flow/steady transport simulation to save time. In this case switch the 'Problem class' to 'Steady flow/steady transport' and change the 'Maximum number of iterations per time step' in the 'Temporal & control data' window to '15'.

4. The Simulator



The Postprocessor

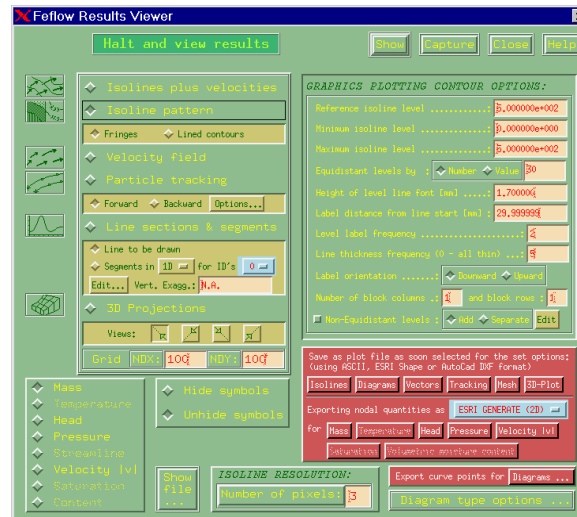
5

In this last step all the postprocessing is done. The results are visualized in 2D as well as in 3D, the budget is computed and particle tracking is performed.

The postprocessor allows the user to evaluate, analyze and export the computational results. Additionally, the Postprocessor Menu controls all saved time steps of the simulation and allows a continuation of the simulation from the last time stage.

To use the postprocessor, you must have saved the simulation results in a so called *.dac file. Because the demo version of FEFLOW does not allow you to save the results, we have prepared the results for you.

-  **Postprocessor** shell menu.
-  **Load and run ...** - The file selector appears.
- Choose **demo_transport_3d.dac** from the *Files* list.
- An alert box pops up asking if you want to abandon the current *.fem file.
- Select  **Yes**.
- **demo_transport_3d.dac** is loaded and you enter the Postprocessor.



5.1 2D Visualization

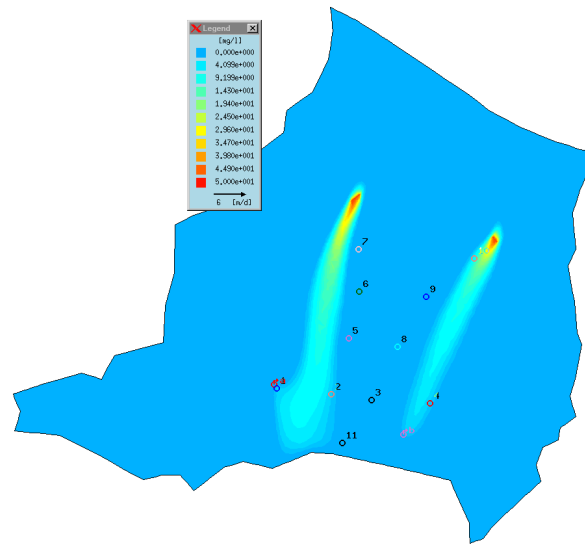
To get an idea of the final results, start with the 2D visualization tools:

-  **View Results at 7.300 e+003 [d]** - The **Results Viewer** appears.

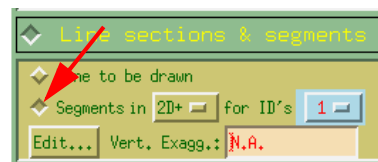
-  **Isoline pattern** in the upper left part of the window
-  **Show** - The mass distribution for the current slice is visualized
- Click the right mouse button. The **Results Viewer** reappears.

5. The Postprocessor

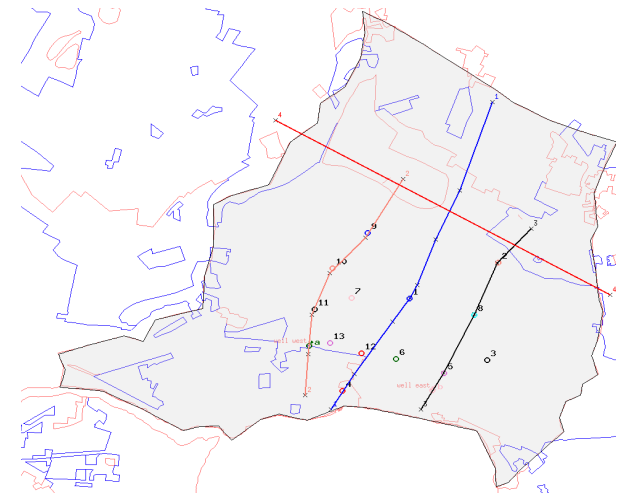
Cross sections can be visualized for all parameter distributions along lines. The lines can be drawn in the working window or imported in ESRI generate format. First draw a cross section:



- Click  **Edit...** below the **Line sections & segments** entry and hold the mouse button.



-  **Draw segments**
- Now you have to draw the line for the cross section(s) on the working window. Refer to the follow figure:

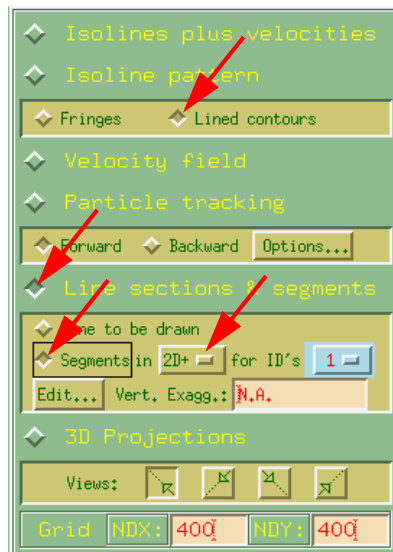


- Click the start point for the cross section on the model. Digitize the cross section by clicking on the model.
- Click the right mouse button to end the editing of a line. The line gets an ID number.
- Click with the right mouse button a second time. The **Results Viewer** reappears.

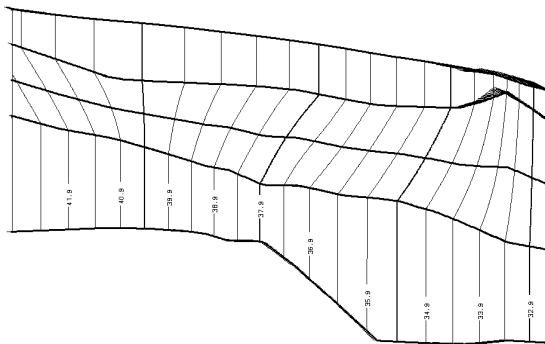
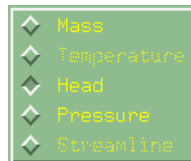
Now we will define an isoline contour for the hydraulic head along the cross section that also shows the velocities at the nodes:

-  **Lined contours** in the upper left part of the window.
-  **Line sections & segments**
-  **Segments**
-  **2D+**. The + indicates additional visualization of velocity vectors at the nodes.

5.1 2D Visualization



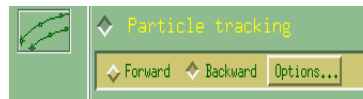
- **Head** in the lower left part of the window for analyzing the hydraulic head distribution.
- **Show** - A vertical cross section of the hydraulic head distribution is displayed.



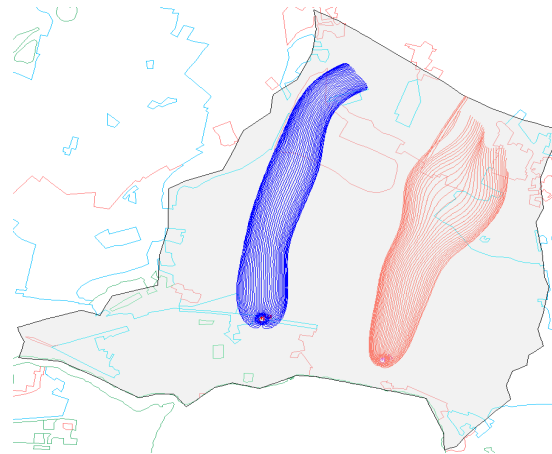
- Click with the right mouse button. The **Results Viewer** reappears.

We will now analyze the flow pathlines to the wells using the particle tracking option:

- Switch to **Slice 3** in the Layers & Slices browser at the lower left side of the screen to start the pathlines from this nodal plain.
- **Particle tracking**



- **Backward**
- **Options...** - The **Pathline editor** appears.



- **multiple pathlines around a single well** in the left row. This option allows you to start multiple pathlines exactly from a well to visualize capture zones.
- **Close** to leave the Pathline editor.

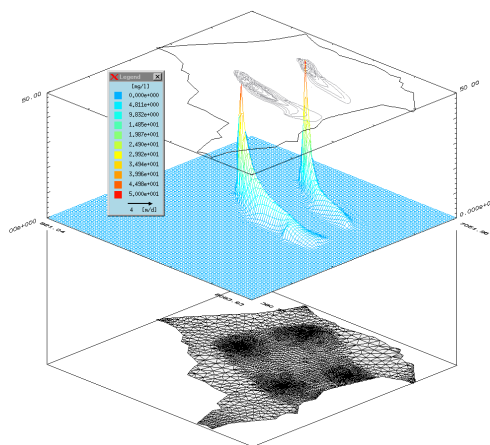
5. The Postprocessor

- **Bu Show** to start the pathline function. The working window displays the border of the model and symbols for the wells.
- Click on a well location to create the backward (reverse) particle tracking.
- Click with the right mouse button to exit the function. The Results Viewer reappears.

At last try out the Pseudo 3D Visualization for parameter distributions:



- **3D Projections.**
- **Mass** in the lower left part of the menu.
- **Bu Show** - The mass distribution is visualized as a 3D plot, where the quantity of mass concentration is visualized along the z axis.
- Click with the right mouse button to exit the function. The Results Viewer reappears.
- **Bu Close** to close the Results Viewer.

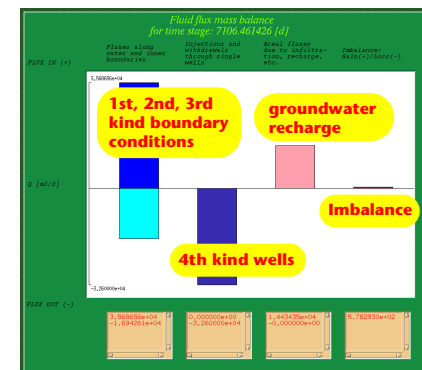


The **Budget analyzer** calculates the water, mass and heat balancing for the entire model, single slices or subregions to be defined by interactive input or import of lines or polygons in ASCII format. The budget can be analyzed for a special time step or over a defined time period.

- **Bu Budget analyzer** - The **Budget analyzer** opens. As default, **Fluid flux mass** and **Total balancing on all inner and outer boundaries (balance checking)** are selected.
- **Bu Start** - The total balancing of the model for the current time step is executed.
- Click the right mouse button to exit the budget function.

Note that FEFLOW has drawn blue and red circles on the model. Red circles show nodes where water enters the model and blue circles indicate water leaving the model. The diameter of the circles shows the relative quantity.

In addition, a window has popped up displaying block diagrams for the in (+) and out (-) flow due to the different boundary conditions and groundwater recharge. The last column shows the imbalance (loss or surplus) of the model.



5.2 3D-Options Menu

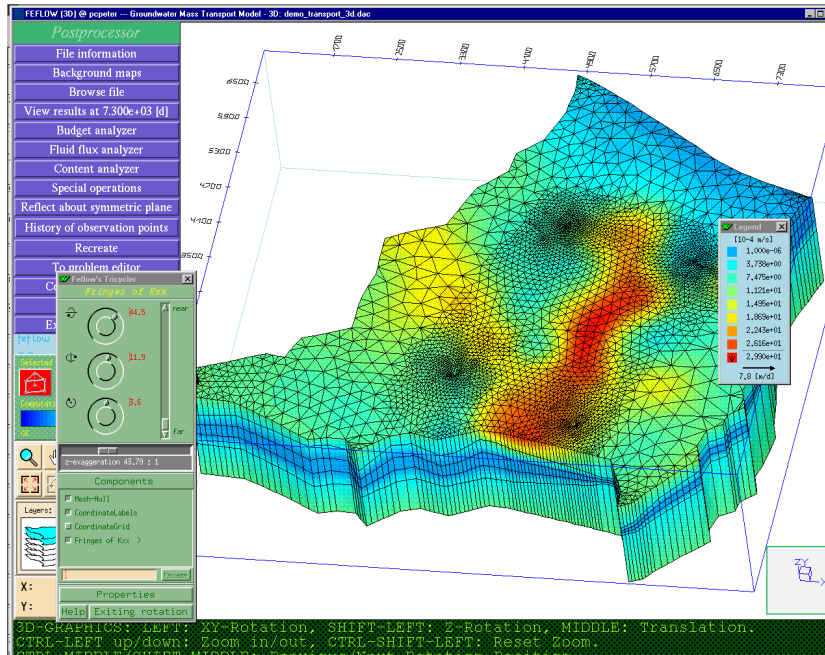
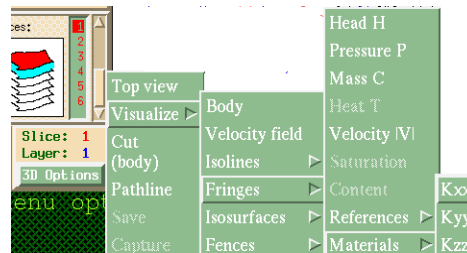
For a detailed 3D description of our model, and to review all parameters, we exit to the main menu.

You can enter the 3D options menu from either the **Shell** menu, the **Postprocessor** or the **Problem Editor**. You enter the 3D Options menu via the green **3D Options** button at the lower left side of the screen, below the Layers & Slices browser.

- Click **3D Options** and hold the mouse button.
- **Visualize**
- **Fringes**
- **Materials**
- **Kxx**

The conductivity in the x-direction is visualized as a 3D object. Additionally the **Tricycler** menu appears.

- Click on the model, hold the left mouse button and move the mouse in order to rotate the model.
- Press the **<Ctrl>** key and move the mouse up and down to zoom in or out. Click the middle mouse button to pan the model.



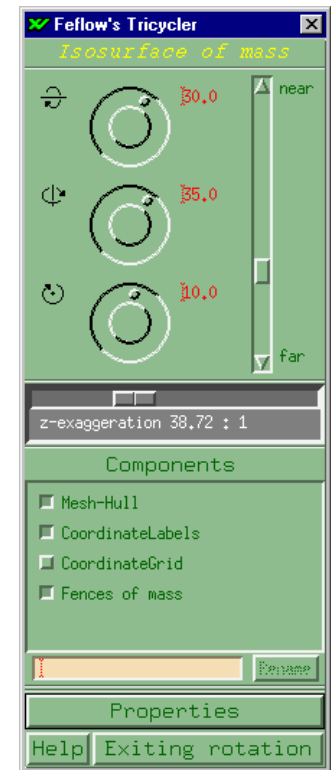
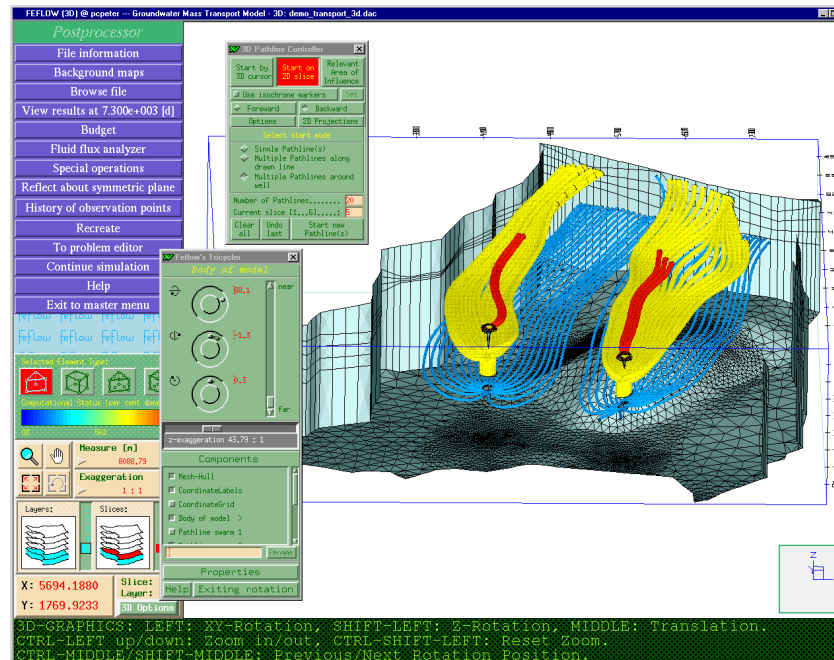
5. The Postprocessor

5.2.1 3D-Pathline Analyzer

One of the main 3D analyzing tools FEFLOW offers is the 3D pathline visualization. You can start the pathlines by positioning the starting point via the 3D cursor on the model (move the red handlers), by starting them from a specified slice or by importing an ASCII file for the computation of the Relevant Area of Influence (RAI) for a well. We will start now some pathlines from the second slice.

- **Bu** **3D Options**, hold the mouse button
- **Pathline** - The 3D Pathline Controller is opened.

- **Bu** **Start on 2D slice**
- Insert **100** **2** in the **Current slice** field
- **Bu** **Start new pathlines** - The selected slice is visualized. The cursor is in the zoom mode.
- Click with the right mouse button to deactivate the zoomer.
- Click any number of arbitrary points in the northern part of the model. Each point will be a starting point for a pathline.
- Click with the right mouse button. The visualization is executed.



5.2.2 3D-Isosurfaces

The visualization of isosurfaces gives the predicted plume of the contaminant plume. Visualize now an **Isosurface** for mass concentration:

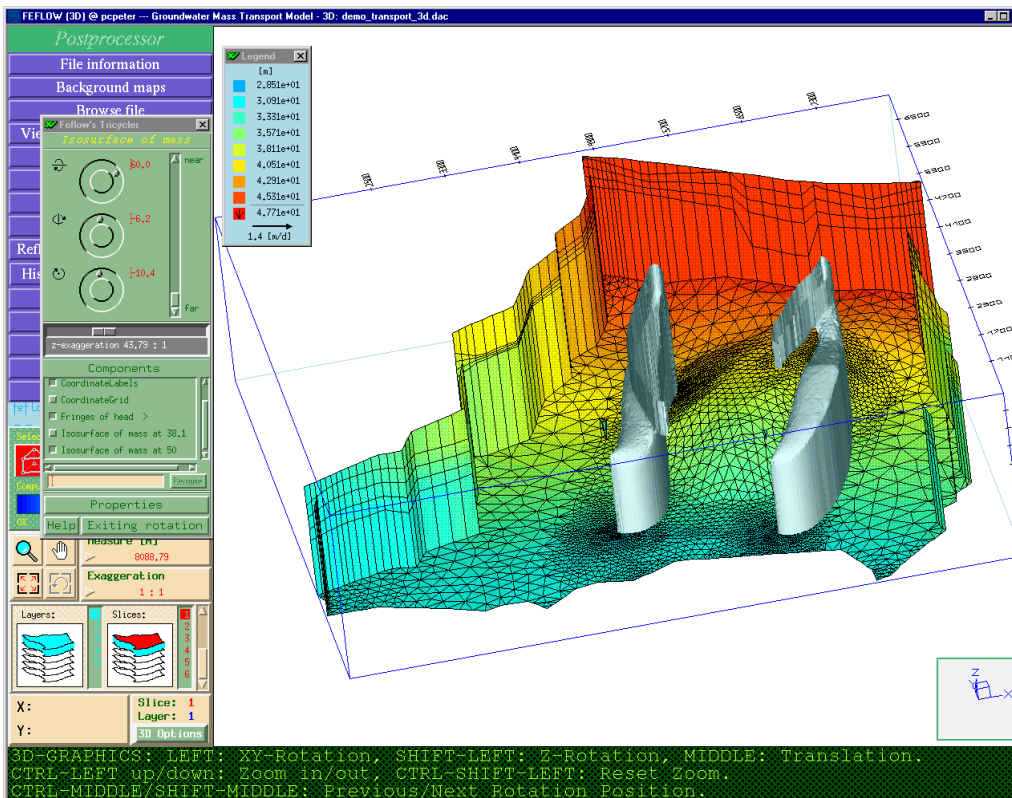
- **Bu 3D Options**, hold the mouse button
- **Visualize** -The 3D Pathline Controller is opened.
- **Isosurfaces**
- **Mass C**

A mean isosurface for the mass concentration is visual-

ized. For defining different isosurfaces just

- click **Bu Properties** in the Tricycler. The Properties menu pops up.
- click the **General** folder and insert the isosurface value, e.g., **100 50 mg/l**.
- press the **Return** key. The isosurface is displayed.

In the **Components** part of the Tricycler you can switch the different isosurfaces on and off. To exit the 3D Options menu, simply click on the **Bu Exiting rotation** button located in the lower right corner of the Tricycler menu.)



5. The Postprocessor

5.3 Final Remarks

This completes the FEFLOW Demonstration Exercise. Please take some time to familiarize yourself with the many features offered by FEFLOW.

As a next step for getting introduced to FEFLOW we recommend to go through the Tutorial, which you can find in the second part of the User's Manual.

For further questions please refer to the online Help found in all menus and windows by hitting <F1> or clicking the Help button.

For information on special topics please have a look at the FEFLOW documentation, which you find on your FEFLOW CD in the 'doc' directory. If you are in possession of a FEFLOW license, you have gotten the doc-

umentation in book-form, too. Pay attention to the following documents in particular:

- **User's manual** ([users_manual.pdf](#)): FEFLOW handling, tutorial for advanced users, introduction to the interface manager IFM
- **Reference manual** ([reference_manual.pdf](#)): all the theory behind FEFLOW
- **White Papers Vol. I and II** ([white_papers_vol1.pdf](#) and [white_papers_vol2.pdf](#)): papers on special topics (benchmarks, numerical methods etc.)

Browse to <http://www.feflow.info> for up-to-date information concerning current releases, FAQs, etc.

If you wish to attend a training course on the application of FEFLOW please contact your local distributor or WASY for proposed dates and course programs.

