

# FEFLOW 7.0

## User Guide

*FEFLOW is a flexible-mesh groundwater flow and transport simulation software, offering convenient pre and post processing options.*



## PLEASE NOTE

### COPYRIGHT

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI. For details please refer to your 'DHI Software Licence Agreement'.

### LIMITED LIABILITY

The liability of DHI is limited as specified in Section III of your 'DHI Software Licence Agreement':

'IN NO EVENT SHALL DHI OR ITS REPRESENTATIVES (AGENTS AND SUPPLIERS) BE LIABLE FOR ANY DAMAGES WHATSOEVER INCLUDING, WITHOUT LIMITATION, SPECIAL, INDIRECT, INCIDENTAL OR CONSEQUENTIAL DAMAGES OR DAMAGES FOR LOSS OF BUSINESS PROFITS OR SAVINGS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION OR OTHER PECUNIARY LOSS ARISING OUT OF THE USE OF OR THE INABILITY TO USE THIS DHI SOFTWARE PRODUCT, EVEN IF DHI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. THIS LIMITATION SHALL APPLY TO CLAIMS OF PERSONAL INJURY TO THE EXTENT PERMITTED BY LAW. SOME COUNTRIES OR STATES DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL, SPECIAL, INDIRECT, INCIDENTAL DAMAGES AND, ACCORDINGLY, SOME PORTIONS OF THESE LIMITATIONS MAY NOT APPLY TO YOU. BY YOUR OPENING OF THIS SEALED PACKAGE OR INSTALLING OR USING THE SOFTWARE, YOU HAVE ACCEPTED THAT THE ABOVE LIMITATIONS OR THE MAXIMUM LEGALLY APPLICABLE SUBSET OF THESE LIMITATIONS APPLY TO YOUR PURCHASE OF THIS SOFTWARE.'

### PRINTING HISTORY

November 2015



## CONTENTS



<b>1</b>	<b>Introduction</b>	11
1.1	Welcome to FEFLOW	11
1.2	The FEFLOW Package	11
1.3	FEFLOW Documentation	12
1.4	Scope and Structure	13
1.5	Notation	13
<b>2</b>	<b>What's New in FEFLOW 7.0?</b>	15
2.1	3D Unstructured Meshing	15
2.2	Revised Model Generation Process	15
2.3	Conversion to Quad-/Hex-Dominant Mesh	15
2.4	Redesigned 3D Layer Configurator	16
2.5	Spatial-Units Panel Split into Selections and Entities	16
2.6	Control-Volume Finite Element Method	16
2.7	Anisotropic Macrodispersion	16
2.8	Parameter Statistics	17
2.9	New Features in FePEST	17
2.10	Additional functionality	18
2.11	Programming Interfaces IFM & Python	18
<b>3</b>	<b>The User Interface</b>	19
3.1	Philosophy	19
3.2	Graphics Driver	19
3.3	Customizing the Interface	20
3.4	View Windows	21
3.4.1	Types of View Windows	21
3.4.2	Navigation	21
3.5	Tutorial	21
<b>4</b>	<b>Working with Maps</b>	25
4.0.1	Maps—What For?	25
4.0.2	Raster Maps/Maps Containing Data	25
4.0.3	Microsoft Excel Worksheets and Access Tables	25
4.0.4	Geodatabases	26
4.0.5	2D/3D Maps	28
4.0.6	Coordinate Systems	29
4.0.7	Georeferencing Maps	30
4.0.8	Handling Maps	30
4.0.9	Map Data	32
4.0.10	Map Export	33
4.0.11	Tutorial	33
4.0.12	Map Layers	35
4.0.13	Map Data	36
<b>5</b>	<b>Starting With a New Model</b>	39
5.1	Model-building Process	39
5.2	Finite-element Mesh Options	39
5.3	Starting a new Model	41



<b>6</b>	<b>Supermesh Design</b>	43
6.1	What is a Supermesh?	43
6.1.1	Supermesh Elements for 2D Meshing	43
6.2	Editing 2D Supermesh Features	44
6.3	Converting Map to Supermesh Features	45
6.4	Export of Supermesh Features	46
6.5	Tutorial	46
6.5.1	Tools	46
6.5.2	Polygons, Lines and Points	46
6.5.3	The Pin Coordinates Toolbar	48
6.5.4	Supermesh Import via Maps	48
<b>7</b>	<b>Finite-Element Mesh</b>	53
7.1	Spatial Discretization	53
7.2	2D Mesh Generation	53
7.2.1	Mesh Generation Algorithms 2D	54
7.2.2	2D Mesh Import from Maps	55
7.3	2D Mesh Editing	55
7.4	3D Mesh Generation	55
7.4.1	3D Mesh Generator TetGen	55
7.4.2	3D Mesh Import from Maps	56
7.5	3D Mesh Editing	57
7.5.1	Modifications of Layered Meshes	57
7.5.2	Pinching Layers with Tetrahedral Elements	57
7.5.3	Re-meshing with Tetrahedral Elements	57
7.6	Mesh-Property Check	57
7.7	Mesh-Element Deactivation	59
7.8	3D Discretization	59
7.8.1	Layered approach	59
7.8.2	Partially Unstructured Approaches	61
7.8.3	Fully Unstructured Approach	62
7.9	Geotransformation	64
7.10	Tutorial	64
7.10.1	Tools	64
7.10.2	Mesh Generation	65
7.10.3	Editing the Mesh Geometry	68
7.10.4	Extending a Model to 3D	71
<b>8</b>	<b>Problem Settings</b>	73
8.1	Problem Class	73
8.1.1	Physical Processes	73
8.1.2	Dimension and Projections	74
8.1.3	Temporal Settings	75
8.1.4	Error Tolerance	76
8.1.5	Free Surface	77
8.1.6	Anisotropy of Hydraulic Conductivity	81
8.2	Equation-System Solvers	82



8.3	Tutorial . . . . .	82
8.3.1	Confined/Unconfined Models . . . . .	83
8.3.2	Unsaturated Models . . . . .	84
8.3.3	Transport Models . . . . .	84
8.3.4	Steady / Transient Models . . . . .	86
<b>9</b>	<b>Working with Selections . . . . .</b>	<b>89</b>
9.1	Introduction . . . . .	89
9.2	Selection Tools . . . . .	89
9.3	Storing Selections . . . . .	91
9.4	Tutorial . . . . .	92
9.4.1	Tools . . . . .	92
9.4.2	General Remarks . . . . .	93
9.4.3	Manual Selection . . . . .	93
9.4.4	Map-based Selections . . . . .	95
9.4.5	Expression-based Selection . . . . .	96
9.4.6	Storing Selections . . . . .	96
9.4.7	Converting Selection Types . . . . .	97
9.4.8	Exporting and Reimporting Selections . . . . .	97
9.4.9	Using Selections as Navigation Reference . . . . .	98
<b>10</b>	<b>Parameter Visualization . . . . .</b>	<b>99</b>
10.1	Introduction . . . . .	99
10.2	View Windows . . . . .	99
10.3	Model Geometry and Data Plots . . . . .	99
10.4	Visualization Options . . . . .	100
10.5	Clipping and Carving . . . . .	100
10.6	Inspection . . . . .	101
10.7	Scene Library . . . . .	101
10.8	Stereoscopic Visualization . . . . .	102
10.9	Tutorial . . . . .	103
10.9.1	View Windows . . . . .	103
10.9.2	Add Model Geometry and Parameters . . . . .	103
10.9.3	Visualization Options . . . . .	104
10.9.4	Clipping and Carving . . . . .	105
10.9.5	Scene Library . . . . .	106
<b>11</b>	<b>Parameter Assignment . . . . .</b>	<b>109</b>
11.1	Introduction . . . . .	109
11.2	Input Parameters . . . . .	109
11.2.1	Geometry . . . . .	110
11.2.2	Process Variables . . . . .	110
11.2.3	Boundary Conditions . . . . .	111
11.2.4	Material Properties . . . . .	116
11.2.5	Auxiliary Data . . . . .	117
11.2.6	User Data . . . . .	117
11.2.7	Discrete Features . . . . .	118
11.3	Assignment of Constant Values . . . . .	118



11.4	Assignment of Time Series Data . . . . .	119
11.4.1	Time Series . . . . .	119
11.4.2	Assignment . . . . .	121
11.5	Assignment of Map Data . . . . .	121
11.5.1	Interactive Data Input . . . . .	122
11.5.2	Automatic Data Input . . . . .	122
11.5.3	Assignment via Quick Import . . . . .	124
11.6	Assignment via Expression . . . . .	124
11.6.1	Assignment . . . . .	125
11.7	Assignment of Lookup Table Values . . . . .	125
11.7.1	Lookup Tables . . . . .	125
11.7.2	Assignment . . . . .	126
11.8	Copying of Data Values . . . . .	126
11.9	Assignment of Multiple Parameters . . . . .	126
11.10	Time-Varying Material Properties . . . . .	127
11.10.1	Manual Assignment . . . . .	127
11.10.2	Manual Assignment from Time Series . . . . .	128
11.10.3	Assignment of Map Data . . . . .	128
11.10.4	Assignment via Expression . . . . .	130
11.11	Use Parameter Expression . . . . .	131
11.12	Interactive 1D Linear Interpolation . . . . .	131
11.13	Units . . . . .	131
11.14	Tutorial . . . . .	132
11.14.1	Tools . . . . .	132
11.14.2	Assignment of Constant Values . . . . .	132
11.14.3	Assignment of Time-Series Data . . . . .	134
11.14.4	Assignment of Map Data . . . . .	136
11.14.5	Assignment via Expression . . . . .	139
11.14.6	Assignment via Copy and Paste . . . . .	140
11.14.7	Assign Multiple Parameters . . . . .	141
11.14.8	Time-Varying Material Property Assignment . . . . .	141
11.14.9	Assignment via Interactive 1D Interpolation . . . . .	142
11.14.10	Multilayer-Well Assignment . . . . .	144
11.14.11	Assignment of Borehole Heat Exchangers . . . . .	145
11.14.12	Discrete-Feature Assignment . . . . .	147
<b>12</b>	<b>Simulation . . . . .</b>	<b>149</b>
12.1	Introduction . . . . .	149
12.2	Model Check . . . . .	149
12.3	Results Output . . . . .	149
12.4	Running the Simulation . . . . .	150
12.5	Convergence . . . . .	150
12.6	Tutorial . . . . .	151
12.6.1	Tools . . . . .	151
12.6.2	Model Check . . . . .	151
12.6.3	Results Output . . . . .	152
12.6.4	Running the Simulation . . . . .	154



<b>13 Results Evaluation</b>	157
13.1 Introduction	157
13.2 Observation Points	157
13.2.1 Scatter Plot	157
13.3 Budget Analysis	158
13.3.1 General Budgeting	158
13.3.2 Subdomain-Boundary Budget	159
13.3.3 Budget Groups	160
13.4 Multilayer Wells: Flow per Layer	161
13.5 Content Analysis	161
13.6 Streamlines and Pathlines	161
13.6.1 Random-Walk Particle-Tracking	163
13.7 Export	163
13.8 Tutorial	164
13.8.1 Tools	164
13.8.2 Observation Points	164
13.8.3 Budget Analysis	165
13.8.4 Subdomain-Boundary Budget	167
13.8.5 Content Analysis	168
13.8.6 Streamlines, Pathlines	168
13.8.7 Random-Walk Particle-Tracking (RWPT)	171
13.8.8 Export of Results	172
<b>14 Animation and Video Export</b>	175
14.1 Introduction	175
14.2 Creating a Presentation	175
14.3 Movie Export	176
14.4 Tutorial	177
14.4.1 Tools	177
14.4.2 Creating a Presentation	177
14.4.3 Movie Export	178
14.4.4 Export Settings	178
<b>15 Groundwater Age Calculation</b>	181
15.1 Introduction	181
15.2 Groundwater Age	181
15.3 Mean Lifetime Expectancy	181
15.4 Exit Probability	182
15.5 Tutorial	182
15.5.1 Tools	183
15.5.2 Setting up a Groundwater Age Simulation	183
15.5.3 Results Evaluation	185
<b>16 Plug-ins and Interface Manager IFM</b>	191
16.1 Introduction	191
16.2 Plug-ins for Users	191
16.3 Technology	192



16.4	IFM for Programmers . . . . .	192
16.5	Tutorial . . . . .	193
16.5.1	Using Plug-ins . . . . .	193
16.5.2	Programming Plug-ins . . . . .	193





# 1 Introduction

## 1.1 Welcome to FEFLOW

Thank you for choosing FEFLOW! You have selected one of the most comprehensive, well-tested and reliable programs for the simulation of flow, groundwater age, mass- and heat-transport processes in porous media.

This manual explains FEFLOW's extensive modeling capabilities so that the easy-to-use intuitive graphical user interface can be used to its full potential.

Please take your time to familiarize yourself with the software to ensure maximum productivity and efficiency in your projects.

## 1.2 The FEFLOW Package

The FEFLOW user interface supports the entire workflow from preprocessing via the simulation run to postprocessing. In addition, there are a number of supporting applications for specific purposes:

### FePEST

A convenient graphical user interface for using PEST by John Doherty with your FEFLOW models for parameter estimation, uncertainty analysis and much more. FePEST runs with a FEFLOW license.

### FEFLOW Viewer

Free visualization and postprocessing tool for FEFLOW files.

### Command-Line Mode

In command-line mode, FEFLOW runs without any graphical user interface. This is especially useful for batch runs or integration into other simulation environments.

### WGEO

WGEO is a geo-imaging software. Its most important fields of application in connection with FEFLOW modeling are georeferencing of raster maps and coordinate transformation.

### FEPLOT

As FEFLOW itself does not provide printing capabilities, FEPLOT can be used to create plot layouts and print maps composed of vector maps, graphical elements, and text.



## FE-LM2

This tool provides functionality for curve fitting, e.g., for obtaining the parameters for parametric relationships in unsaturated flow or for sorption isotherms.

### 1.3 FEFLOW Documentation

The FEFLOW documentation provides an introduction to the practical application of the software as well as a detailed description of the underlying concepts and methods. While you obtained the Installation Guide and Demonstration Exercise in print (DVD booklet), the User Manual is available in pdf format on the installation disk or on our web site.

The comprehensive book “FEFLOW – Finite Element Modeling of Flow, Mass and Heat Transport in Porous and Fractured Media” written by H.-J. G. Diersch represents a theoretical textbook and covers a wide range of physical and computational issues in the field of porous/fractured-media modeling. The book is general and will be useful for both students and practitioners in engineering and geosciences as well as in other fields where porous-media flow dynamics and computational methods are of specific concern.

The book starts with a more general theory for all relevant flow and transport phenomena on the basis of the continuum approach, systematically develops the basic framework for important classes of problems (e.g., multiphase/multispecies nonisothermal flow and transport phenomena, discrete features, aquifer-averaged equations, geothermal processes), introduces finite-element techniques for solving the basic balance equations, in detail discusses advanced numerical algorithms for the resulting nonlinear and linear problems and completes with a number of benchmarks, applications and exercises to illustrate the different types of problems and ways to tackle them successfully (e.g., flow and seepage problems, unsaturated-saturated flow, advective-diffusion transport, saltwater intrusion, geothermal and thermo-haline flow).

The book is available for sale online and is included with every FEFLOW 6.2 license.

This User Manual and the step-by-step Demonstration Exercise contained in the DVD booklet complement the documentation on the more practical side.

A full reference of the user interface elements along with a detailed description of the handling is available in the help system of the graphical interface.

Screen casts for the exercises described in the tutorial sections as well as for the demonstration exercise are available online at [www.feflow.com](http://www.feflow.com).



## 1.4 Scope and Structure

This User Manual is intended as a practical guide to groundwater modeling with FEFLOW. It aims at explaining the essential work steps of model setup, simulation and postprocessing, and at presenting alternative options and settings with their advantages and disadvantages for specific applications. Thus the User Manual can serve both as an introduction for FEFLOW 'newbies', and as a reference for more experienced users. Its position within the complete set of documentation is between the theoretical basis in the FEFLOW book and the detailed description of the user-interface elements and work flows in the help system.

The manual follows a typical modeling workflow—starting from basic maps and finishing with postprocessing and extending FEFLOW's capabilities. Each chapter starts with an introduction to the topic, presents the relevant FEFLOW tools, describes the concepts and workflows, and ends with a tutorial.

## 1.5 Notation

Most of the tutorials are based on prepared files, thus they require installation of the FEFLOW demo data package. <FEFLOW demo> in a file path refers to the folder of the demo data installation. The default installation location may differ between operating systems. On Microsoft Windows, the typical installation location is:

- Windows 7, 8 and 10:  
C:\Users\Public\Documents\DHI FEFLOW 7.0\demo





## 2 What's New in FEFLOW 7.0?

### New features and improvements

FEFLOW 7.0 features a substantially extended range of functionality compared to its predecessors.

This chapter presents the highlights of the new version in an overview of the most important new features and improvements.

### 2.1 3D Unstructured Meshing

With the possibility to use layered, partially unstructured or fully unstructured meshes in 3D, FEFLOW now provides an unprecedented level of geometrical flexibility. Supported by a sophisticated mesh generation algorithm and interfaces to a number of geological modelling frameworks, groundwater models in demanding geological settings can now be set up more easily than ever before. The new meshing options are also especially helpful for the precise mapping of inclined boreholes or other underground structures. Compared with a layered approach, in many cases the total number of calculation nodes may be much lower, thus allowing for a more efficient computational solution.

### 2.2 Revised Model Generation Process

In relation to the new meshing options the entire model generation process was revised. The New FEM Model dialog provides a number of new options:

Model setup supports both 2D/layered 3D meshing and fully unstructured meshes.

Map features can be converted directly to supermesh elements, without the previously required step of using the context menu of the map(s) in the Mapspanel for the actual conversion.

Finite-element meshes (2D and 3D) can be imported directly. The functionality for this (in 2D) was formerly located in the File menu.

The functionality for mesh generation was moved from a toolbar into the new Meshing panel. This provides the means for both 2D meshing and 3D tetrahedral meshing. For the latter, the panel can be used both for meshing a 3D supermesh geometry and for remeshing parts of an already existing 3D finite element mesh.

### 2.3 Conversion to Quad-/Hex-Dominant Mesh

Each triangular mesh (as 2D mesh or in a 3D layered mesh) can be converted to a quad-/hex dominant mesh via the Mesh Geometry toolbar.



Meshes with a large percentage of quad-/hex-elements provide increased numerical stability and can be computationally more efficient due to a reduced element count.

## 2.4 Redesigned 3D Layer Configurator

In the context of adding functionality for layer pinching by using unstructured tetrahedral meshing, the 3D Layer Configurator was completely revised. Slice addition and removal has been redesigned, the option to assign elevations via interpolation from point maps has been added, and an additional Horizon Class for each layer allows to set the priority of each slice in case of intersections.

The ability to record and execute macros allows to easily repeat previously recorded workflows.

Whenever elevations have been assigned within the layer configuration dialog that lead to slice intersections, different options for repairing the mesh geometry, including layer pinching by using unstructured tetrahedral meshing, have been added.

## 2.5 Spatial-Units Panel Split into Selections and Entities

The contents of the former Spatial Units panel can now be found in the new Selections (all current and stored selections) and Entities (all other content) panels.

## 2.6 Control-Volume Finite Element Method

Combining the finite-element and finite-volume approaches, the CVFE method is provided as an alternative to the Galerkin FE approach. Especially for unsaturated and variably saturated conditions this can lead to more stability, faster convergence and a better mass balance.

## 2.7 Anisotropic Macrodispersion

In some natural settings dispersivity does not only depend on flow direction, but at the same time is strongly influenced by the bedding direction. FEFLOW can now consider both influences by using the additional options on the Anisotropy Settings page of the Problem Settings dialog. For Random-Walk particle tracking, the new option can be defined on the Particle-Tracking Computation page of the Problem Settings dialog.



## 2.8 Parameter Statistics

Descriptive statistics can be calculated and graphically displayed for all parameters, either for the entire mesh or for a selection of nodes or elements. The Parameter Statistics dialog is available via the context menus of the parameters in the Data panel.

## 2.9 New Features in FePEST

A description of the new features can be found in the FePEST help system and the FePEST User Manual.

- Support of FEFLOW unstructured models
- Command-line option to start FePEST in server mode
- New pilot points features
  - Tied parameters based on 2D pilot points between several layers
  - Fully 3D pilot points for layer-based and fully unstructured models with 3D Kriging support
  - New methods for pilot point positioning: random distribution with optimization algorithms (Lloyd & Monte Carlo)
  - Enhancement of local heterogeneity by using pilot point multipliers
  - Preferred parameter differences between pilot points: Improve homogeneity in the distribution of parameters
  - Custom regularization input for regularization by using GENREG utility.
- New FePEST operation mode: Pre-/post Monte Carlo Analysis
  - Monte Carlo analysis based on user-defined distributions (mean, standard deviation)
  - Adjustment of realization to an 'almost calibrated' stage using the PEST Null-Space Projection utilities
  - Optimization of each realization sample using FePEST parallelization capabilities
- New charts
  - Parameter Uncertainty: histogram of samples generated by MC analysis
  - Parameter Variance: histogram of variance between MC samples for each adjustable parameter
  - Observation Uncertainty: histogram of observation values as result of each MC sample run
  - Observation Variance: histogram of observation variance as result of each MC sample run
  - Parameter statistics: same as in FEFLOW
- New PEST utilities included:
  - Sensitivity Analysis Utility (SENSAN): Compute sensitivities based on a user-defined parameter set
  - Linear Analysis Utility (GENLINPRED): Compute linear error variance of current FePEST project.
- GUI Improvements:



- Reorganization of the Problem Settings dialog
- Reorganization of FEPEST menus
- Miscellaneous
  - Observation definitions based on values differences between two times (e.g., drawdowns).
  - Parameter and observation definitions as FEFLOW user distributions
  - Calibration statistics
  - Script customization (user-defined script can be run before/after each FEFLOW run)
  - Support of discrete feature parameters as parameter definitions
  - Support of FEFLOW node selections in history-charting mode as observation definitions for steady-state and transient conditions

## 2.10 Additional functionality

- Removal of the Flow-Rate and Flow Volume panels (already deprecated in FEFLOW 6.2), use the Subdomain-Boundary Rate Budget and Subdomain-Boundary Period Budget panels instead
- Option to convert a triangular mesh to a quad mesh in the Mesh Geometry toolbar
- Option for 3D Mesh Smoothing in the Mesh Geometry toolbar
- Mesh smoothing method: new option in Other Settings
- Temporal or spatial growth of dispersivity for random walk in the Particle-Tracking Computation settings
- Global settings for map formats
- Option for additional fixed time steps for the computation (Simulation-Time Control)

## 2.11 Programming Interfaces IFM & Python

IFM/Python do not yet fully support partially and fully unstructured model geometry! Only basic functions will work in such meshes.

New functions for the following topics have been added:

- Anisotropy in dispersivity
- Varying-viscosity models
- Multi-layer wells
- Discrete features
- Selection handling
- Handling expression-based User data
- Handling mesh geometry
- Obtaining diagram data
- Getting data at XYZ coordinates



## 3 The User Interface

### How to use the basic user interface components

#### 3.1 Philosophy

The user interface of FEFLOW 6.2 is designed to provide as many tools as possible without the **need** to open nested dialogs or menus. While allowing an efficient workflow for experienced FEFLOW modelers, the interface might look complex to first-time users.

Therefore only the interface components that are relevant at the current stage of model setup or for the current model class are shown. The five main interface components—menus, toolbars, views, panels and charts—all adapt automatically to the current context.

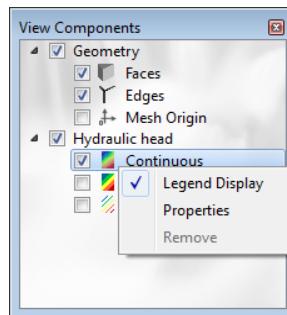


Figure 3.1 Context menu in the View Components panel.

To provide often-needed functionality as quickly as possible, many controls are also accessible via context menus that are available for most user-interface elements, for example, parameters in the **Data** panel or legends in a view window.

#### 3.2 Graphics Driver

The FEFLOW user interface makes use of OpenGL (Open Graphics Library) for visualization. OpenGL is a well-proven standard that gives access to the capabilities of the graphics hardware for accelerated display. To efficiently use OpenGL, a graphics driver provided by the graphics-card or chipset manufacturer should be installed. The standard drivers included with an operating system might not support OpenGL to a sufficient extent. Especially on laptop computers, drivers provided at purchase have been found to contain OpenGL bugs in several cases. We recommend to download the most recent drivers from the graphics-card or chipset manufacturer's web site before using FEFLOW or when problems in the graphical display are observed.



### 3.3 Customizing the Interface

The interface is completely customizable, i.e., the location and visibility of all components except the main menu can be chosen arbitrarily. Components can be docked to a certain main-window location, or they can be floating as separate windows.

To switch between docked and floating status, double-click the header of a panel or move a component to another location by dragging it while pressing the left mouse button. To avoid docking, the `<Ctrl>` key can be pressed before and while moving a panel or chart window.

Panels and charts can also be tabbed so that two or more of these elements are placed above each other. Clicking on one of the tabs brings the corresponding panel or chart to the front.

Floating toolbars, panels and charts can be moved outside the main application window. This is especially helpful to enlarge view windows on one screen while arranging toolbars and panels on another screen.

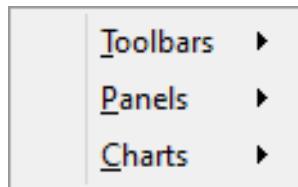


Figure 3.2 Context menu.

View windows can also be set floating with `<Ctrl> + <F11>` and then moved outside the main window (e.g., to a second screen). Full-screen mode is available for all view windows via `<F11>`.

Toolbars, panels and charts can be turned on and off by using the context menu on 'empty' parts of the user interface, i.e., parts where no other context menu comes up (Figure 3.2). Panels and charts can also be closed by clicking on the closing icon in the upper right corner of the element.

While exploring the new interface you may come to a situation where most panels and toolbars are hidden, and the remaining ones are not where you want them to be. In such a case, just switch on **Reset Toolbar and Dock-Window Layout** in the **View** menu, and FEFLOW will come up with the default layout when starting it the next time.



## 3.4 View Windows

View windows contain different views of the model, possibly along with maps and other visualization features. Limited only by the available memory, any number of windows can be displayed simultaneously to show different model components (listed in the **View Components** panel for the active view). Each view has its own settings and components handling.

### 3.4.1 Types of View Windows

FEFLOW has six types of view windows:

To see, go to windows > menu

- **Supermesh** view
- **Slice** view
- **3D** view
- **Cross-Section** view
- **Slice Data-Trace** view
- **Data-Trace** view

New view windows can be opened via the **Window** menu. For opening **Cross-Section** and **Slice Data-Trace** views, a 2D surface line has to be selected in the **Entities** panel. Opening a **Data-Trace** view requires creating and selecting a 3D line first.

### 3.4.2 Navigation

Navigation in view windows is most straightforward by using the left and right mouse buttons and the mouse wheel. By default, the left mouse button is used to pan in **Slice** views and to rotate in **3D** views. Besides invoking the context menu on a view, the right mouse button also allows zooming when a navigation tool is active. In **Slice** views, the mouse wheel has zooming functionality, while in **3D** views it is used for rotation. **<Shift> on the keyboard in combination with the mouse wheel changes the directional exaggeration (in y direction in **Slice** views and z direction in **3D** and **Cross-Section** views).**

**Keyboard shortcuts allow to quickly return to the full view (<Home>), to reset the rotation (<Ctrl>-<Home>), and to reset the scaling (<Shift>-<Home>).**

Additional tools in the **View** toolbar can also be used to return to full view, to return to a preferred view defined via **Positioning** in the **View** menu, and to undo/redo view changes. **View>positioning> undo view change**

## 3.5 Tutorial

When FEFLOW is started it opens with an empty project by default. To create a new model, we click on **New** which opens the **New FEM Problem** dialog. Here, we need to define the initial work area for mesh design. This can either be done via a manual input of the initial domain bounds or with the use



of maps that are loaded in a subsequent step. For a quick start, for example to set up a simple test model in local coordinates, select **Manual input of the initial domain bounds** and accept the default domain bounds with a click on **Finish**. The following components are now visible in the workspace:

New> 2D or layered 3D mesh>manual domain setup>initial mesh extents>get supermesh view.

- the active view window—the **Supermesh** view
- the main menu on top
- a number of panels and toolbars

By default, not all panels and toolbars are displayed. To get familiar with the graphical user interface we now add a further panel to our workspace.

Go to **View > Panels** in the main menu and click on the entry **Plug-ins Panel**. The panel now appears at the bottom right corner of the FEFLOW window.

**View>panel>plug ins**

Change the panel position by dragging it to a different location while holding the left mouse button. Leave the panel as a separate floating window or dock it at a certain location.

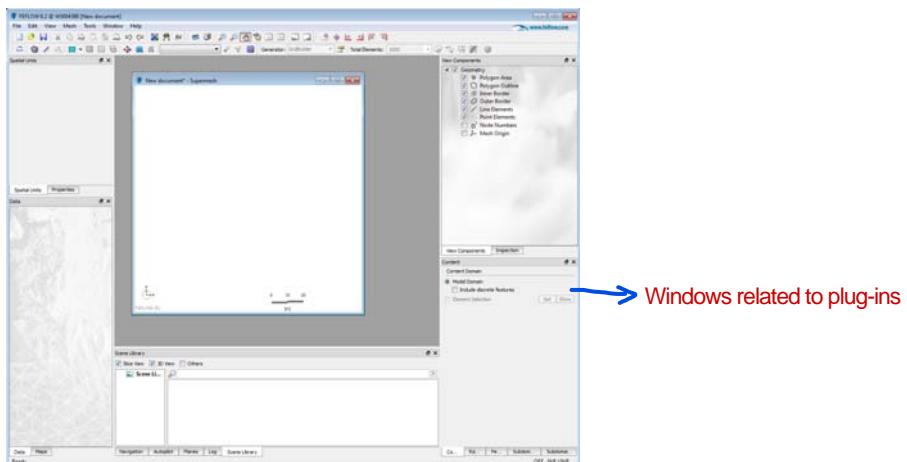


Figure 3.3 The FEFLOW standard layout.

Add another panel using a different method. Right-click on an empty part of the user interface, e.g., in the grey part above the **Inspection** panel. A context menu with the entries **Toolbars**, **Panels**, **Charts** opens up. Go to **Panels** and click on **Map Properties Panel**. The panel now appears in our workspace as a separate floating item. **Dock the panel with a double-click on its header**. The **Selections**, **Entities** and the **Properties** panel are tabbed so that only one is visible at a time. Click on the tab **Properties** to bring this panel to the front. Remove the two panels by clicking on the closing icon in the upper right corner of the respective panel.

Right Click on empty space of the menu bar>panels> map properties.

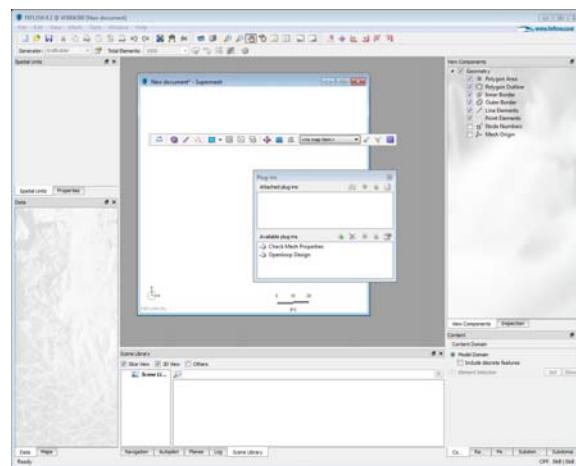


Figure 3.4 A floating panel and toolbar in the workspace.

Only toolbars relevant to the currently active view can be shown. Aside from this restriction, toolbar visibility and position are user-controlled. As an exercise, click on the left border of the **Mesh-Editor** toolbar and drag it to a different location, e.g., into the **Supermesh** view.

To restore the default settings for the graphical user interface go to **View** and select **Reset Toolbar and Dock-Window Layout**. When FEFLOW is started the next time, toolbars and panels will be arranged according to the default layout.

To move the **Supermesh** view outside of the main window, open its context menu with a click on the right mouse button and select **Floating Window**. The view window can now be moved to an arbitrary position on the screen. As per figure 3.5

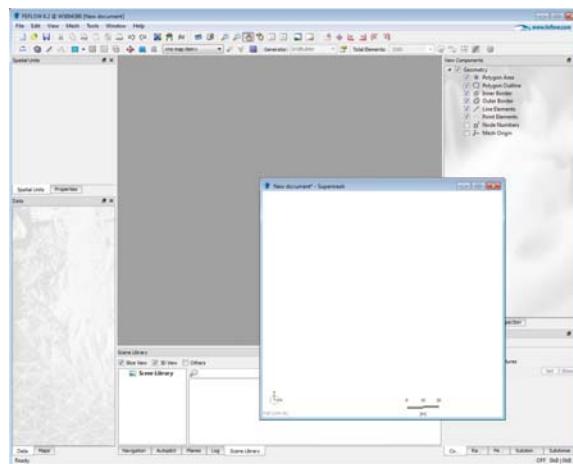


Figure 3.5 The supermesh view as floating window.

To see the **Supermesh** view in full-screen mode, hit <F11>. The full-screen mode is turned off by hitting <F11> once again.



## 4 Working with Maps

Loading and managing different kinds of maps

### 4.0.1 Maps—What For?

Maps are an integral part of all but very simple modeling projects. Their most obvious role is to provide a basis for convenient orientation in the model area. FEFLOW, however, makes much wider use of maps in the modeling workflow. Map geometries can be used to influence the mesh generation process, they can serve to geometrically define the target nodes, elements, edges or faces for parameter assignment, and attributed maps can even provide the input data themselves.

### 4.0.2 Raster Maps/Maps Containing Data

We have to distinguish between raster maps and maps which contain data. Data can be provided in the form of tables or as vector maps if they contain geometry information.

Pixel-based raster maps in formats such as \*.tiff, \*.jpeg, \*.png, or \*.bmp can only provide visual information. Vector maps contain discrete geometries (points, lines, and polygons). Formats supported by FEFLOW include ESRI Shape Files, AutoCAD Exchange Files, DBase Tables, and several ASCII (text) file formats. In addition to geometrical information these file formats also encompass attribute data, i.e., numerical and/or textual information related to certain geometrical features. While some formats like \*.shp support an unlimited number of user-defined attributes, others like \*.dxf only allow drawing attributes such as color or line style, and very simple formats such as \*.trp (ASCII triplet format—XYF) only support one single attribute value.

### 4.0.3 Microsoft Excel Worksheets and Access Tables

Tabular data from Microsoft Excel worksheets and Microsoft Access databases can be imported into FEFLOW as maps. Export of parameter values and chart data into Excel and Access is supported as well. Supported import and export formats include \*.xls, \*.xlsx, \*.mdb and \*.accdb.

View> import settings???  
Did not work!

Each Excel worksheet needs to be imported as a separate map. For workbooks containing multiple worksheets, the worksheet to be imported is selected via an import dialog (see Figure 4.1).

The import of tables from MS Access databases works similarly. After selecting a database, each table needs to be imported as a separate map via a selection dialog.

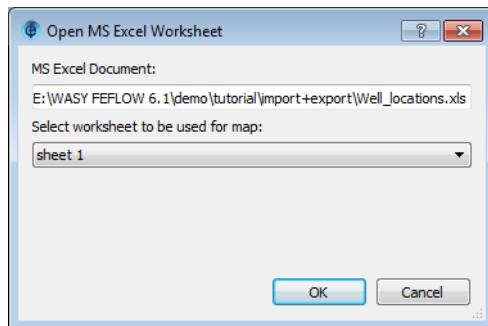


Figure 4.1 Selecting worksheets for import from a MS Excel workbook.

Exported data can either be added as new worksheets or tables to an existing Excel workbook or Access database or they can replace an existing worksheet or table.

#### 4.0.4 Geodatabases

Besides the possibility to load separate maps of various file formats, FEFLOW also provides the option to connect to different types of geographical databases for data import via the **Maps** panel and export via the **Data** and **View Components** panels.

The following database systems are supported:

- PostGIS
- Oracle
- ESRI Geodatabase

Connecting to ESRI Geodatabases is only possible with the 32-bit version of FEFLOW and with an adequate ESRI license.



*Export into an existing table/feature class erases all previous content of the table/feature class. The export option cannot be used to append information to existing tables!*

#### Database Connection

The connection to a geodatabase is managed through a **FEFLOW Database Connection File** (\*.fdb). The connection file contains all settings for the connection to a specific geographic database, i.e., information on the host, port name of the database, user name and password.

An existing connection file can be opened via **Add Map(s)...** in the context menu of the **Maps** panel or by clicking on the search icon in the panel. Creating a new connection file or editing an existing one is possible via the file-selection dialog evoked via **Add Map** or the search option in the **Maps**



panel. Make sure that the displayed file types in the file-selection dialog include the format \*.fdb.

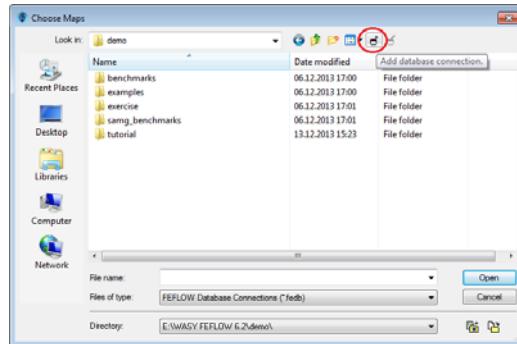


Figure 4.2 Adding a database connection.

As an example, the dialog for creating a connection to a PostGIS database is shown in Figure 4.3.

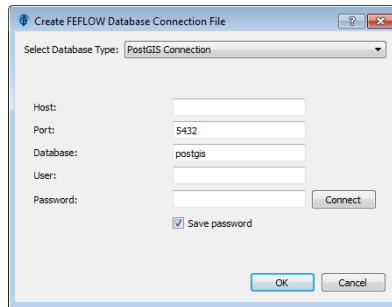


Figure 4.3 Creating a \*.fdb file for PostGIS.

Depending on the database type, different options are available when creating a new connection file. For all details on the available settings, please refer to the FEFLOW help system.

After the connection to a database, the procedure differs depending on the type of database.

### PostGIS and Oracle

?????

For PostGIS and Oracle databases, the **Open PostGIS Table** and **Open Oracle Table** dialogs are opened, respectively. The upper part of the dialog provides a summary of the **Connection Properties** while the lower part gives access to the data: Existing tables can be opened or user-defined **SQL** queries can be used to derive database data. To obtain geometries from the database when using SQL queries, a geometry column needs to be specified in

**SQL**, which stands for Structured Query Language, is a powerful and widely used language for communicating with and manipulating data stored in relational databases. Think of it as your way to talk to and organize that vast library of information!



addition. Clicking on **Verify** in the dialog checks the SQL syntax and displays the available columns so that the one containing the geometry information can be selected.

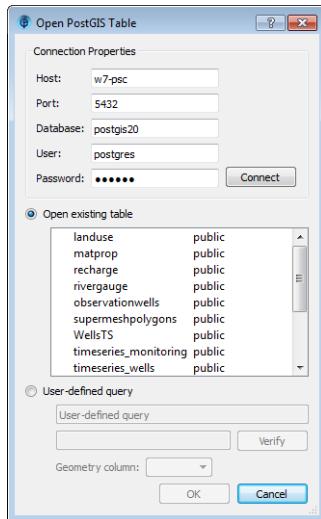


Figure 4.4 Open PostGIS Table dialog.

### ESRI Geodatabases

For ESRI Geodatabases, the standard **OpenARCGIS Table or Feature Class** dialog is brought up after opening a **FEFLOW Database Connection File**. Here, it is possible to select a feature class from the database.

## 4.0.5 2D/3D Maps

ESRI Shape Files, AutoCAD Exchange Files and tabular files (\*.dbf, \*.dat) may contain three-dimensional map information. FEFLOW supports 3D map display in 3D view windows (Figure 4.5) which can be activated via the **Map-Properties** panel.

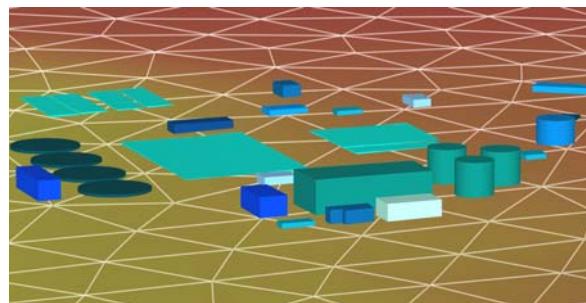


Figure 4.5 3D map in 3D view window.

#### 4.0.6 Coordinate Systems

##### Structure of UTM:

Global Grid: The Earth is divided into 60 north-south zones, each spanning 6 degrees of longitude. Zones are numbered 1 to 60, starting at  $180^{\circ}\text{W}$  and progressing eastward.

Local Projections: Each zone uses a transverse Mercator projection, which minimizes distortion within its limits. This preserves distances and directions accurately for smaller regions.

Eastings and Northing: Locations within a zone are identified by eastings (distance east from the central meridian) and northing (distance north from the equator), both measured in meters.

Dealing with spatial data requires the definition of a unique coordinate system as a reference. FEFLOW can use data in any metric cartesian system, i.e., any system with orthogonal x and y axes and coordinates in meters. The most popular of these systems is the UTM coordinate system.

To achieve better precision in the calculations, FEFLOW always uses a local and a global coordinate system at the same time. The axes in both systems have the same orientation, only the origin of the local system has an offset in global coordinates.

Locations in the local system can be expressed in cartesian or polar coordinates. The coordinate system used in a particular view window can be defined in the **View** menu. The offset of global and local coordinate system is defined automatically via map extents or manually when starting a new model, but can be edited later on in the **Coordinate-System Origin** dialog which is accessed with a click on the **Edit Origin** button in the **Origin** toolbar. In practical cases, it is usually sufficient to deal with the global coordinate system.

In 2D cross-sectional and axisymmetric models the y coordinate refers to the elevation. In these cases an offset between local and global coordinates in y should be avoided so that there is no doubt about the elevation reference. Internally, FEFLOW uses the local y coordinate as the reference for elevation-dependent parameters, e.g., when converting hydraulic head to pressure head and vice versa.

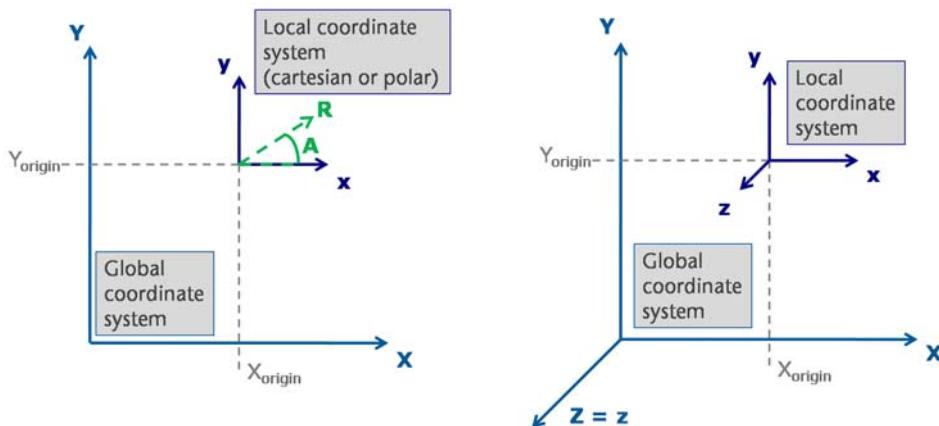


Figure 4.6 Global and local coordinates (2D/3D).

#### 4.0.7 Georeferencing Maps

The WGEO software provided with FEFLOW can add a geographical reference to raster images such as scanned maps in \*.tiff, \*.jpeg, \*.png or \*.bmp format for use as maps in FEFLOW. WGEO can also perform coordinate transformation for raster and vector maps applying a 7-parameter Helmert transformation routine.

In Plus mode (separate licensing required), WGEO also provides functionality for georeferencing of ESRI Shape Files (\*.shp) and AutoCAD Exchange Files (\*.dxf). Additional coordinate transformation routines are also available.

Please refer to the WGEO manual and its help system for a detailed description of the respective work flows.

#### 4.0.8 Handling Maps

The **Maps** panel is used to load and manage raster and vector maps as well as database connections. Available formats are \*.tif, \*.jpg and \*.png, \*.bmp for raster maps, and \*.shp, \*.lin, \*.ply, \*.pnt, \*.trp, \*.ano, \*.dxf, \*.smh, \*.dbf, \*.dat, \*.pow, \*.xml, \*.xls, \*.xlsx, \*.mdb, \*.accdb. To open a connection to a geographical database, an \*.fdb file needs to be loaded. In case of tabular data, the columns containing coordinate values have to be chosen at the time of import, unless they correspond to some defaults like X, Y and Z. The available supermesh (polygons, lines and points) of the current model is also displayed as map data in the panel.

Where to get fdb files??

By default, maps are sorted according to their format in the **Maps** panel. For more convenience especially in large projects, the tree structure can be customized according to the user's needs: The order of folders and maps within



a folder can be changed by drag and drop and maps can be moved to different folders. Existing folders can be renamed by pressing <F2> or via their context menu and new folders can be created via the context menu of the **Maps** panel or by clicking on the search icon at the top of the panel.

The search bar at the top of the Maps panel allows to filter map and folder names by a full-text search.

The relative map locations and all map settings can be exported to \*.mre (FEFLOW map reference) files and thus easily be transferred between different FEFLOW models. Export is available for folders and maps, while the import option is limited to folders.

## Map Layers

While raster maps already contain information about the display color for each pixel, this is typically not the case for features in vector maps. The display information for these kinds of maps is contained in so-called layers. When loading a map, FEFLOW creates a layer named **Default** with just one single style (color, line style, etc.) applied to all features in the map. The properties of the default layer can be edited, and additional layers can be added by using the functions in the context menu of the layer and the map. All layers of a specific map can be exported into an \*.fml (FEFLOW map layer) file via the context menu of the map. The map layer file can then be used to import all layers to the same or a different map in the same or a different FEFLOW model.

## Map Properties

The properties of a map layer can be edited in the **Map Properties** panel which is opened via the context menu of the layer. Basic settings such as opacity, lighting options and 3D drawing options can be applied to all features of the map. The map can be classified based on one of the attribute fields of the map either by applying a different style to each unique attribute value, or by partitioning the overall range of values of a numeric attribute into a number of classes. Predefined color palettes are readily available to be applied to the classes or unique values.

Color and other styles can also be edited manually for any individual class or for a selection of multiple classes.

Via the context menu of a map layer, all styles can be exported into an \*.fms (FEFLOW map style) file and imported again for a specific map layer.

You create a map layer for river networks and stylize it with blue lines, varying widths, and specific labels.

You export these styles to an \*.fms file. Later, you start a new project with different river data.

You import the \*.fms file to apply those same styles, ensuring visual consistency and saving time.

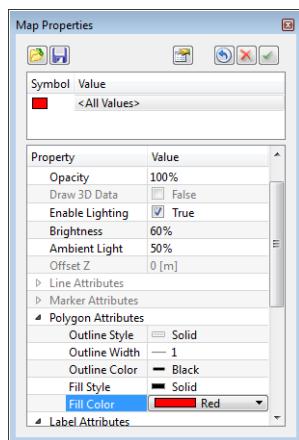


Figure 4.7 Map Properties panel.

#### 4.0.9 Map Data

Vector maps or databases can be applied as a basis for data assignment or data regionalization, deriving the basic data directly from the map file. For details on the assignment of map data, please refer to page 25 of this User Manual.

##### Map Table

The attribute data of vector maps or database files can be shown via **Show Map Table...** in the context menu of a map. The **Map Table** dialog will display the different records and attribute data contained in the map in tabular format.

##### Map Data Selection/Map Joining

Besides using the entire data range provided in a map, it is possible to use only selected map data. To limit the data range of a map, SQL selection statements on attribute data can be used via **Join/Select Map(s) Data...** in the context menu of a map.

The option **Join/Select Map(s) Data...** also provides the possibility to join additional data tables to a map, using a common identifier in both tables.

For tables imported from a geographical database, the context menu of the table header in the **Join Maps** dialog can be used to display the defined relationships with other tables of the database. Selecting a defined relationship automatically imports the corresponding table and relation.



## 4.0.10 Map Export

All the model properties and results can be exported to different kinds of map files, retaining the geographical reference of the model.

Export of parameters is invoked via the context menu of the parameter in the **Data** panel, or via the context menu of the parameter in the **View Components** panel. Both export for the entire model domain and for the selected geometries or the values in the current slice/layer only are supported. In a results (\*.dac) file, process-variable results can be exported for the current time step or for all or a selection of time steps.

Visualization options such as isolines or fringes can be exported to a map file via the context menu of the visualization style in the **View Components** panel.

Exported maps can automatically be loaded to the **Exported Maps** section within the **Maps** panel of the current FEFLOW model.

### Quick Parameter Import

The **Quick Import** option can be used to assign data of a previously exported map to several parameters at once without the need for defining parameter links via the **Parameter Association** dialog first. The import can be limited to specific model parameters and/or to the current slice/layer or the current selection. To match the input data to the mesh elements or nodes, different selection methods are available. A detailed description of this import option and a tutorial are provided in Section 11.

## 4.0.11 Tutorial

In the following exercises we want to get familiar with the handling of maps in the FEFLOW workspace. The most important tool in this context is the **Maps** panel which is used to load and manage maps.

As a first exercise, we load a number of maps of different formats that could be used to set up a supermesh.

Start with an empty FEFLOW project. In the **New FEM Problem** dialog choose the second option and click on **Finish**. In the import + export folder, select the file **SimulationArea.jpg** that is used as a background map for orientation. The map now appears as Geo-JPEG in the **Maps** panel. Double-click on this entry to add the map to the active view.

Next, load some further maps that contain information on the model site. Click on the search icon in the **Maps** panel and choose **Add Map(s)....** Select the files

- **demo\_wells.pnt**
- **model\_area.shp**



- **rivers.shp**
- **sewage\_treatment.shp**
- **waste\_disposal.shp**

We can either load the maps one by one or import them all at once by pressing the <Ctrl> key while we select the maps.

Depending on the respective file formats, the maps now appear in different tree branches in the **Maps** panel, together with a **Default** (layer) entry that FEFLOW creates automatically. Add all maps to the active view with a double-click on their **Default** entry.

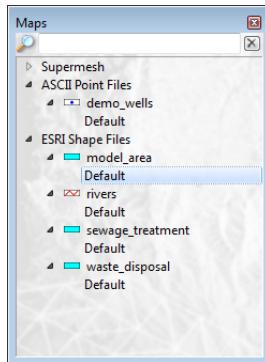


Figure 4.8 Maps panel.

Maps that have been added to the active view also appear in the **View Components** panel. Here, maps can be switched on and off temporarily via the check boxes. Their drawing order follows the tree structure of the **Maps** panel but it can be changed by dragging maps with the mouse cursor to another position within the folder or by dragging entire folders to a different position within the tree. If not all of the loaded maps are visible in the **Supermesh** view change their order in the **View Components** panel to bring maps covered by others to the surface.

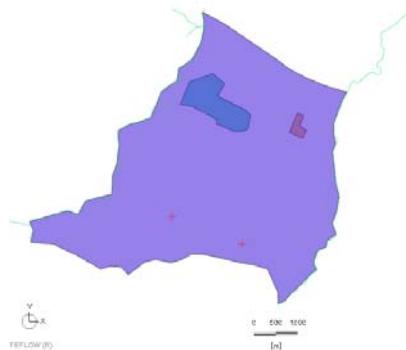


Figure 4.9 Maps displayed in the active view.

We change the sorting within the **Maps** panel by creating new folders and moving several maps to a different folder location.

Click on the search icon at the top of the **Maps** panel and select **Create Folder**. Enter **Contamination sources** as a new name and hit <Enter>. The new folder now appears as the first entry after the **Supermesh** maps. Select both the **sewage\_treatment** and **waste\_disposal** maps while keeping the <Ctrl> key pressed and move the maps to the new folder via drag and drop. Next, click on **ESRI Shape Files**, press <F2> and rename the folder to **Model geometry**. Move the map **demo\_wells** to this renamed folder and then delete the now empty folder **ACII Point Files** by clicking on it and hitting <Del>. Confirm the deletion with **Yes**.

To store all settings of the **Maps** panel in a map-reference file, click on the search icon and select **Export Map Reference(s)...**. Enter **maps** as name for the \*.mre file and confirm with <Enter>. Now delete all loaded maps via **Remove All Maps** in the context menu of the panel. After confirming with Yes, the maps are removed from the **Supermesh** view and the **Maps** and **View Components** panels.

The maps and all settings can now easily be imported again via the previously exported map-reference file. Select **Import Map Reference(s)...** from the context menu of the **Maps** panel and load the file **maps.mre**. The maps and our user-defined folder structure are now automatically reimported. Double-click on each of the default entries again to display the vector maps in the active view.

#### 4.0.12 Map Layers

The map **model\_area** defines the outer boundary of the model area. To change the style of the **Default** layer of this map make a right-click on **Default** in the **Maps** panel and select **Edit Properties** from the context menu. In the upper part of the **Map Properties** panel that now opens click on <All Values> and go to **Polygon Attributes**. Change the fill color of the



polygon and also the outline style. Confirm the new settings with the **Apply Changes** button and close the panel.

For the next map **sewage\_treatment** we create a new layer besides the already existing default layer. Open the context menu of this map with a right-click and select **Create Layer**. A new entry **Layer 1** is now added to the tree of this map. Open the **Map Properties** panel for this layer as previously described and click again on **<All Values>**. Change the fill color and the outline style of the polygon and also the opacity and confirm the settings with a click on **Apply Changes** before closing the panel.

Double-click on **Layer 1** in the **Maps** panel to add this layer to the active view. You can switch between different layers of a map using the check boxes in front of these layers in the **View Components** panel. The last one activated is always the uppermost layer. Figure 4.9 gives an example for a certain style of the imported maps.

The remaining maps that we have loaded contain spatial information on rivers and wells, i.e., line and point structures. To enhance the appearance of the rivers in the active view we change width and color of the line, again using the **Map Properties** panel for the **Default** layer of the map **rivers**.

For the map **demo\_wells** the style settings of the markers and for the labels can be edited separately.

#### 4.0.13 Map Data

Instead of showing both rivers contained in the map **rivers**, we only want to plot the eastern one. To do so, we use a SQL selection statement on the attribute field **NAME** of the map to create a data selection that is limited to the river in the east.

Open the context menu of the map **rivers** with a right-click on the map and select **Show Map Table...** to display the attribute data of the map in tabular format (see 4.10). For the attribute field **NAME** two different data sets exist, **River West** and **River East**. Close the **Map Table** dialog and now choose **Join/Select Map(s) Data...** to open the **Join Maps** dialog. In the input field of **Select record where** enter **NAME="River East"** as SQL selection statement to select the map data for the eastern river only. Make sure to enter the name with a capital R and E as the SQL syntax is case sensitive.



	ID	LENGTH	RIVERS	NAME	_Shape
1	1	169.681	1	River West	<Polyline>
2	2	320.391	2	River West	<Polyline>
3	3	207.734	3	River West	<Polyline>
4	4	627.731	4	River West	<Polyline>
5	5	1126.3	5	River West	<Polyline>
6	6	4718.89	6	River West	<Polyline>
7	7	442.371	7	River West	<Polyline>
8	8	1528.06	8	River West	<Polyline>
9	9	526.16	9	River West	<Polyline>
10	10	759.09	10	River West	<Polyline>
11	11	440.33	11	River West	<Polyline>
12	12	9164.1	12	River East	<Polyline>

Figure 4.10 Map Table dialog for the map rivers.

To ensure that the syntax is correct click on **Preview**. The map table now only shows one record for the **River East** and reflects a reduced data selection according to the SQL statement. Leave the **Map Table** dialog with **Close** and the **Join Maps** dialog with **OK**. The active view now also reflects the reduced map data set by only displaying the eastern river.

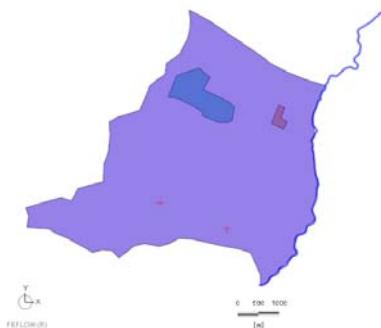


Figure 4.11 Map Rivers after Select Map(s) Data.





## 5 Starting With a New Model

How to setup a FEFLOW model.

### 5.1 Model-building Process

The model-building process in FEFLOW always consists in the same basic steps, no matter how simple or complex the modeling task at hand is. Not all the steps have to be done in the outlined order, though; in some cases iterations between different steps may be necessary, and changes of what has been created in one step may be possible and required later on in the process.

First of all, a finite-element mesh has to be generated that fills the envisioned model domain with **Elements**, at whose corner points the calculation **Nodes** are located.

Once a mesh has been built, the model properties (**Problem Class**, etc.) can be defined. Here, for example, it is decided whether saturated or unsaturated/variably saturated flow is simulated, or whether transport is considered as well.

Based on the finite-element mesh, **Initial Conditions**, **Boundary Conditions** and **Material Properties** are defined.

### 5.2 Finite-element Mesh Options

There are two ways to get to a finite-element mesh in FEFLOW:

- Definition of a Supermesh and mesh generation in FEFLOW
- Import of a finite-element mesh from one or several map(s)

FEFLOW provides different options for finite-element meshes, depending also on the dimension of the model:

#### *2D Model*

- Triangular mesh
- Triangular mesh from quad mesh
- Quad mesh
- Quad mesh from triangular mesh
- Quad-dominant mesh consisting of triangles and quads

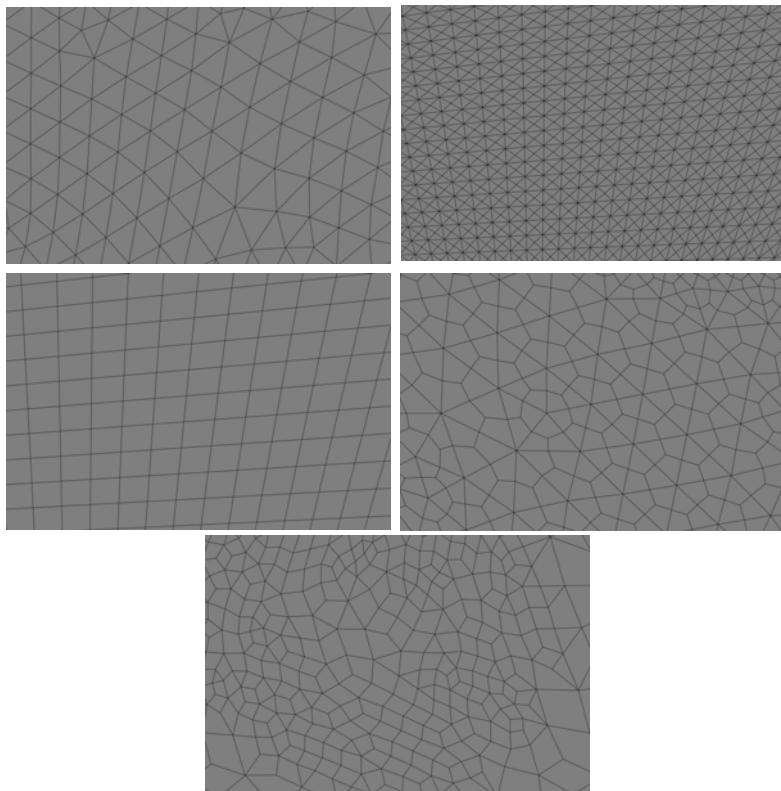


Figure 5.1    Different mesh types for 2D models in FEFLOW

### 3D Model

- Layered mesh consisting of triangular prisms
- Layered mesh consisting of quad-based prisms
- Layered mixed mesh consisting of triangular and quad-based prisms
- Layered mesh with unstructured parts
- Layered mesh with pinched layers
- Fully unstructured tetrahedral mesh

While all layered meshes are constructed by vertically extruding an existing 2D mesh, the fully unstructured option may be set up without this intermediate step.

Figure 5.2 illustrates the different workflows for mesh generation. Typically, mesh generation is done via an intermediate step, the so-called **Supermesh** that defines outer and inner geometries that need to be considered in the mesh. While for setting up 2D and layer-based 3D meshes the Supermesh is 2D and can be designed manually and by using maps, for fully unstructured meshes in 3D the Supermesh is three-dimensional and has to be imported from maps.



As an alternative to mesh generation within FEFLOW, suitable maps can be imported for direct use as finite-element mesh in both 2D and 3D (grey arrows in Figure 5.2).

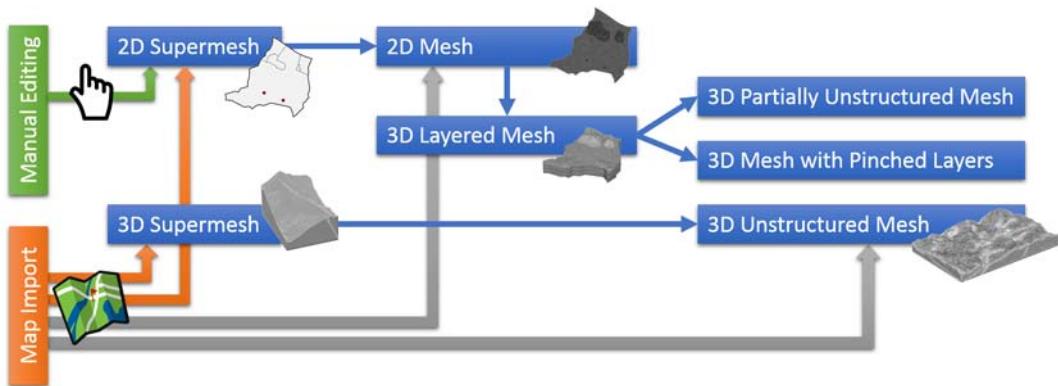


Figure 5.2 Manual and map-based workflows for the generation of different mesh types.

### 5.3 Starting a new Model

A new FEFLOW model is started by using the **New** button in the Standard toolbar, or by choosing **New** from the file menu.

In the wizard that is opened, it has to be decided how the finite-element mesh will be generated. The options follow the different workflows shown in Figure 5.2.

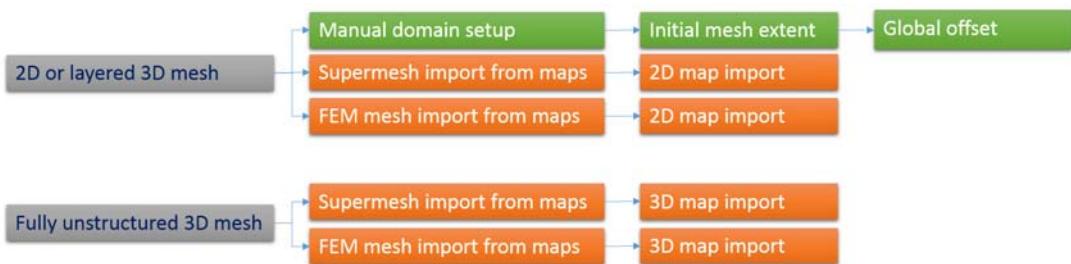


Figure 5.3 Choices for model setup in the New FEM model wizard.

The first choice to be made refers to the setup of a two-dimensional/3D layered mesh, or a fully unstructured mesh. The second option requires that a three-dimensional description of the domain geometry is available in a suitable 3D map format. In most cases, the source for such files will be a geological modeling or 3D drawing software. It is crucial that the geometry to be imported is clean, i.e., that there are no overlaps or intersections of polygons



in the hull and any other geometrical elements used. FEFLOW does not have any correction or editing mechanisms for these 3D geometries before they are used for mesh generation, so all editing/cleaning needs to be done in the source software. As an alternative, a tetrahedral finite-element mesh can be imported from a suitable 3D map file if available.

For a 2D or layered mesh, there are workflows for map-based setup and manual domain definition. The manual workflow (shown in green in Figure 5.3) is especially recommended in cases where all the data input is to be done manually on screen, without using map and file-based data. Even with the choosing the manual setup, however, it is possible to use maps later on. When using maps to define the domain extent, these can also be already converted into supermesh elements from within the wizard. As an alternative, a finite-element mesh can be imported from one or several suitable map files if available, e.g., in case of a requirement to add to an existing mesh.



## 6 Supermesh Design

Setting up the framework for mesh generation

### 6.1 What is a Supermesh?

The so-called **Supermesh** in FEFLOW forms the framework for the generation of a finite-element mesh. It contains all the basic geometrical information the mesh generation algorithm needs.

While in the very simplest case the **Supermesh** only defines the outline of the model area, i.e., consists of one single polygon, the concept offers many more possibilities: **Supermeshes** can be composed of an arbitrary number of polygons, lines and points in 2D and for 3D layer-based meshing, or solids, lines and points when working with unstructured mesh geometry in 3D. Their respective features and purposes are described in the following sections.

#### 6.1.1 Supermesh Elements for 2D Meshing

##### Supermesh Polygons

A subdivision of the model area into a number of separate polygons can be useful for a number of reasons:

- Finite-element edges will honor the polygon boundaries, allowing for example an exact zoning of parameters and setting of boundary conditions at exact locations later.
- The required density of the finite-element mesh can be specified for each polygon.
- The polygons can be used for parameter assignment and results evaluation later.

##### Supermesh Lines

Lines in the **Supermesh** are applied to represent linear structures in the finite-element mesh to be created. Their advantages include:

- Finite-element edges will honor the line, providing for example the basis for later applying the boundary condition for a river exactly along the river axis.
- The mesh may be automatically refined during mesh generation along the line.
- The lines can be used for parameter assignment and results evaluation later.



## Supermesh Points

Points in the **Supermesh** are typically placed at the locations of production or injection wells or at observation locations. They make sure that a finite-element node is set at exactly this location during mesh generation, they allow for local mesh refinement around the point, and they can be used for parameter assignment, e.g., to set the boundary condition for a pumping well.

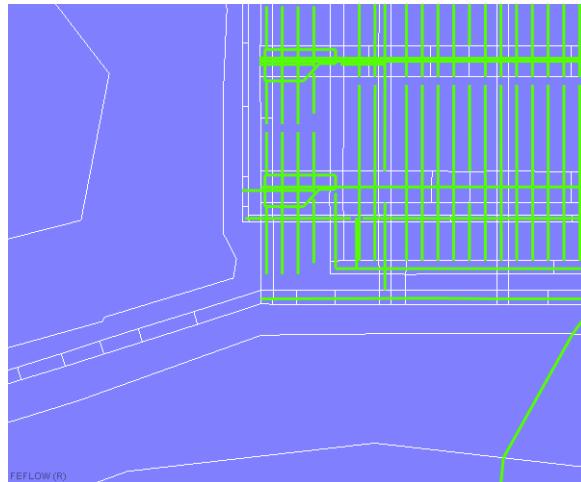


Figure 6.1 Example for a Supermesh detail.

## 6.2 Editing 2D Supermesh Features

Sets of polygons have to fulfil some requirements:

- No overlapping polygons are allowed.
- No polygons can be entirely contained by another polygon.

The user-interface tools ensure that these requirements are met at any time. Internal holes in the supermesh are possible. They are indicated by another color for the internal boundary (Figure 6.2).

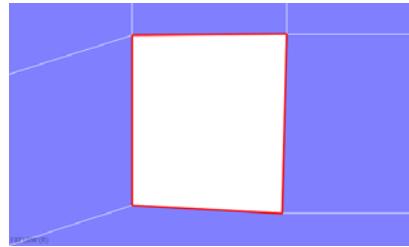


Figure 6.2 Inner Supermesh border.

The **Mesh-Editor** toolbar provides the tools to digitize and edit **Supermesh** polygons, lines, and points.

In **Move Node** mode, both the originally digitized nodes and smaller nodes in between can be moved. Moving the small nodes results in curved polygon edges (parabolic or circular shape) that are typically applied to curved structures such as borehole edges or pipe walls in small-scale models.

When digitizing a polygon next to an existing one, the editor will automatically follow the existing polygon boundary to close the new polygon (Figure 6.3).

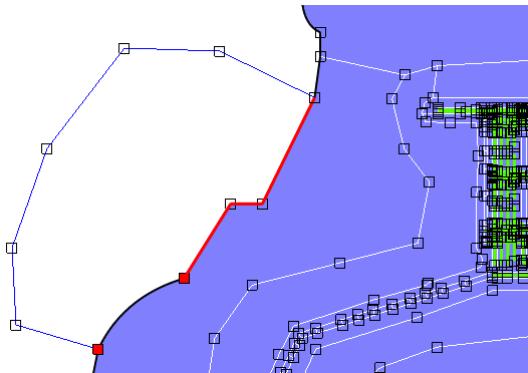


Figure 6.3 Follow existing boundaries.

## 6.3 Converting Map to Supermesh Features

Instead of digitizing **Supermesh** features on screen, they can be imported from background maps. This is done via the **Convert to Supermesh** entry in the context menu on the name of the map file in the **Maps** panel.

All features of the map (or all selected features when applying an SQL selection query on the map) are converted to supermesh features using this approach. Polygons that would overlap with already existing polygons are not converted.



## 6.4 Export of Supermesh Features

Once a finite-element mesh has been generated, all **Supermesh** features (points, lines, polygons) are displayed in the **Maps** panel. The export of **Supermesh** features as maps is available via the context menu of the **Supermesh** entry or of the respective **Supermesh** features in the **Maps** panel.

## 6.5 Tutorial

### 6.5.1 Tools

All of the tools that are used in this exercise are located in the **Mesh-Editor** toolbar.



Figure 6.4 Supermesh toolbar.

### 6.5.2 Polygons, Lines and Points

To get some hands-on training in supermesh design we design a first supermesh that consists of a single polygon.

Open an empty FEFLOW project with a click on **New** and specify the initial mesh extent to define the initial area for mesh design. In the **New FEM Problem** dialog choose **Manual input of the initial domain bounds** and accept the default extent with **Finish**. The domain bounds have now been set to 100 m x 100 m.

Now, click on **Add Polygons** in the **Mesh-Editor** toolbar in the upper left of the window. Set polygon nodes with a left-click in the **Supermesh** view window and finish by clicking on the first node of the polygon again. The finished polygon appears shaded in grey.

Now, add a second polygon that adjoins the first one. Pay attention to how the mouse cursor symbol changes depending on its position in the **Supermesh** view: The cross-hairs cursor indicating that polygon nodes can be set only appears outside the existing polygon. This makes sure that only non-overlapping polygons are created. Clicking inside the polygon FEFLOW does not create a new node. Place the first node of the new polygon on the edge of the existing polygon, continue with some more nodes and set a last node on the polygon edge again. To finish the polygon, use the autoclose function by double-clicking on the last node. FEFLOW now automatically closes the polygon along the existing polygon edge.

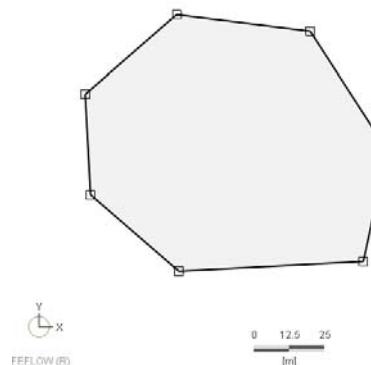


Figure 6.5 A finished polygon in the Supermesh view.

Also, add some line and point features to the supermesh, using the **Add Lines** and **Add Points** tool. Finish a line with a double-click on the last node.

The polygons that we have created can be merged with the **Join Polygons** option. First, both polygons need to be selected. Activate the **Select in Rectangular Region** tool in the **Mesh-Editor** toolbar and click into both polygons while holding **<Ctrl>**. The selected polygons are now highlighted in turquoise. Now the **Join Polygons** button in the **Mesh-Editor** toolbar is activated. Click this button to merge the two polygons.

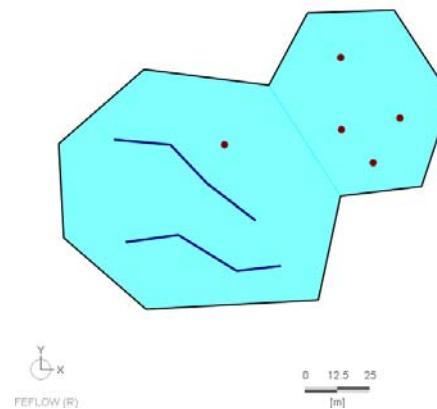


Figure 6.6 Polygons selected for joining.

Polygons, lines and points that have been misplaced can also be deleted. Activate one of the selection tools available in the drop-down list that opens with a click on the small arrow next to **Select in Rectangular Region**.



Click on the merged polygon (if it is not still selected). Then simply press the **<Del>** key to remove the polygon. You may try the same with one or several of the lines or points.

### 6.5.3 The Pin Coordinates Toolbar

Polygon nodes can also be positioned exactly. In the next step, we design a square polygon with the dimensions of **100 m x 100 m**. First, select all remaining polygons, lines and points and press **<Del>** to delete them. Next, click **Add Polygons** and press **<F2>**. The **Pin Coordinates** toolbar appears. Insert **1<sup>23</sup> 0,0** to set the first node and press **<Enter>**. In the same way, enter the coordinates of the remaining three nodes of the polygon. To finish the polygon, enter the coordinates of the first node again or simply click on it.



*Unlike other toolbars, the Pin Coordinates toolbar cannot be opened via the View menu or a context menu. It is called by pressing <F2> whenever the chosen tool supports direct coordinate input.*

If one or several nodes are misplaced while a line or a polygon is drawn, the last node(s) can be deleted by clicking on a previous node of the same line or polygon.

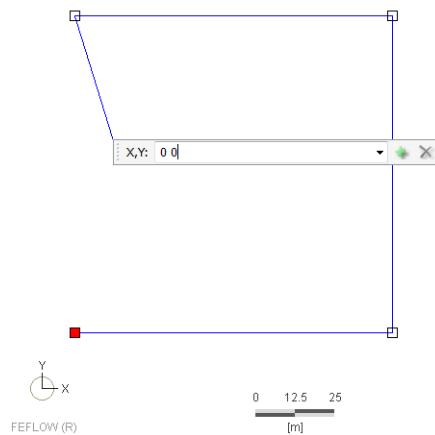


Figure 6.7 Using the Pin Coordinates toolbar.

### 6.5.4 Supermesh Import via Maps

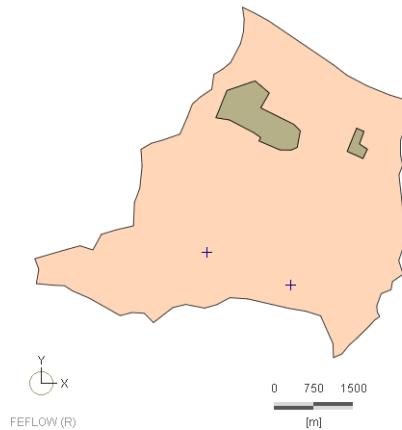
In the next step we do not design a supermesh manually on screen but import the supermesh features from a map. Start an empty FEFLOW project and in the **New FEM Problem** dialog, select the second option to define the initial



work area via imported maps. Click on **Finish** and in the file selection box select the files

- **demo\_wells.shp**
- **model\_area.shp**
- **sewage\_treatment.shp**
- **waste\_disposal.shp.**

These maps are now displayed under ESRI Shape Files in the **Maps** panel. Double-click on the entry **Default** of each map to make all of them visible in the **Supermesh** view. Figure 6.8 shows the loaded maps in the **Supermesh** view.



**Figure 6.8** Maps displayed in the Supermesh view.

First we want to create a polygon that describes the total model area. We import this polygon from the map **model\_area**. Open the context menu of this map with a right-click and select **Convert to > Supermesh Polygons**. When we click on the **Add Polygons** button in the **Mesh-Editor** toolbar, the imported polygon becomes visible.

In addition to this polygon we want to include two well locations as points in our supermesh. The workflow to import these points is completely analogous to the polygon import. The map **demo\_wells** contains the well locations. Right-click on this map to open its context menu and select **Convert to > Supermesh Points**. The two well locations are now visible as red points in the **Supermesh** view.

As a last step we want to include the two contamination sites in the supermesh. These cannot be imported from a map via the **Convert to** option as this would lead to overlapping polygons. Instead, we will split the existing polygon and cut out the contamination sources. Start with the eastern source of contamination. Click on **Split Polygons** and select the map **waste\_dis-**



**posal** from the dropdown list in the **Mesh-Editor** toolbar. To digitize the contamination source accurately we can use a tool that snaps to the fixed points of this map. To activate the snapping click on **Snap to Points** right next to the dropdown list.

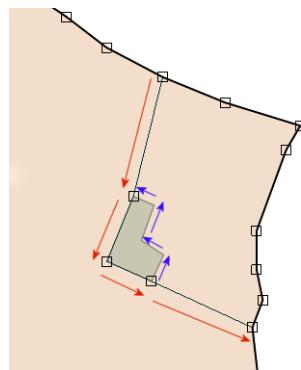


Figure 6.9 Polygon splitting along contamination site.

Polygon splitting must start and end at an already existing polygon border. As the contamination sources are located completely inside the model area two cuts are necessary. Start on an arbitrary point on the model boundary and go halfway around the contamination source. To complete the first cut, return to the model boundary on the other side (see figure 6.9). Complete the polygon with a second cut along the missing parts of the contamination source polygon.

Complete the supermesh by creating the polygon for the second contamination source in the same way, this time selecting **sewage\_treatment** from the drop-down list in the **Mesh-Editor** toolbar.

An exemplary finished supermesh setup is shown in figure 6.10.

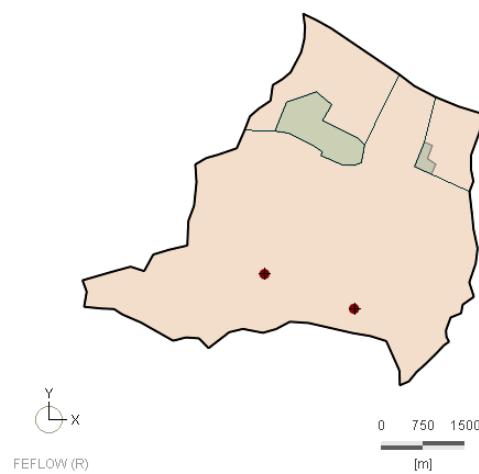


Figure 6.10 Completed supermesh.





## 7 Finite-Element Mesh

Obtaining a suitable spatial discretization of the model domain

### 7.1 Spatial Discretization

This section describes the generation of finite-element meshes. During the simulation, results are computed on each active node of the finite-element mesh and interpolated within the finite elements. The denser the mesh the better the numerical accuracy, and the higher the computational effort. Numerical difficulties can arise during the simulation if the mesh contains too many highly distorted elements. Thus some attention should be given to the proper design of the finite-element mesh. For transport simulations, the Péclet criterion can be useful for determining the required mesh density. The **Péclet Number** is available as elemental distribution in **Auxiliary Data** in the **Data** panel. To assist in creating a well-shaped mesh, FEFLOW offers various tools, including local refinement and derefinement of the mesh and (selective) mesh smoothing. Local refinement during mesh generation will typically lead to a better mesh quality than later subdivision of elements.

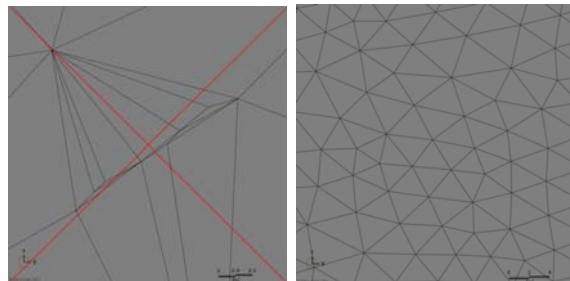


Figure 7.1 Examples for bad and good mesh geometry.

As described in Section 5.2, FEFLOW supports a number of different meshing options. For all 2D applications as well as 3D meshes based on layered meshing, a 2D finite-element mesh is defined. For 3D applications, this is then expanded in the z direction into a layered mesh, which then may be further developed into a partially unstructured mesh by re-meshing parts of the domain or by pinching layers. For fully unstructured meshes, a 3D Supermesh is imported, and an unstructured 3D mesh is generated from this.

### 7.2 2D Mesh Generation

FEFLOW supports the generation of either triangular or quadrangular finite-element meshes. A separate panel is available to support the mesh generation process. The generation is generally based on the input of an approxi-



mate number of finite elements to be generated. The desired mesh density of each supermesh polygon can be edited separately.

Different algorithms for the mesh generation are provided, all of them with their specific options and properties. Some algorithms can consider also lines and points in the supermesh and allow a local mesh refinement at polygon edges, lines and points.

Mesh generation is typically a trial-and-error process. The user hereby iteratively optimizes element numbers, generator property settings and—if necessary—the supermesh until a satisfactory mesh is obtained.

Significant effort can be involved in mesh generation, especially in cases with a large number of geometrical constraints (many polygons, lines and points). In typical cases, however, the effort required for generating a good finite-element mesh saves time at later stages of the modeling process due to reduced risk of instabilities.

## 7.2.1 Mesh Generation Algorithms 2D

There are many different strategies for the discretization of complex domains into triangles and quad elements in 2D and as the basis for . As each has its specific advantages and disadvantages, FEFLOW supports three different algorithms for triangulation and one for quad meshing.

### Advancing Front

**Advancing Front** is a relatively simple triangular meshing algorithm that does not support any lines or points in the supermesh. If present, they are simply ignored in the generation process. Its main advantages are its speed and its ability to produce very regularly shaped elements.

### GridBuilder

**GridBuilder**—developed by Rob McLaren at the University of Waterloo, Canada—is a flexible triangulation algorithm. **GridBuilder** supports polygons, lines and points in the supermesh as well as mesh refinement at points, lines, or supermesh polygon edges.

### Triangle

**Triangle** is a triangulation code developed by Jonathan Shewchuk at UC Berkeley, USA. It is extremely fast, supports very complex combinations of polygons, lines and points in the supermesh, allows a minimum angle to be specified for all finite elements to be created, and provides the means for local mesh refinement with a maximum element size at lines or points of the supermesh.



## Transport Mapping

**Transport Mapping** is the algorithm used in FEFLOW for generating meshes of quadrilateral elements. This option requires that the quad meshing option in the **Mesh** menu is selected and that all supermesh polygons have exactly four nodes.

Lines and points in the supermesh are ignored when generating quadrilateral meshes.

### 7.2.2 2D Mesh Import from Maps

The finite-element mesh geometry cannot only be generated from scratch, but it can also be imported from maps via **2D Mesh Import...** in the **New model** wizard. This is especially useful if the area of an existing model has to be extended and the user does not wish to start over with a completely new model.

It is possible to import an arbitrary number of maps and to combine them to a new 2D finite-element mesh. Only triangular and quad polygons are allowed in the maps. Furthermore, polygons in the imported maps must not overlap, and nodes at common boundaries have to be identical in number and location.

## 7.3 2D Mesh Editing

Some specific modifications of the finite-element mesh are possible at any time after mesh generation, even after model parametrization:

- Deletion of elements
- Mesh refinement by element subdivision
- Mesh de-refinement (after previous refinement)
- Splitting of quad elements into triangles
- Splitting of triangular elements into quads
- Convert triangular mesh to quad-dominant mesh
- Smoothing of the mesh at selected node locations or of the entire mesh
- Flipping element edges (triangular elements)
- Moving nodes within the area of the surrounding elements

All the mesh-editing functionality is contained in the **Mesh-Geometry** toolbar.

## 7.4 3D Mesh Generation

### 7.4.1 3D Mesh Generator TetGen

**TetGen** - A Quality Tetrahedral Mesh Generator and a 3D Delaunay Triangulator is developed by Hang Si at the Weierstrass Institute for Applied Analysis



and Stochastics (WIAS) in Berlin, Germany (<http://wias-berlin.de/software/tetgen>).

TetGen is used by FEFLOW for generating fully unstructured 3D meshes, for creating 3D partially unstructured meshes on basis of a layered mesh via the **Meshing** panel, and for creating 3D meshes with pinched layers. The various options of TetGen available in the **Meshing** panel are described in detail in the FEFLOW help system.

## TetGen Mesh Generation Properties

As for 2D meshing, the mesh quality for tetrahedral meshes (or meshes with unstructured parts) is quite important with relation to numerical stability of the model. Compared to a 2D mesh, a purely visual control of quality is much more difficult to do, and in many cases simply impossible. Thus the Auxiliary Parameters related to mesh quality need to be used to check the mesh created: Condition number, and Minimum and Maximum dihedral angles. The aim is for a low condition number and large minum / small maximum dihedral angles.

To achieve a good mesh quality, the properties for mesh generation with TetGen can be used. As each geometry to be meshed is different, there is no simple cookbook answer on this. However, at least the following properties might be worth to mention here:

- The **Quality-mesh generation** option is used to turn on a large number of constraints on the mesh quality, among them:
  - Radius-edge ratio (values below 2.5 are often considered as OK)
  - Minimum and maximum dihedral angle
  - Global tetrahedron volume constraint
  - Local constraints on area, length and volume for the hull and add-in geometries help to achieve the desired local resolution.
- The **CVT** option allows to add additional add-in points in the domain in order to achieve a more regular distribution of the tetrahedral elements. The more points are inserted, the finer and more regular the mesh will be.
- The option **Display mesh quality statistics** can be turned on to automatically provide statistical data on quality after the mesh generation, without having to manually check the Auxiliary parameters.

As for 2D meshing, and maybe even more so, 3D mesh generation should be seen as an iterative process. The mesh generation properties are changed in the process, until the resulting mesh fulfills the requirements of the modeling project at hand.

### 7.4.2 3D Mesh Import from Maps

The finite-element mesh geometry cannot only be generated from scratch, but it can also be imported from maps via **3D Mesh Import...** in the **New**



**model** wizard. This is especially useful if a tetrahedral mesh has been generated outside FEFLOW, e.g., in a 3D geological modelling software. Some focus should be put onto checking the mesh quality in this case, as requirements of the numerical simulation on the mesh may be different from the ones relevant in the external software.

It is possible to import an arbitrary number of maps and to combine them to a new 3D finite-element mesh. All maps have to contain tetrahedral elements only. The elements in the imported maps must not overlap, and nodes at common boundaries have to be identical in number and location.

## 7.5 3D Mesh Editing

### 7.5.1 Modifications of Layered Meshes

Layered meshes, i.e. meshes without any unstructured parts, can be modified using a number of different operations:

- Mesh refinement by element subdivision
- Mesh derefinement (after previous refinement)
- Splitting of quad elements into triangles
- Smoothing of the mesh at selected node locations or of the entire mesh
- Flipping element edges
- Moving nodes within the area of the surrounding elements

### 7.5.2 Pinching Layers with Tetrahedral Elements

Zones where layers pinch out can be meshed with tetrahedral elements. By using this process, the mesh becomes partly unstructured.

### 7.5.3 Re-meshing with Tetrahedral Elements

Parts of the model domain can be chosen interactively for re-meshing. Re-meshing is done based on tetrahedral elements, while the connection of the domain parts with a structured (layered) mesh and the new unstructured parts is done by using pyramid elements.

## 7.6 Mesh-Property Check

FEFLOW provides some basic tools to check the properties of a finite-element mesh, which are accessible via **Auxiliary Data** in the **Data** panel. The following parameter distributions can be shown in the currently active view:

- Elemental areas



- Max. interior angle of triangles or quadrangles
- Delaunay criterion violations
- Condition number

For 3D models, additional distributions are available:

- Slice distance
- Layer thickness
- Elemental volumes

For transport models, FEFLOW provides additional distributions which are calculated and updated at each time step during the simulation:

- Péclet Number
- Courant Number

Distances and areas in **Slice** and **Supermesh** views and distances in **3D** views can be measured using the **Measure** tool. If the **Supermesh** is the active view, the **Measure** tool is located in the **Mesh-Editor** toolbar. For active **Slice** views, the tool is accessible via the **Inspection** toolbar.

For checking the mesh quality in the entire model or parts of the domain, the option to show descriptive statistics on the above listed parameters can be very useful (available in the context menu of the parameters in the **Data** panel). Figure 7.2 and Figure 7.3 show a comparison of the histograms for two different options of mesh generation.

The closer the maximum interior angles are to 60° and the lower the condition number, the better the overall mesh quality.

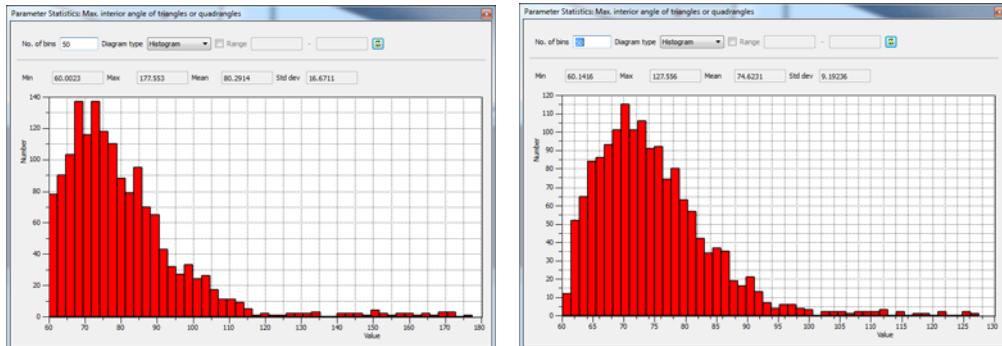


Figure 7.2 Parameter Statistics for Max. interior angle of triangles or quadrangles, left: mesh generated with Triangle without Quality mesh option, right: with Quality mesh option and after additional smoothing.

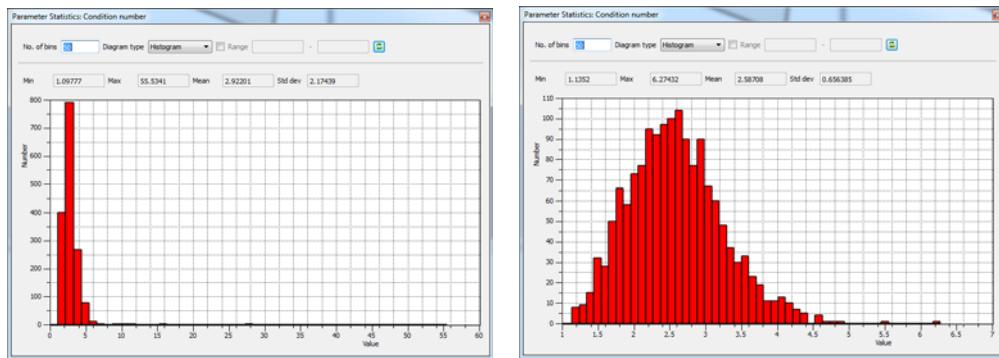


Figure 7.3 Parameter Statistics for Condition number, left: mesh generated with Triangle without Quality mesh option, right: with Quality mesh option and after additional smoothing.

## 7.7 Mesh-Element Deactivation

By default all mesh elements are active and results are computed at each mesh node. Using the **In-active Elements** parameter in the **Geometry** section of the **Data** panel, elements can be deactivated for the entire simulation period or only for specific time steps during a simulation run. In case that all elements surrounding a mesh node are inactive, FEFLOW does not compute results at this node.

With this option, time-varying geometries as well as impermeable structures can easily be considered.

In 3D models, the parameter **In/outflow on top/bottom** is applied to the first set of active elements if elements within the top layer(s) are set to inactive. This ensures that assigned recharge or evapotranspiration values are passed on to deeper locations of the model domain correctly.



*In/outflow on top/bottom is not available in models with partially or fully unstructured mesh. In these cases, a Neumann boundary condition (Fluid-flux BC) has to be used instead. This cannot be automatically inherited to the elements below in case of element deactivation!*

## 7.8 3D Discretization

For 3D models, FEFLOW can apply a layer-based approach, a fully unstructured approach, or a mixture of both.

### 7.8.1 Layered approach

In case of the layered approach, the triangular, quadrangular or mixed mesh is extended to the third dimension by extruding the 2D mesh, resulting in pris-



matic 3D elements. In FEFLOW terminology, all (typically) horizontally adjacent 3D elements comprise one layer, while a slice is either the interface between two (typically) vertically adjacent layers or the top or bottom of the model domain. All mesh nodes are located on slices.

The extension of a 2D model to a 3D model is facilitated by the **3D Layer Configuration** dialog that is accessed via the **Edit** menu. Initially defined layers are plane. Real elevations are assigned like a process variable for each node as discussed in Section 11, and can be attributed directly in the dialog or like any other parameter.

Unless using mixed prismatic/tetrahedral meshing, all the layers in 3D models have to be continuous over the entire model domain. Thus model layers representing lenses or pinching-out stratigraphic layers have to be continued to the model boundary. Typically, they are then assigned a small thickness and the properties of the layer immediately above or below.

3D model setup is in most cases based on a vertical extension of a horizontal mesh. For applications such as modeling of dams where a high level of detail is needed vertically, but less along the horizontal axis, the mesh can be generated in vertical projection and extended horizontally. In the latter case, the y axis in FEFLOW points in the direction opposite to gravity, similar to a 2D cross-sectional model.

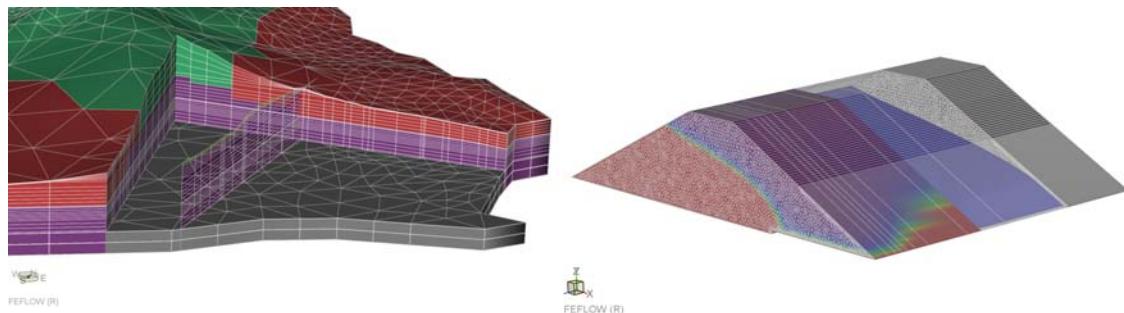


Figure 7.4 Horizontal and vertical layering approaches.

The **3D Layer Configuration** dialog also provides tools to add or remove layers from existing models, and to change layer thicknesses globally. The **Validate** option allows to verify slice elevations with respect to a certain minimum slice distance. Model properties of new layers can be conveniently inherited from already existing layers.



## 7.8.2 Partially Unstructured Approaches

### Partial Re-Meshing

Setting up a layered model in a first step, it is possible to re-mesh parts of the model domain with an unstructured mesh in order to refine the mesh or to honor specific geometrical features in the respective part of the model. The chosen part of the domain is re-meshed with tetrahedral elements in this case, and pyramidal elements are used to connect layered and tetrahedral meshes. At re-meshing, the properties of the existing mesh are inherited to the new elements.

Partial re-meshing is done by using the **Meshing** panel. The elements to be replaced can be either defined by a stored element selection (see Section 9.3), or by using a 3D map (see Section 4). 3D maps can also be used as so-called **Add-ins**, which means that their geometry is taken into account for the mesh generation process.



*The successful use of Add-ins for the generation of unstructured meshes requires that the applied add-ins neither intersect themselves, nor each other or the boundaries of the meshed domain (see Figure 7.5).*

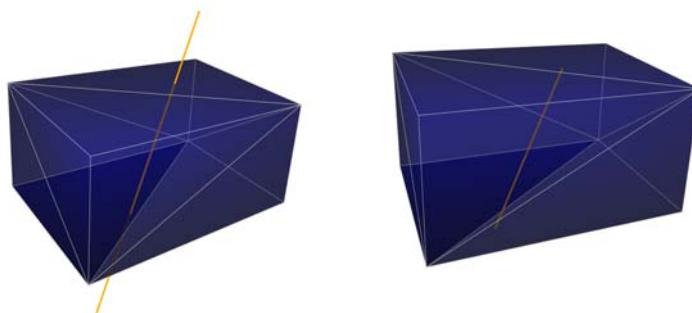


Figure 7.5 Linear add-in, intersecting the boundary of the domain (left, meshing impossible) and located completely inside the domain (right).

### Layer Pinching

Partially unstructured meshing can also be used to implement layer pinch-outs in a layered model. In these cases, a layer pinching out will be meshed with prismatic elements, and only the pinching zone is re-meshed with tetrahedral and pyramidal elements (see Figure 7.6).

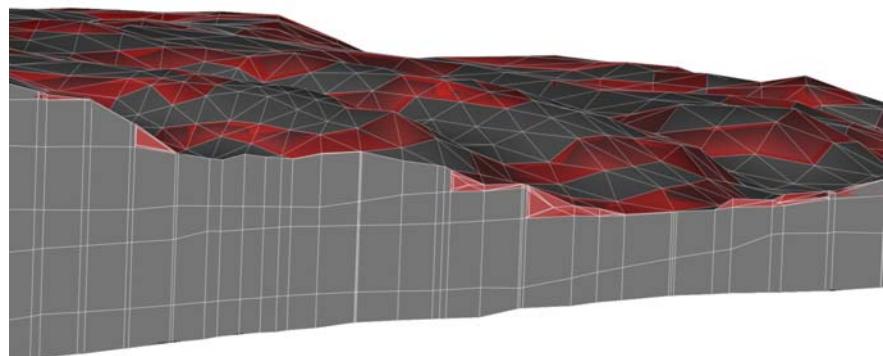


Figure 7.6 Layer pinching: Cut-out of a model with outcropping layers; tetrahedral/pyramid elements in the outcrop areas are highlighted in red.

Layer pinching is achieved by a specific workflow available in the **3D Layer Configuration** dialog.

### 7.8.3 Fully Unstructured Approach

For cases where the geometrical constraints for a model are available in terms of three-dimensional geometries, FEFLOW provides the means to discretize the entire model domain with tetrahedral elements. Major precondition for a successful mesh generation is that the domain and any possible subdomains are defined by closed, non-intersecting and non-self-intersecting surfaces. In typical cases, such geometries are derived from geological modeling software or 3D design software (Figure 7.7).

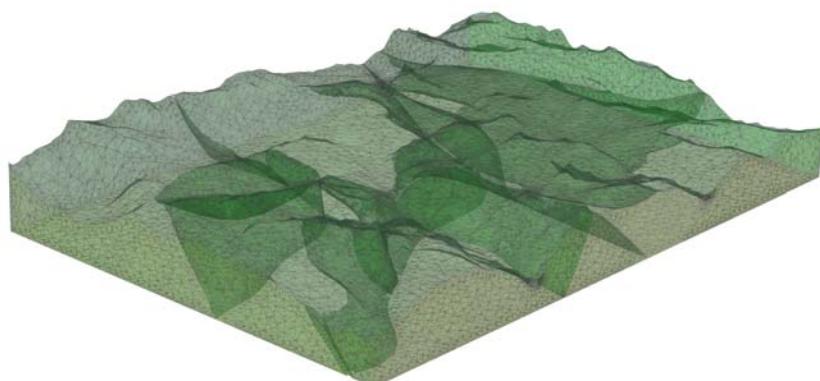


Figure 7.7 3D Map consisting of triangulated surfaces enclosing a number of sub-domains without intersection and self-intersection.



*The successful generation of an unstructured mesh from 3D map data strictly requires the 3D maps to enclose all subdomains without gaps in the surfaces. No intersections or self-intersections of the surfaces are tolerated.*

## 3D Supermesh

Before meshing the domain, the imported geometries are converted into a 3D supermesh. The workflow is supported by the **New problem** wizard. Additional elements can be added directly to the **Meshing** panel. Other than 2D supermeshes (see Section 6) 3D supermeshes also allow to retain certain information contained in the map data for the final mesh. For example, regions can be marked for later grouping of elements. This functionality is essential as there is no other way of selecting all the elements in a certain region during model parametrization.

### Surfaces

Polygon maps are loaded as surfaces. They define the 3D boundaries of model regions. While general polygon maps are basically supported, it is for most applications highly recommended to use triangulated surfaces for this purpose.

### Region and Surface Markers

Region markers (3D points) are used to mark sub-regions of the domain that are enclosed by surfaces. The marker can be at any location within the region. Finite elements created within a region (or within several regions marked with the same marker name/ID) can be grouped together during the mesh generation process. The element groups can be used then to parametrize the model.

Region markers can also be used to constrain the element size for the region, e.g., by using an attribute of the marker point in the map.

### Hole Markers

This kind of marker (3D point) can be used to indicate that a specific region in between the surrounding surfaces is a hole in the domain and is not to be meshed.

### Add-ins

Lines or points can be used as Add-ins, i.e., as additional geometries to be considered during the meshing process. The segment length can be constrained to control mesh refinement at the line.

### Custom

With the Custom option, points can be split up into different kind of markers by using an attribute.



## 7.9 Geotransformation

FEFLOW expects all attached maps and the model itself to share the same coordinate system. Maps using differing coordinate systems can be transformed in advance using the external software WGEO, which is part of the FEFLOW software package.

In case that a geotransformation of the model itself is necessary, the **Geographic Transformation** option in the **Mesh-Geometry** toolbar can be used.

## 7.10 Tutorial

### 7.10.1 Tools

All of the tools used in these exercises are located in the **Mesh-Generator** toolbar

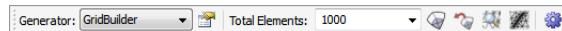


Figure 7.8 Mesh-Generator toolbar.

and in the **Mesh-Geometry** toolbar.



Figure 7.9 Mesh-Geometry toolbar.

Some mesh-editing options also require the tools of the **Selection** toolbar.



Figure 7.10 Selection toolbar.

The first drop-down list in the toolbar gives access to the available selection tools. The third drop-down list allows to switch between different target geometries (nodes, elements, edges, faces) for selections.



## 7.10.2 Mesh Generation

### Triangulation

To get some hands-on experience in how the available mesh-generator algorithms work we apply the three different mesh generators on the same supermesh and study the resulting finite-element meshes.

First, click on **Open** to load the supermesh file **mesh.smh** from the supermsh folder. This supermesh consists of two polygons, one line and three point features.

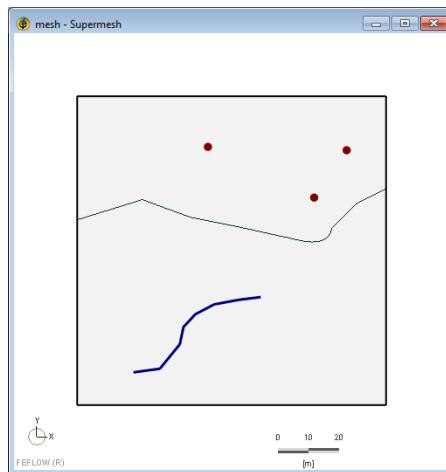


Figure 7.11 Supermesh.

Start with the **Advancing Front** algorithm which can be selected from the generator list of the **Mesh-Generator** toolbar.

Enter a **Total Number** of **12<sup>3</sup> 2000** elements in the input field and click on **Generate Mesh** to start the mesh-generation process. A new window, the **Slice** view, opens with the resulting finite-element mesh.

As figure 7.12 shows, **Advancing Front** ignores the line and point features which are included in the supermesh.

Now, use the same supermesh to generate a finite-element mesh with the **GridBuilder** algorithm. Click in the **Supermesh** view to make the **Mesh-Generator** toolbar visible again and select **GridBuilder**. Without any further changes simply click on **Generate Mesh**.

The resulting finite-element mesh looks similar to the one created with the **Advancing Front** algorithm, except that polygon edges, lines and points are now honored by the mesh.



As a next step, we will refine the mesh around the point and line features. First, activate the **Supermesh** view with a click. The refinement settings are located in the **Generator Properties** dialog in the **Mesh-Generator** toolbar.

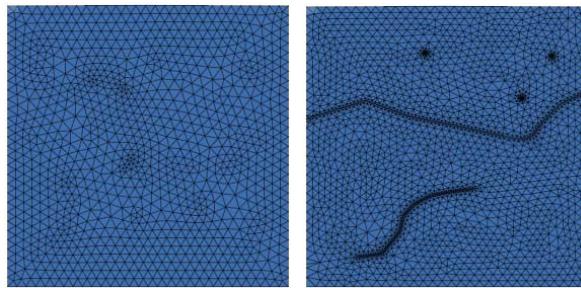


Figure 7.12 Advancing Front and GridBuilder.

Open the dialog and activate a refinement for all three geometrical features. For polygon edges, choose a refinement level of **5**, for lines a level of **8**, and for points a gradation of **10**. Make sure that the option **Apply to SELECTED polygon edges or line segments** is activated before you leave the dialog.

To select the elements to be refined go back to the **Supermesh** view and click on **Refinement Selection**. Click on all segments of the line and of the edge that separates the two polygons. The selected edges are shown in green. Then click on **Generate Mesh** again. The resulting finite-element mesh with local refinement is shown in figure Figure 7.12.

Next, we will apply the **Triangle** algorithm to generate a finite-element mesh.

Choose **Triangle** from the generator list in the **Mesh-Generator** toolbar. Open the **Generator Properties** dialog and activate refinement for polygon edges, line and point elements with the following settings:

Refinement around **SELECTED** polygon edges

- Polygons: gradation: 5, target element size: 1.0 m
- Lines: gradation: 3, target element size: 0.5 m
- Points: gradation: 3, target element size: 0.5 m
- Leave the dialog and use the **Refinement Selection** tool to select the edge between the two polygons in the **Supermesh** view. Clicking on **Generate Mesh** should now produce a finite-element mesh that looks similar to the one on the left in figure 7.14.

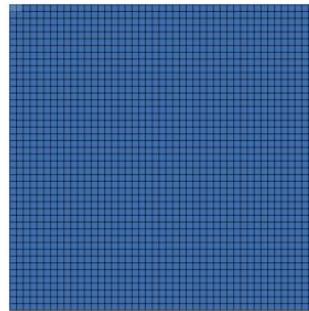


Figure 7.13 Quad mesh.

### Separate Editing of Polygons

For every supermesh polygon the mesh density can be defined separately while the total number of elements remains constant. Go to the **Supermesh** view and activate the tool **Edit Meshing Density** in the **Mesh-Generator** toolbar. FEFLOW now automatically selects a polygon for which a meshing-density factor can be specified. Start with the proposed polygon and enter a factor of **10**. After hitting **<Enter>** the number in the input field turns red, indicating that the meshing density for this polygon has been modified. Proceed with the second polygon. Select it with a single click and enter a density factor of **2** in the input field. Hit **<Enter>** again and start the mesh generation with a click on **Generate Mesh**. The right mesh in figure 7.14 shows the resulting finite-element mesh. To reset the meshing density factors to the default value of 1, click on **Reset Meshing Density**.

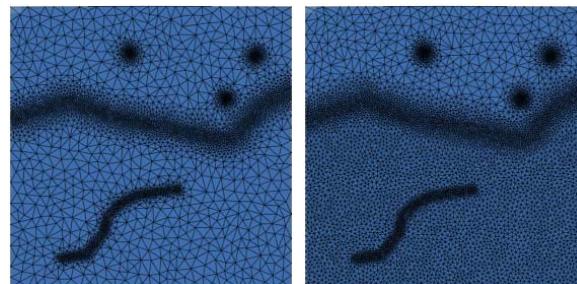


Figure 7.14 Meshes generated with the Triangle algorithm.

### Quad Meshing

A finite-element mesh with quadrilateral elements can be generated using the **Transport Mapping** algorithm.

We load a new file which is similar to the one used for the triangulation exercises, except this supermesh consists of only one polygon that has exactly



four nodes. Click on **Open** and load the file **quadmesh.smh**. **Transport Mapping** requires supermesh polygons with exactly four nodes. Lines and points are ignored in the mesh-generation process.

To enable the quad meshing option go to **Mesh** and activate **Quadrilateral Mode**. Select **Transport Mapping** in the **Mesh-Generator** toolbar, enter a **Total Number** of 2000 elements and click on **Generate Mesh** to start the meshing process. The resulting finite-element mesh is shown in figure 7.13.

### 7.10.3 Editing the Mesh Geometry

#### Triangular Meshes

For this exercise load the file **triangle.fem**.

The geometry of the finite-element mesh can be edited after the mesh-generation process has been finished. All the necessary tools are located in the **Mesh-Geometry** toolbar.

It is possible to refine the mesh globally (entire mesh) or locally (only selected parts). A derefinement option for previously refined parts of the mesh is also available.

If we want to apply local mesh refinement we have to select a target area first. Make sure that the **Slice** view is the active view. All necessary selection tools can be found in the **Selection** toolbar. To create an element-based selection choose **Select Elements** from the drop-down list of target geometries. Then, activate the **Select in Rectangular Region** tool from the drop-down list of selection tools to draw a rectangle around the line feature.

Now click on **Refine Elements** in the **Mesh-Geometry** toolbar. Each selected element is subdivided into four elements. The result is shown in figure 7.15.

The derefinement tool is used similarly; however, only those parts that were previously refined can be derefined.

Elements can also be deleted from the finite-element mesh. Select a couple of elements and click on **Delete Elements** to cut out these elements. On the right hand side of figure 7.15 a (purely illustrative) example for a mesh with deleted elements is shown.

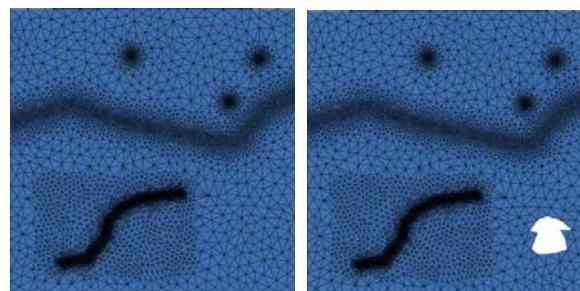


Figure 7.15 Refinement and deleted elements.

To deactivate elements during the simulation we use the parameter **In-/active Elements** in the **Geometry** section of the **Data** panel. Double click on **In-/active Elements** to set this parameter active and use the tool **Select in Rectangular Region** to create an element selection. Make sure that the input field of the **Editor** toolbar shows the setting **inactive** and click on **Assign** to set the selected elements to inactive. The result is shown in figure 7.16.

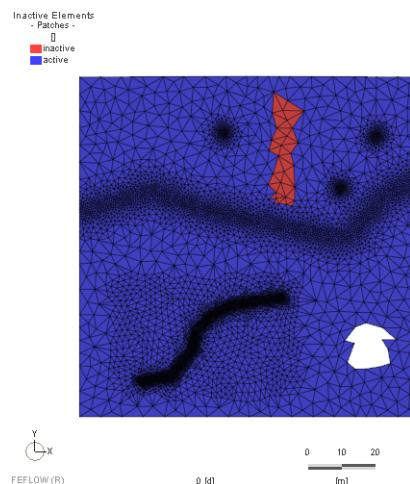


Figure 7.16 Active and inactive mesh elements.



If elements are set to active or inactive via map assignment, the active or inactive status needs to be expressed by numeric values: 1 for active and 0 for inactive.

Mesh smoothing can produce more regularly shaped elements and reduce the number of obtuse-angled triangles. To use the **Smooth Mesh** tool in the **Mesh-Geometry** toolbar, the nodes which are allowed to be moved during the smoothing process have to be selected first. Activate **Select Nodes** in the **Selection** toolbar and draw again a rectangle around the



line feature. Click on the **Smooth Mesh** tool that is now active and clear the selection with a click on **Clear Selection**. Figure 7.17 shows the selected mesh location before and after the smoothing.

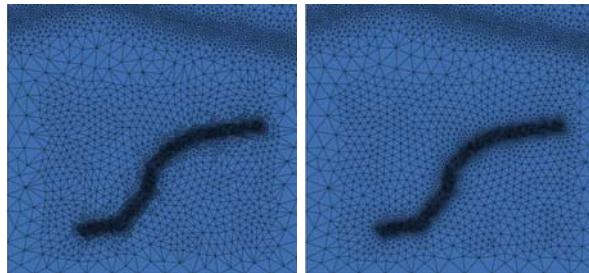


Figure 7.17 Mesh before and after smoothing.

To check for obtuse angles and triangles violating the Delaunay criterion, we use the tools provided in **Auxiliary Data** in the **Data** panel. First, double-click on **Max. interior angle of triangles** to show obtuse-angled triangles in the active view. Pay attention to how the angle distribution changes when we click on **Undo** in the **Standard** toolbar to return to the unsmoothed mesh. Next, double-click on **Delaunay criterion violations** and then click on **Redo** in the **Standard** toolbar to return to the smoothed mesh again. Both distributions indicate that the mesh smoothing has significantly improved the mesh quality.

Use the **Move Node** tool in the **Mesh-Geometry** toolbar and drag a mesh node to change its position within the mesh. Also, use **Flip Edge** and click on an element edge to change the subdivision of two adjacent triangles. Edge flipping will freeze the mesh so that derefinement of previously refined areas will no longer be possible.

## Quad Meshes

Click on **Open** to load the file **quadmesh.fem** for this exercise.

Except for the derefinement and the flip edge tool all editing options are also available for quad meshes.

As an additional option, quad meshes can be transformed into triangular meshes. Four different triangularization methods are available of which we will use one. Select the method **Four Triangles around Center** from the drop-down list in the **Mesh-Geometry** toolbar to subdivide every quad element into four triangular elements. The resulting finite-element mesh is shown in figure 7.18.

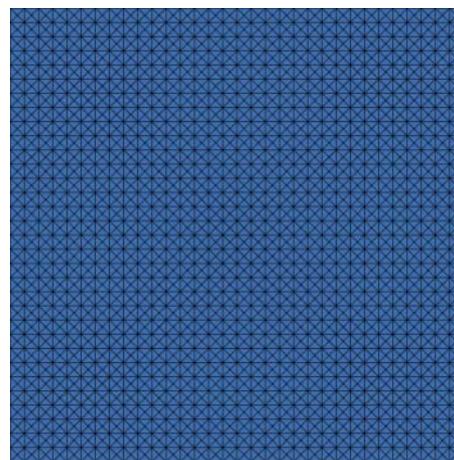


Figure 7.18 Triangulated quad mesh.

#### 7.10.4 Extending a Model to 3D

After the model has been discretized in 2D we now extend it to a 3D model. Click on **Open** and load the file **triangle.fem** for this exercise.

To perform the extension to a 3D model, go to the **Edit** menu and open the **3D Layer Configuration** dialog. The table on the left displays the number of slices and layers, and also the elevation of each slice.

The 3D model shall consist of 3 slices and 2 layers and the top slice shall be located at an elevation of **5 m**.

To set the elevation of the top slice, enter **5** in the **Elevation** input field and hit <Enter>. Increase the number of slices to **3** and hit <Enter>.

The table now shows 3 slices with elevations of **5 m**, **4 m** and **3 m** (see Figure 7.19).

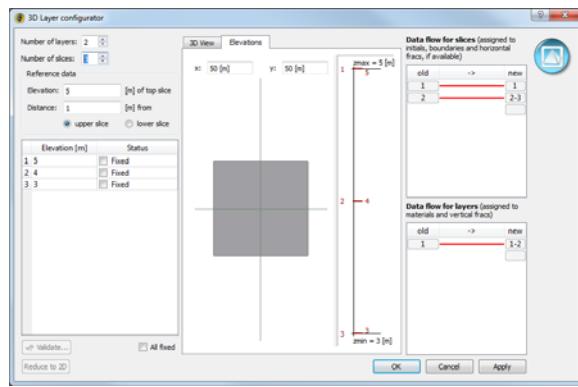


Figure 7.19 3D Layer Configurator.

Click on **OK** to apply the settings and to leave the dialog.

A new view window, the **3D** view, now automatically opens, displaying the model in 3D. The in-slice spatial discretization in plan view remains the same but the previously 2D finite elements have now been extended to 3D prismatic elements.

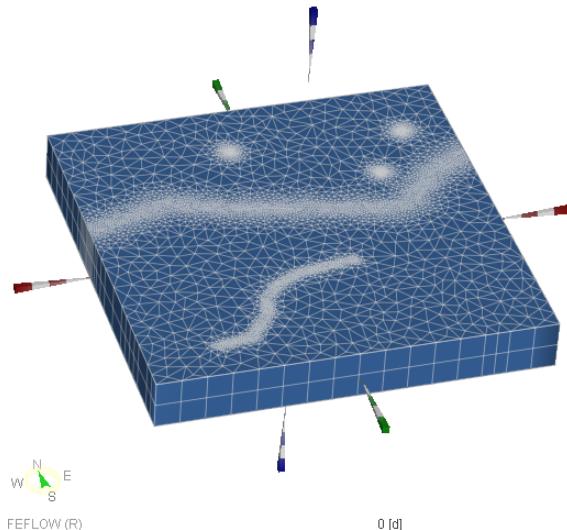


Figure 7.20 The model in 3D view



## 8 Problem Settings

Defining the modeling approach

### 8.1 Problem Class

#### 8.1.1 Physical Processes

FEFLOW allows the simulation of flow, groundwater age, mass- and heat- transport processes in either saturated, or in variably saturated media. The basic settings defining the simulated processes are done in the **Problem Settings** dialog that is accessed via the **Edit** menu.

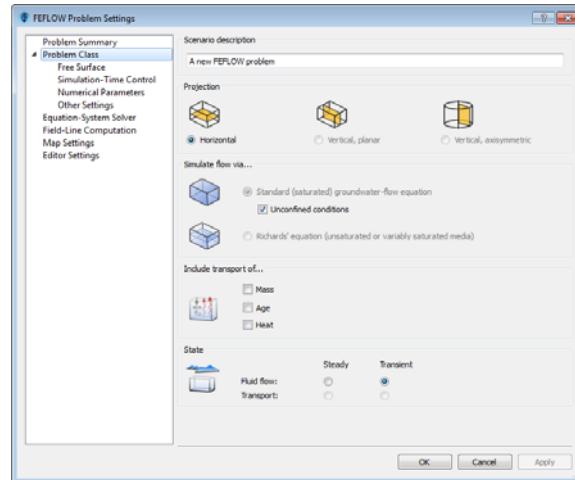


Figure 8.1 Problem Settings dialog.

#### Saturated / Unsaturated

Saturated groundwater flow is described by the equation of continuity with a Darcy flux law. Different options for handling a phreatic surface (for layered 3D meshes) are described elsewhere in this manual.

For unsaturated/variably saturated flow, FEFLOW solves Richards' equation that assumes a stagnant air phase that is at atmospheric pressure everywhere. Substantial computational effort can result from the typically nonlinear relationships between capillary pressure and saturation and between saturation and hydraulic conductivity.

FEFLOW provides two different options to define these relationships:

- **Spline Models**



- ***Empirical Models.***

**Spline Models** are used to derive the parametric relationships from tabular data using one of several different spline interpolation techniques.

Alternatively, the following six ***Empirical Models*** are available:

- ***Van Genuchten***
- ***Modified Van Genuchten***
- ***Brooks & Corey***
- ***Haverkamp***
- ***Exponential***
- ***Linear***.

As the FEFLOW implementation of Richards' equation also includes the proper terms for saturated flow, it is generally applicable to variably saturated conditions.

## Flow / Transport

A transport simulation is always performed in conjunction with a flow simulation. FEFLOW provides capabilities for single-species and multispecies solute-transport simulation, groundwater-age simulation with the possible species age, lifetime expectancy and exit probability as well as heat-transport simulation. A combined mass-and-heat ("thermohaline") transport problem can be simulated, also together with groundwater age.

A detailed discussion of the application and available settings for groundwater-age simulations, can be found in Section 15.

## Steady State / Transient

Transient simulations proceed from an initial condition and cover a specified time period. In contrast, a steady-state solution can be obtained directly and represents the state of a system having been subject to fixed boundary conditions and material properties for an infinitely long time. It is possible to combine a steady-state flow with a transient transport simulation. In such a case, the flow system is solved once at the beginning with all storage terms set to zero to obtain a steady-state solution as the basis for the transient transport calculation.

### 8.1.2 Dimension and Projections

FEFLOW supports 2D and 3D models. Finite elements of a lower dimension (1D in 2D models, 1D/2D in 3D models), so-called discrete features can be added, representing for example fractures or boreholes.



## 2D Models

A newly generated finite-element mesh always represents a 2D model. Two-dimensional models can be of horizontal, vertical, or axisymmetric projection.

A typical application for horizontal 2D models is the setup of regional water-management models without significant vertical flow components. Vertical models are used, for example, for the simulation of unsaturated flow and salt-water intrusion. Axisymmetric models have a radial symmetry such as the cone of a pumping or injection well. Essential for the suitability of an axisymmetric model are material properties and outer boundary conditions that are homogeneous along the circumference of the well cone or mound.

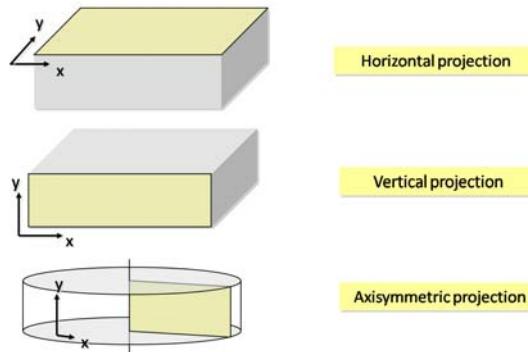


Figure 8.2 2D model projections.

## 3D Models

3D models can be created by expanding the mesh in the third direction into layers via the **3D Layer Configuration** dialog which is accessed via the **Edit** menu or via **<Ctrl>+<L>**. As all layers have an identical 2D discretization, the 3D mesh consists of prismatic or cuboid elements and each layer extends over the entire model domain.

In 3D models, the direction of gravity can be set to match any of the major coordinate directions (see ).

### 8.1.3 Temporal Settings

Corresponding to the discretization in space a discretization in time has to be specified for transient simulations.

FEFLOW supports three different options:

- Constant time steps
- Varying time steps



- Automatic time-step control

As constant time steps have the disadvantage that the most dynamic moment expected during the simulation controls the time-step length for the entire simulation, the definition of varying time steps offers some more flexibility. However, this option requires the specification of the length of each single time step in advance. Thus in most cases FEFLOW simulations use an automatic time-step control scheme, where an appropriate length of the time step is determined based on the change in the primary variables (head, mass concentration, groundwater age, temperature) between the time steps.

## First Versus Second Order Accuracy

The automatic time-stepping procedure in FEFLOW is based on a predictor-corrector scheme.

The second-order accurate, semi-implicit Forward Adams-Bashforth/backward trapezoid rule (AB/TR) is the default for standard flow and combined flow and transport simulations. While the second-order approach applied for the prediction in this method in many cases provides a more accurate estimation of the predicted result for the next time step and thus a faster solution, it also may more easily lead to instabilities under highly nonlinear conditions in unsaturated or density-dependent models. Thus for unsaturated model types the first-order accurate, fully implicit Forward Euler/backward Euler (FE/BE) method is used by default and should be applied for density-dependent flow simulations manually.

### 8.1.4 Error Tolerance

The dimensionless error tolerance is used for two purposes in FEFLOW:

- The determination of convergence in iterative processes based on the change of results between iterations, e.g., the 'outer' iteration within a time step or in a steady-state solution to take care of nonlinearities in the basic equations.
- The determination of an appropriate time-step length in automatic time-stepping procedures based on the deviation of calculated from predicted solutions.

The absolute error (e.g., head difference) is normalized by the maximum value of the corresponding primary variable in initial or boundary conditions (maximum hydraulic head, maximum concentration, etc.).



*Convergence is determined by a comparison of the input error tolerance and the absolute error normalized by the maximum head (typically input in m ASL). Thus the accepted absolute error depends on the elevation of the model. Higher elevations require a lower error tolerance to obtain the same accuracy!*



### 8.1.5 Free Surface

By default, a newly created FEFLOW model reflects a confined aquifer. Saturated simulations of unconfined conditions require specific treatment of the phreatic groundwater table.

#### 2D Model

In unconfined two-dimensional models with a horizontal projection the saturated thickness is iteratively adapted to the resulting hydraulic head. For this purpose, material-property input includes the aquifer top and bottom elevations.

When the hydraulic head exceeds the aquifer top elevation, the model calculations presume confined conditions in the respective area, i.e., the saturated thickness is limited to the difference between top and bottom elevation. Aquifers with partly confined conditions are thus easily simulated.

Two-dimensional vertical cross-sectional and axisymmetric models are always assumed to be completely confined. Modeling of unconfined conditions in these cases strictly requires a simulation in unsaturated/variably saturated mode, hereby applying Richards' equation.

#### 3D Model

In three-dimensional models with gravity in the direction of the negative z axis (default 'top view' models) two different strategies can be applied in FEFLOW for handling the phreatic surface besides simulating in unsaturated mode. It is important to note that these methods were originally designed for regional water-management models. They are clearly limited (with very few exceptions) to cases with a single phreatic surface. Simulations where partial desaturation of the model is expected below saturated parts, e.g., due to drainage in a lower aquifer, have to be simulated in unsaturated/variably saturated mode. In contrast to an often-heard opinion, despite the nonlinearity of Richards' equation unsaturated models can be as computationally efficient or even more efficient than saturated models with phreatic-surface handling if appropriate simplifications are applied to the unsaturated material properties.

As both options for phreatic-surface handling have their specific advantages and disadvantages, the methods applied in FEFLOW will be explained in detail:

##### Free

This mode takes care of the phreatic surface by vertically moving the calculation mesh in a way that the top of the model is exactly at the water-table elevation at any time. For this purpose, layer elevations are changed at each time step (transient simulation) or for each iteration (steady-state simulation). The movement is done by using the so-called BASD technique which places



layer interfaces (slices) at elevations of slices in the original stratigraphy whenever possible.

Material (elemental) properties are mapped from the original stratigraphy to the actual stratigraphy, whereas nodal properties such as boundary conditions and slice-based observation points are moved with the slices.

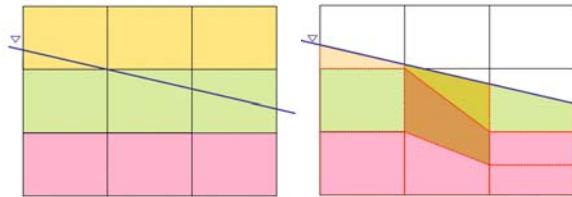


Figure 8.3 Schematic views of mesh movement and parameter interpolation in free mode.

The most important advantage of this method is its ability to exclude all partially saturated or unsaturated parts from the model so that only saturated parts are simulated at all times. The main disadvantage is a consequence of the layer movement: In cases where the actual water table cuts through slices of the initial stratigraphy, model elements may occur that are located in multiple layers (and material zones) of the original stratigraphy. In these cases, a volume-weighted averaging for the material properties is applied, giving rise to material properties that differ from the original input. When interpolating between aquifer and aquitard properties, this approximation might not be acceptable.

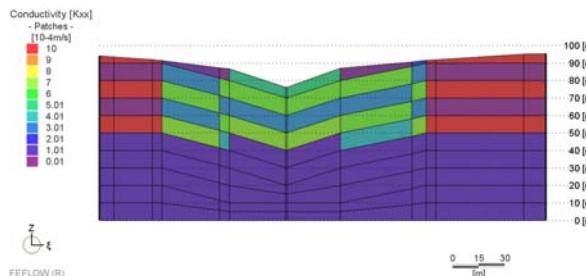


Figure 8.4 Unintended conductivity interpolation in free mode in a schematized layered aquifer (cross-section view).

As a rule of thumb, the free method is particularly suitable for cases where the water table varies within one model layer. It is less applicable to models where steeply dipping phreatic surfaces are expected.



## Phreatic

In phreatic mode, the model stratigraphy is fixed and, as a consequence, elements may become dry or partially saturated. In contrast to unsaturated mode, the calculation of the unsaturated zone is highly simplified and (with rarely occurring exceptions) only one phreatic surface is possible.

For each partially saturated element, the partial saturation is calculated by dividing the saturated thickness of the element by the total thickness of the element. Conductivity values in all directions are then linearly reduced by multiplying them with the partial saturation of the element.

For entirely dry elements (hydraulic head below the element bottom), a residual water depth is applied for the calculation of the partial saturation and reduced conductivity. This residual water depth can be defined on the **Free Surface** page in **Problem Class**.

Groundwater recharge is applied on the top of the model in phreatic mode and therefore has to pass the partially saturated/dry elements before reaching the water table.

The phreatic mode avoids all slice movement and related parameter interpolation and is therefore applicable to water tables with steep gradients that extend over multiple layers. On the other hand, dry elements with low conductivity values can lead to strong contrasts in the model, making the solution more difficult. The default low residual water depth might cause difficulties for the infiltration of recharge into dry soil, especially in cases with time-varying groundwater recharge.



*In models with thick dry layers the residual water depth may have to be increased in order to avoid very low actual conductivity values and increase model stability.*

In phreatic mode, the unconfined storage term is always only applied to the slice set to 'phreatic', i.e., usually to the top slice. The values for drain-/fillable porosity, however, are correctly derived from the layer where the water table is located at a given time. While this simplification has no negative consequences for typical regional models with significant horizontal flow components, it makes the phreatic mode less suitable for problems in which vertical flow is important, such as simulations of drainage of a soil column from the bottom.

Source and sink parameters assigned within phreatic layers can be scaled by the local pseudosaturation if the option **Scale sources/sinks by pseudosaturation** is set active. This will result in reduced source/sink values wherever the hydraulic head falls below the element top elevation (elements are not fully saturated). By default, the scaling of source/sink values is enabled to maintain compatibility with older FEFLOW versions not providing the two options.



## Free Surface Settings

To allow combining of different methods of phreatic-surface handling, the method is set for each slice separately.

Typically, the first slice is set to either phreatic or free, while all the slices below are set to dependent except for the bottom slice which is always set to fixed. Dependent slices are defined by the first slice above that is non-dependent, e.g., dependent slices below a free slice can move if necessary.

Layers whose top slice is set to confined are treated as fully saturated, no matter whether the hydraulic head is above or below the layer. In models using the free approach, fixed is especially useful to avoid slice movement (and possible material-property interpolation) in layers that are known to be saturated during the entire simulation.

## Head Limits for Unconfined Conditions

In unconfined models two particular cases may occur:

- All layers can be dry in a certain location.
- The water level can exceed the top of the model.

FEFLOW provides two options for dealing with a dry model bottom:

- Unconstrained head (default):  
Hydraulic-head values lower than the model bottom are tolerated. The saturated thickness is considered equal to the residual water depth at these locations.
- Constrained head:  
FEFLOW will prevent hydraulic-head values below the model bottom. For this purpose, first kind boundary conditions are internally applied at all bottom nodes that would otherwise fall dry, with a fixed head that equals the bottom elevation plus the residual water depth as defined on the **Free Surface** page.

For the top of the model domain, similar options are available:

- **Unconstrained head** (default):  
Hydraulic-head values exceeding the model top elevation are tolerated, and the parts above the top are treated as part of the first layer, i.e., calculation is done with the properties of the first layer (3D models). For 3D models using the phreatic mode for the free surface, alternatively the aquifer can be treated as confined as soon as it gets fully saturated. In 2D horizontal unconfined models, the model is treated as confined if the water level exceeds the top elevation, which is a material property in this model type and therefore can be set higher if this behavior is not desired.



- **Constrained head:**

FEFLOW will prevent water levels higher than the model surface. If hydraulic-head values higher than the top elevation would occur, FEFLOW sets fixed-head boundary conditions with a value of the top elevation at these locations.

Optionally, the setting **Prevent inflow** can be enabled for the top head limit. With this setting, the fixed-head boundary conditions act as seepage faces, i.e., FEFLOW applies an additional constraint condition that only allows outflow. This causes an iterative setting of the head limits per time step. Only up to 30 iterations are performed. If there are still changes in the location of these additional head boundary conditions after 30 iterations, FEFLOW proceeds to the next time step.

If the **Prevent inflow** option is disabled (default), the checking for nodes with a water level higher than the top is only done once per time step. The **Prevent inflow** option is more accurate, but requires more computational effort as up to 30 iterations per time step might be performed.



*Constraining the water table on top or bottom changes the overall water balance of the model. Especially adding water on the model bottom might not be acceptable.*

## 8.1.6 Anisotropy of Hydraulic Conductivity

### 2D Model

In a 2D model, the anisotropy of the hydraulic conductivity is defined through a set of material properties and not via the **Problem Settings** dialog. **Conductivity [max]** (**Transmissivity [max]** for confined models) defines the maximum conductivity. The direction of the maximum hydraulic conductivity is defined as **Anisotropy angle** (defining the angle between the x-axis and the max hydraulic conductivity). The ratio between minimum and maximum hydraulic conductivity is finally defined as **Anisotropy of conductivity**.

### 3D Model

In a 3D model, the **Anisotropy Settings** page of the **Problem Settings** dialog offers different approaches for the definition of anisotropy of hydraulic conductivity.

The following settings are available:

- **Axis-Parallel Anisotropy** (default):

The three different conductivities (**K\_xx**, **K\_yy**, **K\_zz**) are directed along the x, y and z axes of the model.



- **General Anisotropy with Computed Angles:** The principle directions for the main conductivities  $K_{1m}$ ,  $K_{2m}$ ,  $K_{3m}$  coincide with the layer inclination. The inclination is determined separately for each element.
- **General Anisotropy with User-Defined Angles:** The principle directions for the three main conductivities and the three Euler angles needed for the rotation of the coordinate axes are defined as material properties for each element.

## 8.2 Equation-System Solvers

FEFLOW offers multiple iterative and two direct equation solvers. By default, FEFLOW uses iterative solvers because they are suited for problems of arbitrary size. In advanced selection mode, separate iterative solver types can be selected for the symmetric (flow) and unsymmetric (transport) equation systems. The default options are a preconditioned conjugate-gradient (PCG) solver for flow and a BICGSTAB-type solver for transport. Alternatively, for either type of equation systems an algebraic multigrid solver can be chosen (SAMG). The main advantages of SAMG are its parallelization on multicore or multiprocessor systems, and its more efficient solution algorithm, in particular for steady-state simulations and simulations with large ranges of element sizes in the mesh. As the algebraic multigrid technique is not always the most efficient one, the SAMG solver automatically selects between a CG-type or AMG-type solution strategy according to the current conditions.

As the computational demand of the built-in, Crout-based method direct solver increases with the third power of the number of mesh nodes, the applicability of this solver has a practical limit of about 100,000 nodes. Storage and computational demand of the parallel sparse direct solver PARDISO are directly linked to the numbering of the finite-element mesh (and consequently the bandwidth of the global finite element matrix) and the number of degrees of freedom. As a consequence, PARDISO is particularly well-suited to solve matrices originating from 2D models even up to some millions of nodes (depending on hardware availability). For 3D problems with more than 1 million nodes, the use of PARDISO can still be considered but storage demand and solving time are problem and hardware dependent. Below the practical limit of 1,000,000 nodes, it is recommended to use PARDISO because the solution error then theoretically is at the level of machine precision.

## 8.3 Tutorial

In this exercise we load a number of models with different problem types to get familiar with the available problem settings.



### 8.3.1 Confined/Unconfined Models

We start with a very basic 2D flow model. Start FEFLOW and click **Open** to load the file **quadmesh.fem**. To check the basic settings of the model, open the **Problem Settings** dialog via the **Edit** menu.

On the **Problem Class** page we can see that the standard (saturated) groundwater-flow equation is used for the flow simulation and that confined conditions are assumed. Switch from a **Horizontal** projection to **Vertical, planar** and click **Apply**. The option  **Unconfined conditions** is now disabled as **Vertical, planar** models are always assumed to be confined unless the unsaturated mode is used. Additionally, the **Gravity Direction** page is displayed under **Problem Class**.

Switch back to a **Horizontal** projection and choose  **Unconfined conditions**. Once we confirm the changes with **Apply**, the **Free Surface** page is added in **Problem Class**. In the **Data** panel, the material-properties list automatically adapts to the changed settings. **Transmissivity** is replaced by the parameter **Conductivity [max]**, and **Top** and **Bottom elevation** now need to be defined.

Additionally, for unconfined (phreatic) models **head limits for unconfined conditions** can be set to define the model behavior when the water table touches the top surface, or the model falls dry at the bottom.

Leave the **Problem Settings** dialog with **Cancel** and close the model without saving the changes.

We proceed by studying a 3D flow model. Click **Open** and load the file **free3d.fem**. Again, open the **Problem Settings** dialog via the **Edit** menu. The **Problem Class** page shows that a saturated model type is set.

In contrast to 2D models the projection is fixed for 3D models, but the gravity direction can be changed from the default negative z-axis to a different direction.

On the **Free Surface** page, a **Fully confined system** or the setting **Unconfined aquifer(s)** can be chosen where two different options are available for the latter. Currently, a **Free** approach is chosen. When we change the status of slice 1 to **Phreatic**, we can also define whether the top slice will be treated as confined or unconfined when the water level reaches the top surface.

Leave the dialog with **Cancel**, activate the **3D** view and click **Start** in the **Simulation** toolbar to start the simulation. We can see the mesh moving in such a way that the top of the model is always at the water level elevation.

When the simulation has stopped, leave the simulator with a click on **Stop**. Reload the same file via **File > Recent FEM Problem files**.



Go back to **Edit > Problem Settings** and switch to **Fully confined system** on the **Free Surface** page. Click **Apply** and leave the dialog with **OK**. Start the simulation run for the changed conditions with a click on **Start**. The mesh geometry is now fixed and the resulting hydraulic head distribution differs from the **free** model run.

Terminate the simulation with a click on **Stop** and close the model without saving the changes.

### 8.3.2 Unsaturated Models

We continue with a 2D unsaturated flow model with vertical projection. Click on **Open** and load the file **dam\_seepage.fem**.

Start with the **Problem Settings** dialog located in the **Edit** menu. The **Problem Class** page shows that flow is simulated via Richards' equation and that the state is set to **Steady**.

The page list on the left-hand side now contains a new entry for **Unsaturated Flow**. Here, the basic settings for unsaturated models such as the form of the Richards' flow equation, iteration schemes and hysteresis settings can be changed. Do not do any changes here and switch to the **Problem Class** page again. As we can see, only **Vertical, planar** and **Vertical, axisymmetric** projections are available for 2D unsaturated models. Leave the dialog with **Cancel**.

In unsaturated models additional parameters are available in the **Data** panel. **Saturation** and **Moisture content** are added to the list of **Process Variables** and the **Material Properties** list now contains unsaturated-flow parameters. Double-click on **Unsaturated-flow model type**. The **Slice** view now shows that a **Van Genuchten modified** model has been chosen to describe the capillary pressure vs. saturation and saturation vs. hydraulic conductivity relationships.

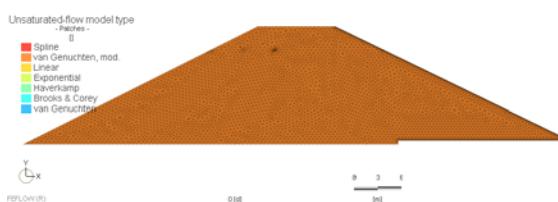


Figure 8.5 Unsaturated-flow model type.

### 8.3.3 Transport Models

To get familiar with the settings for transport simulations load the model **simulation.fem**.



Open the **Problem Settings** dialog via the **Edit** menu and start on the **Problem Class** page. Saturated conditions are assumed and transport of  **Mass** is included. The state for both flow and transport is set to  **Transient**.

In mass-transport simulations, the **Problem Settings** dialog contains two additional pages: In **General settings** on the **Transport Settings** page, we can choose between the  **Convective** and the  **Divergence form** of the transport equation and change the settings for the calculation of fluid viscosity and density. Additionally, the **Reference concentration** for the mass-transport calculation, and the **Reference temperature** for heat-transport problems are specified here.

Switch to the **Chemical Species** page. Here, additional species that are associated with the fluid or the solid phase can be defined if the model requires more than one species. To add or remove species from the list, the **Add species** or **Delete current species** tools can be used.

Proceed to the **Numerical Parameters** page. On this page, the settings for the **Error tolerance** and for the computation of mass matrices are located. Additionally, different upwinding techniques, e.g.,  **Full upwinding** or  **Shock capturing** can be selected to stabilize the transport calculation. As we can see, for the current model the default option  **No upwinding (Galerkin-FEM)** is set. Click **Cancel** to leave the dialog without any changes.

Notice how the **Data** panel has changed compared to a flow simulation: The **Process Variables**, **Boundary Conditions** and **Material Properties** trees now each contain an additional branch for mass transport (see Figure 8.6).

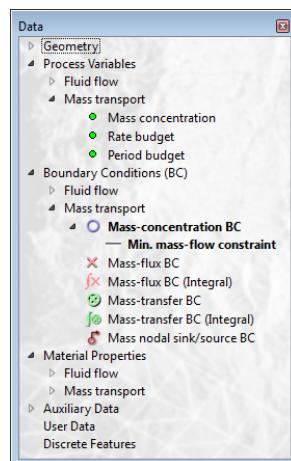


Figure 8.6 Data panel with mass-transport parameters.



In models with multiple chemical species, a separate branch is created for each species so that initial concentrations, boundary conditions and material properties can be defined separately for each species.

Start the simulation with a click on **Start** in the **Simulation** toolbar. The **Rate-Budget** and the **Period-Budget** panels now have two tabs, one for **Fluid** and one for **Mass**. To monitor the mass budget during the simulation, click on the tab **Mass** in both panels and set the check mark in front of  **Active**. If a model contains multiple species, the panels contain a separate tab for each species.

The **Content** panel also contains additional entries for transport. To evaluate the amount of dissolved mass in the model, start the content calculation by checking  **Dissolved Species Mass**. In a multispecies model, the **Content** panel contains a separate tab for each species.

Terminate the simulation with a click on **Stop**.

### 8.3.4 Steady / Transient Models

To study the available settings for simulations in steady and transient state, load the file **enclosed\_valley\_3.fem**.

Access the **Problem Settings** dialog via the **Edit** menu and go to the **Problem Class** page. Flow is simulated via the **Standard (saturated) groundwater-flow equation** and the state is set to **Steady**.

Leave the dialog with **Cancel** and start the simulation with a click on **Start** in the **Simulation** toolbar. After a brief computation the steady-state solution is reached. In addition to the **Hydraulic-Head** chart the **Error-Norm** chart is shown which plots the remaining dimensionless error for every iteration (see Figure 8.7).

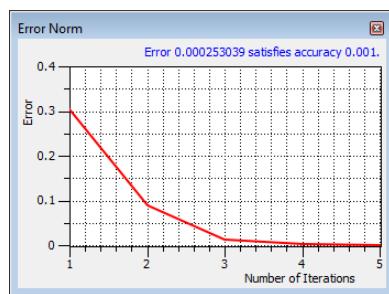


Figure 8.7 Error-Norm chart.

Terminate the simulation with a click on **Stop** and reload the same file via **File > Recent FEM Problem files** without saving the changes. Open the **Problem Settings** dialog again, select **Transient** on the **Problem Class**



page and click **Apply** to confirm the changes. To specify the time discretization for the transient simulation, go to the **Simulation-Time Control** page that now appears in the list on the left-hand side. The available time-stepping options are **Constant time steps**, **Varying time steps** and **Automatic time-step control**. By default, an **Automatic time-step control** based on a **Predictor-corrector scheme** with a **Second-order accurate (FA/BT)** integration scheme is selected. For the automatic time stepping, an **Initial time-step length** and the **Final simulation time** need to be specified. Change the **Final simulation time** to 3650 days and click **OK** to apply the changes and to leave the dialog.

Start the transient simulation by clicking **Start**. The current simulation time is displayed in the **Simulation** toolbar and, optionally, also at the bottom of each view window. All process variables such as hydraulic head can be monitored during the simulation. Instead of the **Error-Norm** chart that is displayed for steady-state simulations, the **Time-Steps** chart appears and shows the elapsed simulation time versus the time-step length (see Figure 8.8).

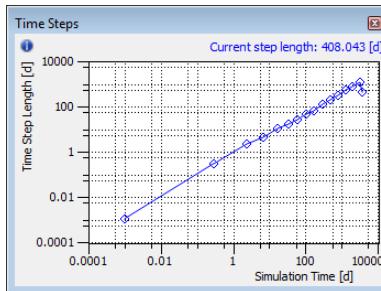


Figure 8.8 Time-Steps chart.





## 9 Working with Selections

Use selections efficiently for parameter assignment and results evaluation

### 9.1 Introduction

Selections of nodes, elements, edges and faces are among the fundamental concepts in FEFLOW. They are the basis for parameter input, visualization and postprocessing.

Selections can be stored with the model for repeated use—for example, to apply flow and mass-transport boundary conditions at the same nodes, or to start pathlines from nodes with well boundary conditions.

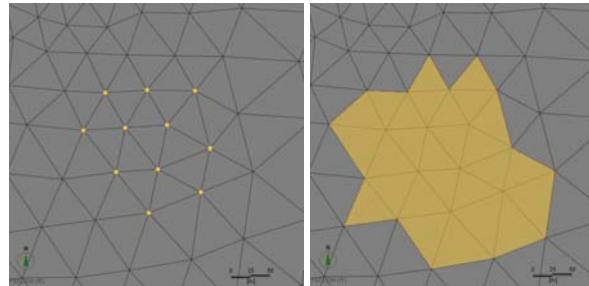


Figure 9.1 Nodal and elemental selections (Slice view).

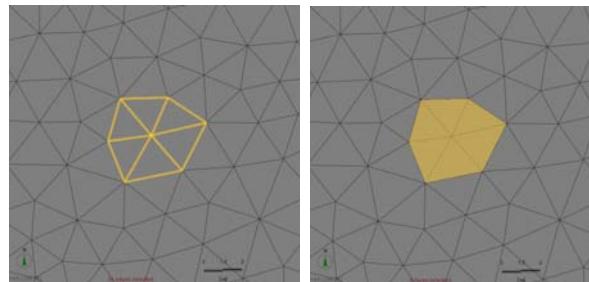


Figure 9.2 Edge and face selections (Slice view).

### 9.2 Selection Tools

Selections can be created by applying one, or a combination, of the available manual or map-based selection tools. The selection tools work on a node, element, edge or face basis, depending on the nature of the parameter that is



currently active in the **Data** panel or on the target geometry that is set in the **Selection** toolbar.



*When switching between different selection types, the last active selection of the current geometry type is restored automatically - unless the selection was cleared manually before.*

The manual selection tools are:

- **Select Individual Mesh Items**
- **Select Complete Layer/Slice (3D view)**
- **Select in Rectangular Region (Slice view)**
- **Select Using a Lasso (Slice view)**
- **Select in Polygonal Region (Slice view)**
- **Select by Map Point**
- **Select by Map Line**
- **Select by Map Polygon**
- **Select Nodes along a Border (Slice view)**
- **Select Arbitrary Node Path**

Map-based selection requires that the desired map is active in the **Maps** panel. An active map is indicated by a map name in bold letters. Selection based on both 2D and 3D maps is supported.

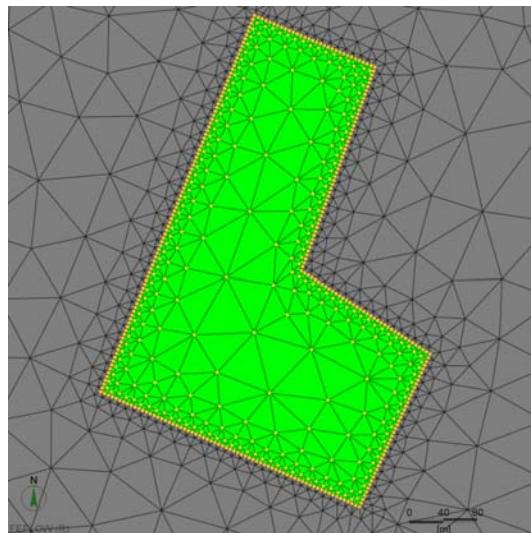


Figure 9.3 Nodal selection based on a map polygon (Slice view).

Interactive selection for a map includes selection based on points, lines, and polygons. The availability of certain selection modes depends on the map type: While for polygon maps polygon, line (edge) and point (node) selection is possible, for point files only point selection is available. In the **Slice** view,



clicking on a map element triggers the selection of all nodes, elements, edges or faces within the snap distance as defined in the **Snap-Distance** toolbar (Figure 9.3). For selections based on 3D map geometries in the **3D** view the chosen map settings in the **Problem Settings** dialog determine how a selection is created. The snap distance is not used in this case.

With the tool **Select by All Map Geometries**, a selection can also be performed for all map elements without user interaction.

For nodes and elements a further selection option exists: FEFLOW's **Expression Editor** can be used to create a selection based on a user-defined expression. The expression can contain process variables, boundary conditions, material properties, mesh data such as node/element numbers or coordinates and also temporal data like time series.

By default, applying a selection tool to a group of nodes or elements toggles the selection state of these nodes, elements, edges or faces, i.e., previously unselected mesh items become selected, and already selected ones are deselected. Alternative selection modes are available via the **Selection** toolbar: Create a new selection every time, always add to the current selection, remove from the selection, or intersect with the current selection (Figure 9.4).

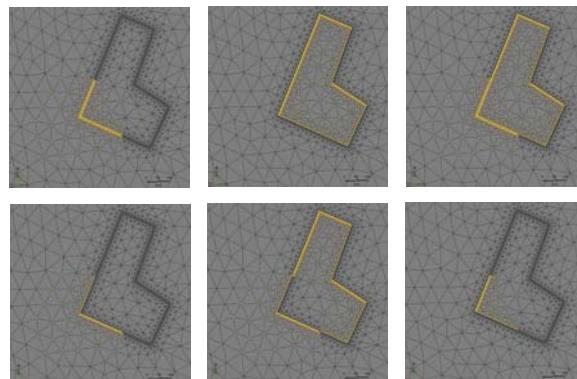


Figure 9.4 Selection modes: start selection, new, add, remove, toggle, intersect (upper left to lower right).



*To avoid that unintentionally previous selections are kept as part of a new selection, it is recommended to clear selections after using them.*

## 9.3 Storing Selections

Using the context menu of either the active view or the **Selections** panel, selections can be stored with the model for later use in parameter assignment, visualization or postprocessing.



Stored selections appear in the **Selections** panel and can be renamed via their respective context menu or set as the current selection. Moreover, the nodes/elements/edges or faces of the stored selection can be added to, removed from, or intersected with the current selection.

Any stored selection can be applied as navigation reference in the active view to redefine the settings for full view and the center of rotation according to the selection.

Stored node selections can also be used as domain of interest for the **Rate-** and **Period-Budget** panels. Additionally, the budget history can be displayed as a time series in the **Rate-Budget** and **Period-Budget** charts.

Stored element selections are available as budget domain for the **Rate-** and **Period-Budget** and **Subdomain Boundary Rate** and **Subdomain Boundary Period Budget** panels, as well as content domain for the **Content** panel.

The current or any stored selection can be converted into a selection of a different geometry type via the context menu of the selection in the **Selections** panel. If a **Slice** view is the active view, the user can choose between converting a selection into a new selection on the current slice/layer (option 2D) or into a 3D selection.

One or multiple selections (keeping the **<Ctrl>** key pressed while selecting) which can be of different geometry types can be exported as **Location Set Collection** via their context menu in the **Selections** panel. A re-import option for the **Location Set Collection** is also accessible via the context menu of the **Selections** panel. This allows for a convenient transfer of stored selections between different model scenarios.

## 9.4 Tutorial

### 9.4.1 Tools



Figure 9.5 Selection toolbar.

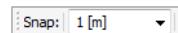


Figure 9.6 Snap-Distance toolbar.



## 9.4.2 General Remarks

Whether nodes, elements, edges or faces are selected depends on the target geometry chosen in the **Selection** toolbar.

In the following exercises all selections are created for nodes. Selections on an elemental basis or edge/face selections can be created following the same principle, though.

## 9.4.3 Manual Selection

As a first exercise, we create a number of selections manually. First, click on **Open** and load the model **selections.fem**. Two views are displayed, a **Slice** view (2D) and a **3D** view window.

### Selections in 2D

Click in the **Slice** view to make it the active view and then browse to slice 1 in the slice list of the **Entities** panel.

Choose the tool **Select Individual Mesh Items** from the **Selection** toolbar, choose **Select Nodes** as target geometry and click on an arbitrary number of nodes on slice 1. The selected nodes are highlighted in yellow. Delete this selection again with a click on **Clear Selection**.



Figure 9.7 Node selection in Slice view.

Instead of selecting individual nodes, we can use **Select All** to create a selection that contains all nodes if a **3D** view is active, or all nodes on the current slice in an active **Slice** view.

After a click on **Clear Selection**, choose the tool **Select in Rectangular Region** and draw a rectangle to create a new selection. Draw a second rectangle that partially overlaps the first one. As we can see, the selection is toggled: The already selected nodes are removed and newly selected nodes are added to the selection. This default selection mode can be changed by



clicking **Toggle in Selection** which exposes all available modes. Click on **Undo Last Selection Step** to reverse the selection created with the second rectangle. Now, activate the **Set New Selection** mode and draw a rectangle similar to the one we just removed. The result is quite different: The nodes in the first rectangle are now deselected and a new selection is set instead. Once again, click **Undo Last Selection Step** and try the other modes available for setting selections.

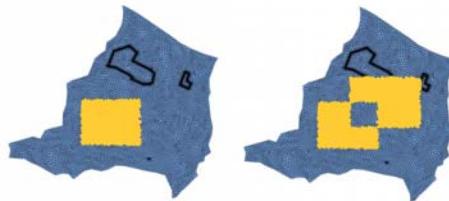


Figure 9.8    Toggled selection.

A tool that is only available for node selections in the **Slice** view is **Select Nodes along a Border**. Clear the current selection and switch back to the **Toggle in Selection** mode. Click and hold the left mouse button on the westernmost node of the southern border. Move the cursor further along and release it at the last node of that border. The selected nodes are now highlighted in yellow.

To also select the corresponding nodes on the slices below, we use the **Copy Selection to Slices/Layers** tool in the **Selection** toolbar. Choose all slices as the target slices. Confirm with **OK**. In the **3D** view we can see that the nodes of the southern border have been selected on all slices.



Figure 9.9    Selected nodes at southern border.

## Selections in 3D

Six selection tools are available in a **3D** view:

- **Select All**
- **Select Individual Mesh Items**
- **Select Complete Layer/Slice**



- **Select by Map Point**
- **Select by Map Line**
- **Select by Map Polygon**

For node selections an additional seventh tool for the assignment of 1D arbitrary **Discrete Features** is available:

- **Select Arbitrary Node Path**

We now create a selection that contains all nodes of the top slice. Make sure that the **3D** view is the active view window. Choose the **Select Complete Layer/Slice** tool from the drop-down menu of the **Selection** toolbar. Placing the mouse cursor over any node in the top slice highlights all its nodes. Create the selection with a single click. The selected nodes are displayed as yellow spheres.

#### 9.4.4 Map-based Selections

A further option to create selections is to use maps. The points, lines and polygons contained in the **Supermesh** are available in the **Maps** panel and selections can be set based on these features.

For this exercise, we use the same FEFLOW model as in the previous exercises. This model contains a number of maps and also its **Supermesh**.

Selections based on 2D maps are only possible in the **Slice** view, so we first have to make sure that this is the active view.

In the first exercise we select the nodes of the well locations contained in the map **demo\_wells**. These locations have also been included in the **Supermesh** and are located in the centers of the refined areas in the southern part of the model area. Browse to slice 4 in the **Entities** panel, then switch to the **Maps** panel and activate **demo\_wells** (not the **Default** entry) with a double-click. Enter **1 m** in the **Snap-Distance** toolbar, activate the tool **Select by Map Point** and click on **Select by All Map Geometries** to create the selection. The selected nodes are now highlighted in yellow. Before we proceed, clear the selection with a click on **Clear Selection**.

In the next step, we create a selection for the nodes within the polygon that is defined in the map **sewage\_treatment**, the refined area on the left side of the model. Activate this map in the **Maps** panel with a double-click, click on **Select by Map Polygon** and then click **Select by All Map Geometries** again. Alternatively, it is possible to select this polygon by clicking on it in the **Slice** view while **Select by Map Polygon** is the active selection tool. Again, finish with a click on **Clear Selection**.

The **Supermesh** can be used to create selections in the same way. Activate **Supermesh > Polygons** in the **Maps** panel. When the mouse cursor is moved over a polygon it is highlighted. Set all nodes within with a single click.



## 9.4.5 Expression-based Selection

The **Expression Editor** can be used to create selections based on a user-defined expression.

Click on **Clear Selection** to make sure that no selection is active and switch to the **3D** view.

To open the **Expression Editor**, click on the tool **Select by Expression**. We want to create a node selection that contains all nodes with a hydraulic-head value of less than 65 m.

First, click into the working window and delete the **true** statement. The table on the right-hand side of the editor contains all input parameters in FEFLOW and also some temporal data and mesh properties than can be used within an expression. Open **Process Variables > Fluid Flow** in the table and double-click on **Head**. Hydraulic head is now added to our expression. To finish the expression that will be evaluated for the selection, click on the **< less-than** operator symbol and then type **65** without a unit. The current user unit for each parameter is displayed in the table on the right-hand side or in the tooltip of the parameter in the work area. User units can be changed via the context menu of a parameter in the **Expression Editor**. The final expression should look like the one shown in figure 9.10. To create the selection based on the defined expression click on **Apply**. Leave the dialog again with **Close** or by activating a different tool.

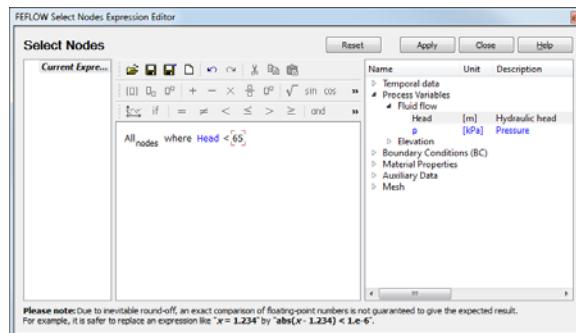


Figure 9.10 Expression Editor for node selection.

## 9.4.6 Storing Selections

Selections can be stored for later use in parameter assignment, visualization or postprocessing actions.

To store the current selection, invoke the context menu by right-clicking in the active view or on an empty part in the **Selections** panel. Choose the option **Store Current Selection** in the context menu and either enter a



new name or accept the suggested default name for the node selection and confirm with <Enter>. The stored selection appears in the **Node Selections** branch in the **Selections** panel.

Clear the selection with **Clear Selection**. To set the selection active again, double-click on it in the **Selections** panel.

#### 9.4.7 Converting Selection Types

Make sure that the **Slice** view is the active view and use the **Slice** entries in the **Selections** panel to switch to slice 1. Activate the stored node selection **Navigation Selection** with a double click. We now convert this node selection into an element selection: Right click on **Navigation Selection** and select **Convert to > Element Selection 2D**. FEFLOW now automatically selects all elements adjacent to the nodes of the stored selection. The result is shown in figure 9.11.

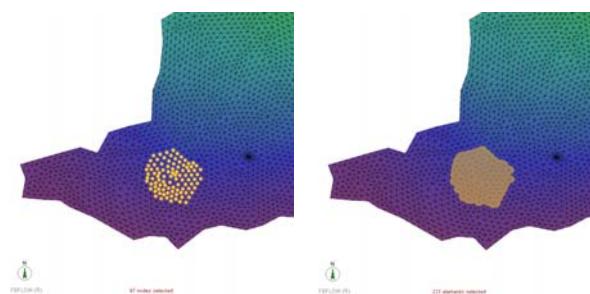


Figure 9.11 Node selection converted into element selection.

#### 9.4.8 Exporting and Reimporting Selections

Selections can be exported and reimported, e.g., for use in alternative model scenarios.

Click on **Navigation Selection** in **Node Selections** in the **Selections** panel, press the <Ctrl> key and click on the name of the node selection which we have stored in chapter 9.4.6. Next, right click on one of the highlighted selections, select **Export Location Set Collection** from the context menu and specify a name for the file to be exported. Confirm with **Save**.

Before reimporting the exported selections, we delete the currently stored node selections. Again, select both selections simultaneously using the <Ctrl> key and then press the <Del> key twice to remove both selections from the **Selections** panel. Now, right-click either on **Node Selections** or in an empty part of the **Selections** panel, select **Import Location Set > Location Set Collection** and choose the file that we have just exported.



The previously exported selections now reappear under **Node Selections** in the  **Selections** panel.

#### 9.4.9 Using Selections as Navigation Reference

Make sure that the **3D** view is the active view and double-click on the stored node selection **Navigation Selection** in the  **Selections** panel to set it active. Next, open the context menu of the selection with a right click and select  **Set as Navigation Reference in Active View**. The settings for full view and the center of rotation are now adapted according to this selection.



# 10 Parameter Visualization

## Plotting input parameters and results

### 10.1 Introduction

Visualization of the model parameters is not only essential during model parameterization but for checking the assigned model properties, for evaluating the model results, and for the presentation of model properties and simulation results.

### 10.2 View Windows

All visualization options refer to a specific view window (see ). Depending on the type of view window, different options for visualization are available. Each view window manages its own list of visualized parameters and visualization styles and settings. The panels controlling visualization always reflect the active view window.

### 10.3 Model Geometry and Data Plots

In the **Entities** panel, the visualization target geometry is chosen. For example, in a **3D** view, a parameter may be plotted on the outer boundary of the model domain, but also on a single slice or the boundary of a parameter zone.

For **Slice** views, the target geometry is a layer or slice, depending on whether nodal or elemental parameters are shown. The **Entities** panel (or the <Pg Up> and <Pg Down> keys) can be used to switch between different layers/slices. **Cross-sectional**, **Slice Data**- and **Data-Trace** views are not affected by the selection of a geometry in the **Entities** panel. In case the active view is a **3D** view, the geometry of any spatial unit can be added to the view by a double-click on the unit, or via the context menu.

To provide more convenience for working with models with a large number of layers/slices and such as stored selections, it is possible to open more than one **Entities** or **Selections** panel at a time via the context menu of the panel. This allows to use one panel for switching between different slices/layers while stored selections or surface and domain locations can easily be accessed via a second panel.

A parameter or parameter group (e.g., the group of all flow boundary conditions) is added to the view by double-click on the parameter in the **Data** panel, or by using the context menu of one or several selected parameters. The parameter will be plotted on the geometry that is active in the **Entities** panel. Parameters can be plotted on more than one geometry at the same time, e.g., on the bottom slice and the vertical hull.



## 10.4 Visualization Options

All geometry (3D view) and parameter information currently available in the active view are listed in a tree in the **View Components** panel. As the tree always reflects the active view, it automatically changes when switching between the view windows. By changing the check box in front of the leaf and branch items in the tree, visibility of the corresponding view components can be toggled.

For parameters, different visualization styles may be available, one of which is active by default. For example, hydraulic head can be shown as continuous plot, fringes, isolines, or isosurfaces in a **3D** view.

The parameters and visualization styles have properties that can be edited in a **Properties** panel. The panel is opened by double-clicking on the parameter or style in the **View Components** panel, or by using the context menu of the parameter or style. The contents of the **Properties** panel depend on the parameter or style, and are described in detail in the FEFLOW help system.

Whenever a new parameter is added to a view, by default the previously shown parameter is removed. While this is a convenient behavior for quickly editing different parameters one at a time, in many cases it is preferable to visualize multiple parameters simultaneously. For this purpose, parameters can be locked via their context menu in the **View Components** panel. Locked parameters are not removed when adding a new component.

## 10.5 Clipping and Carving

Up to six planes can be arbitrarily defined and combined in each **3D** view window to expose data otherwise hidden within the model domain. Each of the six planes of the active view can be accessed via the **Planes** panel. For each plane, the panel provides controls to activate the respective plane and to specify whether the plane is to be considered as part of a cutout combination or as a simple clipping plane. A simple clipping plane clips all parts of the object located on its back side. When using a cutout combination, only those object parts are clipped that are on the back side of all clipping planes of the cutout combination.

Simple clipping planes can be applied in addition to a cutout combination.

The location of each clipping plane can be changed in the **Plane definition** dialog that is opened by clicking on the respective model view icon in the **Planes** panel. While modifying the plane equation, the plane is shown in the active view window.

Whether a visualization item is clipped or not is controlled via its context menu in the **View Components** panel.

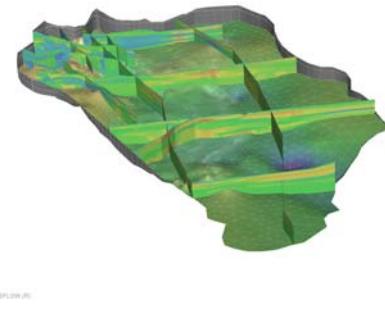


Figure 10.1 Cross-section visualization in 3D.

## 10.6 Inspection

The active parameters in the current view and the attributes of maps shown in the view can be inspected, using the **Inspection** tool which is invoked via the **Inspection** toolbar. The properties of the target location are shown as constant values or curves for transient properties in the **Inspection** panel.

Parameters and map attributes can be displayed in the same list or the panel can be split into two separate parts, the upper one listing the parameters and the lower one displaying the map attributes.

The panel content can be filtered by a full-text search which is accessible via the search bar on top of the panel or in the split-panel sections.

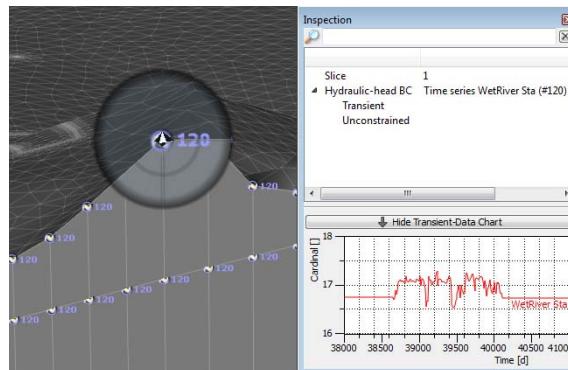


Figure 10.2 Checking boundary conditions with the Inspection tool.

## 10.7 Scene Library

View settings for **Supermesh**, **Slice**, **3D** and **Cross-section** views can be stored in a **Scene Library**. For models with a large number of views, this



bears the benefit of reducing the required memory or CPU load significantly compared to keeping all views open all the time. With the provided export and import functions a large number of identical views can easily be created in different model scenarios.

View settings can be stored as scenes via the context menu of a view or with <Ctrl>-<Shift>-<s>. The stored scenes appear as tiles in the **Scene Library** panel and can be renamed by pressing <F2>. Placing the mouse cursor on a stored scene shows a preview of the scene as a tooltip.

Scenes can easily be restored in existing or new view windows via their context menu. Double-clicking on a scene icon applies the scene to an existing view of the same type or opens a new view window in case that no other view of the same type already exists.

Scenes can be organized in a user-defined folder structure and also be filtered using the check boxes for **Slice View**, **3D View** and **Others** in the **Scene Library** panel or via a full-text search. The entire **Scene Library**, selected scenes or folders can be exported into an \*.xml file via the context menu of a specific entry in the **Scene Library** panel.

Additional **Scene Library** settings are accessible via the menu **Tools** and **Global Settings > Tool Properties**.

## 10.8 Stereoscopic Visualization

FEFLOW 6.2 supports 3D stereoscopic display of model geometries and data. Especially for 3D models with complex geometries, stereoscopic visualization of the model helps to highlight 3D structures due to the enhanced depth perception. Prerequisite for stereoscopic visualization are computer displays, TVs or projectors that support 3D stereoscopic display, and typically also adequate glasses.

The export of stereoscopic images in \*.jps or \*.pns format and animations in \*.avis format is supported. FEFLOW is compatible with common 3D interface standards (shutter and polarizer technology).

Different view settings for stereoscopic visualization and snapshot or movie export are available. FEFLOW supports the following visualization modes:

- Row-interleaved
- Active 3D (Quad-buffered)
- Horizontal Side-by-Side
- Vertical Side-by-Side

These settings can be accessed via **Extended View Settings** in the context menu of a **3D** view window or in the dialogs for movie or snapshot export. The choice of a mode depends on the capabilities of the available 3D hardware.



## 10.9 Tutorial

The visualization options described in the following exercises are available during model setup, simulation run and also in postprocessing.

### 10.9.1 View Windows

Start FEFLOW, click on **Open** and load the file **visualization.fem**. A **3D** and a **Slice** view are displayed. For the **3D** view, **Conductivity K\_xx** is the active parameter. **Hydraulic head** is the active parameter in the **Slice** view.

Pay attention to the changes when we switch between different views: Click in the **Slice** view to set it as the active window. **Hydraulic head** then becomes the active parameter in the **View Components** and in the **Data** panel and **Conductivity K\_xx** is removed from the **View Components** panel. Additionally, the **Entities** panel now shows that we are currently working on slice 1.

### 10.9.2 Add Model Geometry and Parameters

Click in the **3D** view to set it as the active view. Double-click on **Hydraulic head** in the **Data** panel to plot this parameter. Note that the previously displayed parameter disappears from both the active view and the **View Components** panel when a new parameter is plotted.

It is also possible to display different parameters at the same time: Open the context menu of the entry **Hydraulic head** in the **View Components** panel and select **Lock Data View**. The lock icon now appears in front of **Hydraulic head**. In the **Data** panel double-click on **In/outflow on top/bottom** to add this parameter to the active view. Both parameters are now displayed at the same time.

In the **3D** view not only parameters but also additional geometry can be added to the active view. Go to the **Entities** panel and double-click on the entries **Slice 1** and **Slice 4**. These slices are now added to the active view. A new item for each slice appears in the **Faces** and **Edges** branches of the **Geometry** tree in the **View Components** panel. To plot only slices 1 and 4 and not the entire domain uncheck the entries **Faces > Domain** and **Edges > Domain** and also **Hydraulic head > Continuous > Domain** and **In/outflow on top/bottom** in the **View Components** panel. We now want to plot a different parameter on each of the two slices. Start with slice 1. In the **Entities** panel click on **Slice 1** to set it as target geometry for the plot. Then, double-click on **Hydraulic head** in the **Data** panel. Now click on **Slice 4** in the **Entities** panel and double-click **Drain-/fillable porosity**. Not only parameters but also maps can be plotted to selected geometries. In the **Entities** panel, set **Slice 1** as target geometry. Uncheck **Hydraulic head** in the **View Components** panel and double-click on **topogra-**



**phy\_rectified** in the **Maps** panel. The map is now visible on slice 1 in the **3D** view. The resulting plot in the **3D** view is shown in figure 10.3.

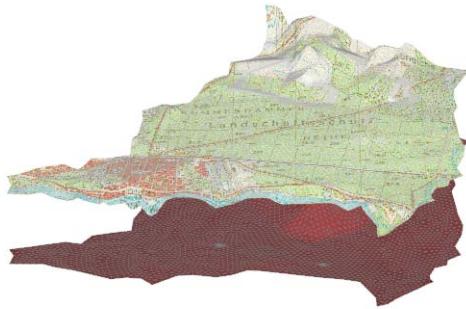


Figure 10.3 Plotting on selected geometries in 3D.

### 10.9.3 Visualization Options

We now select different visualization styles for the parameter **Hydraulic head**. Set the **Slice** view as active view. In the **View Components** panel, three styles are available for this parameter: **Continuous**, **Fringes** and **Iso-lines**. Display the different styles by checking the check box in front of the respective entry.

The properties of each style can be edited in the respective **Properties** panel. Open the context menu of **Hydraulic head > Iso-lines** in the **View Components** panel with a right-click and select **Properties**. Several editing options are available: In the **Iso** tab, the isolines settings can be changed. Use a **Prescribed interval** for isolines, deactivate the **Automatic** option and enter **0.25** as constant isoline interval. Switch to a **Fixed** color source for the labels on the tab **Color** and click **Apply** to confirm the changes.

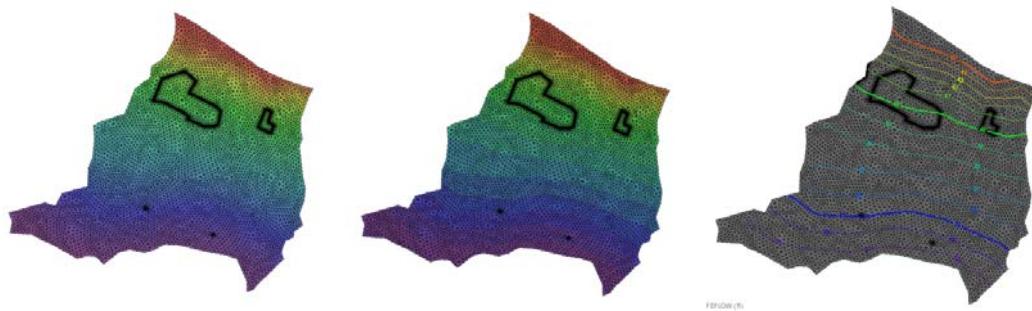


Figure 10.4 Hydraulic head values displayed as continuous, fringes and isolines plot.

To hide the legend of a particular parameter style open the context menu of the style entry in the **View Components** panel and uncheck **Legend Display**.

To edit the number format of the displayed parameter legend select **Number Format** in the context menu of **Hydraulic head** in the **View Components** panel. Here, switch to **Fixed floating-point** and change the number of significant digits to **2**. Clicking **OK** confirms the changes and the legend is now displayed with the new settings.

The data range of the parameter plot and also the color scheme are edited in the **Properties** panel of the parameter. To open the panel right-click on **Hydraulic head** in the **View Components** panel and select **Properties**. Uncheck the box in front of **Auto-update range** and set a new minimum value of **34 m** and a maximum value of **42 m** for the data range. The changes are confirmed with a click on **Apply**. Uncheck **Hydraulic head > Fringes** in the **View Components** panel. At the grey parts of the model the hydraulic-head values exceed the defined minimum and maximum values for the hydraulic-head display. To plot these zones of the model with the same color as the minium and maximum values, check the option **Tolerate Outliers** in the **Properties** panel and click on **Apply**. Both the parameter plot and the legend in the active view reflect the now tolerated outliers.

To change the color style of the parameter plot right-click in the colored sidebar and switch to one of the available presets, e.g., the FEFLOW Classic style. The changes are immediately applied and displayed in the active view.

#### 10.9.4 Clipping and Carving

Go to **Window > New > 3D View** to open a new window. In the **Navigation** panel click on the **Projection** tab on the right-hand side of the panel and move the **Scaling** slider bar upwards to stretch the model and to add some vertical exaggeration for better visibility of the different layers.



In the **Planes** panel the six different planes available for clipping and carving are displayed. Activate the plane in the second box by setting the check mark in front of  **Active**. All parts located on the yellow (front) side of the plane are now clipped away. To change the position of the plane open the **Plane definition** dialog by clicking into its view in the **Planes** panel. Use the slider bars to change the offset or the normal vector. Confirm the settings with **Apply** and leave the dialog with **Close**.

Now activate plane number 4 and select the option **Carving** for both planes. Now, only the features on the yellow sides of *both* planes are cut away.

Visualization examples with carved models are shown in Figures 10.5 and 10.6.

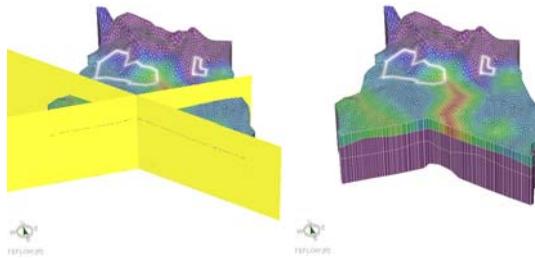


Figure 10.5 Carving planes and carved model domain in 3D view.

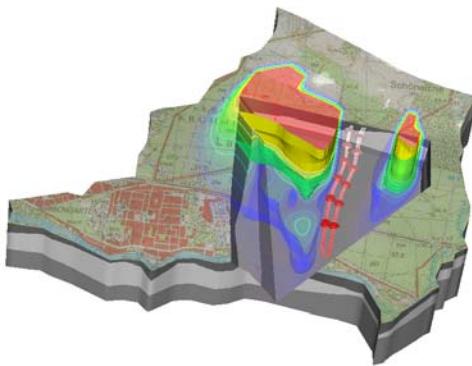


Figure 10.6 3D visualization example.

## 10.9.5 Scene Library

We use the **Scene Library** panel to store the settings of some of the views that we have just created. Right-click into the current **3D** view and



select **Create Scene from Window**. The stored scene now appears as a tile in the **Scene Library**. To show a preview of the scene, place the mouse cursor on the icon. A preview, the name of the scene and the creation date and time are displayed.

To rename the scene, click on its icon and press <F2>. For better identification of the scene, we also add a comment in addition to the name. Right-click onto the icon of the scene and select **Edit Scene Properties**. Add a comment to the scene and confirm with OK.

We now create a new folder in the **Scene Library**. Right-click into an empty part of the panel and select **Create Folder**. To rename the folder, click on it and press <F2>. In order to view the folder structure within the **Scene Library**, select **Show Folders** from the context menu of the panel.

To store a scene in this specific folder, double-click on it before creating a scene. Next, switch to the **Slice** view and use the context menu of the view to create a scene of this view as well. The scene now appears in the subfolder and not in the main level of the **Scene Library**.

To move the stored scene to the main level, click on the scene and move it to the entry **Scene Library** while keeping the <Shift> key pressed. Dragging and dropping a scene to a different location in the library without pressing the <Shift> key creates a copy of the scene instead of moving it.

Close the currently open **Slice** view. To create a new view from the stored slice-view scene, double-click on its icon in the **Scene Library** or use **Create View from Scene** in the scene context menu.

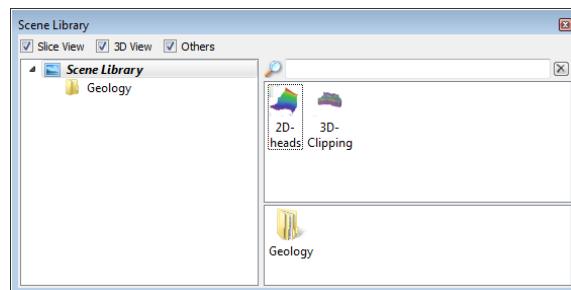


Figure 10.7 Scene Library panel.





# 11 Parameter Assignment

Input of initial and boundary conditions, material properties, user data and discrete features

## 11.1 Introduction

The task of assigning model parameters is in many cases repeatedly performed during model setup and application. After the initial parameterization, model properties may have to be changed during calibration, or parameters need to reflect different scenarios. Thus the work flows for parameter assignment are crucial for the efficient handling of a groundwater model.

Typical assignment procedures are based on the activation of a property to be assigned and the selection of the target nodes or elements. The tools for selecting nodes or elements are described in detail in chapter .

There are some exceptions from this standard workflow: When working with map data, FEFLOW provides the means to automatically activate a parameter and/or select the target geometry based on information in the map or linked to the map.

## 11.2 Input Parameters

FEFLOW distinguishes between seven groups of parameters, all of them visible on the first level of the tree view in the **Data** panel (Figure 11.1):

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

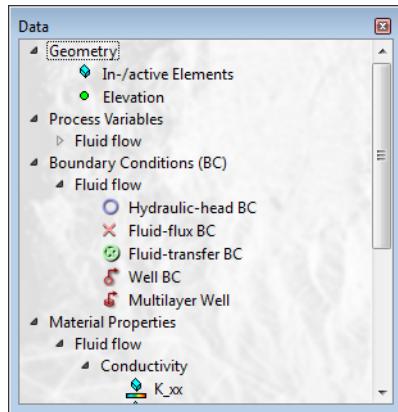


Figure 11.1 Data panel.

The geometry section of the **Data** panel contains the option to set mesh elements active or inactive for a simulation and the node elevation values in 3D models. While process variables (including initial conditions and with the exception of elemental Darcy fluxes) and boundary conditions are defined on the mesh nodes, material properties representing the characteristics of the medium are assigned on an elemental basis. Auxiliary data cannot be edited or assigned but can be used to check some properties of the finite-element mesh or to visualize stream functions in 2D models. User data are additional nodal or elemental distributions that can also be based on user-defined expressions. Assignment and usage are discussed in section . Discrete features can be added along element edges or faces to represent high-conductivity features or they can connect two arbitrary nodes.

### 11.2.1 Geometry

With the elemental parameter **In-/active Elements** mesh elements can be activated or deactivated during the entire simulation or only at specific time stages. By default, all mesh elements are active after mesh generation. More information on the handling of inactive elements can be found in chapter .

In 3D models, **Elevation** is included in the geometry section as a nodal parameter. In models with a fixed mesh, the elevation does not change during the simulation. In models representing a phreatic surface with the free approach (see ), the actual nodal elevation changes during the course of the simulation. The parameter **Reference Elevation** preserves the original node elevations.

### 11.2.2 Process Variables

Process variables include the primary variables hydraulic head, mass concentration, mean age, mean lifetime expectancy, exit probability and temper-



ature (as applicable). When setting up the model, they describe the initial conditions. During and after the simulation these process variables reflect the then-current conditions.

Other process variables contain values derived from the original primary variables. For example, pressure is not stored separately in FEFLOW, but calculated from hydraulic head and elevation ‘on the fly’. When these secondary parameters are used as input variables, the input is converted into the original primary variable based on current conditions.

The third type of process variables are auxiliary variables supporting results evaluation and visualization, such as Darcy flux, rate and period budget and streamlines or pathlines. They cannot be used as input parameters.

### 11.2.3 Boundary Conditions

By default, all model boundaries in FEFLOW are impervious. To allow flows into or out of the model, boundary conditions have to be defined. FEFLOW supports four basic types of boundary conditions for flow, mass, groundwater age and heat transport which can be defined as constant over time or time-varying values. For flow and heat transport, additional fifth boundary condition types are available:

- ***Dirichlet-type BCs:***  
**Hydraulic-head BC / Mass-concentration BC / Age-concentration BC / LTE-concentration BC / Probability BC / Temperature BC**
- ***Neumann-type BCs:***  
**Fluid-flux BC / Fluid-flux BC (Gradient) / Mass-flux BC / Heat-flux BC**
- ***Cauchy-type BCs:***  
**Fluid-transfer BC / Mass-transfer BC / Heat-transfer BC**
- ***Nodal source/sink type BCs:***  
**Well BC / Mass nodal sink/source BC / Heat nodal sink/source BC**
- ***Multilayer Well* (flow)**
- ***Borehole Heat Exchanger* (heat transport)**

For groundwater age, only the first boundary condition type is displayed by default and additional types have to be added via the respective context menu in the **Data** panel.

Except for Borehole Heat Exchangers, the sign convention for boundary conditions in FEFLOW defines flows out of the model as positive (e.g., well abstraction), while inflows are negative (Figure 11.2). Notably, this convention differs from other parts of FEFLOW, such as the **Rate-** and **Period-Budget** or the **Subdomain Boundary Rate** and **Subdomain Boundary Period Budget** panels or the definition of constraints, where inflows are considered as positive, outflows as negative (Figure 11.4).

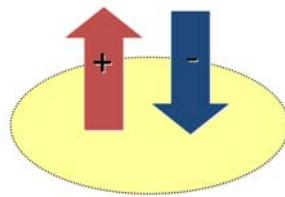


Figure 11.2 Algebraic signs for boundary conditions.

## Dirichlet-type BCs

This boundary condition type - often also referred to as 1<sup>st</sup> kind boundary condition - specifies a time-constant or time-varying value for the primary variable at a node, i.e. hydraulic head for flow, concentration for mass transport, groundwater age and lifetime expectancy, probability of exit and temperature for heat transport. The inflow or outflow to/from the model domain at the node can be calculated from the simulation result.

In certain model types, such as unsaturated or density-dependent simulations, it is possible to derive the boundary-condition value from input such as:

- Pressure
- Saltwater Head
- Saturation
- Moisture Content

While converting input from a different quantity in these cases, and providing the possibility to visualize the boundary conditions in terms of another quantity, the actual boundary condition stored and used during the simulation is always a Hydraulic-head BC.

## Neumann-type BCs

The flux-type boundary condition describes an in- or outflow of water/mass/energy at element edges (2D) or element faces (3D). Though nodally defined, the condition must be applied for at least two adjacent nodes (2D) or all nodes of a vertical or horizontal element face (3D) to be effective. The given value in flow simulations is a Darcy flux perpendicular to the boundary. An optional alternative formulation for unsaturated models allows the definition of a hydraulic-head gradient instead of the Darcy flux.

The successful application of Mass-flux BCs and Heat-flux BCs is linked to the application of the convective or divergence form of the transport equation (). The FEFLOW Book and the help system should be consulted for details.

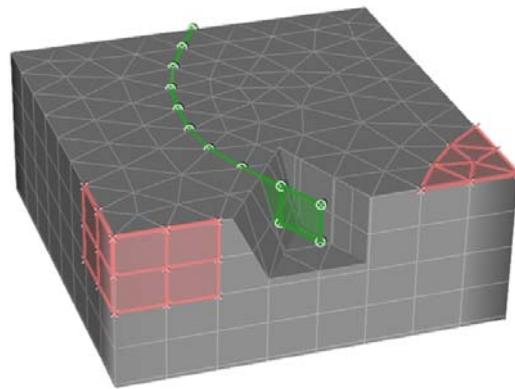


Figure 11.3 Flux and transfer boundary conditions in 3D model.

#### Cauchy-type BC

**Fluid-transfer BCs** can be used to describe rivers, lakes, and known hydraulic heads in a distance from the model boundary (sometimes called 'general head' boundaries). The condition is used to apply transfer properties between a reference value for the primary variable (hydraulic head, concentration, or temperature) and groundwater. Thus the value of the boundary condition is a reference hydraulic head (e.g., river water level), reference concentration, or reference temperature. The hydraulic conductance (e.g., properties of a clogging layer), mass-transport conductance or thermal conductance are defined as material properties on elements adjacent to the boundary condition. Similar to the **Neumann-type BC** the transfer boundary condition must be specified along a line (2D) or for an entire element face (3D—Figure 11.3).

#### Nodal source/sink-type BCs

**Well BCs** and their counterparts for mass and heat transport simulation are nodally applied and represent a time-constant or time-varying local injection or abstraction of water, mass or energy at a single node or at a group of nodes.

If it is intended to simulate a well screened in different layers it is recommended to use the multilayer-well boundary condition.

The successful application of **Mass nodal sink/source BC** and **Heat nodal sink/source BC** is linked to the application of the convective or divergence form of the transport equation ( ). The FEFLOW Book and the help system should be consulted for details.



*The resulting hydraulic head at well nodes depends on the local spatial discretization. Quantitative comparisons should be made with care.*



## Multilayer Wells

In 3D models, multilayer-well boundary conditions can be used to simulate water injection/abstraction via a well screen. The screen can extend over one or multiple model layers.

The well radius and screen top and bottom together with the well capacity are used as input parameters for this boundary condition. Internally, the total injection/abstraction rate is assigned at the bottom-most node of the well and a highly conductive 1D discrete feature is placed along the well screen. Flow within the well along the screened interval is simulated via the Hagen-Poiseuille (cubic) law. The appropriate parameters of the discrete feature are derived from the specified well geometry.

The properties of a multilayer well are defined in the **Multilayer Well Editor** dialog. The dialog is invoked by activating the boundary condition in the **Data** panel and double-clicking on **Multilayer Well** in the **Editor** toolbar.

The partitioning of the injection/abstraction rate to the different model slices/layers that are part of the well screen depends on the surrounding material properties and hydraulic-head distribution and therefore is a result of the simulation. It is calculated from the model results and can be visualized as a property of the multi-layer well during and after the simulation run.

## Borehole Heat Exchangers

Borehole heat exchangers are used to simulate closed-loop geothermal installations. A refrigerant circulates within closed pipes and heat exchange with the surrounding aquifer system is solely driven by conduction of heat.

Borehole heat exchangers are represented as embedded 1D elements and linked to the FEFLOW nodes along join edges in a 3D model.

To define the inflow boundary of a heat exchanger, a time-constant or time-varying flow rate of the circulating refrigerant and the inflow temperature are required. While the flow rate is always directly specified, FEFLOW provides the following four options for the definition of the inflow temperature, each of them possibly also being time-varying:

- Inlet Temperature
- Heat-input Rate
- Temperature Difference
- Power

Additionally, the properties of the refrigerant and the components of the bore-hole heat exchanger need to be specified in a so-called **Borehole Heat Exchanger Data Set** that can be shared by a number of BHEs.

FEFLOW supports four different heat-exchanger geometries:



- Double U-shape
- Single U-shape
- Coaxial shape with annular inlet
- Coaxial shape with centered inlet

The processes within heat exchangers can be modeled via an analytical, quasi-stationary approach in which a local thermal equilibrium between all heat-exchanger components is assumed or via a numerical, fully-transient approach. The latter provides higher accuracy for short-term (time ranges of few hours) predictions especially for conditions with quickly changing inflow temperatures. For long-term predictions with less frequent and only moderately changing inflow temperatures, the quasi-stationary method provides reasonably accurate results at much lower computational cost.

Borehole heat exchangers are defined in the **Borehole Heat Exchanger Editor** which is invoked by activating **Borehole Heat Exchanger** in the **Data** panel and double-clicking on the entry in the input box of the **Editor** toolbar.

For the simulation of arrays of borehole heat exchangers, FEFLOW provides the possibility to connect BHEs serially or parallelly.

## Constraints

In FEFLOW, all boundary conditions can be physically constrained. For example, a fixed hydraulic-head condition at a certain node can be limited by a maximum or minimum flow. Technically, these constraints are realized by temporarily changing the respective boundary condition. If the flow at a fixed head boundary condition would exceed the maximum flow set as a constraint, the corresponding boundary condition is internally transformed into a well boundary condition with the value of the maximum flow, and the time step is repeated. Because this test is performed for all affected nodes at every time step, considerable computational effort can be associated with the use of constraints.

Constraints are typically complimentary to the respective boundary condition, i.e., head-type boundary conditions are constrained by minimum/maximum flow, while flux-type conditions are constrained by minimum/maximum head. Exceptions from this rule include additional constraints for transfer boundary conditions (to limit the infiltration at disconnected rivers) and for all transport boundary conditions (to limit their application to a certain range of hydraulic head).

Constraints are not shown in the **Data** panel by default. They can be added to the tree by using the context menu of the corresponding boundary condition.

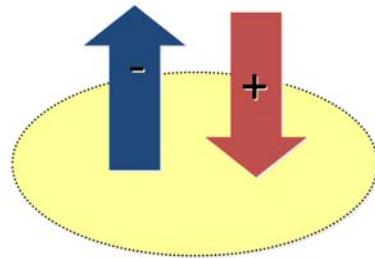


Figure 11.4 Algebraic signs for constraints and budget.

## Modulation Functions

In transient models, boundary conditions can be modified by applying a time series (see section for more information on time series) as a modulation function. The modification is realized by multiplying the time-series values of the modulation function with the boundary condition value at each time step during the simulation.

This functionality can be used, e.g., to modify boundary condition values for calibration purposes or to distribute a total value onto a number of nodes and also to turn boundary conditions off temporarily if modulation functions contain gaps: Multiplying a time-series gap with boundary condition values deactivates the boundary condition during the time period for which a gap has been defined. The same modulation function can be applied to an arbitrary number of boundary conditions with different values or time series at the same time. This reduces the effort for turning a large number of boundary conditions off at the same time significantly compared to defining gaps for each boundary condition separately.

As for boundary constraints, modulation functions are not displayed in the **Data** panel by default but have to be added to the list of available parameters using the context menu of the respective boundary condition(s).

### 11.2.4 Material Properties

The material properties describe the relevant characteristics of the porous medium for the considered flow and transport processes to be simulated. They are defined on an elemental basis.

For the flow simulation, material properties encompass quantities such as hydraulic conductivity in different directions according to the selected anisotropy model, drain-/fillable porosity (specific yield) and specific storage (compressibility), and transfer rates (e.g., to represent river-bed conductance). Parameters such as source/sink or in/outflow on top/bottom (often used for groundwater recharge) are mathematically close to boundary conditions;



however, their typically distributed spatial reference justifies their placement in material properties.

### 11.2.5 Auxiliary Data

In Auxiliary Data, nodal and elemental distributions are available that provide information on different properties of the finite-element mesh. For 2D models, the flow field can also be visualized using stream functions. Auxiliary data cannot be edited by the user, but the distributions are automatically updated according to changes in the model.

In 2D models, the two elemental distributions **Max. interior angle of triangles** and **Delaunay criterion violations** exist which can be used to check the mesh quality. In 3D models, the additional distributions **Slice distance** and **Layer thickness** are provided which can also be used as parameters in the **Expression Editor**. For 2D models, a nodal distribution for the **Vorticity stream function** is available. In transport models, the elemental distribution **Péclet Number** is available which can be used to check whether the spatial discretization is sufficiently fine for a specific transport problem. As the calculation of the **Péclet Number** is based on velocities, parameter values are only available during a simulation or when a full results file (\*.dac) is loaded.

### 11.2.6 User Data

Arbitrary nodal or elemental distributions of user data can be created via the **Data**-panel context menu. They can be assigned and visualized like the 'regular' FEFLOW parameters, but the distributions are currently limited to time constant data.

User data can provide input for the FEFLOW **Expression Editor** (e.g., for defining kinetic reactions in multispecies transport), they can be used for storing the data needed by FEFLOW plug-ins (see Section 16), or for comparing the model results to certain reference conditions.

User data can also be expression-based parameter distributions. In this case, the values of a distribution are calculated on the fly based on the user-defined expression. Expression-based distributions can be used, for example, to dynamically plot the water table drawdown at different simulation times.

FEFLOW offers a set of predefined distributions to display the error norms (normalized predictor-corrector errors) for flow, mass, groundwater age and heat simulations. In transport models with multiple chemical or age species, a separate distribution for each species is available.



### 11.2.7 Discrete Features

Discrete features can be added to models to represent highly conductive one- or two-dimensional features, such as tunnels, pipes, drains, faults or fractures.



*Discrete features always add permeability to the model. They cannot be used to simulate low-permeability structures.*

They are finite-elements of a lower dimension than the basic finite-element mesh, i.e., one-dimensional discrete features can be added in both 2D and 3D models, while two-dimensional discrete features are only available in 3D models.

Discrete features are assigned along element edges (1D) or faces (2D) or they can connect two arbitrary nodes within the mesh. In the latter case, interaction between the arbitrary discrete feature and the finite-element mesh is only possible at connected mesh nodes.

Discrete features can be added via the context menu in the **Data** panel at locations where an edge or face selection is active or where an arbitrary node path has been defined.

For each discrete feature, geometry and flow and possibly also mass-, age- and heat-transport properties need to be defined. For the simulation within discrete features, the user can choose between three different flow laws:

- Darcy
- Hagen-Poiseuille
- Manning-Strickler

For phreatic or unsaturated models using Richards' equation, the page **Other Settings** in the **Problem Settings** dialog provides four different options for the handling of unsaturated discrete features (including Multilayer Wells):

By default, the entire feature is turned off if the pressure of all nodes belonging to a discrete feature drops below zero. In case that at least one, but not all discrete-feature nodes are located in an area with negative pressure, the conductivity of the discrete feature is scaled by a smoothing function.

Alternatively, discrete features can always remain fully conductive or be turned off completely under both partially and fully dry conditions. As a last option, discrete features can remain fully conductive in case of partial saturation and be turned off under dry conditions.

## 11.3 Assignment of Constant Values

The most basic approach for data assignment is the input of time-constant parameter values at the currently selected mesh items (see chapter ). Pre-



condition for the assignment is that a single target parameter has been activated in the **Data** panel by a double-click or via the context menu. An active parameter is indicated by bold letters.

The assignment of a constant value for the parameter to the target selection is executed by typing the value into the input box in the **Editor** toolbar and clicking the green check mark button to the right of the input box (or hitting <Enter> on the keyboard). If the units of the current input value differ from the current parameter units, the unit can be typed in with the parameter. Units recognized by FEFLOW are accepted and the values are automatically converted.

## 11.4 Assignment of Time Series Data

### 11.4.1 Time Series

The input of time-varying boundary-condition data is based on time series. Once defined, a time series can be used for multiple boundary conditions. Time series can also be used for the assignment of time-varying material properties, constraints or modulation functions.

Time series can be imported from files (\*.pow format) or directly be defined and edited in the **Time-Series** dialog that is opened via the **Edit** menu. A third option is to paste tabular data copied from, e.g., Microsoft Excel/Access or a text editor, directly into a time series in the editor from the clipboard.

A time series consists of a unique ID and an arbitrary number of (time, value) data pairs. Optionally, a name can be specified for each time series. The time intervals do not have to be equal. When defining time series, it should be considered that automatic time-stepping procedures (see section ) make sure to meet each time value with a time step, so that short-interval time series may lead to higher numbers of calculation time steps. Interpolation between the data pairs can be defined as linear, constant (step function) or Akima/Akima2.

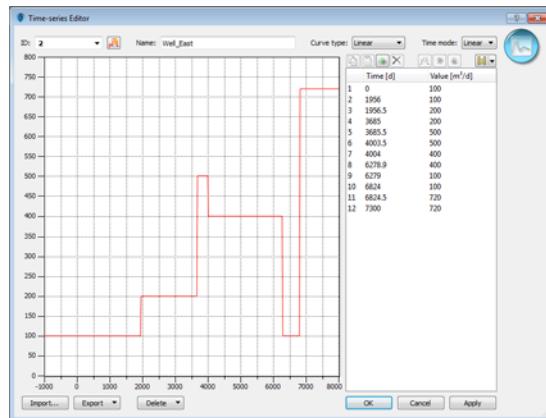


Figure 11.5 Time-series editor.

Time series can be applied to the model cyclically, i.e., a time series defined for part of the simulation time is applied repeatedly until the end of the simulation, or they can be used in linear mode. If the time series only covers a part of the simulation time in linear mode, the first value is used from the simulation start to the beginning of the time-series definition, and the last value in the series is applied until the end of the simulation.

Time series can also contain gaps to turn a boundary condition off during specific time intervals or to limit the application of a boundary condition to a certain time period. Gaps at the beginning and end of a time series can be used to make sure that the time-series values are not applied before the first and after the last defined time-series step as described above for the linear time mode.



As *time-varying material properties need to be defined for the entire simulation time*, FEFLOW does not allow to use time series which contain gaps for material-property assignment.

## Units for Time Series

By default, time series do not contain any information on the units of the time/value pairs and FEFLOW will use the internal unit of a specific parameter for the assignment of the time-series data, e.g., [ $\text{m}^3/\text{d}$ ] for well boundary conditions and [ $\text{m}$ ] for a hydraulic-head boundary condition. It is however possible to define the unit class and user unit for editing the data of a time series. While the unit class for the **Time** data is always time, a time series may contain values of pumping rates, hydraulic head or concentration. Selecting a user unit for **Value** therefore requires prior definition of the appropriate unit class (e.g., pumping rate, length, concentration).



### 11.4.2 Assignment

Time-series data are assigned similarly to time-constant values. Precondition for the time-series assignment is that the corresponding parameter has been activated in the **Data** panel, and that a selection of target nodes or elements has been made. The input box in the **Editor** toolbar has to be switched to time-series mode. The assignment is executed by choosing the ID of the time series to be applied at the selected nodes or elements in the input box in the **Editor** toolbar and clicking the green check mark button besides the input box (or hitting the <Enter> key on the keyboard). FEFLOW only makes the time series with the same unit class as the active parameter or with cardinals available for assignment.

## 11.5 Assignment of Map Data

Map information can be used in different ways for parameter assignment, ranging from using the map geometry to define targets for interactive data input to regionalization based on map-attribute data.



*Elevation in 3D models cannot be varied in time (except by using the programming interface).*

The input options for attribute data depend on the map and target (node/element/face/edge) geometries as shown in Figure 11.6 and Figure 11.7.

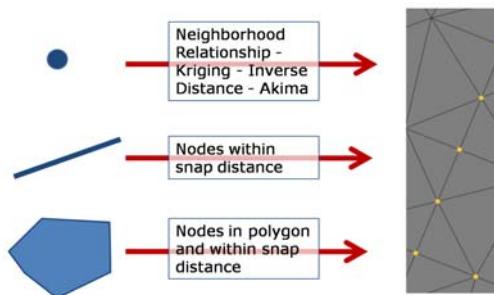


Figure 11.6 Map data input for nodal properties.

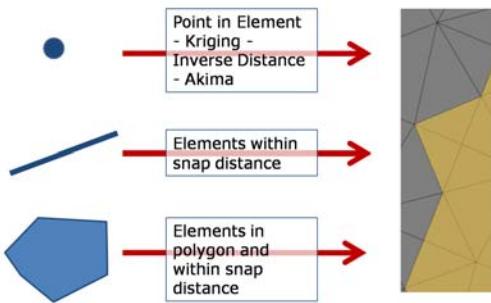


Figure 11.7 Map data input for elemental properties.

### 11.5.1 Interactive Data Input

Interactive data input is chosen in cases where the map geometry (points, lines, polygons) serves as target geometry for data input, but the actual values to be assigned are unrelated to the map.

This mode can be used in conjunction with the input modes for constant data, time series, or lookup tables in the **Editor** toolbar. After hitting the button for point, line, or polygon input the target geometry in the model is chosen by clicking the corresponding geometry. All mesh items within the distance defined in the **Snap-Distance** toolbar are assigned the value. Existing selections are hereby ignored.

### 11.5.2 Automatic Data Input

Many of the map formats compatible with FEFLOW can contain attribute values, e.g., simple triplet-text or ESRI-Shape files. These attributes can be used as the data source for parameter assignment.

Depending on the type of map data (point, line, polygon), the properties may be applied to the target geometries either directly, or via a regionalization algorithm.

#### Parameter Association

For all automatic data input, it is necessary to link the desired map attribute to a target FEFLOW parameter. As this link is stored permanently with the model, the import can be quickly repeated when the map data have changed. The link is defined in the **Parameter Association** dialog that can be accessed via the context menu of the respective map in the **Maps** panel.

Detailed settings for the data transfer can be defined in the properties of the link in the **Parameter Association** dialog. Here, it is defined whether the attribute data are interpreted as distribution of time-constant parameter val-



ues or as the ID of a time series for boundary conditions or constraints or whether time-varying material properties are assigned. For time-constant data and for time-varying material properties, the source data unit can be specified, and, for point data sources, the regionalization method and its properties are set. Furthermore, it is possible to relate data to a certain slice or layer if the slice or layer number is contained in the attribute data of the map. If another map is defined as the selection map, that map defines the target geometry. Thus for example an interpolation from point geometries can be limited to the nodes within a polygon or along a line obtained from another map. Either a selection map or a selection can be stored to easily repeat the data import.

## Regionalization

The regionalization settings are edited in the **Parameter Association** dialog as part of the link properties for point maps. For classic two-dimensional data interpolation, FEFLOW contains three interpolation methods with their respective property settings:

- ***Inverse Distance***
- ***Kriging***
- ***Akima***

The **Neighborhood Relationship** regionalization assigns a property to the nearest node, or to a number of nodes close to a map point. One typical example is the import of pumping wells in cases where the well locations have not been considered in the design of the **Supermesh**. The wells are then applied at the node closest to the original well location (**Single Target**), and if more than one well is mapped to the same node, the pumping rates will be summed up (**Aggregation: Sum**).

Corresponding to the **Neighborhood Relationship**, the **Point in Element** regionalization method for elemental properties assigns a value to those elements that contain a source data point. If more than one point is located within the same element, the point attribute values can be either summed or averaged.

A **1-dimensional linear interpolation** method along lines is available if a line map has been chosen as the link selection. Intended mainly for river water levels, this regionalization provides the means for interpolation between time-constant or time-varying water levels, e.g., for considering the translation of a flood wave along the river. Extrapolation from the last data point in direction or opposite to the line direction as well as in both directions is also supported.

## Assignment

After defining the parameter association link, the actual data assignment is invoked by clicking the green check mark in the **Editor** toolbar presuming the following conditions are met: The input field in the toolbar is in map input mode, the correct map has been selected, the corresponding parameter is



active, and if no selection map has been chosen as a link property, the target nodes or elements are selected. All requirements except the selection can be fulfilled at once by double-clicking on the entry for the link in the **Maps** panel, or, including the selection, by double-clicking on a previously stored node/element selection or selection map entry for the link in the **Maps** panel.

### 11.5.3 Assignment via Quick Import

The **Quick Import** option can be used to assign multiple attributes of a previously exported map to several parameters at once without the need for defining parameter links via the **Parameter Association** dialog first. This option is accessible in the context menu of a map in the **Maps** panel. Prerequisites for successful assignment via this option are:

- The map data has been exported from FEFLOW,
- the export and import model are equal or similar so that slice and layer numbers match for both models,
- the parameter units in both models match,
- additional required information, e.g., time series, Borehole Heat Exchanger datasets, are loaded before performing the quick import.

The import can be limited to specific model parameters and/or to the current slice/layer or the current selection. To match the input data to the mesh elements, one of the selection methods **Select by element number** or **Select by nearest element center(s)** needs to be applied in the **Quick Import** dialog.



*FEFLOW does not perform plausibility checks during Quick Import. The user has to ensure that the data match perfectly before using this option. It is not recommended to use it with maps not previously exported from FEFLOW*

## 11.6 Assignment via Expression

Values for the currently active parameter can be assigned based on user-defined expressions which are specified in FEFLOW's **Expression Editor**. In **Expression** mode, a double-click in the input field of the **Editor** toolbar opens the **Expression Editor** for editing the current expression or any of the stored expressions.



Figure 11.8 Expression Editor.

Arbitrary user-defined expressions are set up in the work area in the center of the dialog. The available model parameters that can be used within an expression are listed on the right-hand side of the dialog. Several toolbars provide access to loading/saving expressions and to mathematical operators and templates that can be used for composing equations.

All parameters and time series as well as the left-hand side of the equation carry units which can be inspected via the tooltip of each parameter in the work area or the list on the right-hand side of the dialog. By default, the editor uses the user-defined unit which has been set via the context menu of a parameter in the **Data** panel. Changing the unit is possible via the context menu of the respective parameter in the work area or in the parameter list in the dialog.

Expressions can be stored or renamed using the context menu of an entry in the list on the left-hand side of the editor.

### 11.6.1 Assignment

Values for the active parameter are assigned at nodes or elements by selecting the currently defined or one of the previously stored expressions from the drop-down list of the **Editor** toolbar and by clicking the green check mark symbol next to the input field.

## 11.7 Assignment of Lookup Table Values

### 11.7.1 Lookup Tables

Parameter lookup tables allow the definition of named properties, e.g., material types, each representing a set of parameters, such as conductivity, drainable porosity, etc. When assigning a parameter, the property name can then be used instead of assigning a value. The definition of the lookup table is located in **Global Settings**, accessed via the **Tools** menu.



The lookup table is independent of the current model, so that the properties can be used in different projects.

### 11.7.2 Assignment

The assignment of parameters from a lookup table is similar to the input of constant or time-series data. The input box in the **Editor** toolbar, however, has to be in lookup-table mode. The properties in the lookup table that contain a value for the current input parameter can be selected from a drop-down list.

## 11.8 Copying of Data Values

Parameter values in the current selection can be copied to the clipboard and pasted either to the same nodes/elements of another parameter of the same type (e.g., **Conductivity K\_xx** to **Conductivity K\_yy**), or to another slice or layer. For time-varying material properties, the entire time-dependent material data can be copied and pasted to another parameter of the same type. Pasting of parameter values to multiple material times of a time-varying parameter is also possible.

The Copy and Paste functions are available in the context menus of the parameters, in the **Edit** menu, and in the **Standard** toolbar.

## 11.9 Assignment of Multiple Parameters

Besides assigning values to different input parameters one after the other, it is also possible to edit multiple parameters simultaneously. This option can be accessed via **Assign Multiple...** in the context menu of parameters in the **Data** panel.

The dialog displays different parameters depending on the entry via which the dialog was opened. Homogeneous values and input units can be changed, the checkmark in front of a parameter determines whether a new value will be assigned or not.

The assignment of new values can be applied to the **entire model domain**, to the **current slice/layer** if the **Slice** view is the active view or to the **current selection** only.

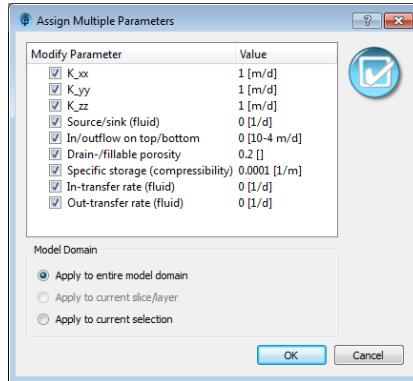


Figure 11.9 Assign Multiple Parameters dialog.

## 11.10 Time-Varying Material Properties

For the definition of time-varying material properties, FEFLOW provides a number of different settings and assignment options. As for time-series, **Linear** and **Akima interpolation** are available for interpolation between given material times. The defined time-varying data can be applied to the model cyclically, or it can be used in linear mode. If the defined time stages only cover part of the simulation time in linear mode, the first material-time value is used from the simulation start until the first defined time stage is reached, and the last material-time value is used until the end of the simulation. Time stages can be added and deleted and a parameter can also be made stationary again, selecting the material time for obtaining a stationary distribution for the parameter. All settings are located in **Edit Time Dependency** in the context menu of a time-varying material property in the **Data** panel. Time-varying material properties are indicated by a blue wave symbol in front of the parameter name in the **Data** panel.

Time-varying material data can be inspected using the **Inspection** tool. The **Inspection** panel displays the current material time and value and also the time-dependency settings (cyclic/linear, Akima/Linear interpolation). The **Transient-Data** chart in the **Inspection** panel shows a preview of the time-series curve for a time-varying parameter.

### 11.10.1 Manual Assignment

To assign time-varying values to a material property manually, the parameter has to be made transient via its context menu in the **Data** panel first. In the **Material Data Time Stages** dialog, a list with material times can be edited manually or imported from a time-series (\*.pow) file (Figure 11.10).

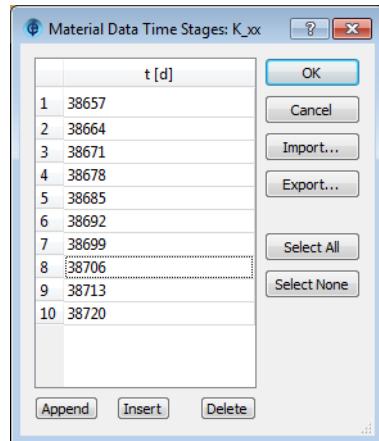


Figure 11.10 Material Data Time Stages dialog.

To assign values to a certain material time, the parameter needs to be active and the material time has to be selected via **Material Time** in the context menu of the parameter in the **View Components** panel. The currently selected material time is displayed in the parameter legend in the active view and shown in the parameter entry in the **View Components** panel. The assignment to the target element selection is executed by typing the value for the selected material time into the input field in the **Editor** toolbar and by clicking the green check mark to the right of the input field (or by hitting <Enter>).

### 11.10.2 Manual Assignment from Time Series

The assignment of time-varying material properties via time series is completely analogous to using time series for boundary-condition assignment (see chapter ). As time-varying material properties need to be defined for the entire simulation period, time series containing gaps are not supported for material properties.

It is only possible to use those time series for material-property assignment which carry the same unit class as the parameter or cardinal units. FEFLOW hides all time series carrying a different unit class and does not display them in the input field of the **Editor** toolbar.

### 11.10.3 Assignment of Map Data

For the automatic assignment of time-varying material properties via maps, it is necessary to create a link between the desired map attribute and the FEFLOW target parameter in the **Parameter Association** dialog that is invoked via the context menu of the input map in the **Maps** panel. To access the different options for time-varying material-data assignment in the **Param-**



**Parameter Association** dialog, the context menu of the material property to be edited has to be used. By default, material properties are assumed to be time-constant. The four different options for linking are described below.

The available settings for the link definition and for regionalization methods are introduced in chapter .

After defining the link in the **Parameter Association** dialog, the actual data assignment is invoked by clicking the green checkmark in the **Editor** toolbar presuming the following conditions are met: The input field in the toolbar is in map input mode, the correct map has been selected, the corresponding parameter is active, and if no selection map has been chosen as a link property, the target elements are selected. All requirements except the selection can be fulfilled at once by double-clicking on the entry for the link in the **Maps** panel, or, including the selection, by double-clicking on a previously stored element selection or selection map entry for the link in the **Maps** panel.

## Assign Material Data to Current Time

With this option, parameter values can be assigned to the currently selected time step via maps.

Prerequisite for this assignment is that the parameter to be edited has already been made transient before accessing the **Parameter Association** dialog. Additionally, the parameter needs to be set active with a double-click on the parameter name in the **Data** panel, and the time stage to be edited needs to be selected via **Material Time...** in the context menu of the parameter in the **View Components** panel.

In the **Parameter Association** dialog, the option **Assign material data to current time** needs to be selected from the context menu of the respective material property. The map attribute that contains the data for the current time stage can then be linked to the FEFLOW parameter. The **Source data unit** can be changed and depending on the format of the input map, further link properties can be defined. The remaining necessary steps to complete the data assignment based on maps are listed in chapter .

## Assign Material Data to Time Stages

Using this option, parameter values are assigned to each time stage of a material property. The input map needs to contain an attribute field for each material time. Selecting **Assign material data to time stages** in the context menu of a material property in the **Parameter Association** dialog invokes the **Material Data Time Stages** dialog in case that the parameter is still time-constant. Material times can be edited manually or imported from a \*.pow file. The time stages are displayed as a list underneath the now time-varying material property. Each attribute field that contains the values for a specific time stage needs to be linked to the corresponding material time. To create multiple links at the same time, all or several attribute fields that contain time-varying data can be selected using the <Ctrl> or <Shift> keys and dou-



ble-clicking on the first time stage. The selection order defines which data set will be assigned to which time stage. It is sufficient to define the link properties in one of the created links, the settings will automatically be transferred to all remaining links.

## Assign Time Series to Material Data

With this assignment method, two map attributes containing the time and corresponding parameter values are linked to a **Time** and **Value** entry of a material property. Instead of time-series files the input map contains one attribute field with all the time stages and a second attribute field with the parameter values for each time stage. Choosing **Assign time series to material data** from the context menu of a material property in the **Parameter Association** dialog adds the entries **Time** and **Value** underneath the material property. A link between the map attribute that contains the time stages and the field **Time** and a second link between the attribute with the parameter values and the field **Value** are required. For both links, the **Source data unit** can be changed.

## Time-Series Joining to Maps

If a map only contains an attribute field with time-series IDs to be used as input for time-varying material properties but not the data directly, the required time series (\*.pow file) can be joined to a map in the **Parameter Association** dialog. Time series can be joined via **Join>Select Map(s) Data** in the context menu in the input map: The field **TimeSeriesID** in the time-series file is linked to the attribute field with the time-series IDs in the input map. Time stages and corresponding values specified in the time-series file are now available as attribute fields that can be linked to a material property. The link is completed following the same steps as described in the previous section **Assign Time Series to Material Data**.

### 11.10.4 Assignment via Expression

Time-varying material properties can also be assigned via user-defined expressions set up in the **Expression Editor**. All time series loaded to a model can be used as parameters within an expression. The steps for setting up a user-defined expression are explained in section 11.6.

The actual assignment of transient material properties via the **Expression Editor** differs from the assignment of time-constant data: First, the material property needs to be made transient. Instead of completing the assignment with a click on the green check-mark symbol, then the option **Assign Expression to Multiple Times...** within the context menu of the input box in the **Editor** toolbar needs to be selected and the time stages to which data are to be assigned have to be chosen.



## 11.11 Use Parameter Expression

Instead of being defined by time-constant or time-varying values, the FEFLOW parameters **Source/sink** and **In/outflow on top/bottom** can also be linked to a user-defined expression to account for dependencies on other parameters. The defined expressions are evaluated iteratively during the simulation run and the parameter values are updated according to the specified dependencies.

**Source/sink** and **In/outflow on top/bottom** are linked to an expression by selecting **Use Parameter Expression** in the respective context menu in the **Data** panel. A link to an expression is indicated by a sigma symbol in front of the parameter in the **Data** panel.

## 11.12 Interactive 1D Linear Interpolation

An alternative option to using maps for the interpolation of values for boundary conditions or constraints along a line is the **Interactive Linear 1D Interpolation**. This assignment tool can be activated in the **Editor** toolbar if a boundary condition or constraint is the active parameter in a **Slice** view. The interpolation between time constant values, time series and also values calculated via the **Expression Editor** is supported.

The line path along which values are interpolated is drawn manually with the tool being active. By pressing **<Ctrl>** while placing a new vertex, a boundary-condition or constraint value can be set together with the vertex, applying the constant value, time series or user-defined expression currently entered in the input box of the **Editor** toolbar.

Additional settings for, e.g., the interpolation of time-varying values or the consideration of already existing data along the interpolation path are accessible via the main menu **Tools > Global Settings > Tool Properties**.

## 11.13 Units

FEFLOW uses a standard unit for each parameter that is listed in the parameter description in the help system. For parameter input and display, other units can be applied in one of the following ways:

- The standard unit for a physical unit (such as velocity, time, etc.) can be changed globally for the user interface via **Units** in the **Tools** menu. One of three predefined systems or a user-defined unit set can be applied.
- For each parameter, the unit for the active model can be chosen via the context menu of the parameter in the **Data** panel. This unit is then the output and default input unit. This setting is stored within the \*.fem file and applied whenever a specific model is loaded.



- The unit of the input values can be defined at the time of input: For manual input, the unit of the input value can be typed in with the value. When assigning the value, the unit is automatically converted. For map-based data input, a unit conversion can be specified as part of the **Parameter Association** link.
- For assignment via user-defined expressions, the user unit of each parameter within the expression and of the parameter to be assigned can be changed in the **Expression Editor**.



*The coordinate system unit in FEFLOW is always meters. Thus maps should be georeferenced in metric systems only.*

## 11.14 Tutorial

In the following exercises we use different methods to assign process variables, boundary conditions, constraints, material properties and discrete features.

Click **Open** and load the file **parameters.fem**. This model is a 3D transient flow and transient mass-transport model.

### 11.14.1 Tools



Figure 11.11 Selection toolbar.



Figure 11.12 Editor toolbar.

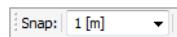


Figure 11.13 Snap-Distance toolbar.

### 11.14.2 Assignment of Constant Values

In the first exercise we assign constant values manually. We start with the assignment of boundary conditions and constraints for flow and mass transport.



Set the **3D** view active, go to the **Data** panel and activate **Boundary Conditions (BC)** > **Fluid flow** > **Hydraulic-head BC** with a double-click. Next, select the nodes to which the boundary condition is to be assigned. We use the node selections displayed in the **Selections** panel to set a selection. Double-click on **Southern Border** to set the selection active. The selected nodes of the southern border now appear as yellow spheres.

Now, enter a value of **32.1** m in the input box of the **Editor** toolbar and click **Assign**. The selected nodes are now surrounded by blue circles, the symbol for a 1<sup>st</sup> kind boundary condition (see figure 11.14). Clear the selection with a click on **X** and proceed with the northern border.

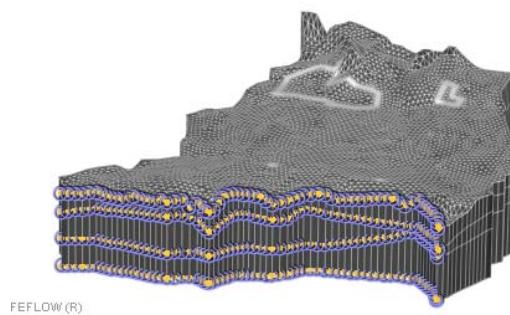


Figure 11.14 Boundary conditions at southern border.

Double-click on **Northern Border** in the **Selections** panel to set this node selection active. Enter **46** m in the **Editor** toolbar and click the green check mark to assign the value.

We proceed with the assignment of mass-transport boundary conditions for the two borders. Go to the **Data** panel and click on **Domain**. In the **Data** panel, activate the parameter **Mass transport** > **Mass-concentration BC**. In the **Selections** panel open the context menu of **Southern Border**. This time, select the option **Add to Current Selection** to select all nodes at both the southern and the northern borders. Type a value of **0 mg/l** in the **Editor** input box and click **Assign**.

To ensure the possibility of a free outflow of mass at these borders we add a constraint. First, click on **Domain** in the **Entities** panel again. In the **Data** panel, open the context menu of **Mass-concentration BC** and select **Add Parameter > Min. mass-flow constraint**. Double-click on **Min. mass-flow constraint**, enter **0 g/d** in the input box of the **Editor** toolbar and assign the value with a click on the green check mark. The constraint will limit the applicability of the 0 mg/l fixed-concentration BC to flow entering the model domain (here treated as positive). At BC nodes with outgoing (here negative) flow, the fixed-concentration BC will be replaced by the constraint. The minimum constraints are displayed as white



bars below the boundary-condition symbols in the model. Make sure to clear any selection before proceeding to the next assignment.

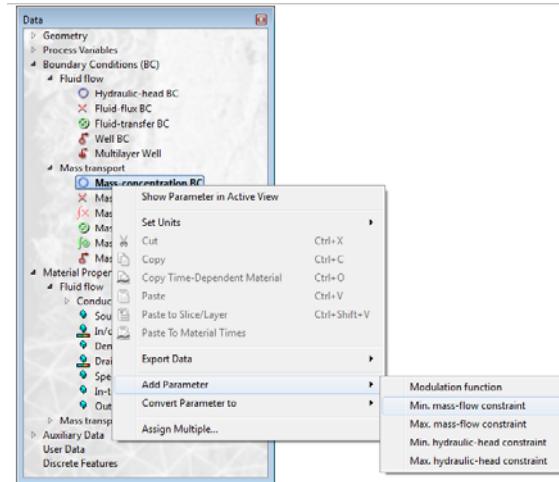


Figure 11.15 Adding constraints.

### 11.14.3 Assignment of Time-Series Data

We assume a mass source that is defined as time-varying mass-flux boundary condition in our model. We therefore use a time series to assign time-varying boundary condition values.

Open the **Time-Series Editor** via **Edit > Time Series**. The ID list on the top left of the dialog is still empty and shows that the model does not contain any time series yet. To load the time series click on **Import** and choose the file **massflux.pow**.

This file contains one times series with the ID 10 that is now added to the time-series list. Before the time series can be assigned, we need to include a gap in the series in order to turn the boundary condition off during a specific time period. Additionally, we need to define the unit class of the time-series values. The time-series graph currently shows a linear increase of the mass flux between days 60 and 90. In order to deactivate the boundary condition between these two time stages, we include a gap after day 60. To do so, click into the line for day 60 in the time/value list on the right-hand side of the editor and then click on **Insert Gap**.

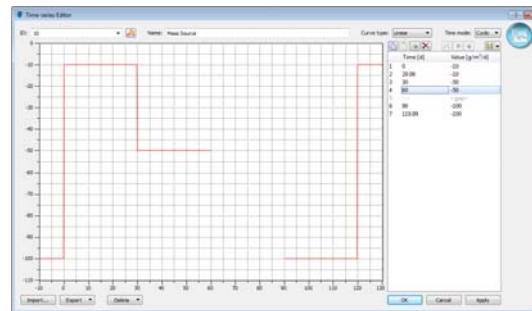


Figure 11.16 Time series including a gap.

Both the time/value list and the time-series graph now reflect that the boundary condition will be turned off between days 60 and 90 (see Figure 11.16).

To change the unit class of the time-series values, right-click on **Value**, select **Change Unit Class** and then click on **Integrated mass rate**. The displayed user unit now switches from cardinal to [g/m<sup>2</sup>/d].

Click on **Apply** and **OK** to confirm the changes and to leave the dialog.

Make sure that the **Slice** view is the active view and that **Slice 1** in the **Entities** panel is selected. Double-click on **Mass transport > Mass-flux BC** in the **Data** panel and also on the stored node selection **Mass Source** to set the selection active. To assign the time-varying mass flux, click on the symbol in the input box of the **Editor** toolbar until it switches to **Time Series** input mode and the time series for the mass-flux boundary condition is displayed. Complete the assignment with a click on **Assign**. The time-varying character of the boundary condition is indicated by a white wave symbol on top of the boundary condition in the active view. The time-series ID is also displayed next to the BC symbol (Figure 11.17).

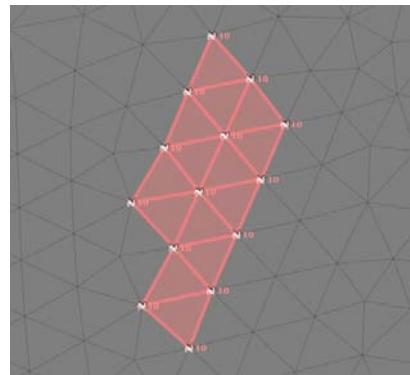


Figure 11.17 Time-varying mass-flux BCs.

#### 11.14.4 Assignment of Map Data

##### Interactive Data Input

To complete the boundary conditions we assign a fixed concentration (1<sup>st</sup> kind Mass BC) to the nodes of the two contamination sites.

Instead of creating a selection prior to the parameter assignment, we perform the assignment using the map geometry as target geometry.

First, click on **Slice 1** in the **Entities** panel when the **Slice** view is the active view. Activate the parameter **Mass-concentration BC** in the **Data** panel. Double-click **Supermesh > Polygons** in the **Maps** panel. Enter a concentration of **500 mg/l** and switch to the now active tool **Assign Data by Map Polygon** located in the map-assignment tools next to the input field of the **Editor** toolbar. Reduce the **Snap distance** to **1 m** in the **Snap-Distance** toolbar to limit the selection to the nodes within the contamination sites. Clear any active node selection with a click on **X**. Move the mouse cursor over the contamination sites until they are highlighted. Assign the concentration with a single click into each of these areas. The boundary conditions now appear as blue circles around the nodes.

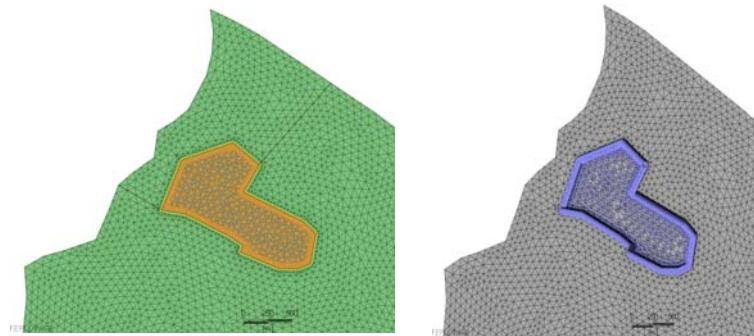


Figure 11.18 Highlighted contamination site (left) and assigned boundary conditions (right).

## Automatic Data Input

### Parameter Association, Regionalization and Assignment

Having the boundary-condition assignment completed we proceed with the assignment of material properties. To assign values for hydraulic conductivity and recharge we use some of the maps that are included with the model. These contain attribute values which have to be linked to the respective FEFLOW parameters.

Start with assigning hydraulic-conductivity values to the top layer of the model. Go to the **Maps** panel and open the context menu of the map **conduc2d** with a right-click. Select the entry **Link to Parameter** to open the **Parameter Association** dialog.

The list on the left hand side contains the attribute data which are stored in the map. Click on **CONDUCT**. To link these data to the FEFLOW parameter now double-click on **Material Properties > Fluid flow > Conductivity K\_xx** in the list on the right hand side. A black line shows the created link. The link properties are edited in the table below.

Leave the **Link Type** as **Time constant data** and set **[10<sup>-4</sup> m/s]** as source data unit for the conductivity values. Do not change the fields **Element/Layer Selection** and **Default Link Selection**.

As the map contains point data, we need to define a regionalization method for data assignment.

From the list of available methods select the **Akima** method. Set a **Linear Interpolation Type** and choose **3 Neighbors** and **0 Over-/Under Shooting**. Confirm the link settings and leave the dialog with **OK**.

To assign the linked conductivity data, make sure that the **Slice** view is the active view and double-click on the entry for the link in the **Maps** panel. **Conductivity K\_xx** now becomes the active parameter in the **Data** panel,



the assignment mode is changed to **Maps** and the correct map for assignment is automatically set in the input field of the **Editor** toolbar.

Browse to slice 1 in the **Entities** panel. Now, select all elements of the top layer with a click on **Select All**. To assign the data click the green check mark in the **Editor** toolbar.

## Assignment via Quick Import

To demonstrate how all or multiple exported model parameters can conveniently be reimported into the same model or an alternative scenario of the same model we export all boundary conditions and then reassign them using the **Quick Import** option.

To export all flow and mass-transport boundary conditions into one file, right-click on **Boundary Conditions (BC)** and select **Export Data > All Nodes** and save the file as **boundary\_conditions.shp**. Confirm the automatic adding of the map to the current model with **Yes**. The map **boundary\_conditions.shp** now appears under **Exported Maps** in the **Maps** panel.

To illustrate that all boundary conditions are assigned correctly via the **Quick Import**, we delete all boundary conditions first. The fastest option to do this is again to right-click on **Boundary Conditions (BC)** in the **Data** panel and to select **Assign Multiple**. Make sure that the setting **Apply to entire model domain** is active and confirm the deletion with **OK**.

Now, go to the **Maps** panel, right-click on the map **boundary\_conditions** and select **Quick Import...** from the context menu. Make sure that all the boundary-condition types that we want to reimport are checked. Choose the option **Select by node number** as **Node Selection Mode** and **Apply to entire model domain** as **Model Domain** (see Figure 11.19). After clicking on **OK** all fluid-flow and mass-transport boundary conditions are reimported.

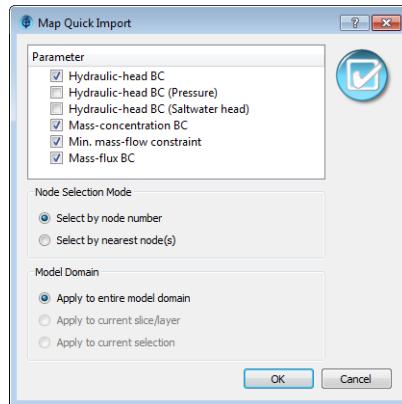


Figure 11.19 Assignment via Quick Import.

#### 11.14.5 Assignment via Expression

The hydraulic conductivity in z-direction is assumed to be smaller by a factor of ten than the conductivity in x-direction. To assign the conductivity values in z-direction, a user-defined expression is applied.

In the **Data** panel, double-click on **Conductivity K\_zz**. Right-click on the symbol in the input box of the **Editor** toolbar and switch to **Expression** assignment mode. Open the **Expression Editor** with a double-click on **Current Expression** in the input box of the **Editor** toolbar. Delete **Current** in the working window and open **Material Properties > Fluid flow > Conductivity** in the list of model parameters on the right-hand side of the dialog. Double-click on **CONDX** and then click on the **Insert a fraction template** symbol. To complete the expression, click into the blue input box and enter **10**. Hydraulic conductivity in z-direction is now linked to conductivity in x-direction.

Leave the dialog with **Close**. Select all elements of the top layer with a click on **Select All** in the **Slice** view and execute the assignment with a click on **Assign**. The completed expression is shown in Figure 11.20.

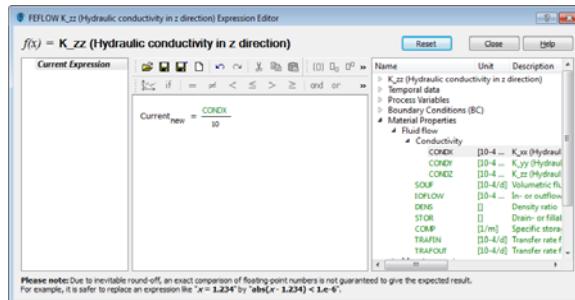


Figure 11.20 Expression-based assignment.

#### 11.14.6 Assignment via Copy and Paste

For the conductivity in y-direction the same values as for the x-direction shall be applied. To assign the values, copy the ***K\_xx*** values to the parameter ***K\_yy***.

First, a selection needs to be created from which parameter values will be copied. Again, select all elements of the top layer with **Select All**. In the **Data** panel, open the context menu of **Conductivity K\_xx** and click on **Copy**. Then, right-click on the parameter **Conductivity K\_yy** and select **Paste**.

Values cannot only be copied to other parameters but also to the same parameter on a different slice or layer. Make sure to click **Clear Selection** before proceeding. Browse to **Layer 3** in the **Entities** panel and select all elements of this layer with a click on **Select All**.

Now double-click on **Drain-fillable porosity** in the **Data** panel and select the option **Copy** from the context menu of the parameter. Open the context menu once more, click on **Paste to Slice/Layer** and select layer 1 as target layer. The values for **Drain-/fillable porosity** are now copied from layer 3 to layer 1.

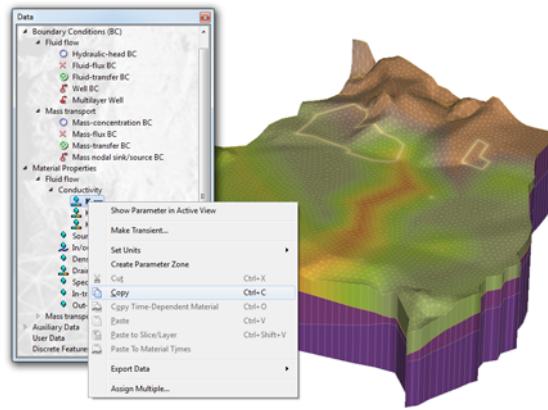


Figure 11.21 Parameter assignment using Copy/Paste.

### 11.14.7 Assign Multiple Parameters

The material properties for mass transport are assumed to be homogeneous throughout the entire model domain. Except for the parameters **Longitudinal** and **Transverse dispersivity** the already assigned values are accepted.

To edit these two parameters simultaneously, open the context menu of **Material Properties > Mass transport** in the **Data** panel, with a right-click and select **Assign Multiple...**. Uncheck all parameters except for the two dispersivities. Double-click into the **Value** field for **Longitudinal dispersivity** and enter a value of **70 m**. Press the **<Tab>** key on the keyboard and enter **7 m** as **Transverse dispersivity** value. Make sure that the option **Apply to entire model domain** is selected and confirm the assignment with **OK**.

### 11.14.8 Time-Varying Material Property Assignment

Continue with assigning recharge which in FEFLOW is treated as a material property. Go to the **Maps** panel and select **Link to Parameter** in the context menu of the map **year\_rec** to open the **Parameter Association** dialog.

The map contains five different data sets which we will assign as cyclic transient recharge data. To define time-varying recharge values, open the context menu of **In/outflow on top/bottom** in the list on the right hand side and select the option **Assign material data to time stages**. In the **Material Data Time Stages** dialog, click on **Import...** and select the file **material-times.pow** to load the time stages for which the input map contains recharge data. Confirm the import with **OK**. The loaded time stages are now listed below **In/outflow on top/bottom**. Click on the entry **ULTRAWET\_Y**, hold the **<Shift>** key and click on **ULTRADRY\_Y**. To set all links between the map



attributes and the corresponding material times at once, double-click on **0 [d]** on the right-hand side. Black lines indicate that five links have been defined. Click on one of the links to display the link settings.

As source data unit for the recharge values keep the default unit of **[10<sup>-4</sup> m/d]**. Do not modify the fields **Element/Layer Selection** and **Default Link Selection**. As this map is a polygon shape file no regionalization is necessary to import the data. All necessary settings are shown in figure 11.22.

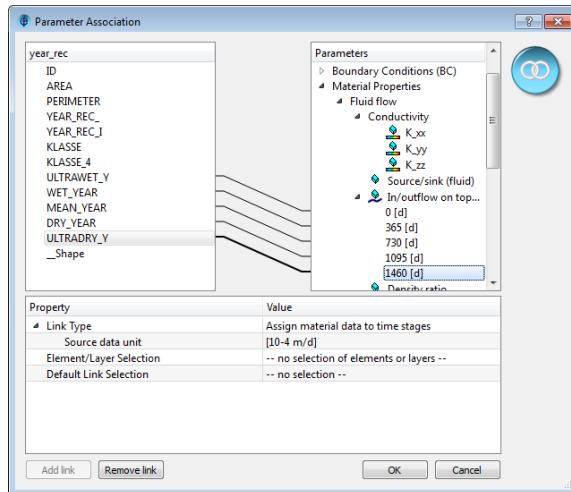


Figure 11.22 Parameter Association dialog for time-varying recharge data.

Click on **OK** to confirm the settings and to leave the dialog. The link between the map data and the FEFLOW parameter is now shown in the **Maps** panel in the tree **year\_rec > Linked attributes**.

To assign the linked recharge data, reduce the snap distance to **0 m** and follow the same steps as for the assignment of hydraulic conductivity in section

The legend of **In/outflow on top/bottom** in the **Slice** view displays the time stage that the plotted parameter distribution belongs to.

To use the defined recharge values in cyclic mode, open the context menu of **In/outflow on top/bottom** in the **Data** panel and select **Edit Time Dependency > Cyclic**.

#### 11.14.9 Assignment via Interactive 1D Interpolation

In the next step a time constant boundary condition is assigned via **Interactive 1D Linear Interpolation**. The boundary condition is assigned along the



outer model boundary but the interpolation at locations within the model domain works completely analogously.

Make sure that the **Slice** view is the active view and navigate to the top slice via the **Entities** panel. Activate the boundary condition **Hydraulic-head BC** with a double click on the parameter in the **Data** panel. Once the boundary condition is activated for assignment, the tool **Linear 1D Interpolation** becomes available in the drop-down list of tools located at the far right in the **Editor** toolbar. To use the tool, simply click on it.

We will define both the geometry and the values to be used for the interpolation interactively. In the input field of the **Editor** toolbar, enter a value of **40 m** and click on a node along the model boundary in the East while you keep the **<Ctrl>** key pressed. Pressing **<Ctrl>** assigns the value, time series or expression currently entered in the **Editor** toolbar at the selected node location while a simple click only adds a vertex along the interpolation path without a value. The assigned head value of 40 m is automatically displayed next to the first vertex.

Place additional vertices along the outer model boundary by clicking the left mouse button: A green line indicates the element edges along which the line geometry will be placed. As the snap distance is used for the selection of vertices, make sure that it is large enough for a convenient use of the interpolation tool.

In case that you would like to remove the last vertex of a line, place the mouse cursor on top of it and then hit **<Del>**.

To finish the line and to start the interpolation between the defined values and locations, enter **38 m** in the **Editor** toolbar, hold **<Ctrl>** and double-click on a node at the outer model boundary which represents the end of the line. To display the interpolated boundary-conditions values, check the entry **Value Label** in **Hydraulic-head BC** in the **View Components** panel. An example for interpolated boundary conditions is shown in Figure 11.23.

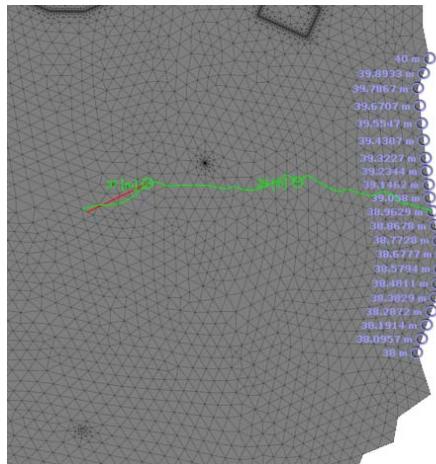


Figure 11.23 Interpolated boundary conditions.

Additional options to ignore boundary-condition values already assigned along the line geometry during the interpolation process, options for the interpolation of time-varying data or extrapolation are accessible via **Tools > Global Settings > Tool Properties > Linear 1D Interpolation**.

#### 11.14.10 Multilayer-Well Assignment

For this example, it is assumed that the pumping rate is time-constant and that the wells are screened over the entire model depth.

After switching to the **Slice** view activate **Boundary Conditions (BC) > Fluid flow > Multilayer Well** in the **Data** panel. In the **Selections** panel, set the join-edge selection **Well West** active with a double-click.

The currently active assignment mode is **Multilayer Well** which is shown by the **Multilayer Well** symbol in the input box of the **Editor** toolbar. Double-click into the input box to open the **Multilayer Well Editor** dialog (see figure 11.24).

The preview on the right-hand side of the dialog shows the well coordinates and the currently selected join-edges along which the well screen will be placed.

As **Capacity**, enter a value of **1000 m<sup>3</sup>/d**. For all other parameters we accept the default values. Set the **Multilayer Well** with a click on **Assign** and leave the dialog with **Close**. A red **Multilayer Well** symbol now indicates the position of the specified boundary condition. Use the **Attributes** tree in the **View Components** panel to display the different attribute values of the **Multilayer Well**.



Proceed with the remaining well. In the **Selections** panel, double-click on the stored join-edge selection **Well East** and open the **Multilayer Well Editor** dialog again with a click into the input box of the **Editor** toolbar.

Change the capacity of the well to **500 m<sup>3/d</sup>** and follow the same assignment steps as described for the western well.

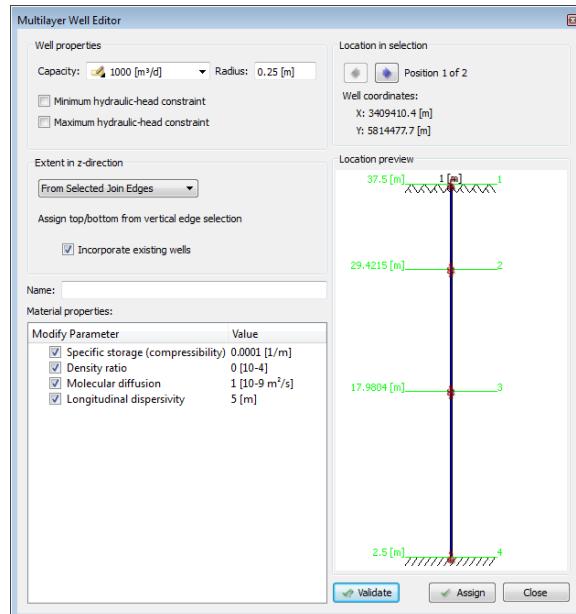


Figure 11.24 Multilayer Well Editor.

#### 11.14.11 Assignment of Borehole Heat Exchangers

Reload the model **parameters.fem** to add a **Borehole Heat Exchanger** (BHE) to the model.

First, open the **Problem Settings** dialog via **Edit > Problem Settings** and switch to **Include transport of... Heat** on the **Problem Class** page. Confirm the changes with **Apply** and leave the dialog with **OK**.

In the **Data** panel, activate **Boundary Conditions (BC) > Heat transport > Borehole Heat Exchanger** with a double-click. In the **Selections** panel, double-click on the stored join-edge selection **BHE** to set the edges active along which the BHE will be placed.

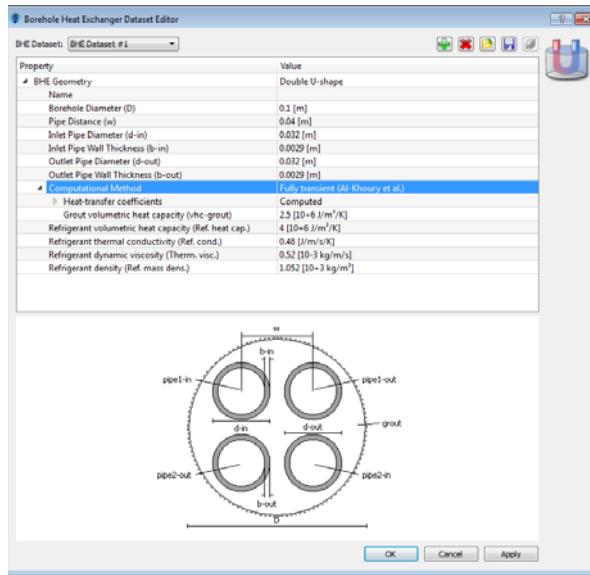


Figure 11.25 Borehole Heat Exchanger Dataset Editor.

As a first step, a BHE dataset has to be created that defines the geometric characteristics of the heat exchanger and also contains the information on the properties of the refrigerant and exchanger components. Double-click on **Borehole Heat Exchanger** in the input box of the **Editor** toolbar to enter the **Borehole Heat Exchanger Editor**. To add a new dataset, click on **Edit borehole heat exchanger datasets** and then on **Add new borehole heat exchanger dataset**. This creates the new **BHE Dataset #1**. Alternatively, the **BHE Dataset Editor** can be accessed via **Edit BHE Datasets** in the **Borehole Heat Exchanger** toolbar if the **Slice** view is the active view.

Leave the **Double U-shape** geometry and only reduce the **Borehole Diameter** to **0.1 m**. The geometry change is automatically reflected in the borehole sketch at the bottom of the dialog window. For the **Heat-transfer coefficients** switch from **User-defined** to **Computed** mode. Confirm the settings and return to the **Borehole Heat Exchanger Editor** with a click on **OK**.

The BHE data set is now available for assignment and the geometric properties are displayed in the dialog. Two remaining properties need to be defined, the flow rate of the refrigerant within the borehole pipes and also the inlet temperature. Instead of applying a constant inlet temperature, we assign a temperature difference between the inlet and the outlet of the BHE. Switch from **Inlet Temperature** to **Temperature Difference** and enter a value of **-3° C**. Next, enter a **Flow Rate** of **30 m³/d**. For the extent in z-direction leave the setting **From Selected Join Edges**. The dialog with all set-



tings is shown in figure 11.26. Assign the **Borehole Heat Exchanger** with a click on **Assign** and leave the Editor with **OK**.

Use the **Attributes** tree in the **View Components** panel to display the different attribute values of the heat exchanger.

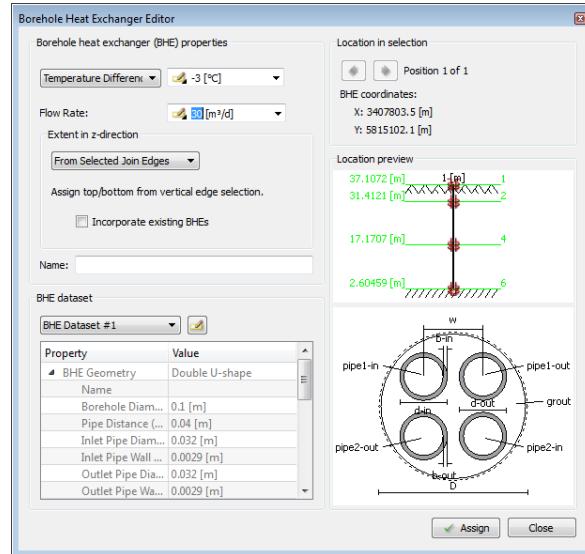


Figure 11.26 Borehole Heat Exchanger Editor.

### 11.14.12 Discrete-Feature Assignment

Reload the model **parameters.fem**.

On slice 3 we assign some one-dimensional discrete features that represent highly conductive fractures. In the **Selections** panel, double-click on the stored slice-edge selection **Fractures**. Use the slice list in the **Entities** panel to switch to slice 3.

To add discrete features at the selected locations, go to the **Data** panel and open the context menu of **Discrete Features**. Select **Add Slice-Edge Feature Element > Hagen-Poiseuille** (see Figure 11.27).

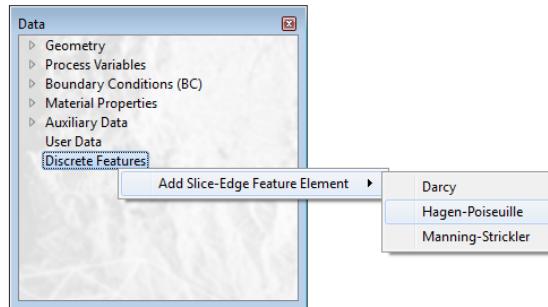


Figure 11.27 Adding Discrete Features.

The discrete feature is now added to the **Data** panel as **Feature 1**. The chosen flow law and geometry type are displayed behind the name of the feature. Change the name of the feature to **Fractures** by selecting **Rename...** in the context menu of **Feature 1**.

To show the discrete feature in the active view, click on **Slice 3** in the **Entities** panel and then double-click on one of the discrete-feature parameters in the **Data** panel.

The values for **Hydraulic aperture** will be edited at all slice-edges belonging to the discrete feature. Double-click on **Hydraulic aperture** and also on **Fractures** in the **Data** panel to set the edge selection active again in case that you have already cleared the selection. Enter a value of **1 m** in the input box of the **Editor** toolbar and click on the green check mark button to assign the value.

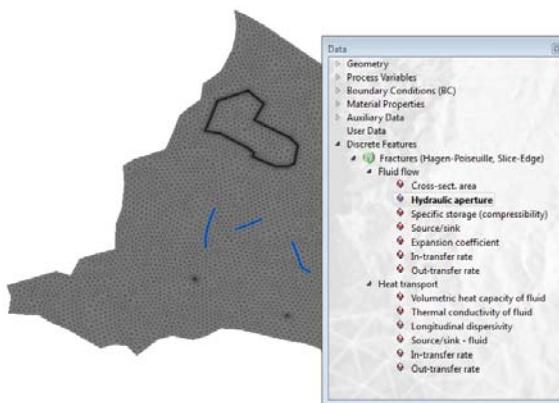


Figure 11.28 Discrete Features in the view and Data panel.



# 12 Simulation

## Running a FEFLOW model

### 12.1 Introduction

The FEFLOW user interface keeps all visualization options available during the simulation. The simulation progress can thus be conveniently monitored and problems can be detected early.

FEFLOW does not separately store initial conditions of hydraulic head or concentration/age/temperature during the simulation. So the hydraulic-head process variable contains the initial head values before the simulation, the current hydraulic head results at each time step of a transient simulation, and the final hydraulic head after the simulation. These final results are also retained when leaving the simulation mode by clicking **Stop** in the **Simulation** toolbar.



*If a given simulation record does not contain all the time steps, postprocessing methods that cover multiple time steps, such as transient pathline analysis or time-integrated budgets, can only give approximate answers.*

### 12.2 Model Check

Before running the model, all input parameters and other model properties should be thoroughly checked. A basic overview of the model characteristics is provided in the **Problem Summary** page in the **Problem Settings** dialog. The icon preceding each material property in the **Data** panel indicates whether the respective parameter is homogeneous or heterogeneous and time constant or time dependent (see ).

### 12.3 Results Output

Simulation results can be stored in two different formats: a reduced results file (\*.dar) and a full simulation record (\*.dac). Both results files can be recorded at the same time.

The reduced format contains text output for the observation points and well locations only. It is especially useful for automatic output processing with user-programmed scripts or optimization software such as PEST. Reduced results files always contain values for all the time steps.

In contrast, the full simulation record contains all nodal values of the primary variables (hydraulic head, concentration, age, temperature) and of the flow velocities. By default, results at all time steps are stored. It is possible, however, to skip a number of time steps between each output, or to specify the simulation moments for which output is desired. In the latter case, the auto-



matic time-stepping scheme ensures that results are calculated exactly at the prescribed output moments.

It is also possible to record a results file that only contains the final time step.

## 12.4 Running the Simulation

The handling of the interface is as easy as starting a tape or video recorder: The **Start** button in the **Simulation** toolbar starts the simulation, the **Pause** button pauses it, the **Stop** button exits the simulation mode, and if the **Record** button has been pressed before starting the simulation, the results are recorded in a file while the simulation runs.

All visualization and also some results-evaluation features (**Rate-** and **Period-Budget** panels, **Subdomain Boundary Rate** and **Subdomain Boundary Period** panels, **Content** panel) can be used during the simulation. It should be considered, however, that some of these features might slow down the simulation. Especially more elaborate visualization options such as isosurfaces or budgeting may require significant calculation effort.

After the simulation finishes, FEFLOW remains in paused simulator mode. Thus simulation results such as flow velocities and the solution of the last time step remain available for postprocessing. Only when exiting the simulation mode via **Stop** these data are removed from memory, and stream-line/pathline calculation or budget analysis are no longer possible. Any postprocessing then requires loading of the full simulation-record file (\*.dac).

## 12.5 Convergence

When solving non-linear problems for steady state, or for transient conditions with constant or varying time steps, an **Error-Norm** chart is shown. Once the change in results between two subsequent iterations is below a certain error tolerance (see ), the simulation is terminated. FEFLOW shows a message in the **Log** panel if no convergence was achieved within the defined maximum number of iterations.

In transient simulations with an automatic time-stepping procedure, the time step is shortened if necessary to achieve convergence. Only if repeated time-step reductions do not improve the situation, a log message is shown indicating that time-step reduction failed. The simulation is continued with the results of the last calculation for the time step. In general, models that show convergence problems even after greatly reducing the time-step length should be checked for errors in the input data or for insufficient mesh resolution.



## 12.6 Tutorial

### 12.6.1 Tools



Figure 12.1 Simulation toolbar.

### 12.6.2 Model Check

Load the file **simulation.fem** via **Open**. Before we start the simulation we want to make sure that all necessary parameters have been properly assigned, that the problem settings have been set correctly and that all necessary settings for later postprocessing have been made.

Start with the problem settings. Go to **Edit > Problem Settings** and open the **Problem Summary** page in the dialog. Here, an overview of key simulation control and model parameters is presented. The problem class settings, information on the mesh quality and boundary condition types and numbers are displayed (see Figure 12.2).

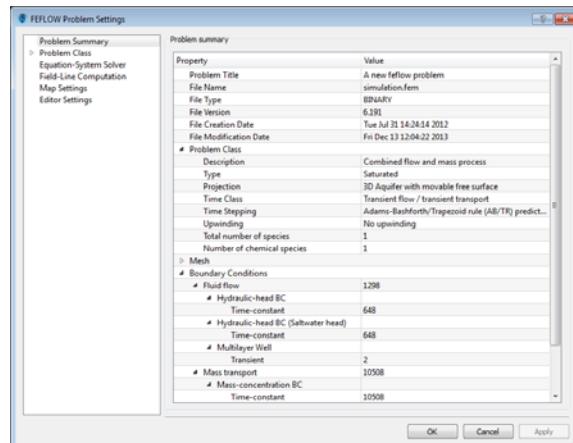


Figure 12.2 Problem Summary.

Leave the **Problem Settings** dialog with **Cancel**.

To check values at specific nodes or elements we can use the **Inspection** tool. Double-click on **Boundary Conditions (BC) > Fluid flow** in the **Data** panel. All fluid-flow BCs are now displayed in the **Inspection** panel. After clicking on **Inspection** move the mouse cur-



sor over the nodes where boundary conditions are set to check the values that have been assigned there. The value for the respective boundary condition, the node and the slice number are shown in the **Inspection** panel.

Both constant values and time series can be checked with the **Inspection** tool. Set the **Slice** view as active view and move the mouse cursor over the nodes where a multilayer well is set. The assigned time series for the capacity is then shown in the **Transient-Data** chart in the **Inspection** panel. The well geometry and further well properties are also displayed in the **Inspection** panel (see Figure 12.3).

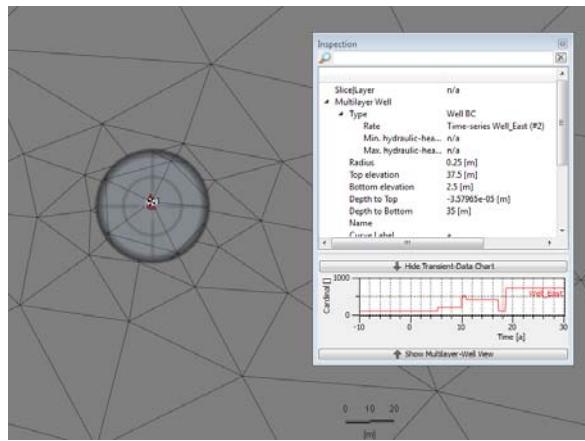


Figure 12.3 Checking boundary conditions with the **Inspection** tool.

The distribution type of a material property can be quickly checked in the **Data** panel. Homogeneous values are indicated by a symbol, heterogeneous values by a symbol in front of the parameter.

### 12.6.3 Results Output

Before starting the simulation we define results saving. Click on **Record** in the **Simulation** toolbar to open the **Record Properties** dialog. Check  **Save complete results (DAC file)** to create an output file. Click on **Browse** to choose the location for the results file and leave the dialog with **OK**.

### Budget Groups

If time series for the fluid- and mass-rate and period budget at selected node locations are to be calculated and recorded, these locations need to be selected and activated for this kind of budgeting before a simulation is started.



In the **Selections** panel, open the context menu of the stored node selection **Wells** and select **Budget-History Charting > BCs** to calculate fluid and mass quantities related to the specified multilayer well BC that leave the model at these locations. Repeat the same steps for the node selection **Southern Border** to calculate the accumulated mass that leaves the model via the southern border. For both selections, a time series will be shown in the **Fluid-** and **Mass-Rate Budget** and **Fluid-** and **Mass-Period Budget** charts.

## Observation Points

We also import observation points to monitor and compare the obtained hydraulic-head results at certain locations against measured water levels.

Set the **Slice** view active. In the **Maps** panel, open the context menu of the shape file **obs\_points** and select **Convert to > Observation Points**.

In the **Observation Points - Column Bindings** dialog, select the field

**SLICE** as **Slice number** and the field **TITLE** as **Label**. For the reference data, set **Hydraulic head** as **Parameter ID** (see Figure 12.4). As the flow equations are solved for transient conditions, the reference data that the results will be compared to are loaded from time series and not as time-constant values. Choose the field **TS\_ID** as **Time series ID**. The time series that contain the reference data have already been loaded to the model. As **Confidence interval**, choose **CONF\_INTER** and confirm the observation-point import with **OK**.

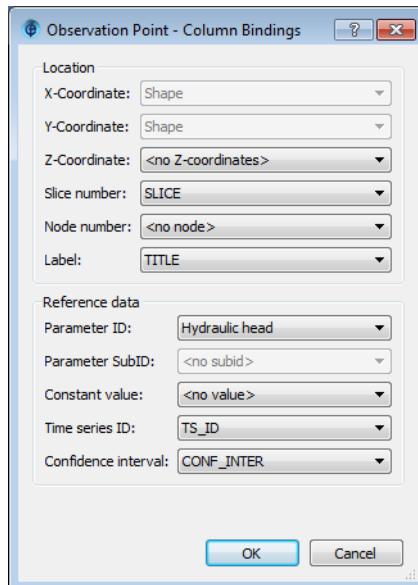


Figure 12.4 Observation Points import.



To check the properties of the observation points, click on **Edit Observation Point Properties** in the **Observation-Point** toolbar. The reference values and confidence interval for each observation point are then displayed in the dialog (see Figure 12.5).

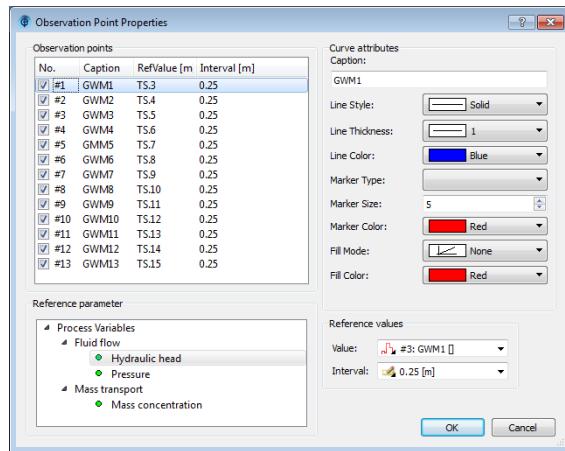


Figure 12.5 Observation Point Properties.

#### 12.6.4 Running the Simulation

After checking the model properties and defining the record location we start the simulation by clicking **Start** in the **Simulation** toolbar.

Five different charts appear: **Hydraulic Head**, **Pressure**, and since we are simulating transient flow and transport, also the **Time-Steps** and **Local-** and **Average-Concentration** charts. Graphs for the results at the well locations and also for the observation points are displayed.

The elapsed simulation time is shown at the bottom of every view window.

During the simulation we can display different parameters in the active view. Go to the **Data** panel and double-click on **Process Variables > Mass transport > Mass concentration** to see how the mass distribution changes with time.

Open the **Rate-Budget** panel via **View > Panels** and set the check mark in front of  **Active** to monitor the fluid-rate budget while the simulation is running. After a short computation time the rate budget for each boundary condition type, for source/sink terms, storage capture and release and the imbalance are displayed.

Now click **Stop** to terminate the simulation. Once the simulation has stopped, notice that the rate budget is no longer displayed in the **Rate-**



**Budget** panel due to the now lacking fluid-flux vector field. Evaluation of flux-based results is still possible by loading the simulation record.





# 13 Results Evaluation

Analyzing the simulation results

## 13.1 Introduction

As the evaluation of modeling results takes place both during and after the simulation, the content of this chapter applies to monitoring the simulation results at runtime, after finishing the simulation, and after loading a simulation record (postprocessing).

Results evaluation also involves all visualization and animation options described in Section 10 and Section 14. This section focuses on the tools specifically designed for output interpretation.

## 13.2 Observation Points

Observation points are used to monitor simulation results at specific locations in the model domain via chart windows. Using the tools in the **Observation Point** toolbar, observation points can be defined either manually in a **Slice** view, or by converting the current node selection into observation points. Features of point maps can be directly converted into observation points using the context menu of the map in the **Maps** panel. The import also allows to obtain a label and slice or node number for the point from the attribute data of the map, as well as reference information, typically from measured field data. Observation points can be placed on a slice or at an arbitrary xyz position.

During the simulation, charts are automatically shown for the results of the primary variables (hydraulic head, concentration, age species and temperature) at the observation points. Additional charts, e.g., for pressure, can be added via the **View** menu.

By selecting **Observation Points** as plotting target (in the **Entities** panel) for a process variable, the deviation between computed and measured values can be displayed in the current view. This includes graphical comparison of the deviation using error bars and the confidence intervals, as well as labels showing the absolute deviation.

Observation points can be added before the start of a simulation or when a results file is loaded.

### 13.2.1 Scatter Plot

The deviation between observed data and computed results can also be shown in **Scatter plots**. **Scatter plots** are invoked via the context menu of the charts for the primary and process variables. FEFLOW automatically uses the loaded observation data to calculate the deviation. Additionally, the



absolute error, the root mean square (RMS) and the standard deviation are displayed.

In a results file, the plot always reflects the currently displayed simulation time.

## 13.3 Budget Analysis

### 13.3.1 General Budgeting

The **Rate-Budget** panel can show the current fluid-, heat-, or mass rate balance for a selected domain of interest (DOI) at the current simulation time. A budget can be calculated for the entire model domain, a selection of nodes or an arbitrary subdomain consisting of elements. Any stored nodal or elemental selection can be used as a DOI either via context menu in the **Selections** panel, or by pressing the green ‘plus-sign’ button in the upper left corner of the budget panel after selecting one or more stored selections in the **Selections** panel. All budgets set active will be computed and dynamically updated whenever FEFLOW is in simulator mode, i.e., when the simulation is running, is paused, or has completed.

The **Period-Budget** panel shows the accumulated, i.e., time-integrated budget for the simulation period specified in the panel. The period-budget calculation can only be activated before a simulation is started, or when post-processing a simulation-record (\*.dac) file. Activating a period budget will implicitly activate the corresponding rate budget as well.

The respective fluid, heat, or mass balance is subdivided into flows related to the various boundary-condition types, contributions from distributed sources/sinks (e.g., groundwater recharge), storage capture (negative) and release (positive), and a remaining imbalance term arising from solver residual error and errors due to the spatial and temporal discretization. When using an element selection (i.e., a subdomain) as DOI, the additional - **Internal Transfer** - category appears in the **Rate-** and **Period-Budget** panels. This category represents the transfer across those parts of the DOI boundary that are inside the model domain. Model-internal flows crossing the subdomain-boundary into and out of the DOI are considered as positive and negative, respectively.

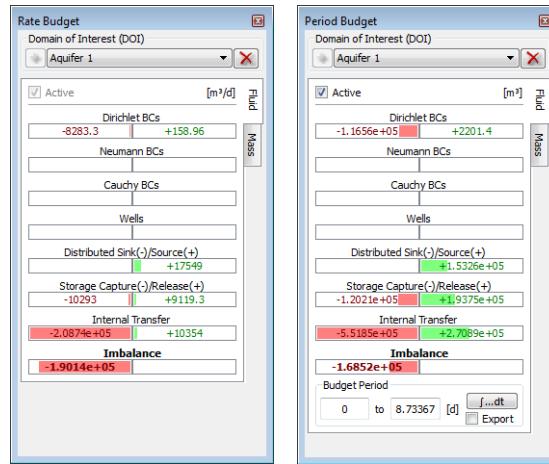


Figure 13.1 Rate-Budget and Period-Budget panels.

By activating the export option in the **Period-Budget** panel before starting the budget calculation, the associated rate- and period-budget values for the specified simulation period can be exported to a (\*.txt) file.

The budget results of the currently selected time step can also be exported via **Export Current Values...** in the context menu of the **Rate-** and **Period-Budget** panels after the budget calculation.



*Calculated storage values represent a lumped result of both confined storage (related to specific storage) and unconfined storage (related to drain-/fillable porosity or unsaturated-flow porosity).*

### 13.3.2 Subdomain-Boundary Budget

To calculate the fluid, heat, or mass flow across the boundary of a specific subdomain within a model, the **Subdomain Boundary Rate** and **Subdomain Boundary Period Budget** panels can be used. While the concept and sign convention of these two panels correspond to those of the general **Rate-Budget** and **Period-Budget** panels, there are some key distinctions:

- Subdomains are always defined as an element selection, so only element selections can be used as DOI for subdomain-boundary budgeting.
- Only the boundary of the subdomain will be considered in the budgeting.
- Values shown in the **BCs** category will only include budget values for those BC-assigned nodes that are located (1) on the subdomain boundary and (2) not shared with any part of the model domain adjacent to the DOI.
- A masking domain can be defined to limit the extent of the subdomain boundary that is considered in the budgeting.



If no masking domain is applied, the values shown in the **Internal Transfer** category will correspond to those in the same category of the general budget panel for a given subdomain, i.e., element selection.

In many cases though, it may be of particular interest to compute the flow through only a part of the boundary of a subdomain. This is accomplished by using a second elemental selection as a **Masking Domain (MD)**. Any stored or the current elemental selection can be applied as a MD by selecting it in the **Selections** panel and pressing the green check mark button in the **Masking Domain (MD)** section of the **Subdomain Boundary Rate** panel.

By default, the masking will constrain the budget evaluation to those parts of the subdomain (DOI) boundary that also belong to the MD. The **MD** panel section also provides an option to invert the masking so that only those parts of the DOI boundary are considered that do *not* belong to the MD.

Subdomain-boundary budgets supersede the now-deprecated **Flow-Rate** and **Flow-Volume** panels whose results are based on approximated Darcy-flux values and which are kept for compatibility and comparison purposes only. As subdomain-boundary budgets are obtained by using the primary solution together with the respective terms of the Continuity Equation, they can be generally expected to be more accurate.

### 13.3.3 Budget Groups

Any stored node selection in the **Selections** panel can be used as a separate *budget group* for which the summed calculated rate and period budget can be displayed as time series in the **Rate-Budget** and **Period-Budget** charts, e.g., to calculate the flow balance along a river or to determine drained water volumes in dewatering scenarios, etc.

To invoke this budgeting option and to display time series for a node selection in the budget charts, **Budget-History Charting** and one of the following four budget-evaluation options need to be selected in the context menu of a stored node selection in the **Selections** panel:

- **BCs**: Flows related to boundary conditions,
- **Distributed Sink(-)/Source(+)**,
- **Storage Capture(-)/Release(+)**,
- **Imbalance**: Sum of flows related to boundary conditions, distributed sinks/sources and storage capture/release.

As an additional fifth option, '**Total Balance**' is available as a legacy mode which includes flows related to distributed sources/sinks + boundary conditions, but no budgeting of storage changes.

The budget time series for a stored node selection can only be displayed in the budget charts if the budgeting option for a node selection was activated



before the simulation was started. Adding budget groups to an existing simulation record is not yet supported.

The calculated summed rate and period budgets can be exported via the context menu of the **Rate-** and **Period-Budget** charts.

## 13.4 Multilayer Wells: Flow per Layer

For multilayer wells, the total capacity is internally associated with the lowermost node that belongs to the well screen. Therefore, the **Rate-** and **Period-Budget** panels only display the total pumping/injection rate or volume and not the rates or volumes associated with each layer.

To check the flow for each layer that belongs to a multilayer-well screen, **Multilayer well** needs to be plotted in the active view. The entry **Flow per Layer** in the list of multilayer-well attributes in the **View Components** panel can then be activated to display the flow for each layer.

## 13.5 Content Analysis

The **Content** panel provides access to the values for various spatially integrated quantities, always at the current simulation time. Depending on the FEFLOW problem class of a given model, the panel contains separate tabs for **Fluid**, **Mass**, **Groundwater Age** and **Heat**. In case of multiple chemical or age species, a separate tab is created for each species. Depending again on problem class and type, content integration can provide quantities such as domain size, pore space, fluid volume, dissolved mass, absorbed mass, thermal energy in fluid and solid phases, etc.

If the entire model domain is considered, discrete features can be included in the calculation as well. Alternatively, the current element selection or any stored element selection (i.e., subdomain) can be used for content evaluation via the context menu in the **Selections** panel.

## 13.6 Streamlines and Pathlines

Streamlines and pathlines can be very helpful in visualizing steady-state and transient flow fields. While a streamline represents the trajectory of a particle in a flow field assumed to be constant in time, a pathline follows a particle in a transient flow field. Pathline computation therefore requires loading a simulation record of a transient model. Pathlines and streamlines are coincident in a steady flow field.

Streamlines and pathlines can be calculated forwards or backwards from the starting point. The starting points (seeds) can be set at points, circles around points, inside or at the surface of a cylinder or sphere or distributed along lines. Distributed seeds can either be equally spaced along the starting line,



or spaced such that the distance between adjacent seeds is inversely proportional to the local flux magnitude.

Streamlines can also be shown during a transient simulation. In this case, the computed trajectory will change with the flow field after each time step.

To visualize travel times of the particles along streamlines and pathlines, isochrone markers or period sections can be used.

The correct calculation of travel times requires the definition of an effective porosity. For this, one of the available porosities for the given model type (specific yield, porosity for unsaturated flow, mass or heat transport porosity) or an arbitrary user-data distribution can be applied.

In mass-transport models FEFLOW additionally provides the option to account for retardation due to Henry-type sorption.

To display streamlines or pathlines, a selection of an appropriate source geometry in the **Entities** panel or in the **Selections** panel (node selection, 3D point set, 3D polyline, 3D loop, cylinder or sphere) is required. Node selections can be created by storing the current selection, tools for generating the other 3D geometries are available in the **Drawing** toolbar.

The second important component for streamline/pathline generation is the **Pathlines** or **Streamlines** entry in the **Process Variables** section of the **Data** panel. Following a double click on either **Forward** or **Backward** the pathlines/streamlines are calculated and added to the active view. Pathline properties are found in two different locations: While the effective porosity for isochrone calculation is selected in the **Problem Settings** dialog, view-specific visualization options such as number of seeds along a line, line width or time periods for period sections are controlled via the **Properties** of the pathlines or period sections entry in the **View Components** panel.

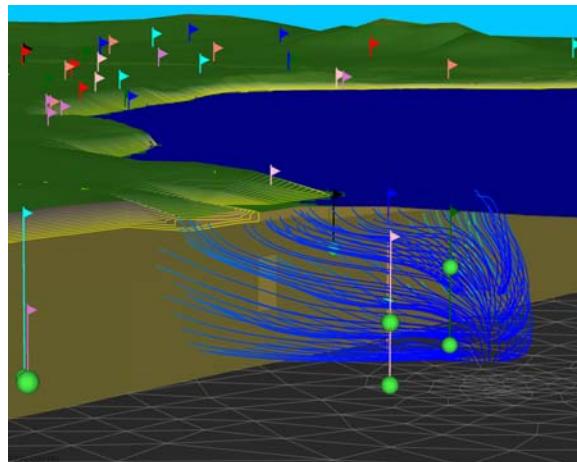


Figure 13.2 Pathline analysis.

### 13.6.1 Random-Walk Particle-Tracking

Random-Walk Particle-Tracking (RWPT) solutions can be obtained by proceeding as with standard advective streamlines and pathlines, and by assigning the required additional dispersive parameters: molecular diffusion and coefficients of dispersivity. RWPT solutions are theoretically consistent with advection - dispersion equation solutions - provided that a sufficiently large number of particles is used.

## 13.7 Export

The FEFLOW export options can roughly be grouped into export of raster images (movies and snapshots of a view), and export of vector data (data and plots).

Snapshots of the active view are exported by using the camera button in the **View** toolbar. A resolution enhancement can be applied to the image and texts (e.g., legend text) to obtain high-resolution output suitable for poster printing. Snapshots of a **Slice** view can optionally be exported as georeferenced images. Additional 3D export settings are used when exporting stereoscopic images. Section 14 describes how movies are created from a view animation.

Data and plot export encompasses a variety of different file formats, ranging from simple column-based text files to GIS and CAD files. Data of process variables, boundary conditions, material properties and reference distributions are exported via the context menus of one or several selected parameters in the **Data** or **View Components** panel. If a full simulation results file is loaded, data can be exported for the current time step, a user-defined selection or for all recorded time steps.



Visualization features such as isolines or fringes are exported by using their context menu in the **View Components** panel. If supported by the output format (e.g., ESRI Shape file, AutoCAD Exchange file), plot files are exported in 3D from 3D models.

## 13.8 Tutorial

In the following exercises we use a simulation record to evaluate the modeling results after the simulation. Start FEFLOW and click **Open** to load the file **results.dac**. The file contains the complete results for every calculated time step of a transient flow and mass transport simulation.

### 13.8.1 Tools

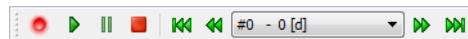
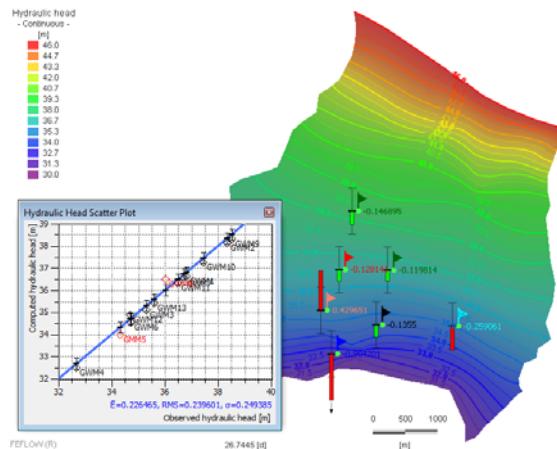


Figure 13.3 Simulation toolbar.

### 13.8.2 Observation Points

For the loaded simulation record, nine different charts can be displayed: **Time Steps**, **Hydraulic Head**, **Pressure**, **Local** and **Average Concentration**, **Fluid-Rate** and **Fluid-Period Budget**, and **Mass-Rate** and **Mass-Period Budget**.

To open a chart, right-click in an empty part of the FEFLOW workspace, select **Charts** and then the chart that is to be displayed. For each observation point a curve is plotted so that the hydraulic head, concentration and pressure changes at these locations can be monitored during and after the simulation.



**Figure 13.4** Observation points with error bars and labels and hydraulic-head scatter plot.

To compare the field data for **Hydraulic head** to the calculated head results at observation-point locations, activate the **Slice** view first, switch to slice 2 and select **Model Locations > Observation Points** in the **Entities** panel. Next, double-click on **Process Variables > Fluid flow > Hydraulic head** in the **Data** panel. The deviations between measured and computed water levels are now shown as colored bars and as labels next to the observation-point locations (see Figure 13.4). The error bars and labels always refer to the currently selected simulation time. Use the tools in the **Simulation** toolbar to switch to different simulation steps to check how the deviations change. Green bars indicate deviations within the confidence interval, red bars indicate that the deviations exceed the confidence interval. The position of an error bar indicates whether the deviation is positive or negative.

In addition to displaying the water-level deviations in the active view, we open a scatter plot to compare measured and computed data. Open the context menu of the **Hydraulic-head** chart and select **Scatter plot**. The plot always reflects the results at the currently selected simulation time, displayed in both the **Simulation** toolbar and the view windows.

### 13.8.3 Budget Analysis

We can use the **Rate-Budget** panel to monitor the fluid and mass flows that enter and leave the model domain, or only specific areas.

For mass-transport simulations, the **Rate-** and **Period-Budget** panels contain two tabs, one for fluid and one for mass (see Figure 13.1). To start the budget calculation for the entire model domain set the check mark for  **Active** in the respective tab for fluid and mass. After a brief computation, the **Rate-Budget** panel shows the flow rate for each boundary condition type, for sources and sinks, storage capture and release and the imbalance



term representing the remaining residual error. Flows that leave the model have a negative algebraic sign while flows that enter the model have a positive algebraic sign.

Use the **Period-Budget** panel to calculate the time-integrated budget the first 1000 days of the simulation: Switch to day 1000 via the drop-down list of stored time steps in the **Simulation** toolbar and then click on in the panel to start the calculation.

We now calculate the fluid budget for the border in the south instead of taking the entire model as budget domain. Go to the **Selections** panel, open the context menu of **Southern Border** and select **Use Selection as Budget Domain**. The domain of interest is automatically switched from **Domain** to **Southern Border** in both the **Rate** and the **Period-Budget** panels. Start the rate-budget calculation with **Active**. The rate budget is now calculated for the selected nodes only. The budget shows that water is leaving the model domain through the border in the south.

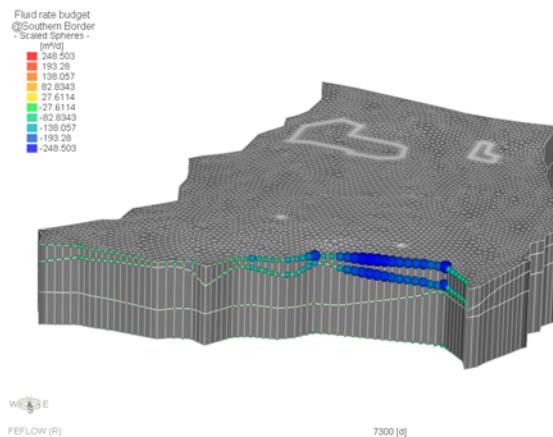


Figure 13.5 Scaled budget spheres along the southern border.

Additionally, the in- and outgoing flows can be visualized as scaled spheres. Set the **3D** view as active view and click on **Southern Border** in the **Selections** panel.

Now, double-click on **Fluid flow > Rate Budget** in the **Data** panel. Blue and green spheres of different sizes are plotted at nodes along the southern border where water leaves the model. We now increase the relative size of the spheres in order to make smaller spheres visible. Open the context menu of **Fluid rate budget > Scaled Spheres** in the **View Components** panel and select **Properties**. In the **Size** tab increase the relative size to **3.0** and click **Apply**. Figure 13.5 shows the resulting plot in the **3D** view.



To check how much mass has left the model via the border in the south over time, we use the **Mass-Period Budget** chart. Open the chart via **View > Charts > Mass-Period Budget History** and check the curve **Southern Border**.

### 13.8.4 Subdomain-Boundary Budget

To evaluate how much mass enters the lower aquifer via the two contamination sources, we apply the **Subdomain Boundary Budget** panels.

First, we define layer 5 as domain of interest for the budget calculation. In a subsequent step, we create a second element selection to apply as masking domain. This step is required as we would like to compute the mass flow through only those boundary parts of layer 5 which are in direct contact with the contamination sites.

Activate the **Slice** view and switch to **Layer 5** in the **Entities** panel. Make sure that the geometry type is set to **Select Elements** in the **Selection** toolbar and select the entire bottom layer with a click on **Select All**. Right-click into the **Slice** view and select **Store Current Selection**. Rename the selection to **Lower Aquifer** and finish with <Enter>. The stored selection now appears in the **Selections** panel. To use this selection as domain of interest in the **Subdomain Boundary Budget** panels, right-click on **Lower Aquifer** and select **Use Selection as Boundary-Budget Domain**. The selection is now available as domain of interest in both the **Subdomain Boundary Rate** and **Period Budget** panels. In the **Subdomain Boundary Rate Budget** panel, switch to the **Mass** tab and start the budget calculation with the **Active** check box. The panel now displays the mass rate entering and leaving this subdomain at the currently selected simulation time.

As we are only interested in the amount of mass entering the lower aquifer via the bottom sections of the two contamination sites, we now apply a second selection as masking domain. First, clear the current selection with a click on **Clear Selection**. Now switch to **Layer 4** in the **Entities** panel and activate the **Supermesh Polygons** map for selection via a double-click on this entry in the **Maps** panel. Then, reduce the **Snap distance** to **1 m** in the **Snap-Distance** toolbar and switch to the **Select by Map Polygon** tool in the drop-down list in the **Selection** toolbar. Move the mouse cursor over the model domain and create the two selections with a single click into the two contamination sites. Use the context menu of the **Slice** view to store the selection as **Contaminations**. To apply this second selection as masking domain, click on **Contaminations** in the **Selections** panel and then click on the green check-mark symbol in the **Masking Domain** section in the **Subdomain Boundary Rate Budget** panel. The mass-flow calculation is now limited to these element faces which are shared by both selections (see Figure 13.6).



The settings of the **Subdomain Boundary Period Budget** panel follow those of the **Rate** panel. To invoke the time-integrated budget calculation, set the check mark in front of  **Active**, enter the starting time for the budget calculation and click on . After a brief computation time, the mass entering and leaving the domain of interest via the selected subdomain boundaries is displayed (Figure 13.6).

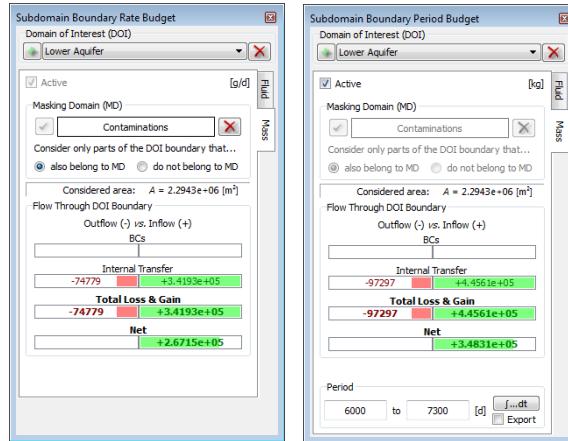


Figure 13.6 Subdomain Boundary Rate and Period Budget panels with results for the selected DOI and MD.

### 13.8.5 Content Analysis

To check how much mass the model domain contains at different time steps we use the **Content** panel. Switch to the tab **Mass** and activate the content calculation for  **Dissolved Species Mass**. The currently contained amount of mass is now displayed in grams. Browse the time-step list in the **Simulation** toolbar to check how the mass content changes with time.

### 13.8.6 Streamlines, Pathlines

To visualize the steady and transient flow fields we plot both streamlines and pathlines. Start with a 3D plot of streamlines that start at the two well locations.

First, set the **3D** view as active view and deactivate **Geometry > Faces** in the **View Components** panel as otherwise the streamlines located below slice 1 would be concealed.

Now, click on **Node Selection > Wells** in the **Selections** panel and double-click on **Streamlines > Backward** in the **Data** panel. Start the plot by checking  **Traces** in the **View Components** panel. After a brief computation the streamlines are shown in the **3D** view.



To visualize the flow field in the vicinity of the wells we increase the radius for the streamline seeds around the wells. To change the radius and other streamline properties, open the context menu of **Travel time, backward streamlines seeded@Wells** in the **View Components** panel and select **Properties**. In the upcoming **Properties** panel type **1<sup>23</sup> 30 m** as radius, hit <Enter> and click **Apply**. Several streamlines are now plotted around the two wells. Figure 13.7 shows a 3D plot of streamlines around the wells.

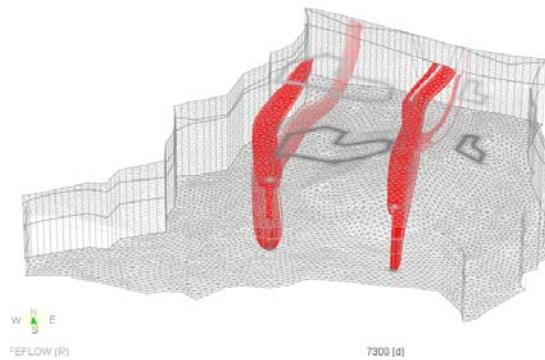


Figure 13.7 3D Streamlines around the wells.

To visualize the transient flow field with pathlines, go back to the **Data** panel and double-click on **Pathlines > Backward**. To start the plot, repeat the same steps as previously described for the streamlines. Furthermore, select time step **#66** in the **Simulation** toolbar.

We also add **Period Sections** to visualize particle travel times. In the **View Components** panel, uncheck  **Traces** and check  **Travel time, backward pathlines seeded@Wells > Period Sections** and then double-click on this entry to edit the period-sections settings in the **Properties** panel. On the **Iso** tab, deactivate the  **Automatic** option, enter an interval of **1<sup>23</sup> 365 d** and confirm the changes with **Apply**.

To plot each period section in a different color, open the **Properties** panel of **Travel time, backward pathlines seeded@Wells** via its context menu in the **View Components** panel. Right-click into the color bar on the left-hand side of the panel and select **Presets > Rainbow**. Figure 13.8 shows the resulting plot.

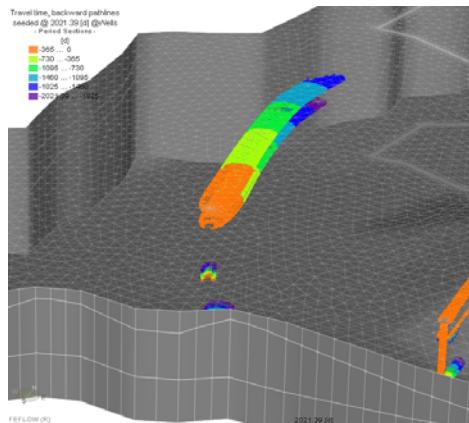


Figure 13.8 Period sections.

Streamlines or pathlines can also start from polylines. To draw a polyline for streamline plotting, open the context menu of **Streamlines > Forward** and select **Draw seed line/points > Draw a 3D Line**. Start an arbitrary line on the top slice of the model with a single click. Extend the line by adding points and finish with a double-click.

The line is now displayed in the active view and is shown as **Domain Locations > 3D Polyline #1** in the **Entities** panel.

Now, click on **3D Polyline #1** in the **Entities** panel and start the streamline plot with a double-click on **Streamlines > Forward** in the **Data** panel. Figure 13.9 shows an example for a streamline plot from a polyline.

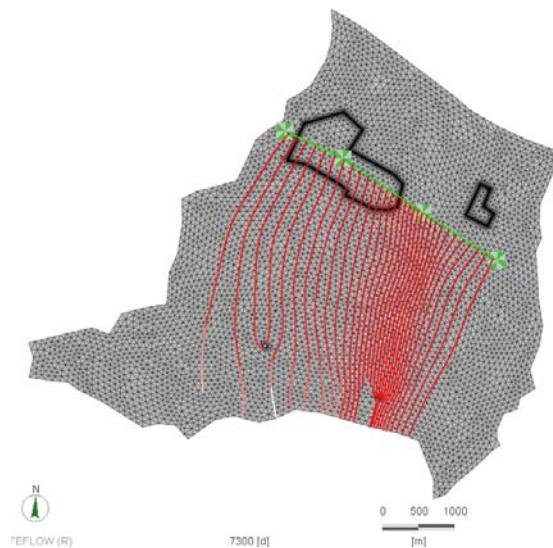


Figure 13.9 Streamlines starting at a polyline.

### 13.8.7 Random-Walk Particle-Tracking (RWPT)

To illustrate RWPT select **Backward Random-Walk Tracks** in the Data panel, for the same node locations as in the previous section.

Go to Edit > Problem Settings > Field-Line Computation > Random-walk tracks and use the default, homogeneous dispersive parameters for molecular diffusion and the coefficients of dispersivity.

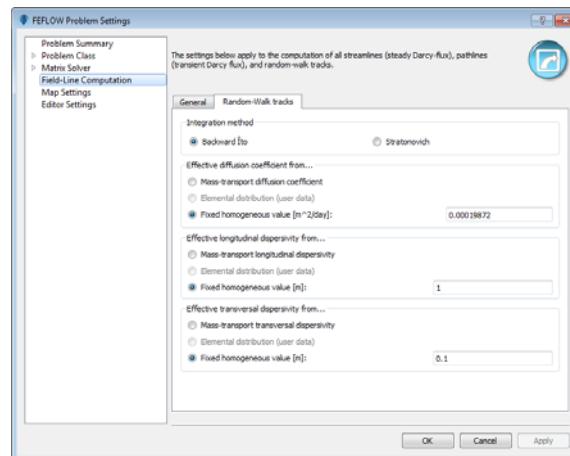


Figure 13.10 Defining RWPT parameters.



Note that dispersivity values are small. As opposed to standard advection-dispersion solution techniques, the RWPT does not suffer from numerical dispersion effects and stability constraints. Thus, there is no need to artificially increase dispersion coefficients according to mesh size. In other words, RWPT can be solved with arbitrary dispersive coefficients, in preference close to their physical values, even if the mesh is very coarse.

In order for RWPT solutions to be consistent with advection-dispersion solutions, a sufficiently large number of input particles must be used. The greater the amount of used particles the more the RWPT and advection-dispersion solutions will be comparable. However, RWPT solutions can be very memory-demanding when many particles are used so it is recommended to (i) start with a relatively small amount of particles (e.g., 100), (ii) analyze the runtime and memory demand, and (iii) subsequently increase the number of used particles if possible. Most problems will require a minimum number of about 10,000 particles for RWPT solution to be able to explore all possible advective-diffusive/dispersive paths and thus be consistent with advection-dispersion solutions. Note that there is a control during the RWPT solution phase on memory usage to avoid memory overflow situations.

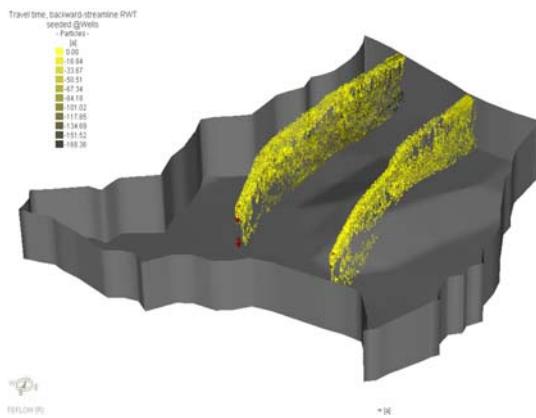


Figure 13.11 Backward RWPT solution seeded at the wells.

### 13.8.8 Export of Results

To export the mass distribution obtained at the end of the simulation browse to the last time step in the **Simulation** toolbar and double-click on **Mass concentration** in the **Data** panel. Open the context menu of **Mass concentration** in the **View Components** panel and select **Export Data > Current Time #116 7300 d > All Nodes**. Select a file name and choose the ESRI Shape File 3D format (\*.shp). The exported data can be loaded again as a map and assigned as initial mass distribution in subsequent simulations. To automatically add the exported map to the current model, confirm with **Yes** in the dialog appearing after the map export.



Plots can be exported in the same way as data. To export the isolines plot open the context menu of **Mass concentration > Isolines** in the **View Components** panel and select **Export Plot**. Choose a file name and format for the exported isolines and finish the export with **Save**.

A further option to export simulation results is to take snapshots of the active view window. Click **Snapshot of the Active View** in the **View** toolbar and choose a resolution enhancement factor for the exported snapshot. After clicking **OK**, choose a name and file format for the snapshot.





# 14 Animation and Video Export

Up-to-date presentation of input data and simulation results

## 14.1 Introduction

Dynamic visualization is essential when presenting model results to clients, authorities and the public, and facilitates understanding of complex processes.

## 14.2 Creating a Presentation

A dynamic presentation can be set up for each view window using its autopilot. The autopilot of the active view window is accessible via the **Autopilot** panel where up to four separate categories, **Simulation Time**, **Position**, **Clipping**, and **Visibility** can be set up along a common presentation-time line. Presentation time is real time as observed on a wrist watch during presentation play-back. The available categories depend on the type of the respective view window. Each category has an **Engage** checkbox that acts similarly to a clutch. When engaged, the autopilot completely controls the respective category. For example, engaging the autopilot **Position** category will disable user-controlled navigation via the **Navigation** panel or mouse tools and all object positioning becomes autopilot-controlled.



Figure 14.1 Autopilot panel.

The total length of the presentation (in seconds) can be entered in the **Autopilot** panel. The displayed percentage value corresponds to the position of the timeline slider. It can be edited to explicitly set a timeline position.

A play-back function is provided to preview the autopilot animation sequence.

A simple movement is produced by defining two object-position keys, one at the beginning and one at the end of the presentation-time line. Whenever the autopilot **Position** category is engaged (e.g., during autopilot play-back and movie export), all intermediate object positions are automatically interpolated between the bracketing keys, resulting in a continuous object movement. To



produce more complex paths, additional keys can be freely placed along the presentation-time line.

When visualizing FEFLOW simulation records (\*.dac files) for transient problems, the simulation-time axis (often covering days, months, or years) has to be mapped to the presentation-time axis (typically, seconds or minutes long). The autopilot performs this mapping simultaneously with the dynamic object positioning.

While the autopilot **Simulation Time** category is engaged, the candidate time step shown in the **Simulation** toolbar is obtained by interpolating the user-defined simulation-time keys on the presentation timeline. By default, beginning and end of the simulation period are mapped to beginning and end of the presentation timeline, respectively. Thus, in the simplest case, no additional simulation-time keys have to be specified at all.

To obtain animation effects such as a gradually extending cut-out section progressively exposing internal isosurfaces, the autopilot of a 3D view window can dynamically interpolate between predefined (key) clip settings.

While the autopilot **Clipping** category is engaged, the settings for clipping and cutout planes are obtained by interpreting the user-defined clipping keys on the presentation timeline. At a given presentation moment, the status of each plane ('Disabled', 'Clipping', or 'Cutout') will be given by the respective previous clipping key on the presentation timeline while position and orientation of all planes are continuously interpolated between the respective bracketing clipping keys.

The autopilot can dynamically apply predefined feature-visibility settings to control the display of multiple data sources in sequence or to display features such as the finite-element mesh or a surface map during only a portion of the entire presentation.

While the autopilot **Visibility** category is engaged, the feature-visibility settings are obtained from the user-defined visibility keys on the presentation timeline. At a given presentation moment, the visibility status of each feature will be given by the respective previous visibility key on the presentation timeline.

## 14.3 Movie Export

Export of the autopilot presentation of the active view to a movie is evoked via **Export AVI from Active View** in the **File** menu or the **View** toolbar. Any of the video compression codecs available on the operating system can be chosen for the export. As formats, the standard \*.avi format can be selected or the animation can be exported as 3D stereoscopic movie in \*.avis format.



## 14.4 Tutorial

### 14.4.1 Tools

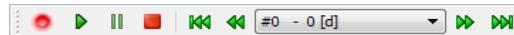


Figure 14.2 Simulation toolbar.

### 14.4.2 Creating a Presentation

In this exercise we create a presentation from a results file of a transient free simulation and export the presentation as a movie. First, click **Open** and load the file **animation.dac**.

We start with a very simple presentation in the **3D** view in which we show how the mesh stratigraphy changes during the simulation. Switch to the **Autopilot** panel located below the view windows and make sure that the **3D** view is the active view.

By default, beginning and end of the simulation are mapped to beginning and end of the presentation timeline, respectively. The two green bars at beginning and end of the **Simulation Time** line show that time keys are set at these positions.

For showing the mesh movement during the simulation no additional settings are necessary. Simply click **Preview** to preview the first presentation.

In the next step we extend the animation to 20 seconds and add further time line-keys to slow down the animation at the stages where strong changes are visible.

First, enter **20 sec** as total presentation time and hit <Enter>. In the upcoming **Expand presentation** dialog select the option **Rescale all existing key timings** and click **Rescale**. Now, browse to time step **40** in the time step list in the **Simulation** toolbar, type **8.0 sec** in the left input field of presentation time and hit <Enter>. Alternatively, the white slider bar can be moved manually to **8.0 sec** of presentation time. To set a key at this time step click the green checkmark in the **Simulation Time** line. A green bar appears indicating that a key has been set here.

Repeat the same steps to set the second time key for time step **65** at **16 sec**onds of presentation time. After this time key has been set, move the white slider bar to the beginning of the presentation and click **Preview** again to see the presentation.

In addition to presenting the mesh movement during the simulation we want to change the model position in the active view during the presentation.



The model is to move from the upper left corner towards the center of the active view during the presentation. An increasing zoom factor and some rotation are also desired.

First, press the middle mouse button and drag the model to the upper left corner of the view. Now, use the right mouse button to zoom out and then, with the left mouse button, slightly rotate the model backwards and to the left. To set the first key at this position move the white slider to the beginning of the presentation time and click the green checkmark in the **Position** line.

For the second key, position the model in the center of the view, zoom in and rotate the model to the right. Move the slider to the end of the presentation and click the green checkmark in the **Position** line again. The position of the model will now dynamically change between these two prescribed positions during the presentation.

We complete the presentation by removing the handles from the active view. Right-click in the view window and deselect **Show > Handles**. Now click **Preview** again to see the final presentation.

#### 14.4.3 Movie Export

When all presentation keys are set we proceed with the export of the presentation as a movie.

In the **Autopilot** panel, choose a **Frame Rate** of **20 fps** and click **Export AVI from Active View** in the **View** toolbar. Choose a name for the movie and click **Save**. Finally, select one of the available compression codecs and the desired quality and click **OK** to start the movie export.

#### 14.4.4 Export Settings

Presentations can also be exported and reloaded again. The presentation settings are exported together with the settings of the active view. After creating the presentation right-click in the active view window and select **Export Settings** to save the presentation and view settings as \*.xml-file. Alternatively, the presentation can be stored as a scene in the **Scene Library** panel and be restored. To store the presentation settings, right-click into the active view and select **Create Scene from Window**. Enter **Presentation** as name and finish with <Enter>.

Reloading the animation and view settings works in the same way: Go to **Window > New** and select **3D View**. Right-click in the new **3D** view to invoke the context menu, select **Import Settings** and load the file that we have just exported. The view and animation settings are now adopted from the \*.xml file.



Alternatively, right-click onto the stored scene in the  **Scene Library** panel and select  **Create View from Scene**.





## 15 Groundwater Age Calculation

Crucial information on flow dynamics and mixing, and for capture zone analysis and risk assessment

### 15.1 Introduction

FEFLOW provides the capabilities to conveniently calculate groundwater age, lifetime expectancy and exit probability. These three properties are rooted in the same theoretical framework of *component exposure time* and are obtained by solving standard mass-transport equations for which specific coefficients and boundary conditions must be defined. Transport of age therefore is available as a new problem class, similar to the transport of mass or heat. Solving for lifetime expectancy and exit probability in addition to groundwater age requires the definition of these additional **Age Species** in the **Problem Settings** dialog and the assignment of the corresponding transport parameters. The simulation of groundwater age is based on the forward solution of the transport equation while for lifetime expectancy and exit probability the backward transport equation is used.

The solutions of groundwater age, lifetime expectancy and exit probability can be used to perform specific diagnoses on a flow field, such as the analysis of the flow dynamics and mixing processes, the estimation of outlet vulnerability, or the evaluation of outlet capture zones and the origin of water.

### 15.2 Groundwater Age

Mean age represents the elapsed average time spent in the aquifer since groundwater has entered the model domain, i.e., it indicates the travel time from the inflow boundary to the current location. Typically, it is assumed that groundwater 'life' starts at inflow into a model and FEFLOW therefore automatically assigns a Dirichlet boundary condition of 0 days at all inflow boundaries. If groundwater is not assumed to be 'born' at inflow boundaries, an age-concentration BC with the specific age value needs to be assigned at these locations manually.

Age solutions provide information on underground travel times and mixing processes and the results on mean groundwater age can be used for calibration purposes if age data are available, e.g., from isotope analyses.

### 15.3 Mean Lifetime Expectancy

Lifetime expectancy is defined as the average time for groundwater still needed before exiting the domain via an outlet. It therefore corresponds to the expected travel time from the current location to an outflow boundary. As groundwater is assumed to 'die' at outflow boundaries, FEFLOW automati-



cally assigns a LTE-concentration BC with a value of 0 days at these locations.

Results for lifetime expectancy can be used for risk-vulnerability assessments and the analysis of groundwater dynamics: Zones with longer lifetime expectancy indicate groundwater divides whereas areas close to outflow boundaries show shorter travel times.

## 15.4 Exit Probability

The parameter exit probability can be used to calculate the probability of water exiting the model domain at specific locations. In contrast to age and lifetime expectancy simulations, boundary conditions for exit probability are not applied automatically. Instead, outflow locations for which the exit probability is to be calculated have to be selected manually. As the probability of exit is 100% at an outflow boundary, a probability BC with a value of 1 is typically assigned there.

Assigning a probability boundary condition of 1 at a pumping-well location allows to delineate the capture zone of the well and also to determine the origin of the pumped water. Compared to standard particle tracking, exit probability can provide much more information on a capture zone: Heterogeneity effects can be considered via dispersion and flow times can be determined through temporal spreading of the probability plume. In combination with mean lifetime expectancy, expected travel times towards a well can be determined.



*To calculate exit probability for multiple outflow boundaries with overlapping capture zones simultaneously, multiple age species of type **Exit Probability** can be used.*

Evaluating exit probability via the **Content** panel provides information on the amounts of water related to the capture zone of the well. The proportions of water extracted can be calculated according to their origin of infiltration using the Rate- and Period-Budget tools.

## 15.5 Tutorial

In the following exercises we set up a groundwater-age simulation problem and solve for **Mean Age**, **Lifetime Expectancy** and **Exit Probability**. The results will be used to delineate the capture zone for a pumping well and to determine the contributions of the different sources to the total pumped water.



### 15.5.1 Tools



Figure 15.1 Selection toolbar.

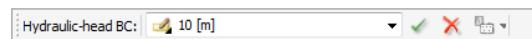


Figure 15.2 Editor toolbar.



Figure 15.3 Simulation toolbar.

### 15.5.2 Setting up a Groundwater Age Simulation

Start FEFLOW and click **Open** to load the file **age.fem**. To setup an age, lifetime expectancy and exit probability solution problem, open the **Problem Settings** dialog via the **Edit** menu. On the **Problem Class** page check the option **Include transport of...  Age** and confirm with a click on **Apply**. By doing so a new page named **Age Species** is created in the tree on the left-hand side of the dialog. Enter the **Age Species** page and click **Add species** twice to create three age species in total. Change the **Type** of the second species to **Lifetime Expectancy** and the third species to **Exit Probability** (Figure 15.5). Confirm the changes with **Apply** and leave the dialog with a click on **OK**.

Each declared age species is now defined as a transport property and transport parameters such as porosity, molecular diffusion and coefficients of dispersivity need to be defined accordingly. By default, FEFLOW only displays the Dirichlet-type boundary condition for each age species: **Age-concentration BC**, **LTE-concentration BC** and **Probability BC**. Additional boundary-condition types can be added via **Add Parameter** in the context menu of the specific boundary-condition section if needed.

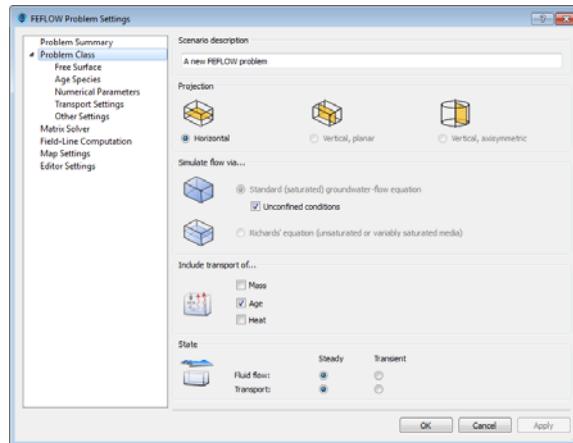


Figure 15.4 Activating groundwater age as problem class.

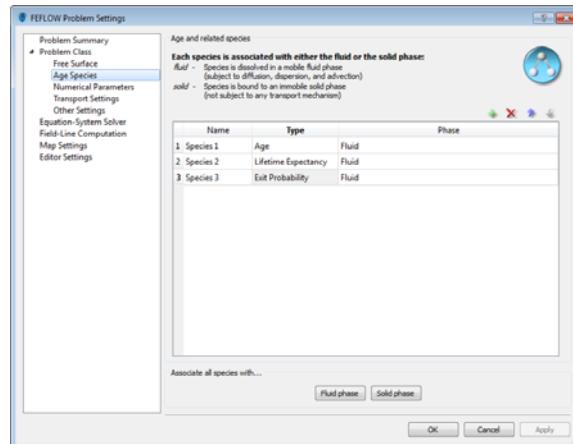


Figure 15.5 Defining age species in the Problem Settings dialog.

The boundary conditions for fluid flow consist of a fluid-flux BC in the upper west model region, a fluid-transfer BC representing a river and two pumping wells. Additionally, water enters the domain via recharge.

To determine the capture zone for the upper pumping well, we need to assign a **Probability BC** of unity at this node location. Double-click on **Probability BC** in the **Data** panel and then double-click on the stored node selection **Pumping well** to set the selection active. Enter a value of **1** in the input box of the **Editor** toolbar and hit **<Enter>** to complete the assignment. Boundary conditions for mean age and mean lifetime expectancy do not have to be assigned as these will be set automatically once the simulation starts.



The transport parameters for the different age species do not have to be changed as we will use the default parameters.

To start the simulation, click on Start in the **Simulation** toolbar.

### 15.5.3 Results Evaluation

#### Travel-Time Analysis

##### Mean Age and Mean Lifetime Expectancy

The results for mean age and mean lifetime expectancy are displayed in units of time. To plot these nodal distributions in the active view, go to the

**Entities** panel and click on **Domain**. Next, double-click on **Mean age** in the **Process Variables** section in the **Data** panel. Water enters the model domain with an age of zero and locations close to inflow boundaries therefore show short travel times. To identify the nodes at which water enters the model domain, double-click on **Mass transport - age#1 (f) > Age-concentration BC**. Inflow locations are now indicated by an age-concentration BC with a value of 0 days.

For better visibility we uncheck the  **Edges** entry in the **View Components** panel. To plot the remaining travel time for water molecules to reach an outflow boundary, double-click on **Mean lifetime expectancy** in the **Data** panel. As we would like to combine this plot with the exit-probability distribution in a later step, lock this parameter in the view by right-clicking on **Mean lifetime expectancy - Iter#1 (f)** in the **View Components** panel and selecting **Lock Data View**. Uncheck the  **Continuous** plot and activate  **Isolines** instead. To change the interval to be used for the isoline plot, double-click on **Isolines**. This opens the **Properties** panel and we remove the check mark in front of  **Automatic**. Change the **Prescribed interval** to **2 [a]** and confirm the changes with **Apply**. The resulting plot is shown in Figure 15.6.

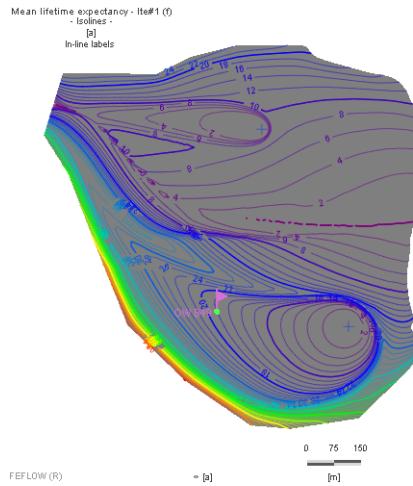


Figure 15.6 Distribution of Mean Lifetime Expectancy.

### Total Travel Time

To determine the total travel time from inlet to outlet, **Mean age** and **Mean lifetime expectancy** need to be summed up. The sum can be calculated and displayed using a nodal expression.

The model already contains a prestored nodal expression called **Total travel time** which can now be edited. Open the context menu of **Total travel time** in the **Data** panel with a right click and select **Edit Parameter Expression**. Delete the right side of the expression and double-click on the process variable **Mean age**, then enter **+ 123** + and double-click on **Mean lifetime expectancy** in the list on the right-hand side of the dialog. The expression should look like the one in Figure 15.7. Leave the dialog with **OK**.

Double click on **Total travel time** to plot the total travel time from model inlet to outlet.

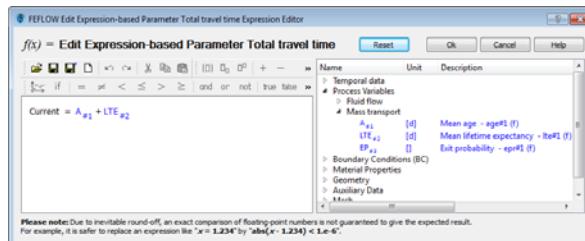


Figure 15.7 Nodal Expression for total travel time.



## Capture Zone Delineation - Exit Probability

The distribution of exit probability represents the capture zone of the target which in our example is the pumping well. A typical limit for capture-zone delineation is the 0.5-probability contour.

To plot the capture zone for the well, double-click on **Exit probability** in the **Process Variables** section in the **Data** panel. To limit the plot to probability values between 0.5 and 1, double-click on **Exit probability - epr#1 (f)** in the **View Components** panel. In the **Properties** panel of this parameter, uncheck the option  **Auto-update range**, enter **0.5** as the minimum value and confirm the changes with **Apply**. The colored area now represents the area with a probability of larger than 50% that water ends up in the well - which is interpreted as the capture zone of the well.

By examining exit probability in combination with the isolines for mean lifetime expectancy it is possible to determine the remaining travel time from a specific location within the capture zone to the outlet.

Figure 15.8 shows the capture zone of the well considering a probability of 0.5 together with the mean lifetime-expectancy contours.

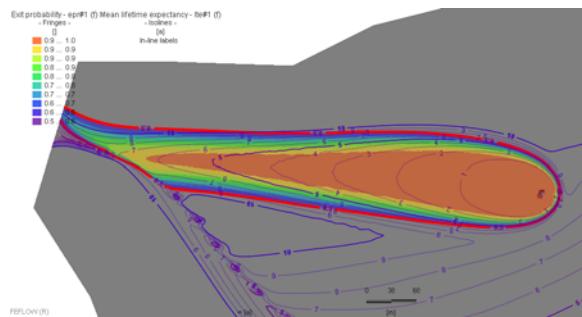


Figure 15.8 Well-capture zone and plot of remaining travel times.

Outflow nodes other than the well can be displayed by plotting the **Probability** boundary condition. FEFLOW automatically assigns a boundary condition with a probability value of 0 at these node locations.

## Capture-Zone Content, Source Partitioning and Origin of Water

To calculate the capture-zone water content, activate  **Exit Probability** **Domain Fluid Volume** in the **Content** panel. The displayed value represents the water content of the entire capture zone (equivalent to probability values larger than 0 and smaller than 1). If the capture zone is limited to probability values between 0.5 and 1, it is necessary to create an element selection for this limited probability range and to calculate the content for this element selection only.



Budget calculations for exit probability can be used to determine the different sources of water at an outflow boundary, as well as the proportions of the respective sources.

In the **Rate-Budget** panel, switch to the tab **EP - epr#1 (f)** and start the mass-budget calculation for exit probability by checking  **Active**. As the flow field is inverted for the exit-probability solution, positive values indicate outflow and negative values indicate inflow boundaries.



Figure 15.9 Mass budget returning the proportions of the pumping-well rate with respect to their origin of infiltration.

The displayed budget indicates that the pumping well captures water which entered the model domain via fluid-flux and fluid-transfer boundary conditions and also via recharge.

The source locations can be displayed in the active view by double-clicking on **Mass transport - epr#1 (f) > Rate budget** in the **Data** panel (see Figure 15.10).

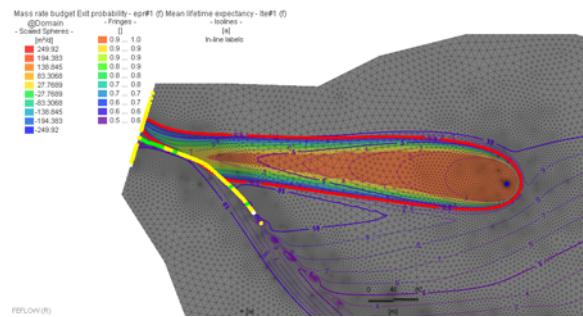


Figure 15.10 Budget spheres for exit probability representing source locations.





# 16 Plug-ins and Interface Manager IFM

Extending FEFLOW's functionality by using and programming plug-ins

## 16.1 Introduction

FEFLOW covers a broad range of functionality for porous-media flow and transport simulation, accessible via a comprehensive user interface.

Nevertheless there are cases in which extended user control over the internal processes in the software is desired and useful. Typical examples include data import from user-defined sources, coupled simulations with other software, extended interdependencies between parameters, specific physical processes, and user-specific output formats.

Many of these cases can be accommodated by the FEFLOW plug-in concept and its open programming interface controlled by the FEFLOW Interface Manager (IFM).

While the use of available plug-ins does not require any programming skills, the IFM also provides support for coding of own plug-ins by advanced modelers with some programming experience.

## 16.2 Plug-ins for Users

Plug-ins for FEFLOW typically come as a set of three files:

- a \*.dll (Windows) or \*.so (Linux) file
- a \*.html file with a description
- a \*.txt file with a copyright statement

For the plug-in functionality, only the first file is important. Plug-ins are automatically registered to FEFLOW when they are located in the modules32 (32-bit version of FEFLOW) or modules64 (64-bit version) folder in the FEFLOW installation directory.

The **Plug-ins** panel is used to manage the plug-ins. Additional plug-ins in other than the above described locations can be registered, and any registered plug-in can be added to the current model. There is no limitation in the number of plug-ins in a model. Their execution order is based on the list order in the **Plug-ins** panel.

Some of the plug-ins have functionality that is invoked by clicking the **Edit** button in the panel or by choosing **Activate** from the context menu of the plug-in. Plug-ins with their own user interface typically bring up the interface when hitting the **Edit** button.



Plug-ins also have the ability to store their own data into the \*.fem file. Thus data input to the plug-in can be saved with the model file.

When deactivating a plug-in the panel, it is no longer executed, but its data are kept. In contrast, removing the plug-in from the current model also removes its data from the FEFLOW model.

## 16.3 Technology

Plug-ins are compiled into Dynamic Link Libraries (DLL) on Windows operating systems, and into Dynamic Shared Objects (DSO) on Linux. With these, it is possible to extend the capabilities of FEFLOW without modifying its source code. The Interface Manager (IFM) uses a technique based on jump-tables that are provided in source code, which supersedes the use of object code or libraries. The jump tables are internally implemented by FEFLOW, so that plug-ins have no external references to FEFLOW's code or data. The plug-ins are thus largely shielded from changes within FEFLOW. In particular, they are not bound to a specific version of FEFLOW, making plug-in management independent from FEFLOW.

The set of API functions provided by the IFM ensures read and write access to nearly all parameters and settings in FEFLOW, along with functionality to influence the course of simulation runs.

## 16.4 IFM for Programmers

The backbone of the programming interface are the above-mentioned API functions together with callbacks, the possible entry points for plug-in code. Each of the callback functions is called at a specific point of FEFLOW's internal code sequence. For example, the PreTimeStep callback function is called immediately before a new time step of the simulation is started.

For each callback to be implemented by the plug-in, a separate C/C++ function stub can be automatically generated. While C and C++ are the 'native' languages for coding FEFLOW plug-ins, code in other programming languages can also be included via mixed-language programming.

On Windows systems, Microsoft Visual Studio is often used for plug-in development, either as the commercial or as the free Express edition. To be able to develop plug-ins, the IFM Development Kit has to be installed as part of the FEFLOW installation. With this software development kit (SDK), a wizard for Microsoft Visual Studio is installed. This wizard provides a graphically supported, efficient way to generate the frame code for a new FEFLOW plug-in project, including all required project and solution settings. The programmer 'only' has to fill in the user-specific code and compile the plug-in. Even the registration of the plug-in to FEFLOW is done automatically during the compilation process.



## 16.5 Tutorial

### 16.5.1 Using Plug-ins

To show the capabilities of plug-ins in FEFLOW, we attach and use the **Check Mesh Properties** plug-in, which is part of the FEFLOW installation. The plug-in calculates the occurring minimum and maximum angle of each finite element of a 2D mesh to check the mesh quality.

Start a new FEFLOW document and create an arbitrary supermesh polygon using the **Add Polygons** tool. When the polygon is finished, click **Generate Mesh** to generate a finite-element mesh.

Open the **Plug-ins** panel via **View > Panels**. In the list of available plug-ins, right click on **Check Mesh Properties** and choose **Attach** from the context menu. The plug-in will now appear in the list of attached plug-ins.

When the **Check Mesh Properties** plug-in is attached, it creates two reference distributions **MeshProperties:MaxAngle** and **MeshProperties:MinAngle** in the **Data** panel and two additional charts are displayed.

Double-click on one of these reference distributions to visualize the angles in the active view.

Whenever the mesh is changed (e.g., when applying **Smooth Mesh** in the **Mesh-Geometry** toolbar), the values in the reference distribution are updated.

If a mesh contains a large number of elements, the computation time for the updating process may become unacceptably long. In this case, the plug-in can be temporarily disabled by removing its checkmark in the **Plug-ins** panel.

### 16.5.2 Programming Plug-ins

The widely used Microsoft Visual Studio development environment has been found suitable for the development of plug-ins. The basic version, Visual Studio Express, is available free of charge (<http://www.microsoft.com/express/>). For Visual Studio 2012 and 2013, the Express version for Windows Desktop (Express for Windows is not sufficient) needs to be installed.

The following example is based on Microsoft Visual Studio 2010 Standard/Professional Edition. The approach differs slightly from other versions of Microsoft Visual Studio and the free Microsoft Visual Studio 2010 Express Edition.

Before starting the plug-in development, the following items have to be installed:



- Microsoft Visual Studio Standard/Professional or Microsoft Visual Studio Express Edition with C++ support
- FEFLOW IFM SDK (the respective package in the FEFLOW setup has to be selected)



*Microsoft Visual Studio has to be installed prior to the IFM SDK to allow the installation of the template.*

## Starting the IFM Wizard

To start with a new plug-in, open Visual Studio 2010 and choose **New > Project** from the **File** menu.

In the tree on the left side of the dialog, click on **FEFLOW 6.2 Projects**. Next, click on the **FEFLOW 6.2 IFM Plug-in** template. Remove the **Create directory for solution** check mark at the bottom of the window. After entering the name (HelloWorld) and the target location for the new plug-in, click **OK** to proceed.

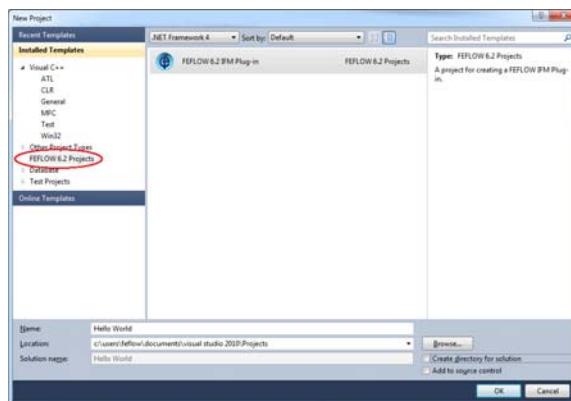


Figure 16.1 Choosing an IFM plug-in template.

## Initial Plug-In Settings

On the **Welcome page**, click **Next**.

The following **Plug-in Name** page asks for a plug-in ID (which is its unique identifier), its display name (the name to be shown in FEFLOW) and the plug-in version number.

By default, **Plug-In ID** and **Display Name** are set according to the project name given before. We accept the default. The value of the **Register Proc** field should not be modified.

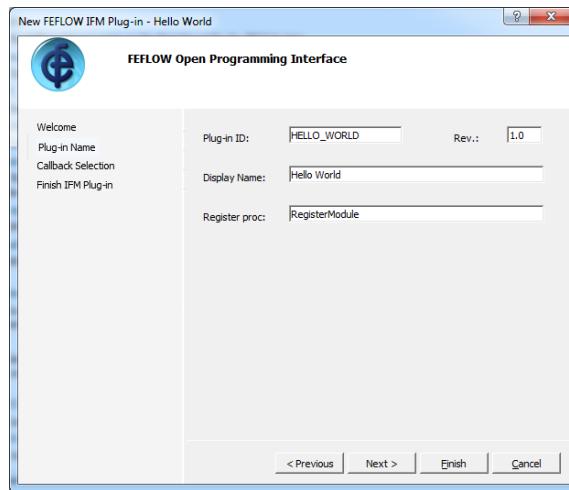


Figure 16.2 Choosing an IFM plug-in template.

Clicking **Next** leads to the *Callback Selection* page. Place the mouse cursor over a callback to get detailed information in a tool tip window. Activate the callback **OnEditDocument**.

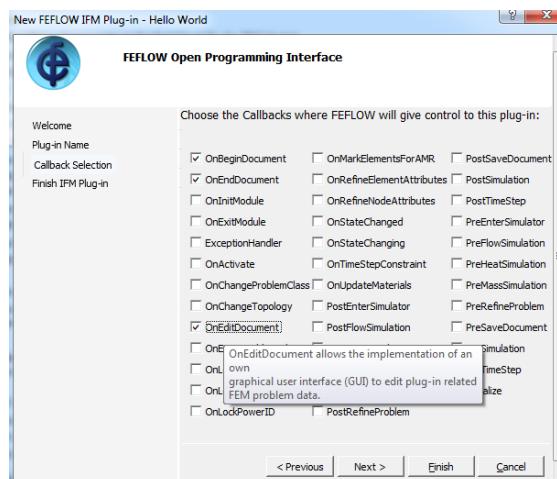


Figure 16.3 Callback Selection page.

Click on **Finish** to start the generation of the plug-in frame code and of the Visual Studio files.



## Implementing the Functionality

With a double-click, open the primary source code file **HelloWorld.cpp** from the tree in the **Solution Explorer** panel.

Scroll down to

```
// Callbacks  
void CHelloWorld::OnEditDocument...
```

and replace the lines

```
/*  
 * TODO: Add your own code here...  
 */
```

by the command

```
IfmInfo(pDoc, "Hello World!");
```

according to Figure 16.4.

```
// Callbacks  
void CHelloWorld::OnEditDocument (IfmDocument pDoc, Widget wParent)  
{  
    IfmInfo(pDoc, "Hello World!");  
}
```

Figure 16.4 Example code.

## Compiling and Debugging

Build the plug-in by pressing <F7> and then start debugging with <F5>. Visual Studio starts FEFLOW as a child process in debugging mode. FEFLOW automatically lists the plug-in in the **Plug-ins** panel.

Load an arbitrary FEFLOW model, attach the plug-in as described in 16.5.1: *Using Plug-ins*, p. 193 and following., and click **Edit plug-in properties** in the **Plug-ins** panel. The message “Hello World” appears in the **Log** panel.

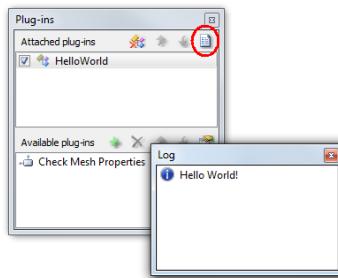


Figure 16.5 Log window message “Hello World!”.

### Compiling release and x64 versions

By default, a debug dynamic link library is compiled, which allows debugging (e.g., using breakpoints in the plug-in code), but has reduced performance.

To build a more efficient release version, change the **Configuration** as shown in Figure 16.6. In case that FEFLOW is still open, make sure to close it in order to return to Visual Studio.

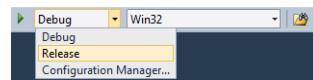


Figure 16.6 Changing the configuration.

In the same way the platform can be changed to x64, providing the means to build plug-ins suitable for the 64-bit version of FEFLOW (Figure 16.7).

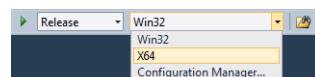


Figure 16.7 Changing the platform.





# INDEX

**E**

Export FePEST Results . . . . . 94

**F**

FePEST File Configuration . . . . . 79

**I**

IFM Implemented . . . . . 83

Interpolation from Pilot Points . . . . . 83

**J**

Jacobian Matrix . . . . . 99

JROW2VEC . . . . . 99

**K**

Kriging Configuration . . . . . 86

Kriging Interpolation . . . . . 84

**N**

NOPTMAX . . . . . 81

**O**

Objective Function . . . . . 93

Observation Definition . . . . . 87

Observation Weights . . . . . 87

**P**

Parallelization . . . . . 89

PEST Parameter Types . . . . . 86

PEST Termination Criteria . . . . . 81

Pilot Point Generation . . . . . 84

**R**

Regularization Setup . . . . . 90

Residuals . . . . . 95

Running PEST . . . . . 92

**S**

Sensitivity . . . . . 99, 100

**T**

Tikhonov Regularization . . . . . 91

**V**

Variograms . . . . . 85







**Numerics**

3D Discretization . . . . . 56

**A**

Advancing Front . . . . . 54

Age . . . . . 175

Anisotropy . . . . . 75

Auxiliary Data . . . . . 111

**B**

Borehole Heat Exchanger . . . . . 108

Boundary Conditions . . . . . 105

Boundary Constraint Conditions . . 109

Boundary-Condition Deactivation . 114

Budget Analysis . . . . . 152

Budget Groups . . . . . 154

**C**

Capture Zone Delineation . . . . . 181

Clipping . . . . . 94

Command-Line Mode . . . . . 13

Constraint Conditions . . . . . 109

Content Analysis . . . . . 155

Convergence . . . . . 144

Coordinate Systems . . . . . 33

**D**

Database . . . . . 30

Dimension . . . . . 68

Discrete Features . . . . . 112

Discretization . . . . . 53

Documentation . . . . . 14

**E**

Element Deactivation . . . . . 56

Error Tolerance . . . . . 70

ESRI Geodatabase . . . . . 30

Exit Probability . . . . . 176

Export . . . . . 37, 157, 176

Expression Editor . . . . . 118

**F**

FEFLOW Book . . . . . 14

FE-LM2 . . . . . 14

FePEST . . . . . 13

FEPLOT . . . . . 13



Free Surface . . . . . 71

## G

Geodatabases . . . . . 30  
Georeferencing . . . . . 34  
GridBuilder . . . . . 54

## H

Head Limits . . . . . 74

## I

Inactive Elements . . . . . 56  
Inspection . . . . . 95  
Interactive 1D Interpolation . . . . . 125  
Interface Manager IFM . . . . . 185  
Internal Flows . . . . . 155  
Interpolation . . . . . 117

## L

Lifetime Expectancy . . . . . 175  
Lookup Tables . . . . . 119

## M

Map Data . . . . . 36  
Maps . . . . . 29  
Material Properties . . . . . 110  
Mean Age . . . . . 175  
Mean Lifetime Expectancy . . . . . 175  
Measure Tool . . . . . 56  
Mesh Editing . . . . . 55  
Mesh Generation . . . . . 53  
Mesh Import . . . . . 55  
Mesh-Property Check . . . . . 55  
Modulation Function . . . . . 110  
Multilayer Well . . . . . 108

## N

Notation . . . . . 15

## O

Observation Points . . . . . 151  
Oracle . . . . . 30

## P

Parameter Assignment . . . . . 103  
Pathlines . . . . . 155



Péclet Number . . . . .	111
Phreatic . . . . .	73
Planes . . . . .	94
PostGIS . . . . .	30
Process Variables . . . . .	104
Programming Interface . . . . .	185
Projection . . . . .	68

**Q**

Quick Import . . . . .	118
------------------------	-----

**R**

Random-Walk Particle-Tracking . .	157
Raster Maps . . . . .	29
Regionalization . . . . .	117

**S**

Scatter Plot . . . . .	151
Scene Library . . . . .	95
Selections . . . . .	83
Stereoscopic Visualization . . . .	96
Subdomain-Boundary Budget . . .	153
Supermesh . . . . .	43
Supermesh Export . . . . .	45

**T**

Time Series . . . . .	113
Time Stepping . . . . .	69
Time-Series Gap . . . . .	114
Time-Varying Material Properties .	121
Transport Mapping . . . . .	54
Travel Time . . . . .	179
Triangle . . . . .	54

**U**

Units . . . . .	125
Use Parameter Expression . . . .	125
User Data . . . . .	111

**V**

Vector Maps . . . . .	29
View Windows . . . . .	25, 93
Viewer Mode . . . . .	13
Visualization . . . . .	93



**W**

WGEO . . . . . 13

**Numerics**

3D Discretization . . . . . 56

**A**

Advancing Front . . . . . 54

Age . . . . . 175

Anisotropy . . . . . 75

Auxiliary Data . . . . . 111

**B**

Borehole Heat Exchanger . . . . . 108

Boundary Conditions . . . . . 105

Boundary Constraint Conditions . . 109

Boundary-Condition Deactivation . 114

Budget Analysis . . . . . 152

Budget Groups . . . . . 154

**C**

Capture Zone Delineation . . . . . 181

Clipping . . . . . 94

Command-Line Mode . . . . . 13

Constraint Conditions . . . . . 109

Content Analysis . . . . . 155

Convergence . . . . . 144

Coordinate Systems . . . . . 33

**D**

Database . . . . . 30

Dimension . . . . . 68

Discrete Features . . . . . 112

Discretization . . . . . 53

Documentation . . . . . 14

**E**

Element Deactivation . . . . . 56

Error Tolerance . . . . . 70

ESRI Geodatabase . . . . . 30

Exit Probability . . . . . 176

Export . . . . . 37, 157, 176

Expression Editor . . . . . 118

**F**

FEFLOW Book . . . . . 14

FE-LM2 . . . . . 14

FePEST . . . . . 13

FEPLOT . . . . . 13



Free Surface . . . . . 71

## G

Geodatabases . . . . . 30  
Georeferencing . . . . . 34  
GridBuilder . . . . . 54

## H

Head Limits . . . . . 74

## I

Inactive Elements . . . . . 56  
Inspection . . . . . 95  
Interactive 1D Interpolation . . . . . 125  
Interface Manager IFM . . . . . 185  
Internal Flows . . . . . 155  
Interpolation . . . . . 117

## L

Lifetime Expectancy . . . . . 175  
Lookup Tables . . . . . 119

## M

Map Data . . . . . 36  
Maps . . . . . 29  
Material Properties . . . . . 110  
Mean Age . . . . . 175  
Mean Lifetime Expectancy . . . . . 175  
Measure Tool . . . . . 56  
Mesh Editing . . . . . 55  
Mesh Generation . . . . . 53  
Mesh Import . . . . . 55  
Mesh-Property Check . . . . . 55  
Modulation Function . . . . . 110  
Multilayer Well . . . . . 108

## N

Notation . . . . . 15

## O

Observation Points . . . . . 151  
Oracle . . . . . 30

## P

Parameter Assignment . . . . . 103  
Pathlines . . . . . 155



Péclet Number . . . . .	111
Phreatic . . . . .	73
Planes . . . . .	94
PostGIS . . . . .	30
Process Variables . . . . .	104
Programming Interface . . . . .	185
Projection . . . . .	68

**Q**

Quick Import . . . . .	118
------------------------	-----

**R**

Random-Walk Particle-Tracking . .	157
Raster Maps . . . . .	29
Regionalization . . . . .	117

**S**

Scatter Plot . . . . .	151
Scene Library . . . . .	95
Selections . . . . .	83
Stereoscopic Visualization . . . .	96
Subdomain-Boundary Budget . . .	153
Supermesh . . . . .	43
Supermesh Export . . . . .	45

**T**

Time Series . . . . .	113
Time Stepping . . . . .	69
Time-Series Gap . . . . .	114
Time-Varying Material Properties .	121
Transport Mapping . . . . .	54
Travel Time . . . . .	179
Triangle . . . . .	54

**U**

Units . . . . .	125
Use Parameter Expression . . . .	125
User Data . . . . .	111

**V**

Vector Maps . . . . .	29
View Windows . . . . .	25, 93
Viewer Mode . . . . .	13
Visualization . . . . .	93



**W**

WGEO . . . . . 13

**Numerics**

3D Discretization . . . . . 56

**A**

Advancing Front . . . . . 54

Age . . . . . 175

Anisotropy . . . . . 75

Auxiliary Data . . . . . 111

**B**

Borehole Heat Exchanger . . . . . 108

Boundary Conditions . . . . . 105

Boundary Constraint Conditions . . 109

Boundary-Condition Deactivation . 114

Budget Analysis . . . . . 152

Budget Groups . . . . . 154

**C**

Capture Zone Delineation . . . . . 181

Clipping . . . . . 94

Command-Line Mode . . . . . 13

Constraint Conditions . . . . . 109

Content Analysis . . . . . 155

Convergence . . . . . 144

Coordinate Systems . . . . . 33

**D**

Database . . . . . 30

Dimension . . . . . 68

Discrete Features . . . . . 112

Discretization . . . . . 53

Documentation . . . . . 14

**E**

Element Deactivation . . . . . 56

Error Tolerance . . . . . 70

ESRI Geodatabase . . . . . 30

Exit Probability . . . . . 176

Export . . . . . 37, 157, 176

Expression Editor . . . . . 118

**F**

FEFLOW Book . . . . . 14

FE-LM2 . . . . . 14

FePEST . . . . . 13

FEPLOT . . . . . 13



Free Surface . . . . . 71

## G

Geodatabases . . . . . 30  
Georeferencing . . . . . 34  
GridBuilder . . . . . 54

## H

Head Limits . . . . . 74

## I

Inactive Elements . . . . . 56  
Inspection . . . . . 95  
Interactive 1D Interpolation . . . . . 125  
Interface Manager IFM . . . . . 185  
Internal Flows . . . . . 155  
Interpolation . . . . . 117

## L

Lifetime Expectancy . . . . . 175  
Lookup Tables . . . . . 119

## M

Map Data . . . . . 36  
Maps . . . . . 29  
Material Properties . . . . . 110  
Mean Age . . . . . 175  
Mean Lifetime Expectancy . . . . . 175  
Measure Tool . . . . . 56  
Mesh Editing . . . . . 55  
Mesh Generation . . . . . 53  
Mesh Import . . . . . 55  
Mesh-Property Check . . . . . 55  
Modulation Function . . . . . 110  
Multilayer Well . . . . . 108

## N

Notation . . . . . 15

## O

Observation Points . . . . . 151  
Oracle . . . . . 30

## P

Parameter Assignment . . . . . 103  
Pathlines . . . . . 155



Péclet Number . . . . .	111
Phreatic . . . . .	73
Planes . . . . .	94
PostGIS . . . . .	30
Process Variables . . . . .	104
Programming Interface . . . . .	185
Projection . . . . .	68

**Q**

Quick Import . . . . .	118
------------------------	-----

**R**

Random-Walk Particle-Tracking . .	157
Raster Maps . . . . .	29
Regionalization . . . . .	117

**S**

Scatter Plot . . . . .	151
Scene Library . . . . .	95
Selections . . . . .	83
Stereoscopic Visualization . . . .	96
Subdomain-Boundary Budget . . .	153
Supermesh . . . . .	43
Supermesh Export . . . . .	45

**T**

Time Series . . . . .	113
Time Stepping . . . . .	69
Time-Series Gap . . . . .	114
Time-Varying Material Properties .	121
Transport Mapping . . . . .	54
Travel Time . . . . .	179
Triangle . . . . .	54

**U**

Units . . . . .	125
Use Parameter Expression . . . .	125
User Data . . . . .	111

**V**

Vector Maps . . . . .	29
View Windows . . . . .	25, 93
Viewer Mode . . . . .	13
Visualization . . . . .	93



**W**

WGEO . . . . . 13



## Numerics

3D Discretization 40

## A

Advancing Front 38

Age 127

Animation 123

Anisotropy 53

Auxiliary Data 81

## B

Borehole Heat Exchanger 79

Boundary Conditions 76

Boundary Constraint Conditions 79

Boundary-Condition Deactivation 83

Budget Analysis 112

Budget Groups 113

## C

Capture Zone Delineation 131

Clipping 68

Combined Regularization 183

Command-Line Mode 9

Constraint Conditions 79

Content Analysis 114

Convergence 106

Coordinate Systems 24

## D

Database 22

Dimension 48

Discrete Features 81

Discretization 37

Documentation 10

## E

Element Deactivation 39

Error Tolerance 49

ESRI Geodatabase 22

Exit Probability 128

Export 27, 115, 128

Export FePEST Results 185

Expression Editor 86

## F

FEFLOW Book 10



## Index

FE-LM2 10  
FePEST 9  
FePEST File Configuration 173  
FEPLOT 9  
Finite-Element Mesh 37  
Free Surface 50

### G

Geodatabases 22  
Georeferencing 25  
GridBuilder 38  
Groundwater Age 127

### H

Head Limits 52

### I

IFM Implemented 175  
Inactive Elements 39  
Inspection 69  
Interactive 1D Interpolation 91  
Interface Manager IFM 193  
Internal Flows 114  
Interpolation 85  
Interpolation from Pilot Points 175

### J

Jacobian Matrix 190  
JROW2VEC 190

### K

Kriging Configuration 177  
Kriging Interpolation 176

### L

Lifetime Expectancy 128  
Lookup Tables 87

### M

Map Data 26  
Maps 21  
Material Properties 80  
Mean Age 127  
Mean Lifetime Expectancy 128  
Measure Tool 39  
Mesh Editing 39



Mesh Generation 37  
Mesh Import 38  
Mesh-Property Check 39  
Modulation Function 80  
Multilayer Well 78

## N

NOPTMAX 174  
Notation 11

## O

Objective Function 184  
Observation Definition 179  
Observation Points 111  
Observation Weights 179  
Oracle 22

## P

Parallelization 180  
Parameter Assignment 75  
Pathlines 114  
Péclet Number 81  
PEST Parameter Types 178  
PEST Termination Criteria 174  
Phreatic 51  
Pilot Point Generation 176  
Planes 68  
PostGIS 22  
Problem Settings 47  
Process Variables 76  
Programming Interface 193  
Projection 48

## Q

Quick Import 85

## R

Random-Walk Particle-Tracking 115  
Raster Maps 21  
Regionalization 85  
Regularization Setup 181  
Residuals 189  
Results Evaluation 111, 127  
Running PEST 183

## S

Scatter Plot 111



## Index

Scene Library 69  
Selections 59  
Sensitivity 190, 191  
Simulation 105, 135  
Singular Value Decomposition 183  
Stereoscopic Visualization 69  
Subdomain-Boundary Budget 113  
Supermesh 31  
Supermesh Export 33

### T

Tikhonov Regularization 182  
Time Series 82  
Time Stepping 49  
Time-Series Gap 83  
Time-Varying Material Properties 88  
Transport Mapping 38  
Travel Time 130  
Triangle 38

### U

Units 91  
Use Parameter Expression 91  
User Data 81  
User Interface 17

### V

Variograms 177  
Vector Maps 21  
Video 123  
View Windows 18, 67  
Viewer Mode 9  
Visualization 67

### W

WGEO 9

