SPRING

Simulation of Processes in Groundwater

Tutorial

Professional Water Systems Modelling Software.

- Groundwater flow modelling
- Contaminant transport modelling
- Heat transport modelling

SPRING – Brought to you by the Engineering Consulting Firm delta h Ltd.

Authors:

Prof. Dr.-Ing. Christoph M. König,

Martin Becker, Annette Diehl, Torsten Seidel, Dr.-Ing. Britta Rosen, Dr. Otto Rüber, Barbara Werth, Christian Zimmermann,
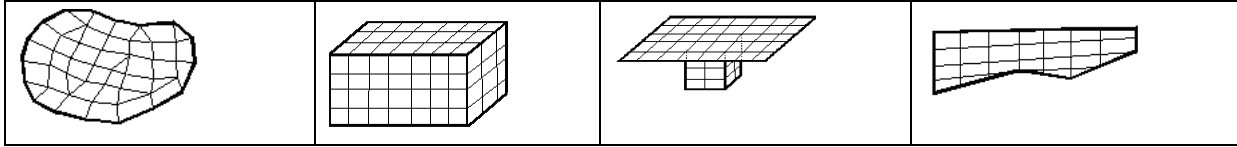
1 Contents

2 Compilation of Different Model Types

The following chapters show the user how to compile different groundwater models containing all the required data step-by-step. Practical examples illustrate the procedures.

The chapter Compilation of a 2D-Model allows the user to compile a horizontal groundwater model containing all the required data step-by-step.

Before you get started with it, you should import the required sample files from our website http://spring.delta-h.de in the menu Download & Support. You find them in the zip-file under: Tutorial (EN): „Tutorial_bsp_files/Tutorial_2d_bsp_files" ( ).

2.1 Compilation of a 2D-Model
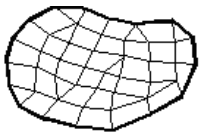
The central files in SPRING are the ASCII model files *name.net* and *name.3d*. The *name.net* file contains all the information for a 2D model or the data of the topmost layer for a 3D model. The *name.3d* file contains the information for the deeper layers. These files are usually created or modified by SPRING interactively. However, they can also be edited using any editor.

During the *model checking*, the model files are read in and written to binary background files. The subsequent calculation modules will read the data required for you from these background files and also save the results there. For the graphic visualisation of the input or results data (*plot generation*), the data are read from the background files and used for generating the relevant plot. This can be followed by having SPRING redisplay, re-edit and print out the plots on a plotter or printer.

The SPRING interface is used for compiling the model, calculating the flow and generating the plot and is

opened by double-click with the left mouse button on the icon   on the desktop.

2.1.1 Step 1: Generating an FE mesh by automatic mesh generation

Compilation of a groundwater model first requires creating a Finite Element mesh. To do so, the module SPRING is started by double-click. By clicking on *File → New*, the following window for creating a new project appears. Basic data (type of model, time unit, scale, extension in x-y direction) can be predefined here. In the example, the predefined settings are applied except for the max. y-coordinate.
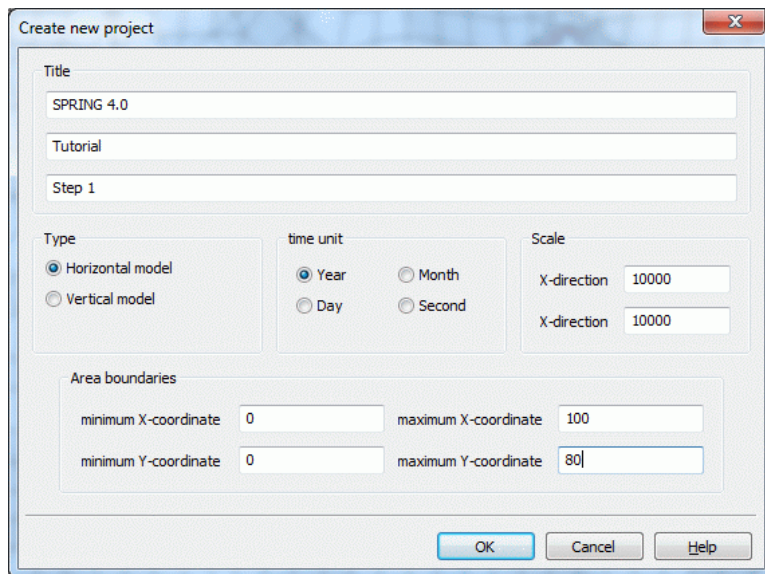
Fig. 1: Creating a new project

In the example, the maximum y-coordinate is set to 80.0 m.

After closing the window by pressing OK and double-click with the right mouse button, the program returns to the surface of the SPRING module. After clicking on *Mesh → Mesh generation → Regular mesh*, a further menu opens, in which the dimensions for the Finite Element mesh to be generated can be entered.
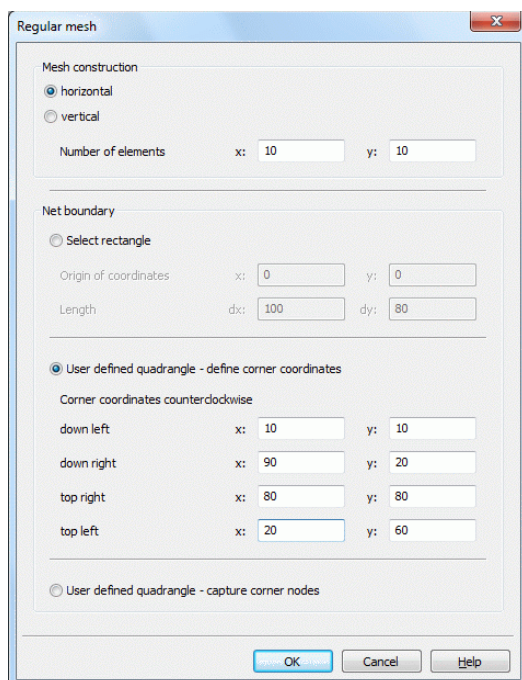
Fig. 2: Generating a regular FE-mesh

In this dialogue the number of elements in x-and y-direction and the direction (horizontal or vertical) is set. Furthermore the mesh boundary can be set in three different ways:

■   Select rectangle:

Here the origin of coordinates and the lengths in the x-and y-direction are to be specified. It is important to ensure that the selected origin and the dimensions are within the selected area when creating the model boundaries.

■   User defined quadrangle – define corner coordinates:

In this method the four vertex coordinates are set. It is important to ensure that the selected origin and the dimensions are within the selected area when creating the model boundaries.

■   User defined quadrangle – capture corner nodes:

Here, the corner coordinates of the FE mesh can be selected interactively in the GUI by selecting with the left mouse button. The corner coordinates are set counterclockwise. This feature is currently not implemented.

Upon applying these settings by pressing the OK button, the following FE mesh is generated, which can be displayed either by clicking on *View → Elements* or by activating the corresponding button ([icon]):
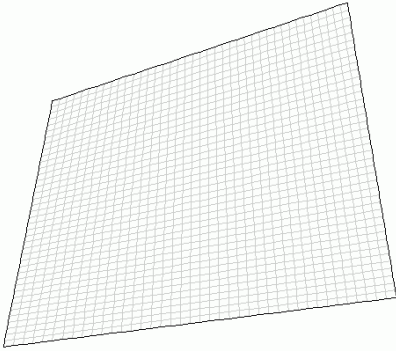

Fig. 3: Automatic generated FE-mesh

During compilation of the model, it is recommended saving the intermediate version from time to time. The first time, the file will receive a name of its own when clicking on *File → Save as* …. During the further course of the project, it is sufficient to save the project.

Now the data assignment (p. 18) would be the next step, but because the automatic mesh generation is insufficient for most hydrogeological problems, step 2 is carried out next: Generation of an FE mesh having irregular constraint geometries

2.1.2 Step 2: Generation of an FE mesh having irregular constraint geometries

2.1.2.1 Description of the investigation area

The area under investigation of the model to be generated looks as follows:
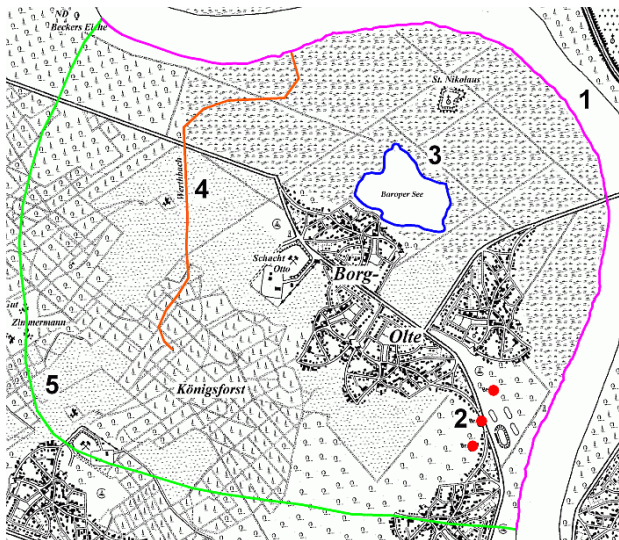

Fig. 4: Area under investigation

The geo referenced tiff file "topo1.zip" and all other necessary files can be downloaded from our homepage: under Download&Support: Tutorial (EN) "Tutorial_bsp_files.zip" ([icon]).

The investigation area can be described as follows:

A large-sized main water course system, into which a smaller-sized water course flows, extends from southeast to northwest. The area under investigation also contains a lake with groundwater contact. There are also three pumping wells that deliver groundwater in the southeast. Since these constraint points affect the local groundwater situation, they must be included in the area of the model.

2.1.2.2 Issues on a groundwater model

A groundwater model is created when a variety of factors influence the groundwater situation and complex boundary conditions are in place and an analytical solution of the problem is no longer possible.

Possible questions for a groundwater model include:

- How is the groundwater flow affected by a change in the withdrawal amounts?
- Which depths of groundwater level does it have?
- What is the impact of subsidences caused by mining on the present depths of groundwater level?
- What are the consequences of a flood wave and how change the flood plains?
- What impact do interventions in the water course have: stream relocation, changes to the stream bed?
- What happens when constructing new production wells and what is their effect on the existing situation?
- etc. etc. etc.

These and many other questions can be answered by a groundwater model.

2.1.2.3 Data retrieval

The following parameters are required as data basis for the following calculation:

Main water course system: Water levels in m (in most cases m above sea level) at least at two points: fixed potential boundary → POTE

Secondary water course system: Water level in upper reach (m), if required, discharge measurements for determination of leakage → VORF and LERA

Withdrawal wells: Withdrawal amounts → KNOT (negative sign, convert in m³/time unit of the mesh file)

Groundwater recharge: Precipitation, soil type, soil use, gradient for calculating recharge → FLAE(mm/time unit of the model file)

Geology: Permeability values in m/s → KWER

Ground level surface: → GELA, and, if required, data on an impermeable layer in the underground → UNTE, otherwise calculation using the thickness

Thickness: →MAEC

Measured values of the groundwater for determining the calibration status: Groundwater surface → EICH

Remark: The time unit of the model file is set when you create a new project (*File → New*).

These data can be delivered by the following sources:

- Ground level elevations and TIFF-files: Land Survey Office, Land Registry
- Groundwater measuring points, levels on water courses: Federal environmental office, water boards, water works, etc.
- Withdrawals: Well operators
- Geology: geological maps, exploration drillings, pumping tests
- Leakage: for example calculation via K values of the surroundings and discharge measurements
- Groundwater recharge: for example on the basis of RVR data, according to Schroeder and Wyrwich or Meßer

2.1.2.4 Determining the model boundaries

Before a new project can be created, the area of investigation boundary of the FE model is determined. For this purpose, first the model boundaries are set.

In the western and southern area under investigation, there is a divide, i.e., a Q=0 boundary can be defined here as boundary of the model. The main receiving water course in the north and east is assumed to another model boundary.

In the chapters that follow, the area under investigation will be reproduced step-by-step in a groundwater model.

2.1.2.5 Creating a new project

As described in Step 1 (regular mesh), for the creation of a new project the minimum and maximum coordinates of the study area are required.

Using *File → Open → Data type selection Bitmap (*.tif)* you open the already saved file "topo1.tif" (directory: „…/Tutorial_bsp_files/Tutorial_2D_bsp_files/topo1.tif").

The coordinates of the mouse pointer are displayed in the status bar of the screen.

The display area of the map can be moved by pressing the middle mouse button. By rotating the mouse wheel can be zoomed into or out of the presentation. Pressing the right mouse button takes you back to the long shot.

The coordinates of the topographic map and the selected model boundary can thus be determined for

 the lower left corner of the map to x = 800, y = 120 and for the upper right corner of the map to x = 7800 y = 6100.

To return to the mode model file SPRING is opened again. In the dialog *File → New* the three comment lines and the coordinates determined for the boundaries are entered:
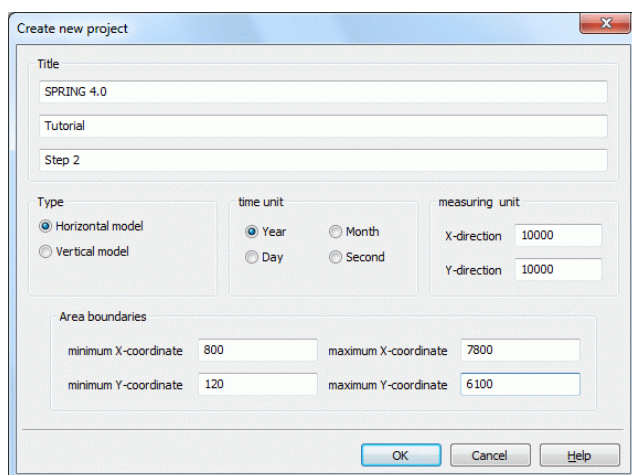

Fig. 5: Coordinates of the new project

After confirming with the OK button the project is created and should be named with *File → Save as….*

2.1.2.6 Creating and importing structures

Normally the user must read in these data him-/herself via a geoinformation system or digitise them on-screen (e.g. via *Structure → New → Points* on the basis of a Tiff-file).

In chapter "SPRING Menues - Structure" in the manual possibilities of creating and importing structures are described in detail.

The necessary model structure data are already digitized and are available in the already downloaded directory under *…./Tutorial_2D_bsp_files/s1_daten1.str*. Via *Structure → Import… →* (file selection) *.str* the file "s1_daten1.str" can be read.
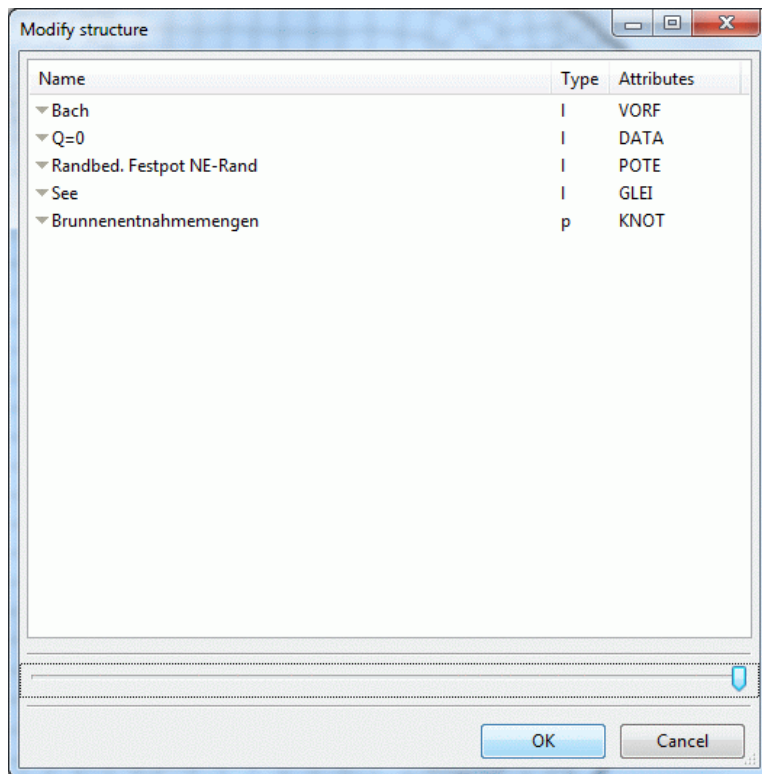
Fig. 6: Imported structures of the file s1_daten1.str

If it does not yet contain any nodes or elements, the project must be saved and opened again, before the structures can be displayed.

After activating the display under *View* → *Structures* → *All* or pressing the corresponding button

( ), the following figure appears:
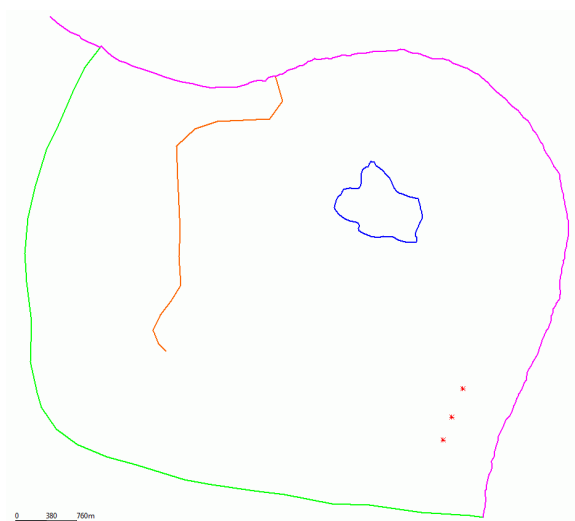
Fig. 7: Existing structure data

2.1.2.7 Modifying structures

Some structures have been data obtained which get lost by a processing (e.g. delete points of the structure).

To prevent this all structures are copied (*Structure* → *Copy*) which should be used later as contours and which possess structure points outside the model boundary.

It is recommended to rename the copied structures (*Structure* → *Edit*) and to add the words "for contours" in the information text.

The copied structures can now be edited (*Delete points, Move or Add*), without losing valuable data.

Via *Structure → Edit → Pick* or activating the corresponding button in the symbol bar (🖐) the desired struc-
ture can be selected. First, the northeastern model boundary (pink) is selected. The following dialog appears:
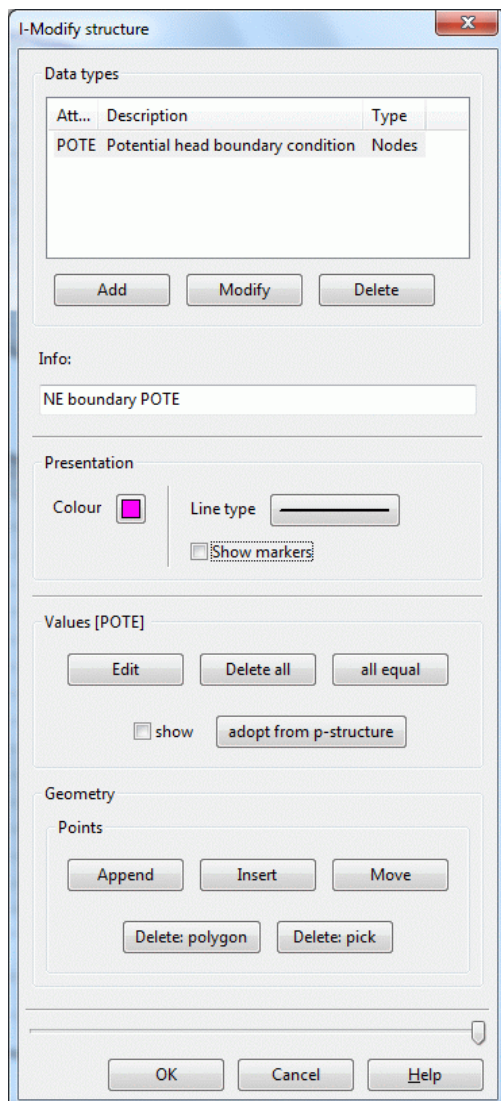

Fig. 1: Modifying l-structures

After *Show marker*, the superfluous structure points at the western end of the structure can be eliminated via
the following items: *Delete: Pick or Delete: polygon.* The actual deletion of the points will not take place until
the structure menu is finished by pressing OK.

You should also check whether the endpoints of the north-eastern and south-western model boundary coin-
cide. If this is not the case, the endpoints can be moved toward each other by using the menu item *Points:
Move* of the *Structure* menu. In the selection window, you will be asked whether the point is to be moved
"free" or onto a "Point". Select "Point".

In this menu, it is also possible to append or insert points into the existing structures. This is required for the
watercourse located within the model (orange). This structure should be attached to another point that lies on
the main water course. In the structure of the main water course should then be a further point inserted at this
point (Capturing mode: Point).

2.1.2.8   Generating contours

All input data are kept available in the structure data. Not all structure data are automatically constraint geom-
etries for the FE mesh (e.g. measuring points or geological explorations outside the area under investigation).
Only all constraint geometries of the structures are transferred. Contour editing is the basis for a homogene-
ous FE mesh. The individual structure points can be visualised while they are still in the Structure menu by
clicking on *Marker → Show*.

Initially not all structure points of the north-eastern boundary should be transferred as contour points.

This is why the distance of the contour points is increased by clicking on *Contour → New → From structure* and then entering the distance in the input window that appears:
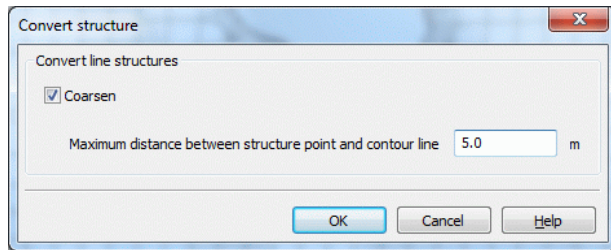

Fig. 8: Converting structures into contours

The maximum possible distance to the original boundary is set to 5 m. To generate a regular distribution of the boundary nodes, the distance of the contour points set to 300 m by clicking on *Contour →Edit l contour → Define maximum distance → Polygon.*
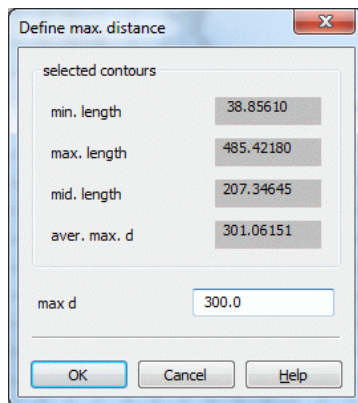

Fig. 9: Setting the maximum distance

This corresponds approximately to the distance of the structure points of the south western boundary. The same procedure is recommended for the lake, although in this case a smaller distance of the contour points should be selected (approx. 35-50 m), in order to obtain later on a smaller element edge length.

The remaining structures are included in the list by clicking on *Contour → New → From structure → Capture / List.* When selecting the list, you can also convert several structures simultaneously by pressing the CTRL key and selecting the desired structures simultaneously.

At the south-eastern structure boundary and at the water course, the distance of the points can be reduced for individual contour elements. This is done via *Contour → Edit l contour → Define maximum distance → Capture.* This will select the contour elements that seem "too large" to the user. An input window appears in which the desired maximum distance can be entered.

At the point where the water course flows in the main water course system, the contiguous contour of the main water course must be divided once (*Contour → Edit l contour → Split into n parts: 1*), to ensure that the contour point of the water course flows in at the division point (*Contour → Move points*).

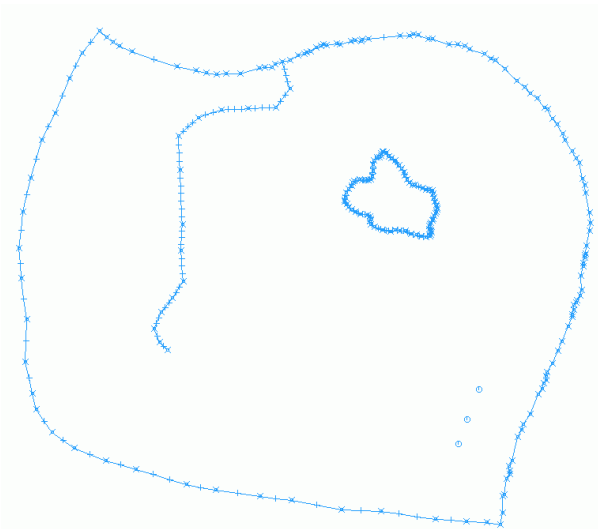This gives the contours shown in the following figure:

Fig. 10: Existing contours

For better visualisation of the logarithmic cone of depression around a production well, the contour points of the wells must be assigned suitable well parameters. This prevents oscillations caused by local changes in potential in the proximity of the well. After *Contour → Edit p-contour → Well parameters*, the following input window appears:

Fig. 11: Setting well parameters

For a detailed description of the parameters, please refer to chapter: Data structure of the groundwater model: Local discretisation: Special case: Well discretisation.

Before node generation, you have to check whether the contour boundary is closed, and the distances of individual contour points are not too small. This is done via *Contour → Optimise*. The setting of the accuracy parameters is confirmed by pressing "OK". If the check is free of errors, the comment "Boundary closed" appears at the bottom edge of the graphic interface. If errors occur, they are displayed and must be eliminated by changing the accuracy parameters.

Note: For these contours as well, the following applies: When the project is reopened, the contours must be displayed by clicking on *View → Contours* or the corresponding button (⊞).

2.1.2.9   Generating nodes

In node generation, first nodes on and in the area of the contours are generated by clicking on *Mesh → Mesh generation → Generate boundary nodes → Global*. The number of passes establishes how many node rows are to be generated in parallel to the contours. 2-3 passes are recommended. Selected in the example: 3 passes.

The contour-free areas are then filled with regular meshes by clicking on *Mesh → Mesh generation → Generate Raster → Global*. The approximate node distance is defined in the following input window:
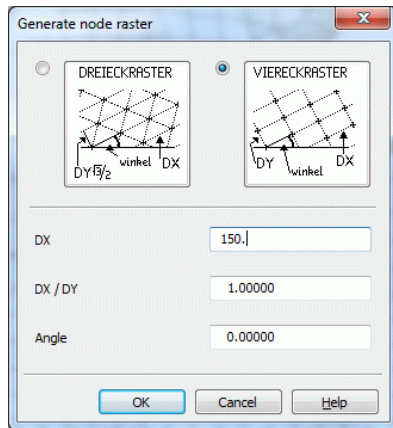
Fig. 12: Generating raster nodes

In the *DX* box, the average node distance of already generated nodes appears as default. It can be modified by the user. In the example, DX is set to 150.0 m. Next the generated regular mesh is displayed.

You now have to check whether the position of the individual nodes produces a homogeneous pattern. Nodes can be added to voids (*Mesh → Mesh generation → Insert nodes*), too many nodes can be deleted (*Mesh → Nodes → Delete*) or nodes can be combined from a defined distance by clicking on *Mesh → Node → Merge → According to distance*.

Note: To zoom, the current function (e.g. "Insert nodes") is finished with the right mouse button. The desired zoom level is drawn with the left mouse button. Press the F12 key, to reopen the current function. (This new feature applies to all editing functions!)

2.1.2.10 Generating elements

Automatic generation of elements is done by clicking on *Mesh → Mesh generation → Generate elements → Triangles and quadrangles*. To release the mesh, for example, twenty times, in order to obtain optimum angles between the element sides, press *Mesh → Unbend ….*
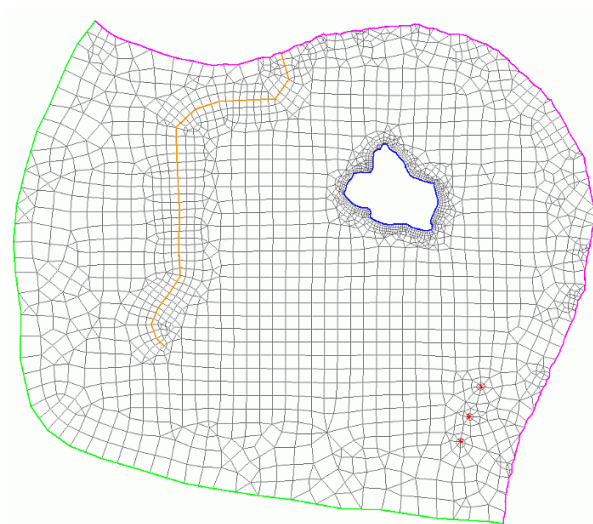
Fig. 2: Finished FE-mesh

In the above picture, for the sake of clarity, nodes and contours have been hidden (*View → Node* or *Contours* or press the corresponding buttons).

The markers on the structures were deleted by pressing *Structure → Edit → (→ Click on structure)* Show *Marker* or by deactivating the Marker button.

Finally the nodes and elements inside the lake are deleted via *Mesh → Nodes → Delete → Area*, because the lake is treated as an internal boundary in this model.

The last step in mesh generation is to carry out mesh controls (*Mesh → Checks…*) and eliminating the errors that may have occurred, before step 3: Data assignment is started!

The finished mesh and the structure file are available to download on our or was already downloaded (…/Tutorial_bsp_files/Tutorial_2D_bsp_files/schritt2_tutorial.zip). It can be used for further processing.
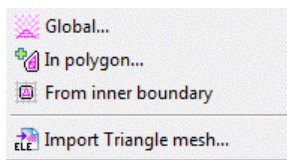
2.1.2.11 Using the external TRIANGLE mesh generator

It is also possible to generate a mesh by means of the external Triangle mesh generator. The advantage of the mesh generator is that the user can predefine, for example, the minimum area or the minimum angle of a triangle element, in order to comply directly with form factors for the element quality (ratio of internal circle to external circle).

Before that, the program Triangle must be installed on the computer (http://www.cs.cmu.edu/~quake/triangle.html).

Since Triangle is not integrated directly into SPRING, SPRING must first create input files for Triangle and then start it as an external program. The output generated by Triangle must in turn be read in by SPRING. This will overwrite the existing mesh information.

To open the relevant dialog, go to the menu item *Mesh → Mesh generation → Triangle mesh generator*: Another menu opens:

With the help of the menu item "Import Triangle mesh" a mesh (*.ele) generated with the external mesh generator Triangle can be imported.

After selecting one of the first three menu items (Mesh generation globally, in polygon or from an inner boundary) opens the associated dialog:
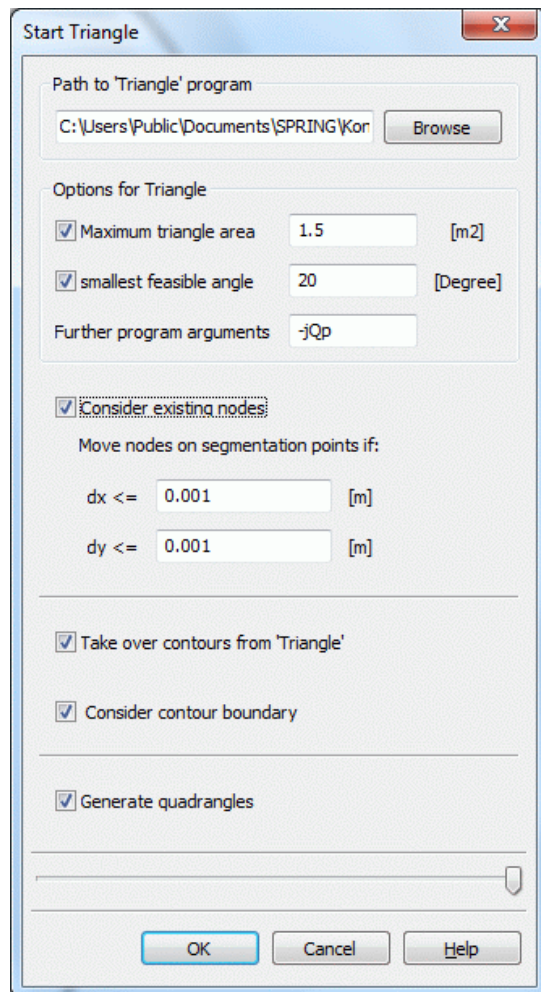
Fig. 13: Dialog for Triangle mesh generator

"Path for the program Triangle":

Here the path is selected under which Triangle has been installed. During the SPRING installation it is installed in the folder "C:\Users\Public\Documents\SPRING\Konfig" by default.

Note: In the operating systems Windows Vista or Windows 7, the file "triangle.exe" must NOT be present in the folder "C:\Programs\SPRING\bin", as there is no writing authorisation in this folder.

"Maximum triangle element": (Triangle Option "-a")

This option makes Triangle generate additional nodes, preventing the generation of triangles whose area is larger than the specified value.

"Smallest feasable angle": ()

This option makes Triangle generate additional nodes, preventing the generation of angles smaller than the specified value. The default is 20°. This option generates high-quality meshes, but can result in Triangle not being finished correctly.

"Further program arguments": ()

Triangle offers many more options of influencing mesh generation. The corresponding parameters for loading Triangle can be entered here.

"Consider existing nodes":

This writes the existing nodes as additional constraint geometries into Triangle's input file. Thus, for example, first a regular mesh can be generated using SPRING's regular mesh generator, which is then meshed by Triangle.

"Take over contours from Triangle" and "Consider contour boundary":

If Triangle generates additional nodes, in order to comply, for example, with the angle criterion, these new points will also be generated on the constraint geometries below (contours). The option "Accept contours from Triangle" makes SUSI, after a successful Triangle run, read in not only the mesh geometry, but also overwrite the existing contours with the one modified by Triangle.

"Generate quadrangles":

This will start SPRING's quadrangle generator, following a Triangle run.

2.1.2.12 Inserting a new structure in an existing model

In this chapter, a concept for later incorporation of a new structure in an already existing mesh is presented to the user.

Before starting with the edition of the mesh geometry, it is advisable to make copies of the original mesh and structure file. All attribute data affected by deletion of an element area should be saved by clicking on *Attributes → Export*. A list of all available attribute data is displayed. By selecting the individual attributes, they can be saved as independent "KENN.str" files and then assigned again later on to the modified mesh (interpolated).

Attention! When working with the 3D model, do not forget the data of the 3D file!

After the preliminary work, the stepwise conversion of the model file is started.

Generating the new structure:

Structure → New → Lines (at the water course or sheet piling wall)

To visualize a sheet piling wall, it is possible to automatically generate an identical second structure in parallel to the first structure. The distance between the two structures is freely selectable. To do so, select the structure under the menu item *Structure → Generate parallel → Catch/List* and determine the side on which the parallel is to be generated with the left mouse button (for area structures, click it on the inside or outside). An input window appears in which the desired distance is entered:
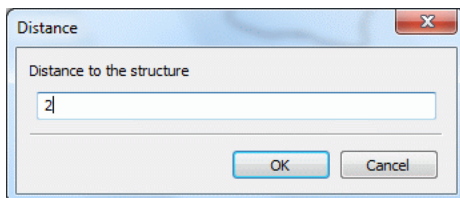

Fig. 14: Distance between parallel and structure (m)

After confirmation with right mouse click, the structure appears in the mesh.

The following picture shows a model detail of the new structure along with its parallel (elements and nodes have already been deleted):
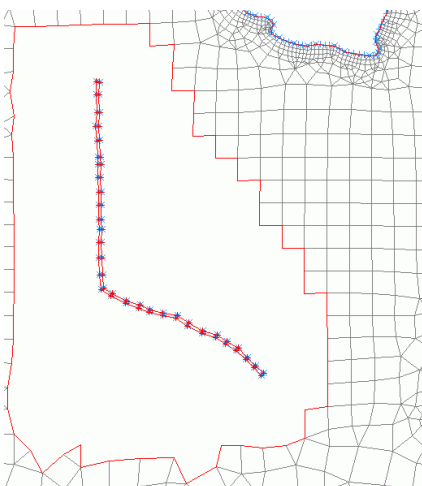
Fig. 15: Model with new structure and already assigned contour (blue)

(To make the parallel visible, a distance of 20 m was entered here.)

Deleting elements in the wide-space area of the new structure:
Mesh → Nodes or Elements → Delete → Area

To apply the new structure as contour:
Contour → New → From structure → Capture

Generating new mesh nodes:
Mesh → Mesh generation → Bitmap generation→ Area: *Catch mesh nodes:* Here it is important to catch the nodes of the existing boundary! The node distance dx should be approximately equal to the existing side length of the elements (in the example, dx = 120.0 m). For neat element grouping, some nodes are inserted manually.

Note: If a closed contour boundary is present, nodes can only be generated within this contour boundary.

Example: An area in an existing model is deleted, for example, to insert a new building. For this building, a closed contour will be created. Now, it is only possible to generate nodes within this contour. Only if the contour is deleted, nodes can also be generated outside the building.

Generating new elements:
Mesh → Mesh generation → Element generation→ Triangles and quadrangles from inner edge: The area in which new elements are to be generated can be selected immediately by a single mouse click.
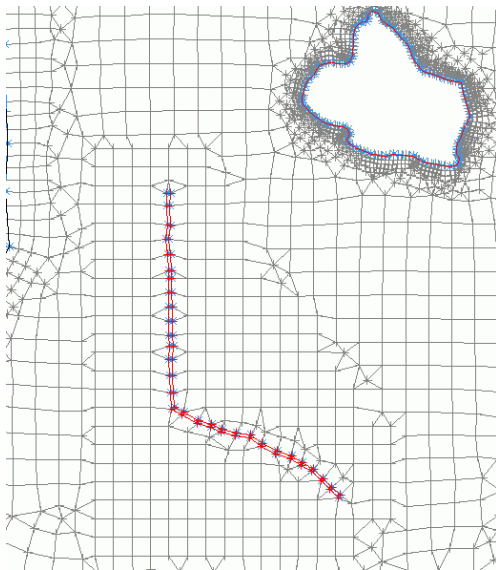

Fig. 16: New nodes and elements in the area of the new structure

After completion of the mesh, geometry checks have to be carried out, and the data must be saved!
In the area of the newly generated elements, gaps have been formed in the database.
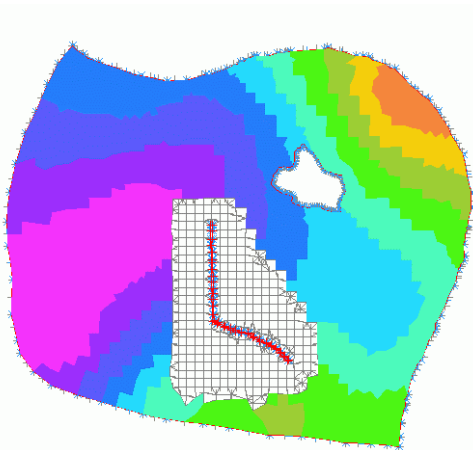
Fig. 17: Missing data in the area of the new elements, using the attribute "UNTE" as an example

The missing data must be completed by new assignment of the attributes.

Attributes → Assign → By interpolation: In the case of large data volumes the saved attribute files must be reduced to the area in question before the new assignment. This is done via *Structure → Import → *.txt*, allocation of the columns and attribute code, and then *Structure → Edit → Delete points:* in the area. Pressing the "Invert selection" button in the Catch mode is sufficient for selecting the area to be assigned, while the remainder will be deleted.

When prompted for the interpolation algorithm, you should select "Gauss", if possible, "Include existing data" and "Filter data with small distance".

Note:

If a local mesh modification has changed the value of only a few nodes, the following procedure is recommended:

If the existing node data of the unmodified nodes are sufficient for interpolation, interpolation without an ASCII file and without additional data from structures is in general only sufficient with "Include existing data". Even when the number of nodes is large, using the Gauss algorithm is suitable for the node data, since prior to interpolation all existing node data that are not in the vicinity of the nodes to be interpolated will be deleted from the interpolation points.

If, due to local mesh modifications, element data (e.g. recharge rates or permeabilities) that are constant within areas must be assigned new values, the following procedure is recommended:

The values can be interpolated without using further ASCII files and without additional data from structures only if the option "Include existing data" is used. By using distance weighting, setting a very large search radius and setting the number of minimum and maximum interpolation points to 1 (exactly one interpolation point), you can ensure that the corresponding elements receive the value of the element assigned with data closest to them.

2.1.3 Step 3: Data assignment

In chapter „Data structure of a groundwater model – Description of the data types" of the manual all attributes of a SPRING groundwater model are described. To start a flow calculation, at least the following data are required:

- Permeability values for all elements: attribute KWER [e.g. 0.0001 m/s]
- Thicknesses for all elements: attribute MAEC [e.g. 10 m]
- Fixed potential head at minimum 2 nodes (potential slope): attribute POTE[e.g. 5-1 m NN]

2.1.3.1   Direct assignment of existing structure data

After completion of the FE mesh, the required model data are assigned. They include, for example, the fixed potentials as boundary condition of the 1st type (POTE), water course potential levels and leakage coefficients on the courses of streams (VORF and LERA), groundwater recharge rates (FLAE), permeability values (KWER), withdrawals from wells (KNOT) or areas with identical potentials (GLEI).

Please note that data required in the entire model area, such as permeability values, lower boundaries or ground level elevations, must be assigned by interpolation, to ensure that all elements or nodes are completely assigned with values.

Data required only locally or in the form of lines, such as leakage coefficients, withdrawals from wells or fixed potentials at the model boundary, can be assigned directly.

The existing structure data file *schritt2_netz.str* contains the data types POTE, VORF, KNOT and GLEI, which are assigned by the following procedure:

Upon clicking on *Structure → Assign → List* or *Attributes → Assign → From structure…→ List*, the following menu appears:
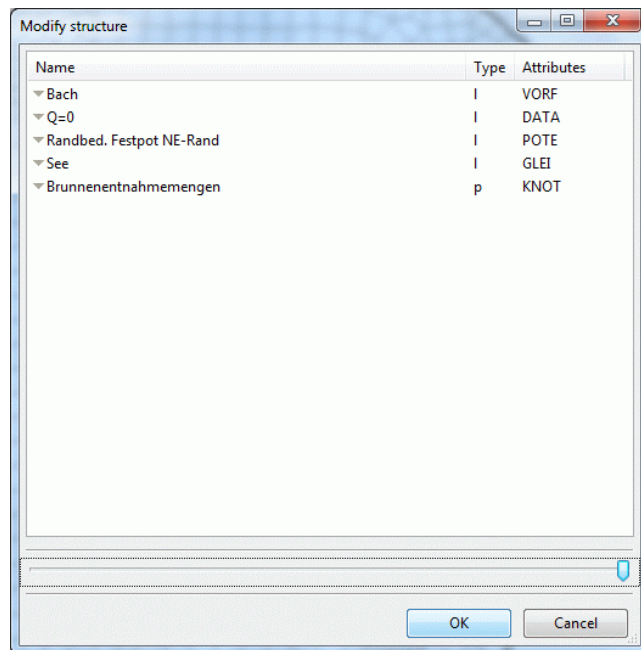

Fig. 18: Assigning structure data

Once all structures have been selected by left mouse click and pressing the Up key and confirmed by pressing "OK", another menu appears in which the accuracy parameters of the assignment can be defined. (Note: The data type DATA is always ignored during the assignment.)
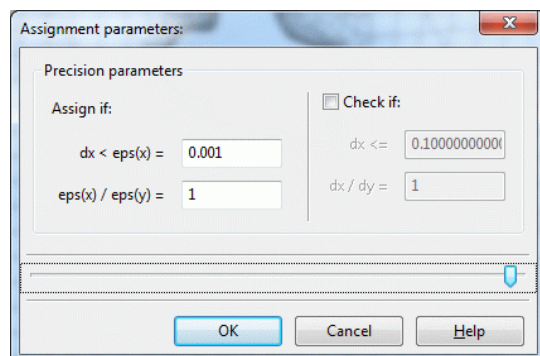

Fig. 19: Assignment parameters

If no further entries are made, the data will be assigned, after pressing the OK button. To click on individual nodes for which the attribute POTE appears at the northeastern edge, the attribute VORF appears at the nodes of the course of the stream, the attribute GLEI appears at the nodes of the lake and the attribute KNOT appears at the three wells, use the menu item *Attributes → Edit nodes → Capture.*

Upon visualisation of the potentials under *View →Show Attributes → Isolines/area plots/values*, using the "Circles" display type,

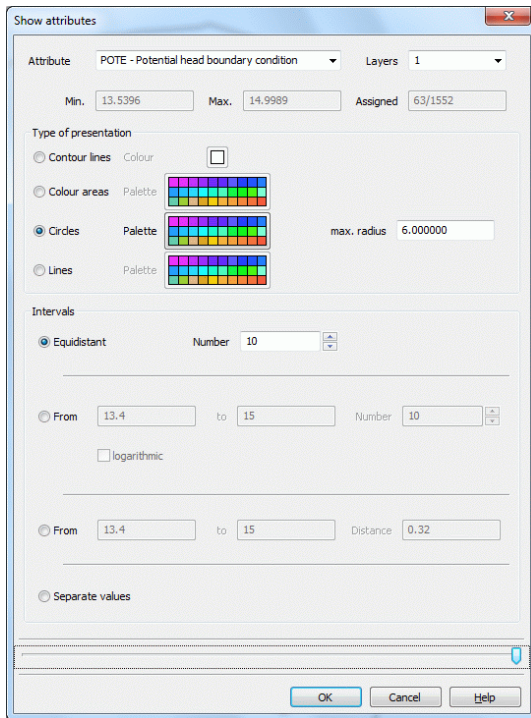Fig. 20: Display of attributes

it becomes apparent that not all nodes of the north-eastern edge have a potential value. This is due to the accuracy parameters of the data assignment!
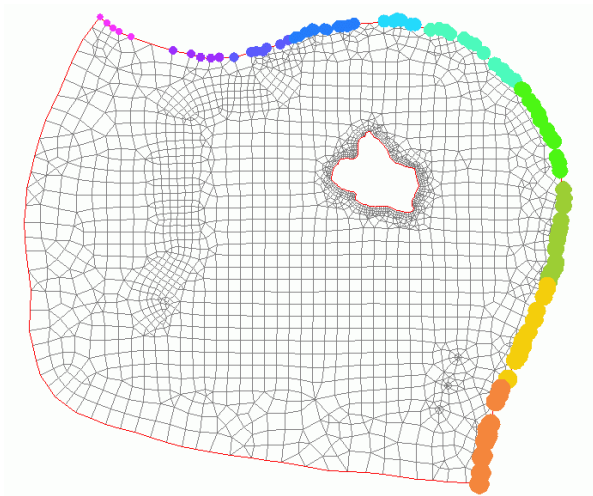

Fig. 21: Display of the assigned (not yet complete) potentials

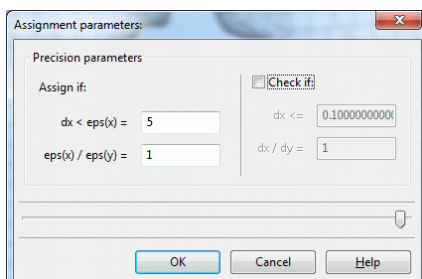Reassignment of POTE from the structures (*Structure → Assign → List)* with the following parameters:

Fig. 22: Entering the accuracy parameters

and confirmation of the prompt "Overwrite all" assigns all nodes of the north-eastern boundary a potential value! (For control, the display can be repeated as described above.)

For the GLEI data type lake boundary nodes, the completeness of the assignment should also be checked. By default, GLEI cannot be displayed with circles. Using the detour via a different data type (e.g. 1KON, KKKK), it can be displayed in spite of that. Use *Attributes → Copy → Attribute-wise* to copy GLEI, for example, to KKKK and then display it. After control, the auxiliary data should be deleted again.
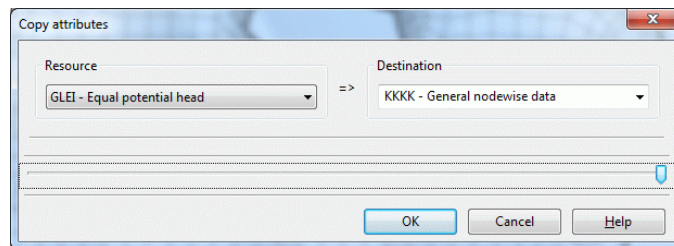

Fig. 23: Copying attributes

2.1.3.2   By interpolation of external data formats

It is also possible to read structure data from "non-project" files (external format") into an existing project. These files do not have to be present in the *.str format. Different formats can be read in by clicking on *Structure → Import ... → .str, .txt, .csv, .arc, .shp*. In the example, the structure data file s3_daten2.str was selected. (It was already downloaded from our).

By saving the project (if desired, under a new name), the "old" and "new" structure data are written to the existing structure data file.

The new displayed structures are: Depth data of the lower edges of the model (UNTE), a MARK of the course of the stream and of the line-related leakage values of the course of the stream (LERA). Whereas the attributes MARK and LERA can be assigned directly as described in Direct assignment, the attribute UNTE must be interpolated in the present example element by element.

The entries for the individual interpolation methods are described in detail in chapter Calculation - Interpolation.

Via *Attributes → Assign → By interpolation*, the following menu appears:
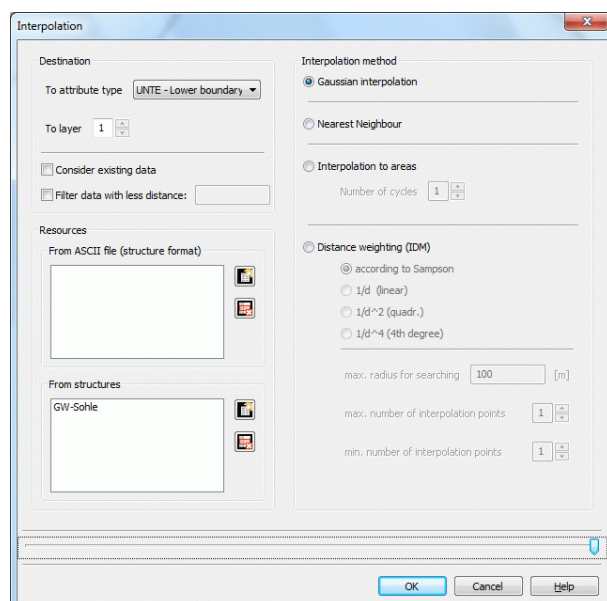

Fig. 24: Interpolation of element data

When selecting *Sources*: "*From structures*" *Add entry* (button), another selection menu appears in which all existing structure data are listed. "UNTE" is selected. The results data type selected is also "UNTE". After pressing the OK button, the data will be interpolated and, after successful calculation, the following control window appears:
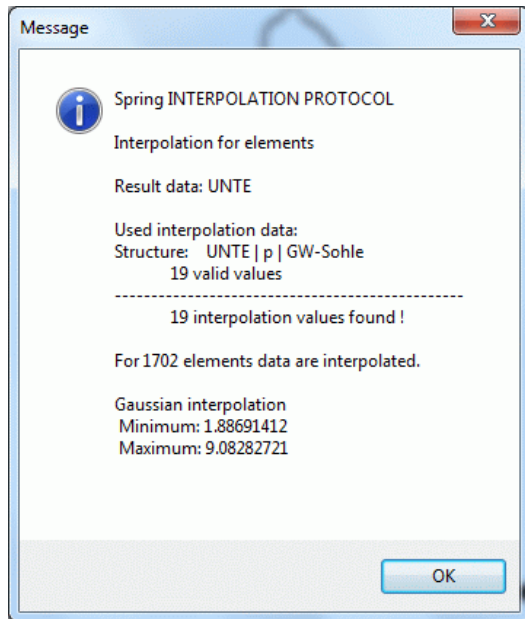
Fig. 25: Gaussian-Interpolation

The interpolation of the calibration potentials must be performed with particular care, since the calibration potentials are one of the most data types for calibrating the groundwater model. The groundwater measuring points in the area under investigation usually contain measured values at the time of calibration which have been saved to a file. Information regarding water levels of the main water course (POTE) and of the secondary water course system (VORF) is also known. These three data types are interpolated to give a calibration state which, after its first display, must be revised manually.

An example of a required revision are interpolated groundwater isolines which show infiltration into groundwater where there is none. This situation is produced when the bed height of small secondary water course systems is set as the water course height, even though they are not carrying any water at all. In this case, the water course heights at the nodes for interpolation must be deleted.

To include the attributes VORF and POTE (they were already assigned in the section Direct assignment), they are transferred from "VORF" or "POTE" to "EICH" via *Attributes → Copy → Attribute-wise*, in order to incorporate them in the interpolation as "existing data".

The menu for the interpolation of node data is available via *Attributes → Assign → By interpolation*. There the file s3_gwmesspkte_neu.txt (download on our !) is selected under *Sources: "From ASCII file" Add entry*. In addition, the menu item "*Include existing data*" is activated. This will include the already existing data under data type selected as results data (here EICH) in the interpolation points.

Here the results data are the calibration potentials (EICH). The data type and, for a 3D project, the layer number are defined under which the interpolated are data to be saved. The data type may differ from the codes of the structure data that may have been selected for interpolation!
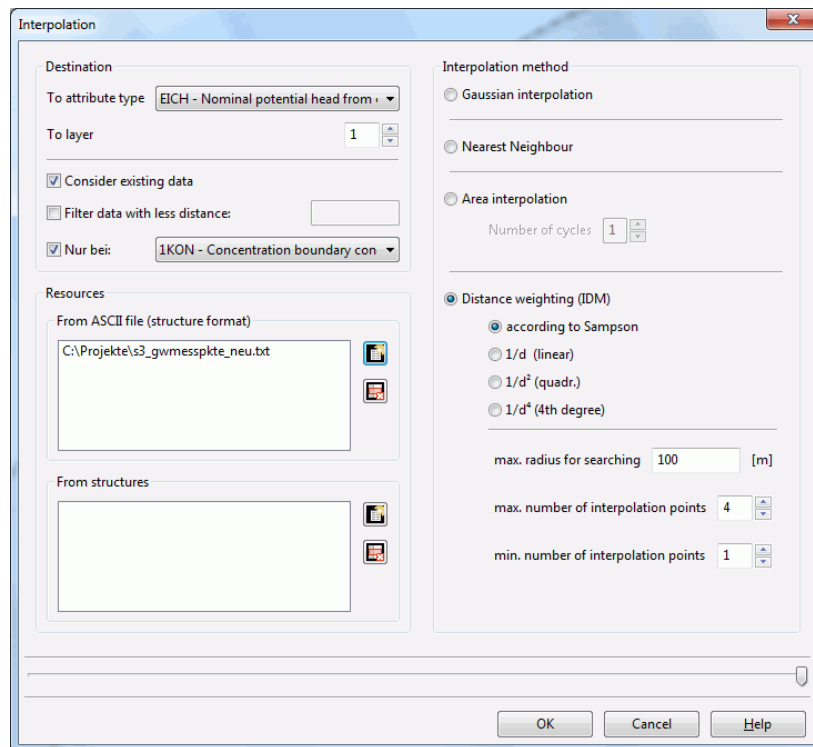
Fig. 3: Interpolation of node data

When the Gauss interpolation algorithm is selected, the following must be taken into account:

The Gauss algorithm does not allow interpolation points with identical coordinates. This can be excluded by checking the check box "*Filter data with smaller distance*". This will check the distance between all interpolation points for the minimum distance entered in the text box. If two interpolation points have a smaller distance, the second interpolation point will not be used. First the existing data, then the structure data according to their sequence in the list and then the data from the ASCII files will be checked.

By selecting the check box *only by* node data can be interpolated in the area in which the selected data type is assigned.

Moreover, the measured values and the position of the measuring points can be displayed at any time in the following input window by clicking on *File → Import → Overlay file → *.txt*.

First, the "Prev" button is clicked. At the bottom of the dialog then the columns are selected. In addition, a marker can be selected (type, height and colour).
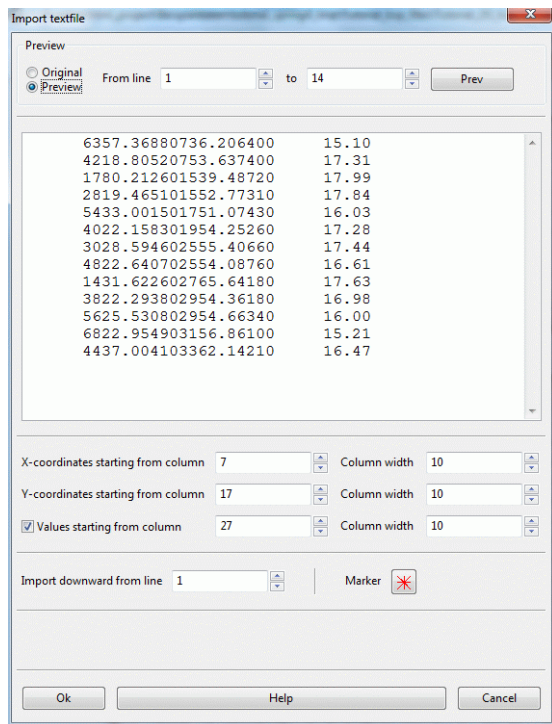
Fig. 26: Importing a text file

After successful interpolation, the calibration potentials (EICH) can be displayed via *View → Show attributes → Isolines/Area plots/Values* in the following form (Isolines in interval 12.50 to 20.0 and distance 0.5 m):
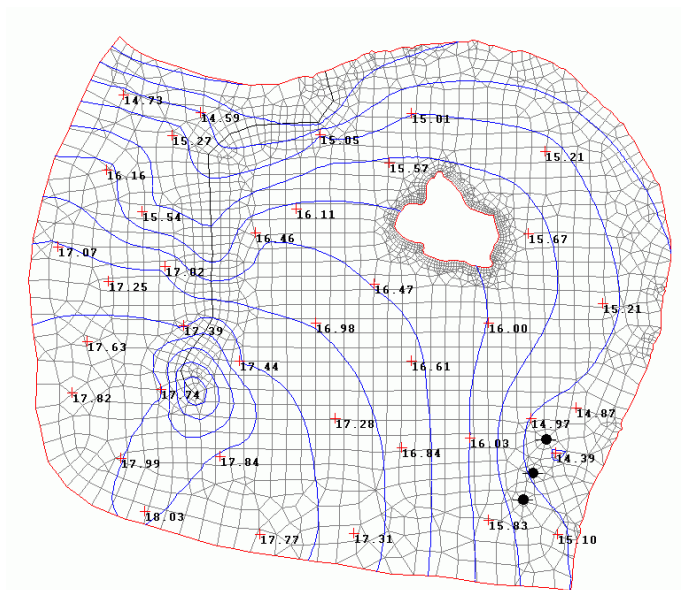
Fig. 27: Interpolated initial potential heads, contour lines with a distance of 0.5 m

In the upper reach of the stream, the previously described, but not really present, groundwater infiltration of the stream can be seen, which is made visible by means of a "groundwater elevation". For calibration, the data were processed as follows: At 17 nodes of the upper reach of the water course system, the water course system potentials (VORF) are deleted, and the interpolation of the calibration potentials (EICH) described above is carried out again.

Fig. 4: Interpolated initial potential heads considering the real groundwater situation

This is followed by reassigning the complete water course potentials (VORF) stored under an auxiliary attribute (e.g. KKKK).

Due to the low data density in the area of the pumping wells, their groundwater depletion is not reflected in the curve of the isolines.

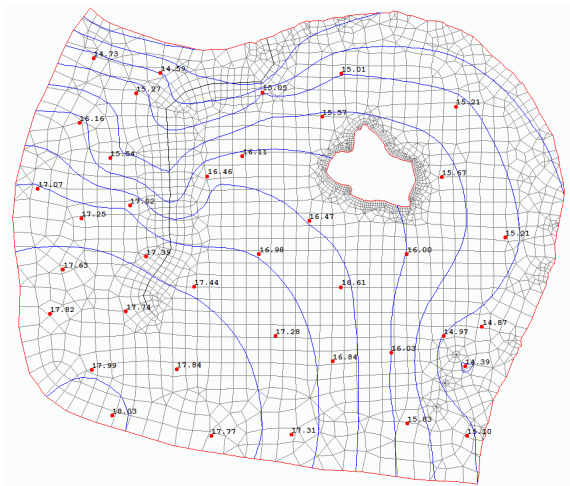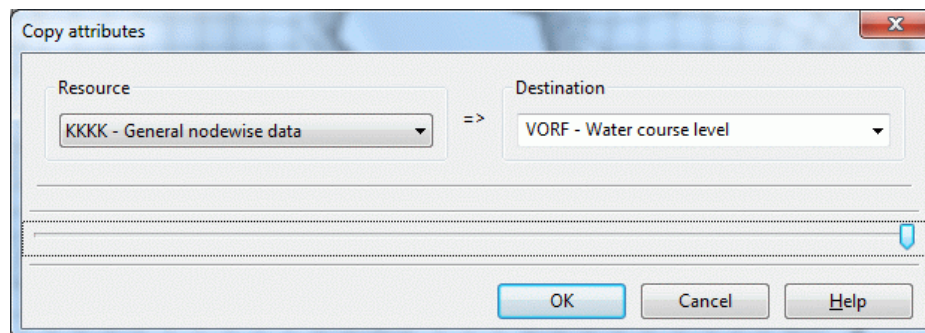Moreover, the ground level elevation must also be assigned. In most cases, ground level elevations are available as ASCII data records, which first have to be read into the structure data file: here "s3_gela.txt (zip)" (download on our and unzip!).

The data are again assigned via *Attributes → Assign → By interpolation, Sources: From ASCII file: .txt*.

At this point, the exact position and length of the columns for the X, Y and Z coordinates must be entered as specified in the file. After selecting the data type "GELA" and the interpolation method, the data will be interpolated to the nodes. Due to the data volume of a DGM grid (Digital Ground Model), it is recommended selecting distance weighting for interpolation (e.g. search radius = 100.0 m, max. number of interpolation points = 4, min. number of interpolation points = 2).

Attention: In distance weighting, you have to ensure in the interpolation protocol that all FE nodes receive a value. If not, the "search radius" must be increased.

2.1.3.3 Manual attribute assignment

For an easy overview, the marker "120" (typical well marker) is assigned to the pumping well as follows:

The withdrawal nodes can be displayed via *View → Show attributes → .../Values → KNOT* and assigned the corresponding mark via *Attributes → Assign → Direct: MARK = 120.*

For the subsequent flow calculation, the groundwater recharge (FLAE) and the permeabilities still have to be assigned. These data types can be assigned globally to all elements by clicking on *Attributes → Assign → Direct*. In the example, the groundwater recharge for the entire area is set to FLAE = 0.200 with the added character " * ", which corresponds to a recharge rate of 0.200 m^3/m^2/time unit or 200 mm/year.
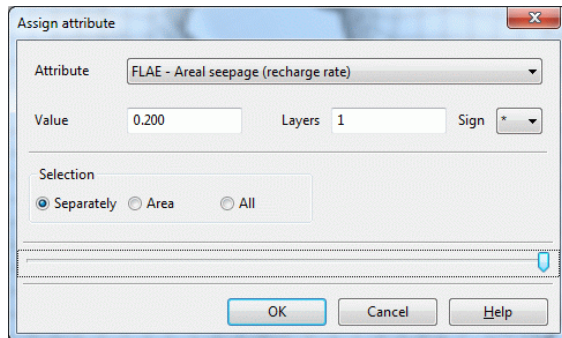
Fig. 28: Direct attribute assignment

The same is now done for the definition of the permeability values (KWER) of the model area, but without additional character in the 15th/16th column. The starting K values before starting calibration are set to 0.00005 m/s. In addition, an upper (KMAX = 0.002 m/s) and a lower (KMIN = 0.000001 m/s) limit for calibration are introduced for the K values, which must also be assigned to all elements.

The assigned marks can be displayed via *View → Markings*, or the corresponding button ( ) can be pressed.

The mesh generated up to this point, including the structure data file, was already downloaded from our in the directory: ".../Tutorial_bsp_files/Tutorial_2D_bsp_files/s3_schritt3_3.zip".

2.1.4 Step 4: Calibration of a groundwater model

2.1.4.1 Visualisation of the initial state

If the FE mesh has been generated and the data assigned, a model inspection has to be carried out prior to calibration (and each time data of the mesh file are changed). This is done via *Calculation → Model checking.* For quicker processing, the model data are stored in so-called background files (aaa, bbb, ccc).

2.1.4.2 Generating a model data plot

Following the model inspection, plots can already be generated from the model data. To illustrate the previous model version, the calibration potentials (EICH) can be displayed in an isoline diagram. Upon clicking on *File → Plot generation → Top view/Map generation*, the following input window appears:

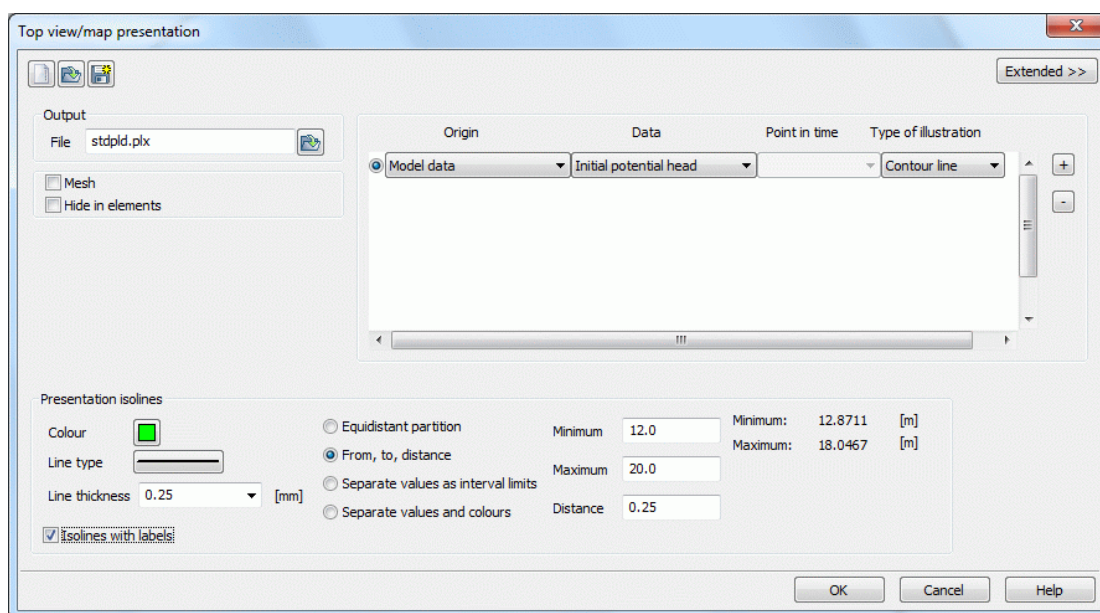Fig. 29: Plot generation

After selecting the data type *Model data → Initial potential heads (EICH) → Contour lines,* the display *From to at a distance:* 12.0 - 20.0, 0.25 m, *Colour*: green is selected. Then a name is entered for the plot

("eich_intpol.plx"), and the calculation run is started. To view the plot, press *File → Open: *.plx*. The following picture appears:
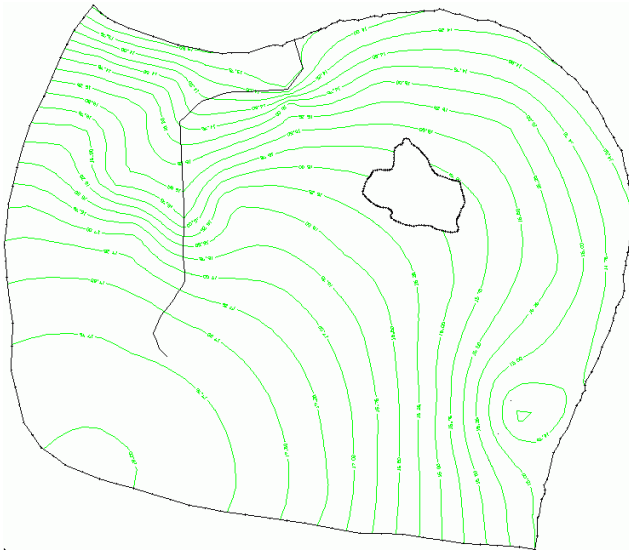
Fig. 30: Contour lines of the interpolated initial potential heads

2.1.4.3 Calibration by the gradient method

First, the existing (starting) K values are used to start an iteration run: *Calculation → Model calibration*. The following dialogue appears:
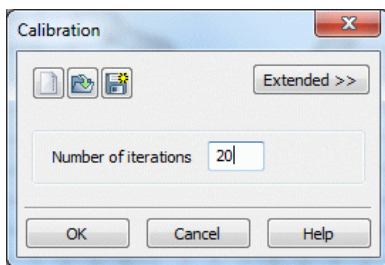
Fig. 31: Calibration

After opening „Extended" further settings can be made:
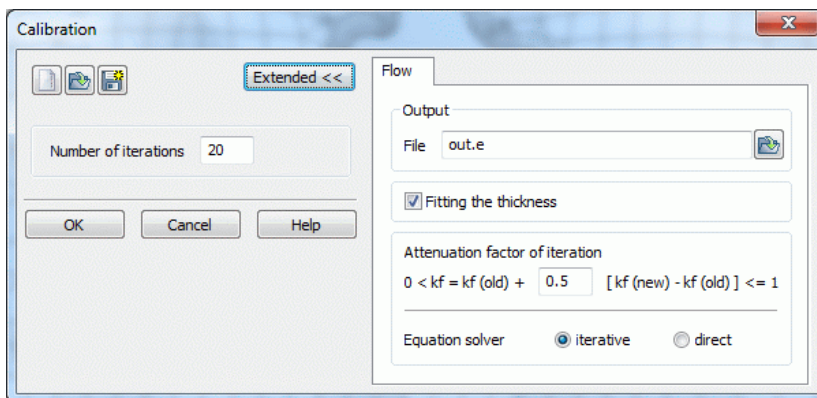
Fig. 32: Extended parameters

After the calculation run, the isolines interpolated from the measured values are contrasted with the isolines obtained from the K values.

To do so, the model data type EICH and the resulting potentials are displayed after calibration (red) in the input window under *File → Plot generation* "*Top view/Map generation*". Upon closing the module, the settings for this plot are saved to the batch file "plo.bpl" (predefined).

The following plot shows the result of the first calibration:
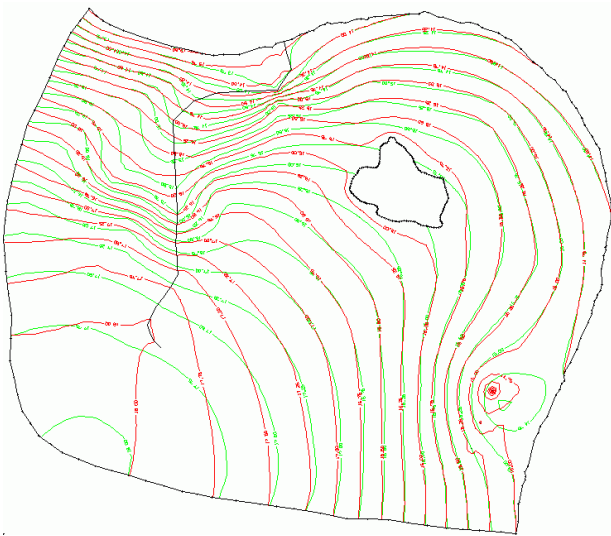
Fig. 33: Calculated (red) and interpolated (green) potential heads after the 20th calibration

2.1.4.4   Generating a measured data difference plot and saving the data

When a measured data file is available, the differences between measured and calculated groundwater levels can be displayed at the measuring points (usually groundwater measuring points) by "*Differences as circles*".

This is done during the plot display by selecting "Measured data" under data origin. An input window appears in which the file containing the measured values and the data to be used for calculation are defined:
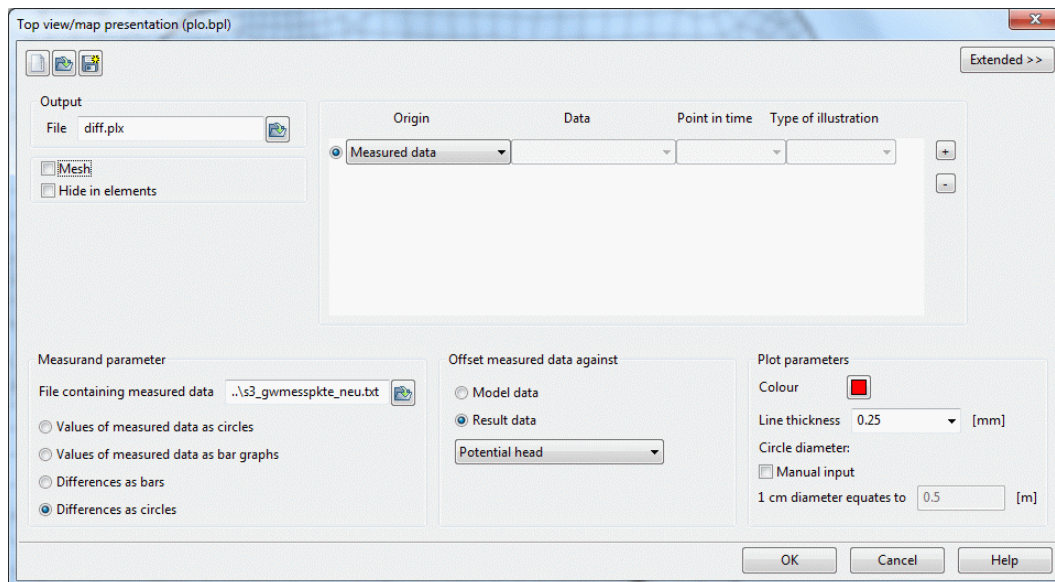
Fig. 34: Generating a difference plot

The file containing the measured data was already made available for download in Step 3: Data assignment.

In the input window, differences are selected as circles, giving the following picture:
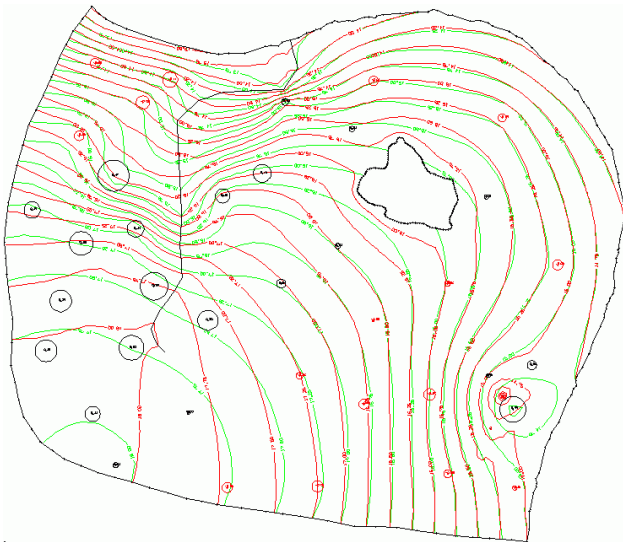
Fig. 35: Contour lines with measured differences

The calculated isolines already reflect relatively closely the curve of the isolines determined by measurement, in particular in the eastern part. The deviation at the measuring points is at most 0.47 m, being on average 0.10 m. (These deviations are displayed in the protocol window of the plot display. The deviations in the calibration protocol window refer to all nodes and are at most 1.24 m, on average 0.11 m). The calculated differences are shown for each measuring point:

Positive values (black) mean: The model calculation produces larger values than the measurement. In this example, that means: In the model, there is too much water at this position. Negative values (red) mean that the calculation produces smaller values than the measurement. In this case, that means: In the model, there is too little water at this position.

Note: If the calculated differences are to be saved as a file for later evaluation, this can be done under *File →
Export → Difference to measured data:*
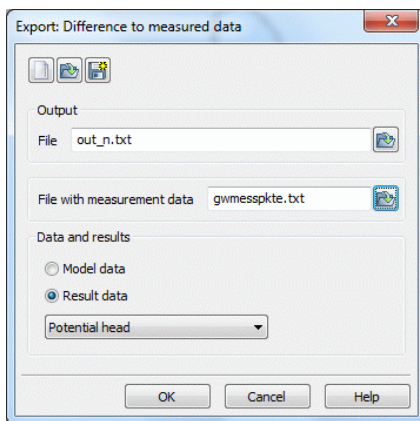
Fig. 36: Export of measured data differences

2.1.4.5 Calibration of the example in three steps

The first calibration produces a good starting base for the K values. They are imported to the mesh file via *Attributes → Import model data/calculation results*.
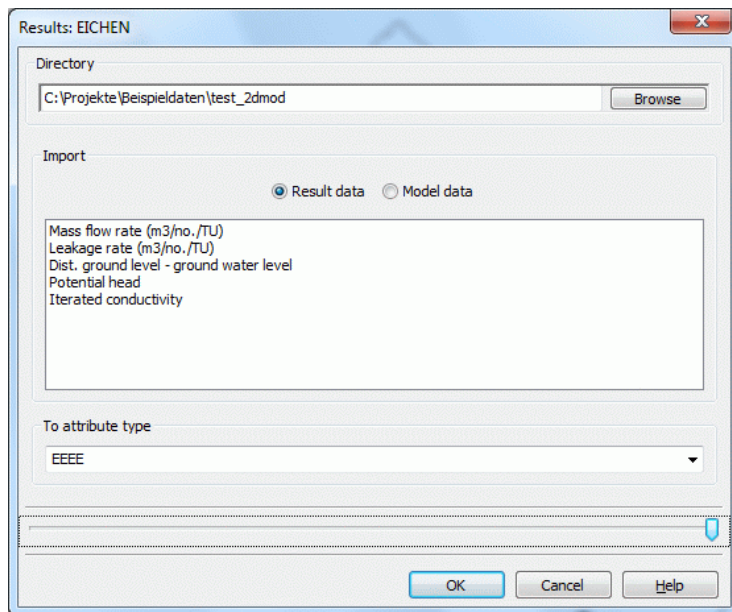
Fig. 37: Import of result data

When the attribute "KWER" is selected, the existing K values are overwritten. To avoid large jumps of the K values in adjacent elements, the K values are smoothed as follows:

Attributes → Computation → Smooth: "KWER". The weighting factor is set to 0.5, and the weighting type is preset to "linear". The mesh file is saved. Then the model is inspected (*Calculation → Model checking*), a flow calculation (*Calculation → Stat. flow*) is carried out with 5 iterations and the module GEONEU (*→ Advanced settings*), and a new plot is generated using the previously saved "plo.bpl" batch file. The result is shown on the following picture:
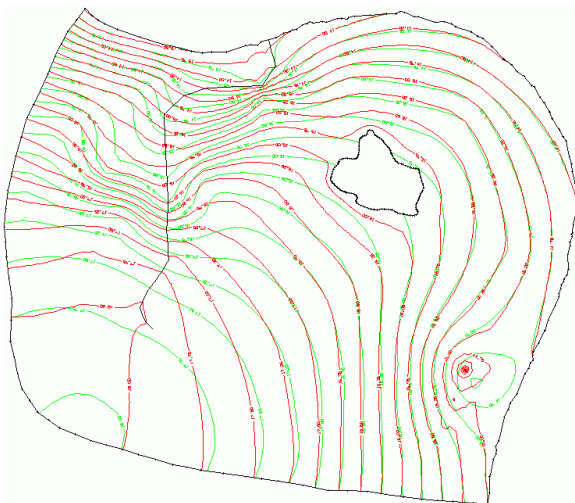
Fig. 38: Contour lines with difference plot after smoothing of the K-values

The maximum deviation at the measuring points is 0.50 m, the average being 0.09 m.

The curve of the isolines in the upper reach of the water course indicates an infiltration of the stream water into the groundwater. Upon "crossing" the water course, the isoline "17.75 (red)" forms a tip in the direction of flow of the water course.

Since this is not in line with the actual situation (stream bed is probably dry), the parameter MXKI is assigned the value 0.0 at the last 14 nodes, i.e., an infiltration is prevented (*Attributes → Assign → Direct*).

After performing the model inspection, carrying out a steady-state flow calculation and generating the plot, the following picture is obtained:
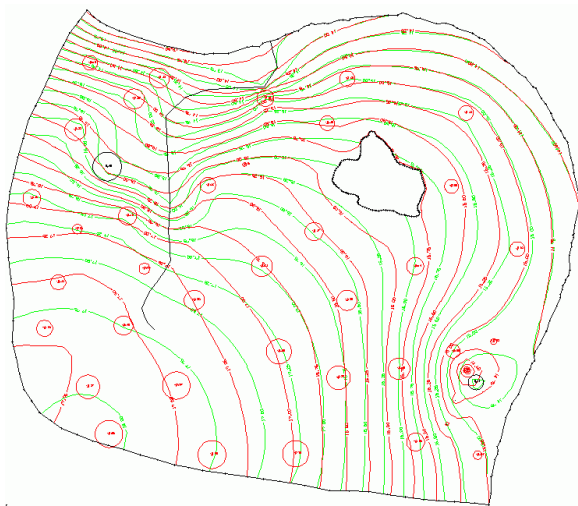
Fig. 39: Contour lines with difference plot after introduction of "MXKI" at 14 nodes

The infiltration in the upper reach of the water course has been eliminated. The curve of the isolines is more plausible, but the average deviation of the measuring points has deteriorated. The maximum deviation is now 0.39 m and the average 0.17 m. A large portion of the model seems to contain too little water!

It seems logical to start another calibration using the parameters modified up to now. The result is shown on the following picture:
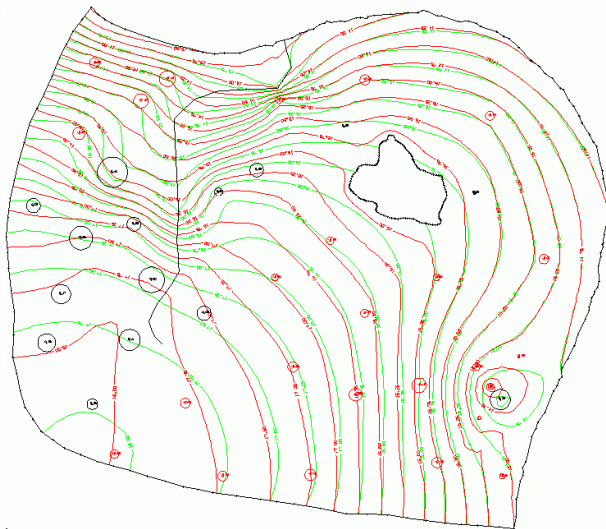
Fig. 40: Contour lines with difference plot following a new calibration

Finally the iterated K values are copied to the mesh file, and a new flow calculation is carried out.

The updated mesh file "s4_kalibriert.zip" (Download on our , *.zip) gives the following picture:
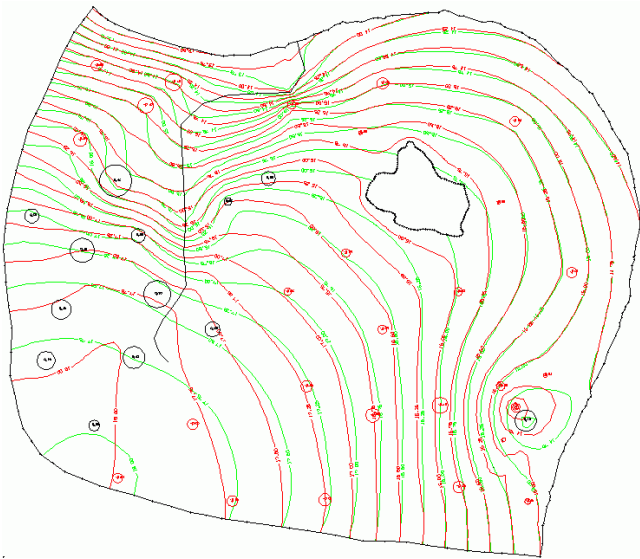
Fig. 41: Contour lines with difference plot of the calibrated model

The deviations at the measuring points are at most 0.41 m and on average 0.08 m. This is considered sufficiently accurate for the example model.

The distribution of the permeabilities now looks like this:
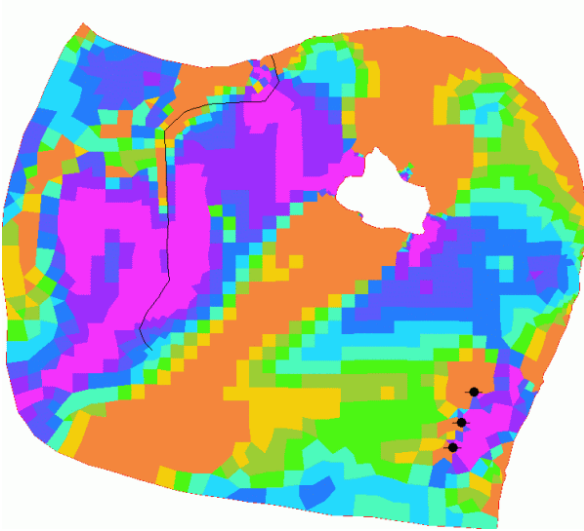
Fig. 42: Distribution of the calibrated permeabilities

The permeabilities were displayed in the interval 1.0e-06 to 0.002 within 10 ranges of values (via *View → Show attributes → Isolines/Area plots/Values*). Upon clicking on *View → More windows → Project manager* the layer list appears in a new window in which the arrangement of the individual layers can be changed via drag&drop.

(Note: "Top" in the layer list corresponds to all the way at the back, "Bottom" corresponds to all the way at the front). To display the legend of the corresponding layer, click on the check box.
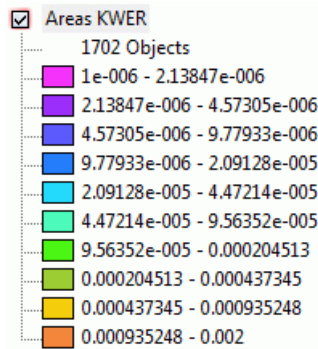
Fig. 43: Legend of the permeability values [m/s]

A stepwise model calibration can look like the one described, but it always requires knowledge by the user on the local situation and cannot be carried out by the book. Moreover, a calibrating should always end with a plausibility check of the (iterated) parameters.

2.1.4.6 General information on calibration

The K values iterated during model calibration are saved to the output file out.e in the file format of the *.net file and can be copied from there to the mesh file. The procedure (*Attributes → Import background data ...*) was already explained at the beginning of chapter Calibration of the example in three steps .

Due to the complexity of a groundwater model, the automatic change of the permeabilities does not necessarily lead to a plausibly calibrated model state. This is why the model must be recalibrated "by hand". The basic procedure is as follows:

2.1.4.7 Schematic procedure during calibration

1. Copy the automatically iterated K values to the mesh file
2. Model checking
3. Flow calculation
4. Comparison of the groundwater isolines from the measured data with the calculated groundwater isolines, if available, it will include the display of differences (circle plot) at the measuring points,
5. Change in the permeabilities by area according to the isoline curves

    - if the calculated potentials are too high, then the ground is too impermeable, i.e., it may be necessary to increase the K values

    - if the calculated potentials are too low, then the ground does not hold enough water, i.e., it may be necessary to set the K values to less permeability
6. In addition, there are, of course, other parameters which likewise can be changed by area:

    - increasing or reducing the leakage factor of the water courses

    - Check inflows from the boundary

    - Smoothing of the K values via *Attributes → Computation → Smooth:* Select KWER and set weighting factor, for example, to 0.5. This will balance jumps in adjacent mesh elements.

After changing the parameters (tip: do not change too many parameters at a time!), the model is inspected, and then a flow calculation is carried out. Then step 4of the schematic procedure is started again. This process is repeated until sufficiently accurate deviations are obtained, and the isoline curves seem plausible.

2.2 Creating a 3D-Model

The zip file „s4_2D_schulung3D.zip" includes the files used in this chapter. You can download it from our homepage (http://spring.delta-h.de)at Download & Support: Tutorial (EN): Tutorial_bsp_files.zip ( ).
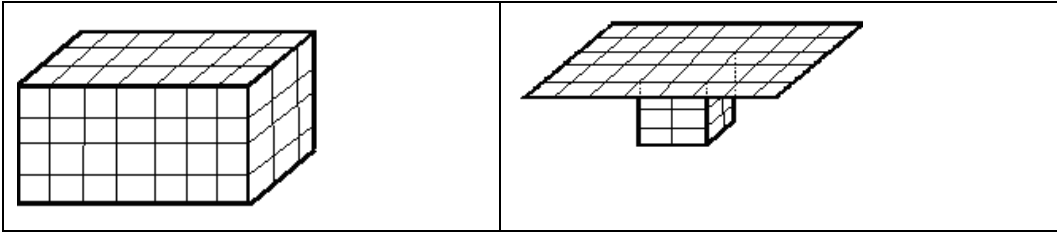
During an investigation the local conditions often request the modification of a two dimensional model into a 3D model, because of geological faults, a second aquifer or a groundwater withdrawal from a deeper groundwater layer. Besides it is always useful to build up an entire 2D model first and then to change it into a 3D model after resolving all geometrical errors. Based on the existing model "s4-2D_schulung3D.net" the way how to create a 3D model is described below.

2.2.1 Creating an entire 3D model
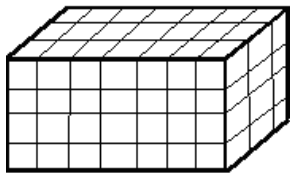
Fig. 44: Example of a homogeneous 3D-model

2.2.1.1 Theoretical background

SPRING has certain rules to be followed in three-dimensional modelling. The main concepts which simplify the 3D mesh generation are:

- A 3D mesh is always based on the 2D mesh of a corresponding horizontal model.
- All nodes in one line over the depth of the model have the same x and y coordinates.
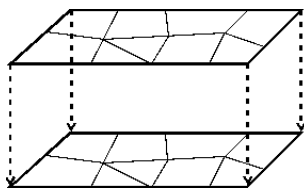
Fig. 45: Vertical projection of a horizontal mesh

A completely variable 3D finite element mesh with nodes and elements arranged freely in 3D space would require complex data management involving considerable memory space and computing time. With this type of mesh there is a risk that the user can lose track of the data. The decoupling of the planar and spatial mesh generation is actually the better and more efficient way to create a 3D mesh. First, equivalent to a planar horizontal model, the 2D finite element mesh is generated. Don't forget the compulsory points in the vertical projection, e.g. exfiltration wells, gauge wells and boundaries of geological layers. In a second step, the 2D mesh is vertically projected into the third dimension.

Therefore, all nodes and elements lie above or below corresponding nodes and elements. All 3D elements have the same projection on the horizontal plane. The user needs only one mesh structure for locating any node or element in any given layer.

Beyond that, the basic 2D horizontal model has two main advantages. The user can complete the planar mesh and check it for errors before projecting it into the vertical direction. Already existing two-dimensional models can easily be enlarged into the 3rd dimension for the case that the calibration or the flow calculation shows that the 2D model cannot exactly model the real processes.
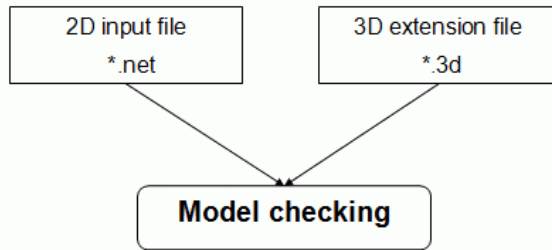
Fig. 46: Separated 2D and 3D data

The three-dimensional mesh generation is done during the model checking. The additional 3D-file contains the necessary parameters. Because of the simplification, there are only a few additional input data. In general, two different algorithms are available:

- 3D-enlargement by definition of the number of layers and proportion factors (3DSH)
- 3D-enlargement by the definition of z-coordinates (ZKOR)

2.2.1.2 3D-Enlargement by the definition of the number of layers and proportion factors (3DSH)

The bottom of the aquifer, known from the main input file, represents the lowest node layer. In general, the ground level (GELA) defines the topmost node layer. In case no GELA data exist, the defined ground water level (EICH) defines the topmost node layer. In case no GELA and no EICH data exist, the model boundary at the top is interpolated to nodes (OBER). By specifying the proportionality factors which define the ratio between the layer thicknesses, the layers between the lowest and the topmost layer can be generated.
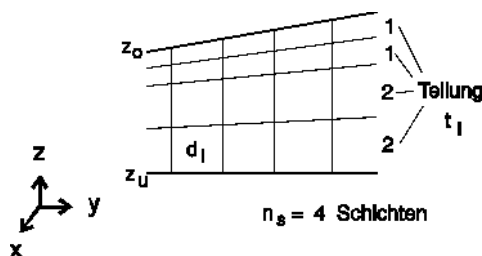
Fig. 47: Generation of a 3D mesh by the definition of proportionality factors

The thickness between the lowermost and the uppermost layers is usually different for every node of the mesh. It is divided into the defined parts and the corresponding z-coordinates are determined:

$$d_i = \frac{z_o - z_u}{\sum\limits_{j=1}^{n_s} t_j} * t_i$$

The above algorithm is useful for relatively homogeneous aquifers without complex geological layers where the spatial flow has external reasons, e.g. a partially penetrating well, flow around a building or beneath a river.

2.2.1.3 Realization in SPRING

After opening the existing project in SPRING you define the boundary of the 3D area with *Mesh → 3D → 3D-Boundary → Mesh-Boundary = 3D-Boundary*. With *View → More Windows → Project information* you see that the model type has changed to "3D-model".
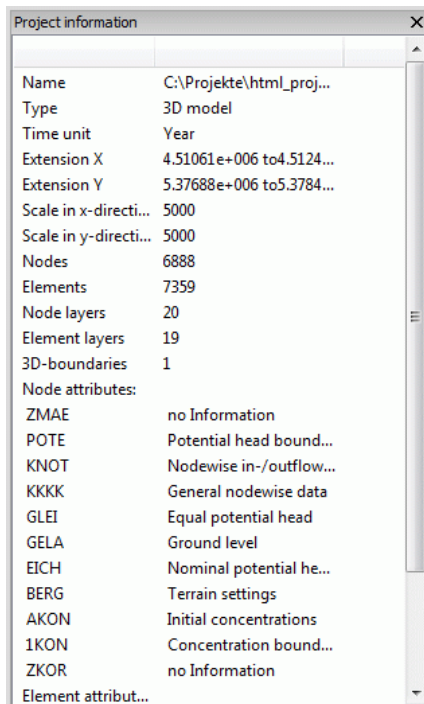
Fig. 48: Project information

Next you have to arrange the new layers with *Mesh → 3D → New layer arrangement…* in the following dialog box:
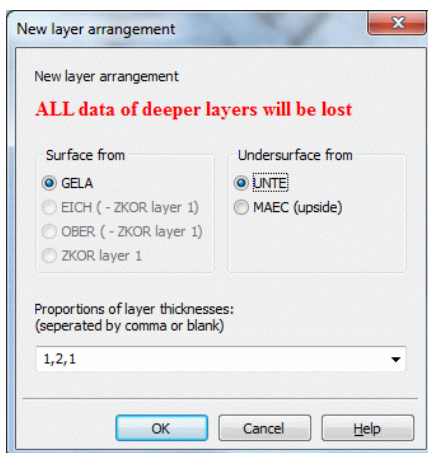
Fig. 49: Arrangement of new layers

If you have considered which and how much element layers are necessary (e.g. dependend on exploratory drillings) these numbers are entered in the input field. They have the following meaning:

■ The number of the figures (here: 3) defines the number of new element layers.

■ The sum of digits of the numbers (here: 1+2+1=4) defines the partial thickness of each element layer. That means, the entire thickness (= GELA - UNTE) is divided by 4 and multiplicated with the given numbers. So the first element layer gets a thickness of 1/4th of the entire thickness, the second gets a thickness of 2/4th of the entire thickness and the third gets a thickness of 1/4th of the entire thickness.

The following vertical section shows the proportions of each generated layer:
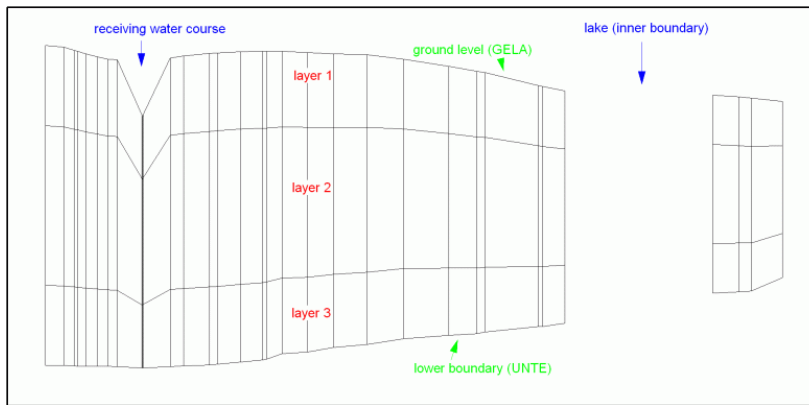
Fig. 50: Vertical section of the new 3D model (hundredfold super-elevation in Y-direction)

It is possible to create a user-defined section at every time with *View → Vertical section*: You select two points by pressing the left mouse button and the vertical section appears in an additional window.

This feature is independent of the model type.
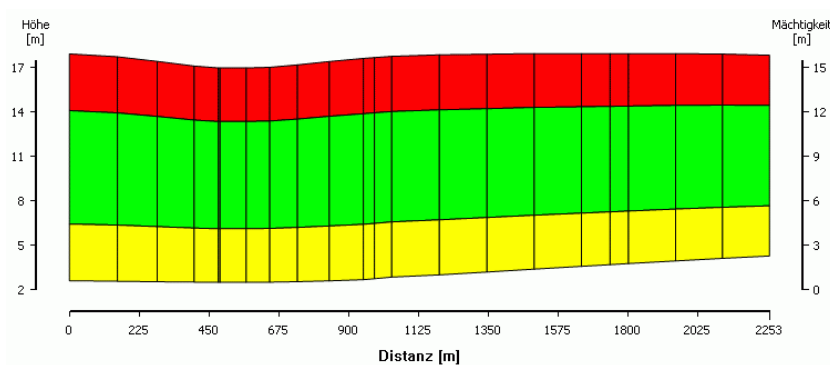
Fig. 51: User-defined vertical section through a 3D-model

The layers of the 3D-model can be visualised with *View → More windows → 3D-View*. An additional window opens, in which you can move the model in all directions by pressing the left mouse button:
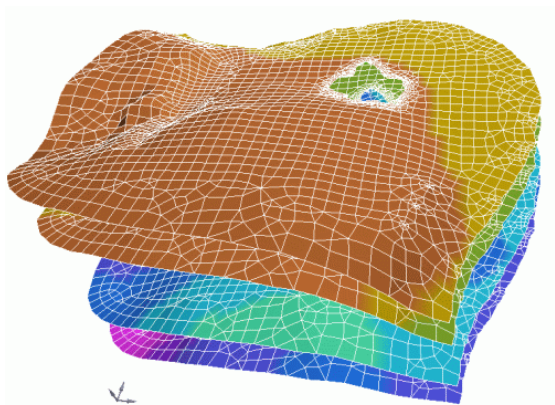
Fig. 52: Three dimensional view of the model

After saving the project you get the files "schulung_3d.net" and "schulung_3d.3d". You can download them from our for further proceedings in the next chapter: "schulung_3d.zip"

For a common flow computation it is sufficient to assign the permeability KWER to the elements of the new layers with *Attributes → Assign → direct*. For difficult hydrogeological conditions a calibration for each layer is requested and other necessary attributes have to be assigned to the generated nodes and elements (e.g. saturation parameters or impermeable layers, etc.).

2.2.1.4 3D-Enlargement by the definition of Z-coordinates (ZKOR)

This algorithm requires a z-coordinate at each node, except in the topmost layer (= ZKOR from the 2D input file or = GELA or EICH). The interpolation module (*Calculation → Interpolation*) can be used to facilitate this input. It can interpolate planar information (e. g. the ground level) from data at points or along lines. The interpolation can be used to compute the z-coordinate at each node of the horizontal layers. The x- and y-coordinates are known from the 2D model. Because of the variable z-coordinates, the horizontal element boundaries can follow geological layers.
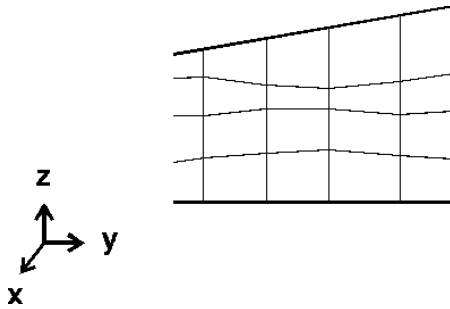
Fig. 53: Generation of a 3D mesh by the definition of z-coordinates

2.2.2 Creating partial layers in a 3D model

SPRING gives you the possibility to create partial layers and to do local mesh refinements in a model.
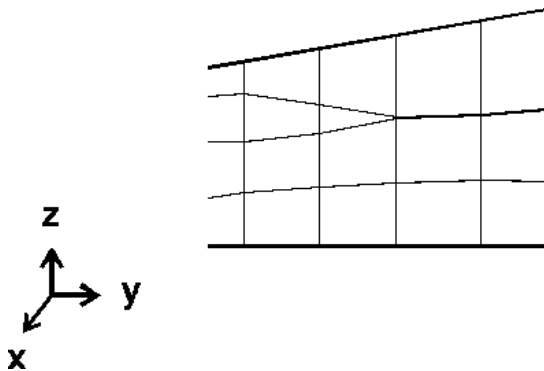
Fig. 54: Partial layers

2.2.2.1 Theoretical background

To consider geological faults or local limited aquitards in a 3D model a vertical mesh refinement is requested. By using the function *Mesh → 3D → Divide layer* in SPRING you can refine the mesh in the vertical direction. The attributes for the new elements and nodes are automatically generated (equivalent to the horizontal mesh refinement).

Only with the help of the exact z-coordinates can partial layers, which do not run through the entire three-dimensional model, be possible. To create the end of a layer, the z-coordinate of the lower node layer must be equal to that of the upper layer, for all nodes along the layer boundary. Afterwards, the model checking elimi-

nates the superfluous nodes and elements. Then, the calculation modules will manage only the absolutely necessary data.

A parameter in the input window of the model checking decides wether the z-coordinates are merged or not. A value of 0.01 m is set, that means, if the vertical spacing of two nodes is lower or equal 1 cm the nodes are merged to one node.
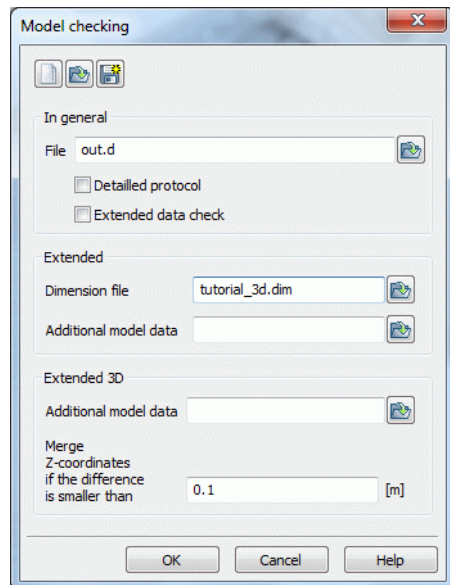
Fig. 55: Model checking with 3D extension

2.2.2.2 Realization in SPRING

Creating local vertical mesh refinements in SPRING are done with *Mesh → 3D → Divide layer*. Data of the new nodes and elements are generated automatically.

After pressing the button *Divide layer* the following dialog box appears:
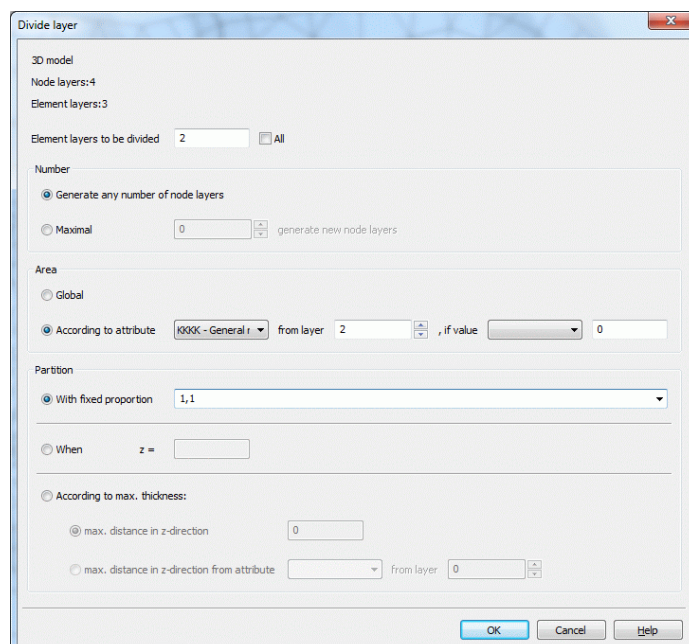
Fig. 56: Divide layers for a local mesh refinement

To show how to divide a special layer by attributes the attribute KKKK was assigned to some nodes of the second element layer.
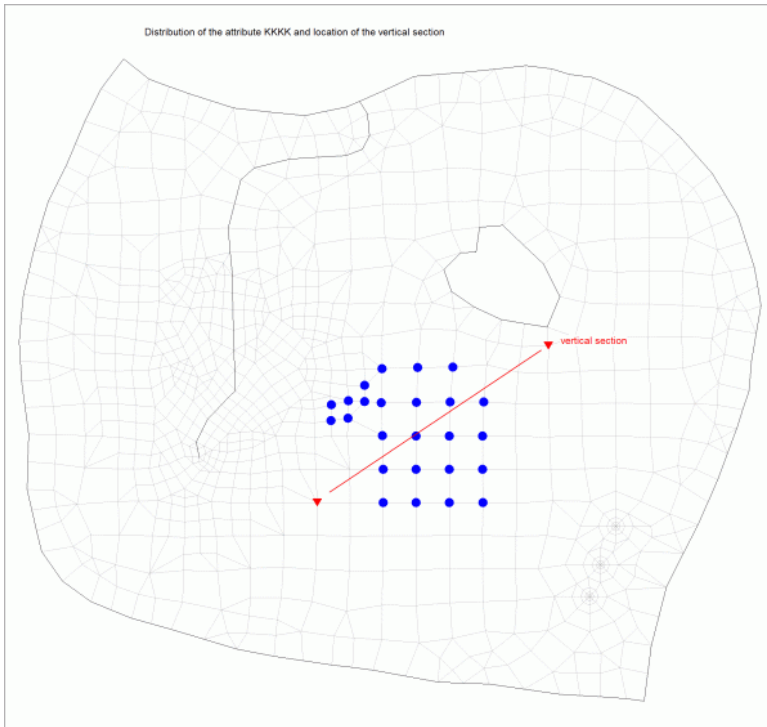
Fig. 57: Horizontal view of the nodes with the attribute "KKKK" and the location of the vertical section

First the element layer for dividing must be defined. Here layer 2 is selected. Now a new element layer is generated at all nodes to which the attribute KKKK is assigned. You must define how the selected layer is to be divided. There are two possibilities: either you define the number of new layers to be generated by using partition factors (equivalent to 3DSH), or you define a maximum layer thickness. In the latter case, the number of layers to be generated depends on the layer thickness specified. If you define partition factors (*Divide with fixed proportion*) for the division of the computed total layer thickness (equivalent to the attribute 3DSH), the number of newly generated layers is constant at all nodes. If you define the maximum layer thickness (*According to max. layer thickness*), the number of new layers depends on the specified layer thickness.

Here *Divide with fixed proportions* was selected with the partition factors "1, 1". This input generates a uniform partition into 2 element layers.

After saving the project the user can visualise the new layer partition with *View → Vertical section*. Result is the following picture:
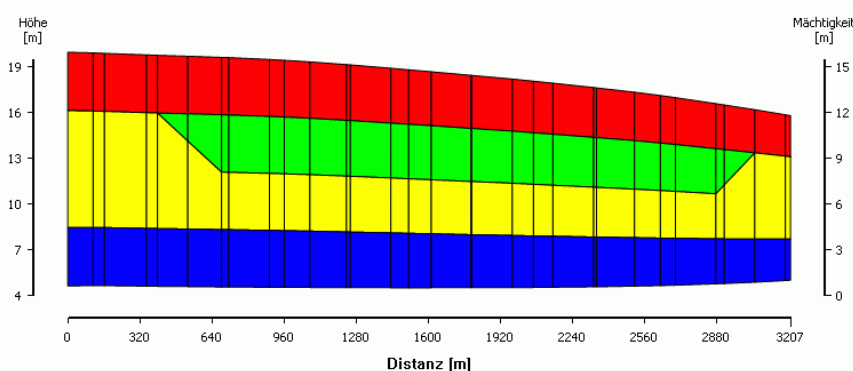
Fig. 58: Local vertical mesh refinement of the 2nd element layer, where the attribute „KKKK" is assigned (green area)

If the selected attribute exists only at a few isolated nodes the following vertical section results (partition factors: 1, 3):
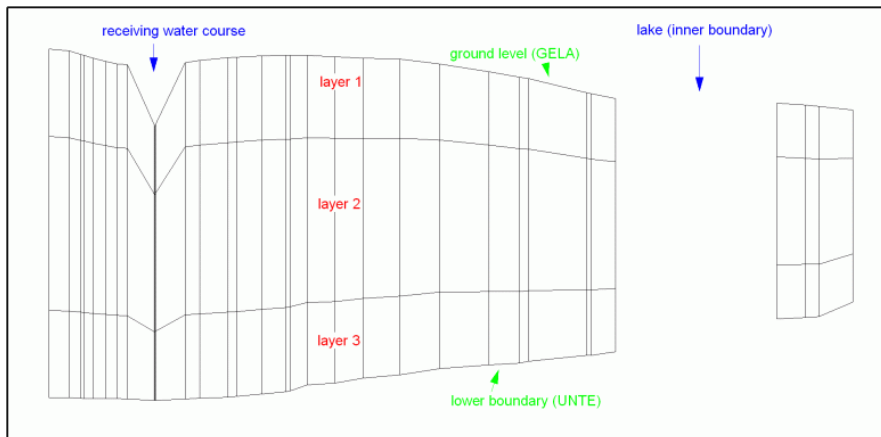
Fig. 59: Local vertical mesh refinement of the 2nd element layer, when the attribute KKKK is assigned only to a few isolated nodes

So you see that it is not always useful to define the partial layers by attributes. Mostly a manual treatment of the mesh based on drilling profiles is necessary.

The following figure displays a 3D model with several partial layers to show the complexity of geological conditions.
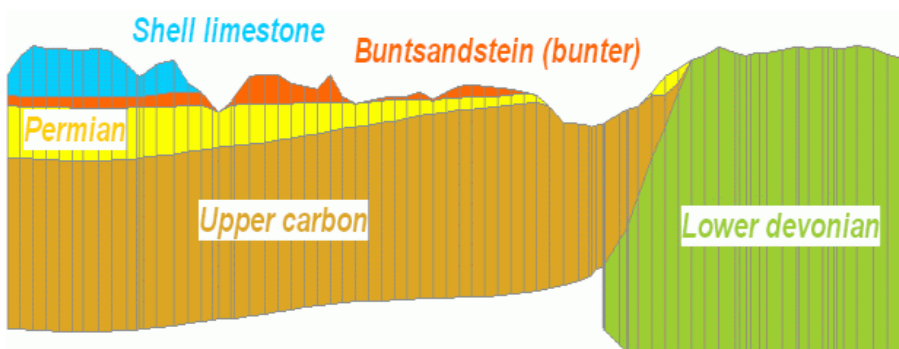
Fig. 60: Partial layers in complex geological conditions

2.2.3 Creating a 2D model combined with a partial 3D model

It is possible in SPRING, to extend a two dimensional horizontal model into a partial 3D model.
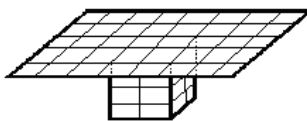
Fig. 61: Example of a horizontal model with a three dimensional part

2.2.3.1 Theoretical background

In most cases, three-dimensional groundwater models are quite small. Predominantly, the local three-dimensional flow is limited to a small area, but requires a fine discretisation into the third dimension too. This causes large numbers of nodes. Hence, the problem is to find correct boundary conditions which do not change during the simulation. Two-dimensional models can be enlarged until some natural boundary such as a watershed divide or a receiving water course is reached. For three-dimensional models, the number of nodes would grow quite fast. Otherwise, oversized horizontal elements in 3D models cause numerical problems. The best way of solving the problem is represented by a large 2D model combined with a partial 3D model.

The partial 3D mesh is generated automatically, and is equivalent to the complete three-dimensional model. In addition, the definition of one or more closed boundaries for the 3D area by node numbers (3DRA) is required. The program identifies the elements inside the boundaries and projects them into the 3rd dimension.

2.2.3.2 Transition zone between the 2D model and the 3D area

The critical zone in a combined 2D-3D model is the transition zone between the two model types. Every two-dimensional horizontal model assumes (based on Dupuit) exclusively horizontal flow. Hence, the partial 3D model must be large enough that the vertical velocity (direction z) in the transition zone is equal 0.

Specifying equal potential heads over the full height at the boundary nodes of the 3D area is a useful method for the transition condition. It means that the potential head at the nodes is unknown and must be computed during the calculation. But there is a second condition that defines a constant potential head for all nodes, one below the other (see identification GLEI). Mathematically, in the equation system, the single nodes are summed and represented by one representative node. The other nodes are eliminated.

The program automatically uses this procedure. Comparing the boundary of the 3D area (3DRA) with the 2D model, it can identify the 3D model as complete or partial. Equal potential head can never be used for the boundary nodes of a complete 3D model. Instead, explicit boundary conditions of 1st order (potential head), or rather of 2nd order (mass flow rate), are required.
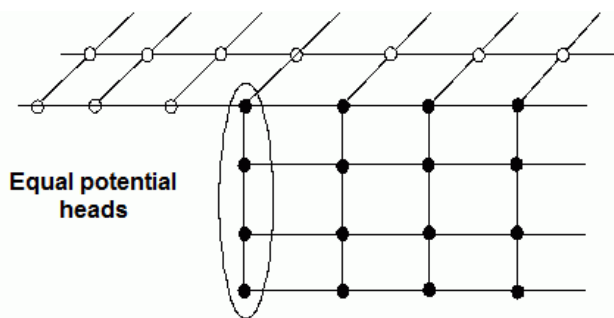
Fig. 62: Transition condition between the 2D and the 3D area

2.2.3.3 Realization in SPRING

For realising a partial 3D model in SPRING you have to define the boundary of the 3D part first with *Mesh → 3D → 3D-boundary → Generate*. All boundary nodes have to be captured.

After confirming the boundary with the right mouse button the new layers have to be arranged (see 3D-Enlargement by the definition of the number of layers and proportion factors (3DSH)).

Here the partition factors "1, 3, 2" were defined. Resulting are 3 layers with a partial thickness of 1/6, 1/2 and 1/3 of the entire thickness.

With *View → 3D-boundary* you can display the new 3D-boundary. Because the predefined way of representation is not very obvious change it with the Object toolbar (*View → Toolbars → Object attributes*). Change the colour and line width and assign them to the object.

You get the following figure:
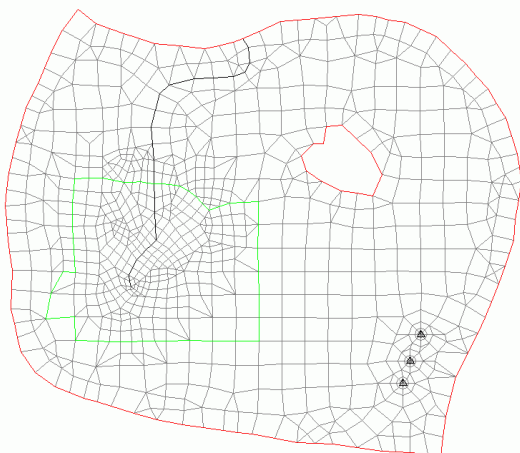
Fig. 63: Displaying the 3D boundary in SPRING

At least, the permeability (KWER) has to be assigned to the new generated elements to run a flow computation.

Besides after running the model checking it is possible to display the modified model with *File → Plot generation → 3D wireframe*. The following input parameters were defined:
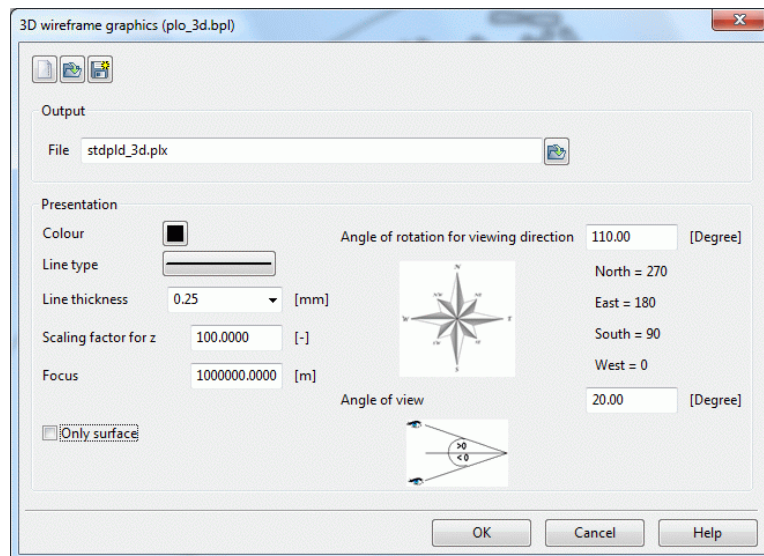
Fig. 64: Input for a 3D wireframe

The following figure of the partial 3D model results:
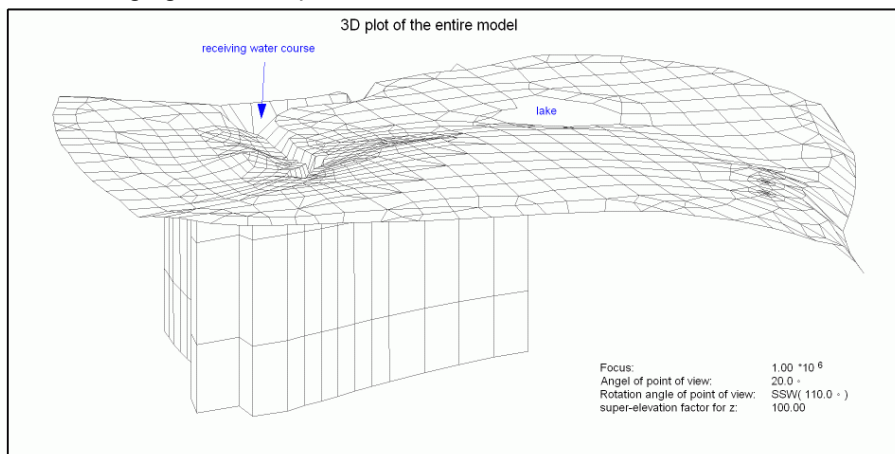
Fig. 65: Mesh plot with partial 3D area

2.2.3.4 Automatic numbering

The numbering of nodes and elements in the deeper layers is carried out automatically. By using arrays, no continuous numbering is necessary. The user can individually define the constant difference between the numbers of nodes and elements in two layers, one below the other. The default offset is 10,000. It allows maximum 9,999 nodes, or rather elements, per layer. The node or element number in the next layer is obtained by adding 10,000.
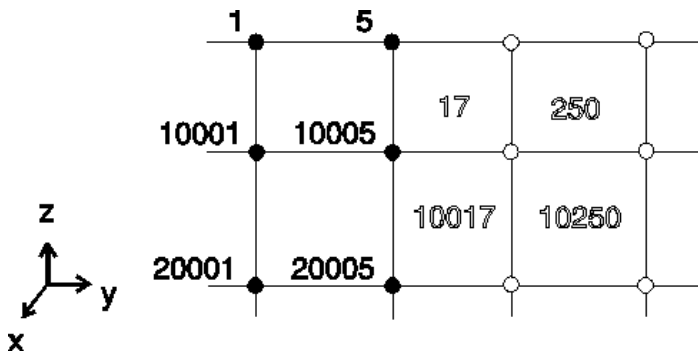
Fig. 66: Numbering of nodes and elements

The numbering with the help of a constant offset has the advantage that the general mesh structure can be obtained from the topmost layer and its node and element numbers. Any node or element in deeper layers can be identified by adding (n-1)*10,000 where n represents the layer number.

Large models can easily have more than 9,999 nodes or elements per layer. In such cases, an offset of 10,000 is not enough. With the help of the attribute 3DNR in the main input file, the value can be edited. For easier management, it is recommended to use always multiples of 1,000.

The input parameters (e.g. hydraulic conductivity or storage coefficient) which are assigned to the 2D model are assigned to the projected layers of the three-dimensional model (3D elements) as well.

2.2.3.5 Automatically assigned data

For a fast and effective data input for 3D models, the model checking automatically assigns the majority of the necessary data. Each layer receives the existing data from the layer above.

The following data types are assigned automatically

- Initial potential heads (EICH)
- Permeabilities (KWER, KWEV, KFVH)
- Limitation of the permeabilities (KMIN, KMAX)
- Dispersivities (DISP)
- Porosities (PORO)
- Storage coefficients (SPEI, KSPE)
- Initial concentrations (AKON)
- Anisotropic conductivities (Z-KA)
- Compressibility of the entire system (KOMP)

For instance, the input file for a 3D model with 6 node layers contains initial potential head for the first and third layers. Therefore, it follows that the second layer gets the values of the first one, while the fourth, fifth and sixth layers get the values from the third one.

If there are no vertical permeabilities or rather no ratio between the horizontal and vertical values specified (KFVH), the ratio is set to the default value 0.1, that means KWEV = 0.1*KWER.

2.2.4 Import of a submesh in an existing model

For many projects, the task is first to set up a large-scale overall model. Later a partial model is cut, refined and edited for detailed questions. In the following the detailed results are to be imported back into the overall model.

The following chapters provide guidance which steps have to be done before and after importing the partial model into the overall model.

2.2.4.1 Deleting the partial model area in the overall model

In the first step, the model nodes that lie within the partial model edge are deleted from the overall model. The following figure (Fig. 67) shows the area of the partial model before importing it into the overall model.
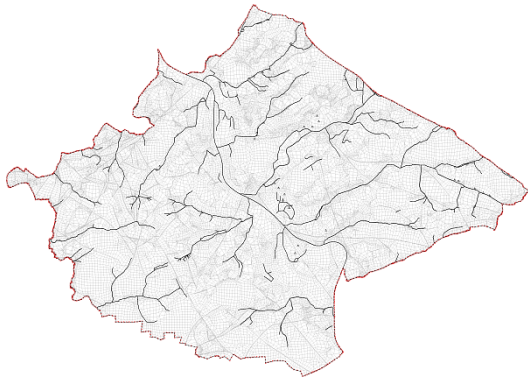
Fig. 67: Partial model area before importing it into the overall model

Along the outer edge of this model, the nodes and elements from the existing overall model are removed (*Mesh → Nodes → Delete → All inside selected structures* or *Mesh → Element → Delete → All inside select- ed structures*). The following figure (Fig. 68) shows the remaining mesh of the overall model.
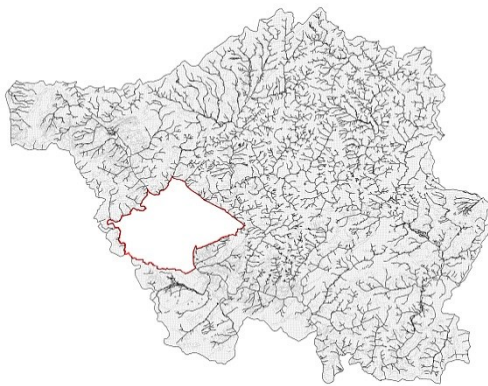
Fig. 68: Partial model area, removed from the overall model (red)

In case of concave edge gradients it happens that elements remain in the area of the partial model because their corner nodes lie on the model edge (Fig. 69, cyan highlighted elements that are outside the red border line). These elements have to be removed manually.
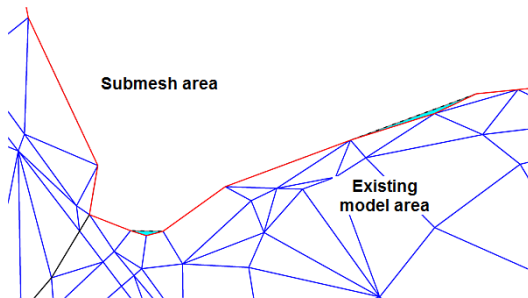
Fig. 69: Elements, which are not automatically removed

A second step is to examine whether linear data structures (e.g., water courses) intersect the model edge multiple times. If this is the case, the lines have to be separated between the point at which the line leaves the remaining model area and the point at which the line runs again into the model area.

The following figure (Fig. 70) shows such a situation. The image (a) represents the initial situation in the over- all model. In image (b) the mesh along the broken red line was cut. In this case a blue line runs out of the remaining mesh and at another point back inside the mesh. The locations where such a situation occurs, have to be identified and must be adjusted by separating the lines (c).

This step is not only required for a correct presentation (e.g. markings), but also for a proper consideration of line-related leakage boundary conditions.
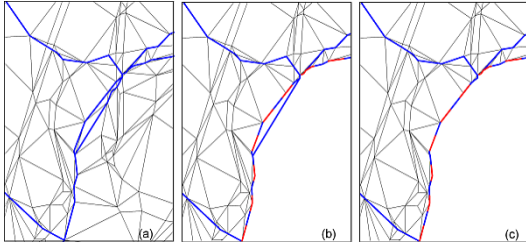
Fig. 70: Correction of line data

2.2.4.2 Horizontal border adjustment

Since the mesh of a partial model is refined for detailed questions, an adjustment of the boundary nodes between partial and overall model is required. For this, the boundary nodes of the partial model are reduced to the boundary nodes of the overall model. The advantage of this approach is to develop triangular elements from quadrilateral elements or elements fall away completely. The result is in this case that there is no need to create a new element, which has to be parameterized. The following figure illustrates this step:
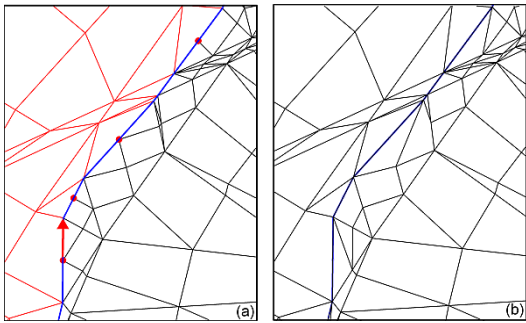
Fig. 71: Adjustment of the boundary nodes

Image (a) shows in black the elements of the refined partial model and in red the elements of the existing overall model. The blue line indicates the seam line of the two models. The red dots mark those boundary nodes of the partial model, which have no counterpart in the overall model. Thus, these nodes have to be eliminated before the integration of the two models. Simply deleting these nodes is not useful, as well as the adjacent elements are deleted, too. SPRING supports manually moving nodes to adjacent nodes (red arrow). By using the menu sequence *Mesh → Nodes → Merge → Capture two* the nodes are merged and the affected elements are automatically removed. The image (b) shows the corrected meshes:

It is important for the numerical calculation that no unfavorable or illegal element geometries remain (Fig. 72).
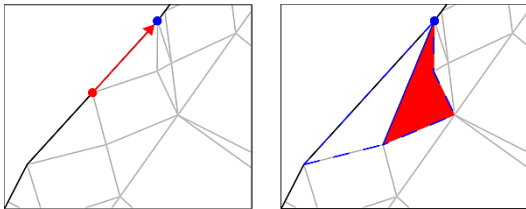
Fig. 72: Unfavorable or illegal element geometries

This has to be considered for each merging of nodes. In very few cases, if both possible directions of merging lead to illegal elements, a manual shift of a single node or dividing a quadrilateral element is essential.

Note:

To avoid this step, it is generally recommended not to perform the mesh refinement (both horizontally and vertically) up to the boundary nodes.

2.2.4.3 Vertical border adjustment

In dealing with a partial model the need can arise, to make changes to the layer structure of the model. The reasons for this can be of different nature:

■ Recent studies and drilling results require a change in the depth position of layer boundaries.

■ The processing of detailed questions in the partial model requires a more differentiated analysis of individual geological units, so that a further subdivision of certain model layers is required.

■ For the correct representation, for example, of imperfect wells or built components (sheet piles, tunnels, etc.) is another vertical division of individual model layers necessary.

Since the added vertical layers of the partial model have no equivalent in the overall model such an assignment has to be created. The following cases are differentiated:

■ In the partial model has been no change in vertical layers. In this case the boundary nodes correspond to the intersection of the two meshes, and the layer mapping is automatically correct. Image (a) of the following figure illustrates this situation.

■ In the partial model, the depth position of layer boundaries, for example, was changed due to new drilling data. However, a vertical mesh refinement was not carried out, so that the number of layers and the assignment between partial and overall model is identical. In this case, the assignment automatically leads to a correct result. If the depth position of layer boundaries was also changed to the edge of the partial model, the assignment nevertheless leads to a correct result. Since there is a unique layer assignment, the depth of the layer boundaries is defined by the imported model part. This means if the overall model is opened first and then the partial model is imported the depth of the boundary nodes of the partial model overrides the depth of the overall model. This situation is shown in image (b).

■ One or more layers in the partial model were vertically refined, except for representing a more differentiated geological structure or a proper hydraulic consideration of measures (eg imperfect wells, vertical wells, sheet piles, tunnels, etc.). Such a situation is schematically illustrated in image (c). In this case, a layer assignment must be performed at the import of the partial model. In the general case the vertical refinement is carried out to the edge of the partial model, so that the depth of the existing model layers has changed there. In this case, an automatic assignment of the model layers is not clearly possible.
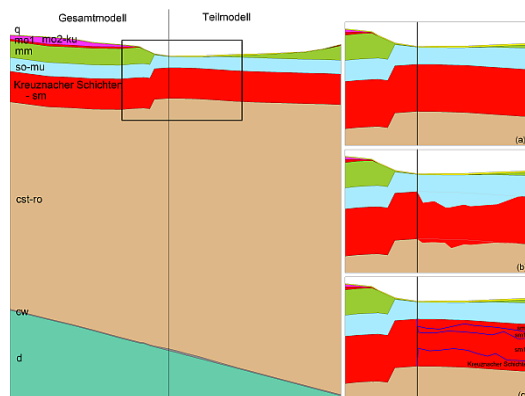
Fig. 73: Assignment of layers between partial and overall model

In the third case, the number of layers in the partial model has changed. The added layers of the partial model must exist formally in the overall model after the import, ie outside the partial model they merge geometrically together with hitherto existing layers. The assignment of the boundary nodes is controlled during the import of the partial model.

Although these eliminated layers are actually not considered in the simulations, they must be maintained for the modelling and for the node and element numbering.

Thus, if another partial model is imported into an overall model in which a vertically refined layer already exists because of a previous import, should as many of the existing coincident layers be used.

Conclusion: In general, the refinement of the layers of the partial model should not be performed on the model boundary. Thus, the depth position of the added layers can be properly assigned to the layer boundaries of the overall model.

2.2.4.4 Installation of the partial model in the overall model

In the previous steps, it was ensured that the boundary nodes of the partial model correspond geometrically with the inner boundary nodes of the overall model. Now, the attribution of the boundary nodes of the partial model is examined in view of the effect in the combined model.

- To the boundary nodes of the partial model is the in- and outflow assigned which was determined in the overall model. This boundary condition has to be removed from the partial model. Otherwise nonexistent in-or outflows are simulated.
- For the partial model has to be checked, whether line data extend beyond the boundary, too.
- Further is to examine whether the same attributes are defined on the corresponding nodes in the partial- and in the overall model. For data types that can be present on a node only once, this step is of minor importance, since in this case the value of the imported mesh overwrites the existing value of the opened mesh.
- For attributes that are allowed several times on a node (eg, line-related leakage coefficient), the corresponding attribute is to be removed in one of the two meshes. Otherwise, for example, has a receiving water course, which runs on the model boundary, after the integration of the partial model, a double-leakage effect. This needs to be avoided in any case.
- The user should have an overview of the existing attributes on the coincident boundaries to recognize present contradictions and to clarify the cause before certain data on the total mesh is definitively lost.

If the nodes of the coincident boundaries are adjusted in both models, both in relation to the geometrical location and on the attribution, the models are combined with SPRING.

The following strategy is advantageous:

First, the mesh of the overall model is opened, from which the area of the partial model was removed. Then the partial model is imported (*File → Import → Submesh ...*). If an attribute is already defined on a boundary node, the value of the partial model overwrites the existing value from the overall model. Since the partial model is usually up-to-date, this allocation is desirable.

If node and element numbers shall remain unchanged in one of the models, this model is opened first and then the other model part is imported, because the node numbering of the imported model is changed.

If the partial model is opened first and the overall model is imported, the 3D boundary must be recalculated (*Mesh → 3D → 3D-Boundary → Mesh boundary = 3D-boundary*).

After saving the integrated model, a new model exists without internal boundaries.

2.2.4.5 Implementation of consistency and plausibility checks

2.2.4.5.1 Geometry an data checks

After importing the partial model, it is recommended to check the geometry of the integrated mesh (*Mesh → Checks ...*). If errors in the mesh occur, this is usually due to the fact that one of the steps described above has not been carried out correctly.

The most common source of error results from the fact that two nodes do not coincide exactly at the inner boundary. Especially for large models, it may happen that a boundary node of the refined mesh was not removed during the manual correction. This node then lies on the model boundary of one mesh, but has no counterpart in the other. These errors are identified by the geometry checks and can then be corrected.

It has proven to correct the errors in the respective models, and then perform the import again. After this two intact models exist. Advantage of this method is that the detailed model can also be used for future considerations further and a new import into the overall model can be done with reduced effort. For a new simulation with the partial model is to be noted that the already deleted boundary condition of in- and outflow has to be calculated and assigned again.

The model checking (*Calculation → Model checking...*) gives possibly indications of inconsistencies in the model data which are not identifiable in the pure geometry checks. If there are no relevant data errors detected, the actual model calculation can be performed.

2.2.4.5.2   Comparison of the calculation results

The next step is to carry out with the geometrically correct overall model a flow calculation. To obtain comparable results, it should be ensured that in the calculation of the partial model and the overall model the same combination of calculation module and parameters are used.

The validation is done by comparing the groundwater contours of the free surface of the integrated overall model with those calculated with the old overall model and the partial model.

As expected, differences will occur around the boundary of the partial model. Since the recalibration of a partial model is not restricted generally to the inner model, changes occur in the flow situation, which also have an influence on the model boundary.

The constant boundary in- and outflows lead to changes in groundwater levels in the peripheral region during the calibration adjustments of the partial model.

The prescribed boundary condition is no longer being forced when calculating the total model and the model can achieve a balance between the situation in the total model and the new situation in the improved partial model.

A comparison of the given boundary in- and outflow of the partial model with the amounts that flow in the new overall model over this surface can also be performed.

2.2.4.6   Notes and recommendations

In a large-scale principle model, detailed questions cannot always be managed efficiently and economically. Caused by the size of the model high computation times and the discretization are often not detailed enough.

One approach is to separate and to consider the region of interest as a partial model. The coupling of the partial model and the overall model is done by given boundary conditions, usually by mass flows. The advantages of this approach can be summarized in the following points

- Large flexibility of choosing a region of interest
- Mesh changes and the varying of parameters can be handled efficiently.

To keep the influence of the model changes on the boundary conditions low, the boundaries of the partial model should be defined far away from the region of interest. When defining the model boundary the following criteria should also be considered:

- Define boundaries as far as possible along stream lines,
- sufficient distance to groundwater withdrawals,
- no open water areas on the boundary,
- the model boundary of the partial model must be always on element edges of the overall model, there must be no quadrilateral elements cut,
- the partial model area must be large enough so that changes in the transition region between partial and overall model which are usually not completely preventable during the import, have no influence on the results of the region of interest.
- In order to allow a smooth import of a partial model in the overall model, the model boundary of the partial model has to remain geometrically unchanged. It may not be refined either horizontally or vertically. If this condition is not observed, the reimport requires not only additional work by the need for a boundary node cleanup, but also an interpretative effort in the case of a vertical refinement. In this case has to be determined how the refined layers are correlated with the layers of the overall model. Vertical mesh refinements, based on local needs (eg installation of sheet pile walls, tunnels, wells and locally common hydrogeologic inhomogeneities) should always be carried out only in the environment of the concerned region.
- The partial model should always be calculated using the same program module as the overall model. Also the calculation parameters such as the number of iterations or the attenuation factor should be maintained.

2.2.5 Generating fractures in a 3D model

With the menu item *Special features → Fractures → 2D-fractures from LERA* vertical and horizontal fractures can be inserted in a 3D model. This is based on the Python script "lera2kluft.py", which runs automatically when you click this menu item.

If the data type LERA (leakage for a polygon) is available the 2D fractures are written with their opening width in the * .3d file of the model.

The opening widths correspond to the values of the respective LERA-range. They have to be assigned by the user with the appropriate values.

2.2.5.1 Assign attribute LERA

The attribute LERA is assigned to any future fracture line.

The easiest way to find the fractures again is when they are numbered and the number reflects the value of the LERA attribute. It makes sense to remove all LERA-attributes that are not used for fracture generation before or to store them as another data type.
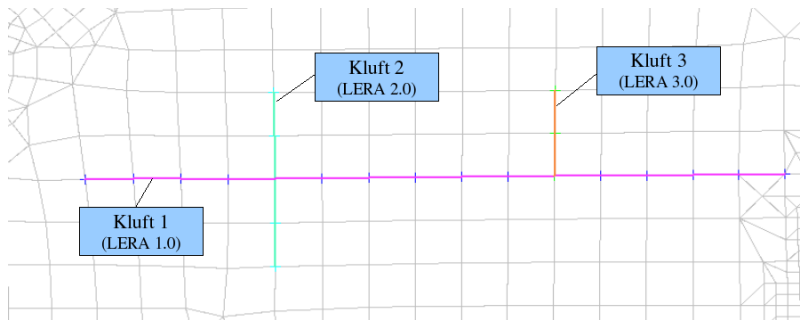
Fig. 74: Displaying the fracture system as LERA attributes

2.2.5.2 Editing the output in the *.3d file

After executing the menu item, the so generated 2D fractures (attribute 2DEL) are written in the * .3d file of the model. The fracture opening widths should be adjusted (data type 2DBR) to the actual fracture opening widths.

Since the LERA attributes were used only as auxiliary attributes, they should be deleted after restart of SPRING. Previously defined LERA attributes are assigned again.

2.3 Creating a transient transport model

The files of this example can be downloaded on our homepage (http://spring.delta-h.de) at Download & Support: Tutorial (EN): Tutorial_bsp_files.zip ( ).

The zip-file contains in the directory "Transport_2d" the model file, the transient input file "instat_RB.txt", the file "sitr" and the corresponding batch file „sitra.bsi" for a transient mass transport calculation.

In this chapter the technical steps will be presented which are required to create a transient transport model. A groundwater model is a transport model, when concentrations are present (attributes 1KON, AKON or KONZ).

2.3.1 Model description

The aquifer is homogeneous, isotropic and confined. The question is, how the pollutant disperses over the time. The length of the model is L = 300 m and it has a width of B = 100 m. The model area will also have the following characteristics:

Permeability	$k_f = 5*10^{-4}$ m/s	KWER
Thickness	m = 18 m	MAEC
Porosity	$n_e = 0.2$	PORO
Dispersivities	$\alpha_L = 7$ m $\alpha_T = 0.7$ m	DISP (sitr-parameters)
Initial potentials	$H_l = 36.0$ m $H_r = 30.0$ m	POTE

Concentrations of the pollutant	c_0 = 240 mg/m^3	1KON
Factor of delay	R = 1	-
Decomposition constant	λ = 0	-

The pollutant entry is made permanent and constant. The location of the source is located at the boundary of the model (node nr 156), so there was a particularly fine discretization chosen.

The following figure shows the model mesh with the pollution source
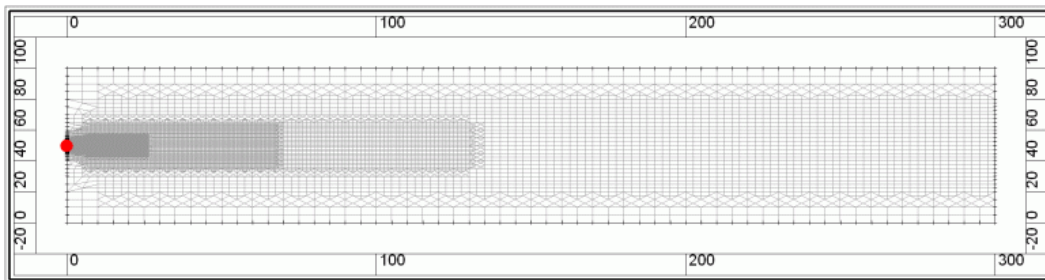
Fig. 75: FE-Model with pollution source

2.3.2 Necessary data for the mass transport

Initial and boundary conditions

- Fixed concentrations: attribute 1KON, Boundary condition of first order, set fixed concentrations. In the example, the choice of this boundary condition is required to represent the permanent constant source of pollution.

- Inflow concentrations: attribute KONZ [kg/kg]: With the data type KONZ concentrations (node wise) are assigned to entered inflows (from KNOT, FLAE, RAND, RANQ, RANX) and calculated flow quantities (leakage rates, mass flow rates from fixed potential heads). Example: In the surface of a landfill is a known quantity of pollutant (KONZ), which is solved but only by a certain amount of rain (FLAE).

- Initial concentrations: attribute AKON [kg/kg]: The initial state (node wise) for a transient calculation is preset. AKON functioned analogously to the initial potential heads (EICH) of the steady state flow.

Material data

- Porosity: If the porosity PORO , the default value of PORO = 0.2 is used for the calculation.

- Dispersivity: The longitudinal dispersivity α_L and the transversal dispersivity α_T are defined explicitly together in the menu of the *material transport calculation* under "Extended" → "Transport".

The following input window shows the possible entries in the extended settings of the material transport:

Fig. 76: Parameters of the material transport

The attributes 1KON (or KONZ), DISP and PORO are absolutely necessary for a mass transport calculation.

2.3.3 Transient input for the mass transport

After the *model checking* follows the calculation of the *mass transport*. The following input window appears:
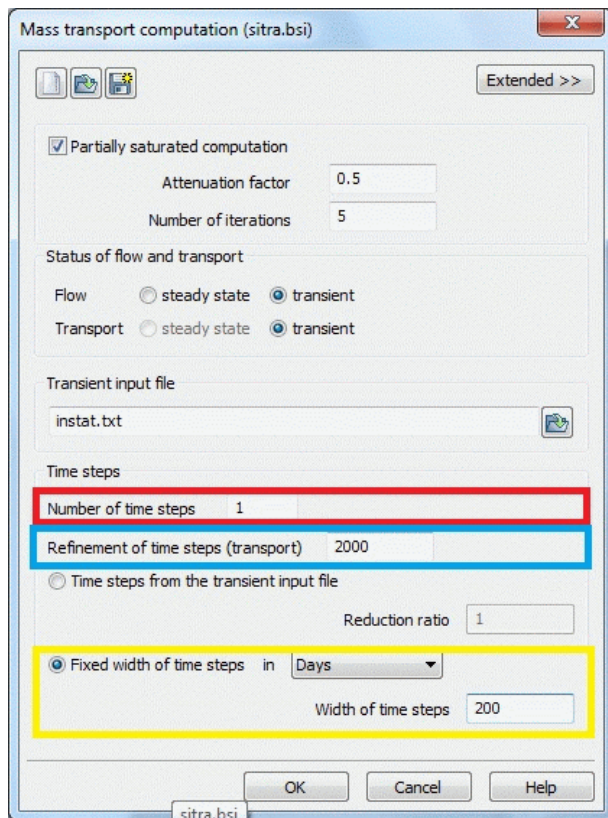
Abb. 77: Transient parameters (3D)

Explanation of the time steps

Red marking: The number of time steps, here 1, determines the duration of the transport calculation. Duration = number of time steps * time step width (here: 200 days).

Blue marking: Time step refinement, will indicate the subdivision of the time step size in time steps of the transport calculation (here: 200 days/ 2000 = 1/10 day).

Yellow marking: The time step width defines the duration of one time step of the flow calculation.

The quantity-based results are stored to the same time steps in which the potentials are saved in the background files ("Extended" → *Output*):
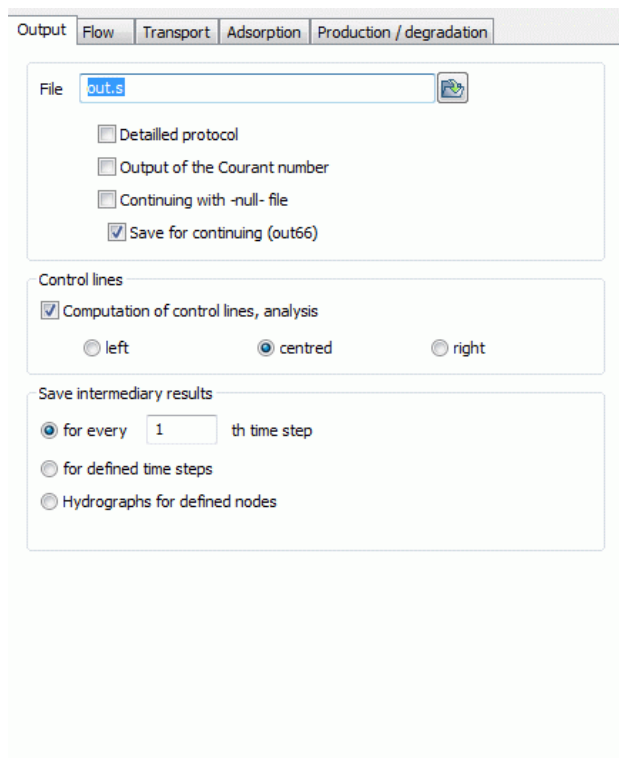
Fig. 78: Output parameters

After completion of the *model checking*, *flow calculation* and *plot generation* you obtain the following concentration distribution after 200 days:
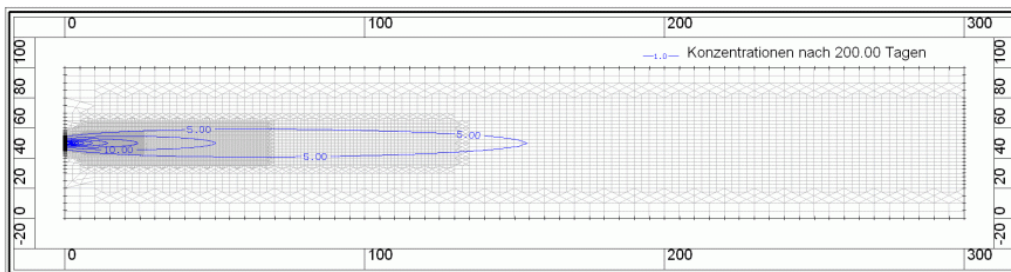
Fig. 79: Concentration distribution after 200 days, isolines in an interval 5 to 50 mg/m^3 with a distance of 5 mg/m^3

2.3.4 Transport with inverted flow

The procedure of the "Transport with inverted flow" is described in chapter How To – Catchment areas.

2.4 Creating a 3D vertical model

In a vertical model, the geometric elements are discretized in the vertical in contrast to a horizontal model. The construction of a vertical model is, however, by the same rules as that of a horizontal model. Typical examples of the use of a vertical model are dams, retaining walls of dams or a flow-related consideration of tunnel or main cross-sections in the vertical.
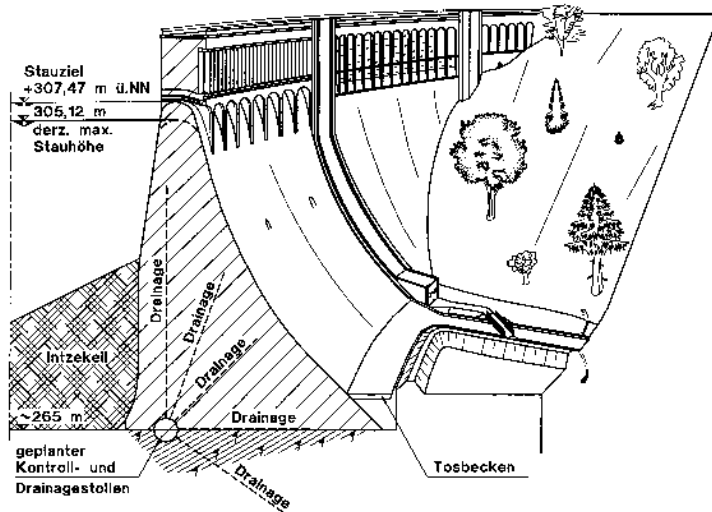
Fig. 80: Cross-section of a retaining wall of a dam

The discretization of this dam as a two-dimensional vertical model looks in SPRING like this:
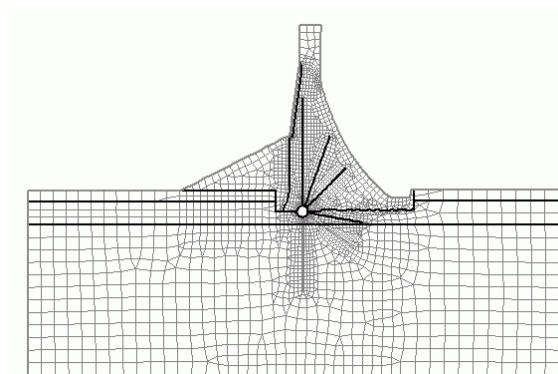
Abb. 81: Discretization of a vertical model in SPRING

To generate the 3D mesh, the 2D mesh is projected in the third dimension ("backwards"):
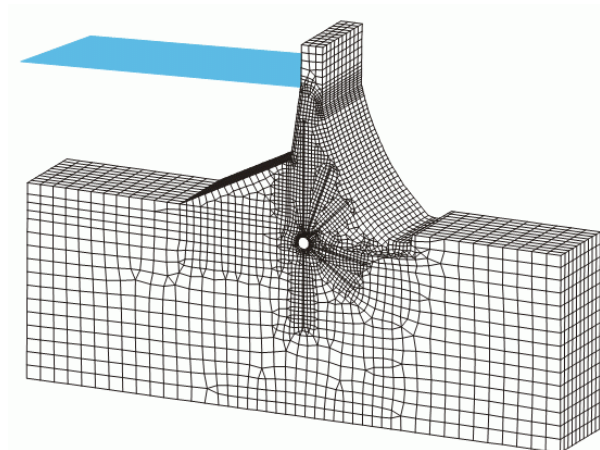
Fig. 82: Discretization of vertical 3D model (model type V-3D)

The corresponding boundary conditions in the present case look like this:
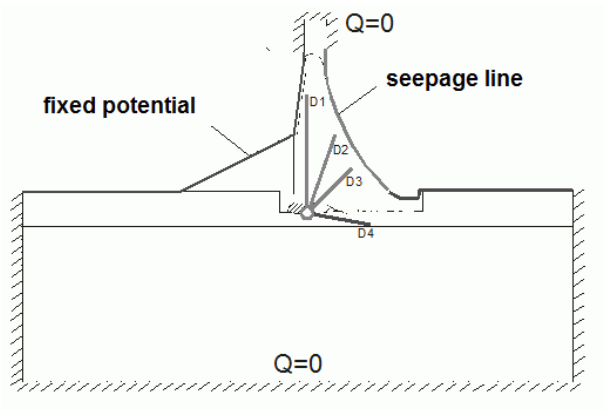
Fig. 83: Boundary conditions of a dam

2.4.1 Special features in the 2D vertical model

In the vertical model describes the X coordinate the position on the section line. The Y-coordinate describes the layer height. The specification of an areal recharge rate (attribute FLAE) in this model type makes no sense, but instead the attributes RAND or RANX are used. The thickness (attribute MAEC) describes the depth perpendicular to the vertical mesh.

The two figures below show how the coordinate systems are defined in reality (left) and in SPRING in the vertical model (right):
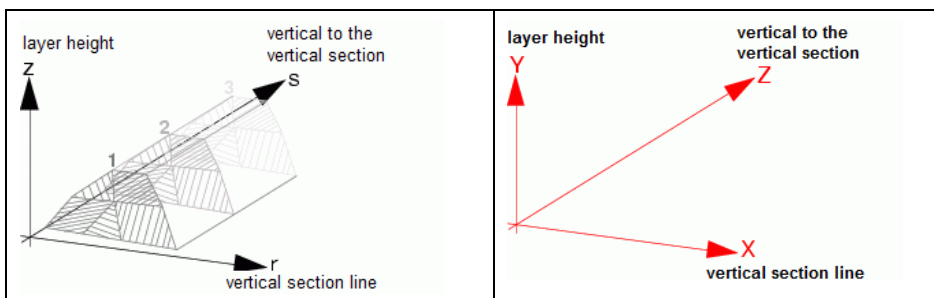
Fig. 84: Coordinate systems in reality (left) and in a vertical model in SPRING (right)

2.4.1.1 Creating a 2D vertical model from an existing 3D model

In SPRING, it is possible to "extract" a 2D vertical model from an existing 3D horizontal model.

For this, the background files of the 3D model as well as a previously created file *name.bpl* of a vertical section matching the model are required. From that, the information of the node coordinates is taken. With the following command used in a DOS shell:

>   *plogeo -mesh name.bpl*

the 2D vertical mesh is created. By assigning the attribute MAEC and the new assignment of any necessary flow-relevant attributes the 2D vertical model will be complete.

2.4.2 Extending to three dimensions

Perpendicular to the intersection, the 2D-vertical model is extended into the third dimension. Then the thickness is divided into 3D layers (*Mesh → 3D → Divide layer...*), so that the Z-coordinate (ZKOR) represents the depth. In contrast to horizontal 3D models, the Z-coordinates of the node layers of a vertical 3D model lay side by side and not above each other. The following two figures show the different layer arrangement and the associated coordinate system:
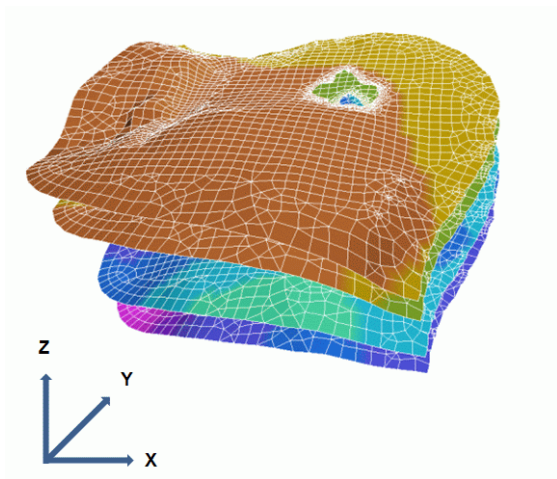
Fig. 85:
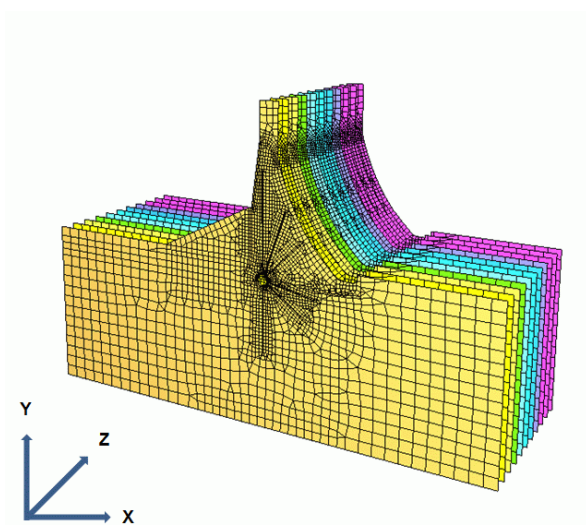
Fig. 86: Layer and coordinate axes in a vertical 3D model

The first node layer of the Z-coordinates (= GELA) lies on the vertical section line and has the value GELA = 0.0. After saving the finished 3D model the model type in the model file is set to "V-3D" with an editor. This is necessary because only the user (and not the software) can decide whether it is a 3D horizontal or a 3D vertical model. Since a 3D vertical model is a very specialized application, no query is done during saving the 3D model which type you have. This manual change of the model type is needed only once.

2.4.3 Presentation of sections in a 3D vertical model

The usual presentation of data and finite element mesh is done in 3D models in a top view/map presentation or in a vertical view / cross section presentation (Plot generation). These are presented for 3D vertical models in SPRING as follows:

2.4.3.1 Top view / map presentation in a 3D vertical model

Should the data of a deeper layer be presented in a 3D vertical model, corresponds this to the top view ("horizontal section") in a horizontal 3D model. The FE mesh is identical in these sections!
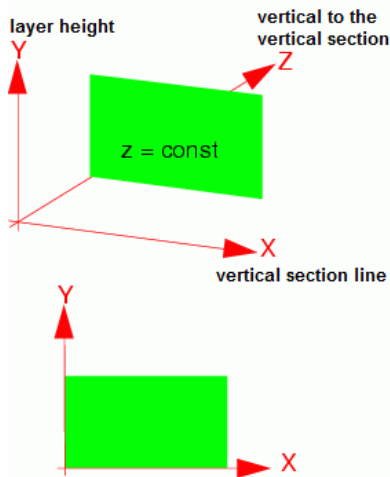
Fig. 5: Section parallel to the X-axis
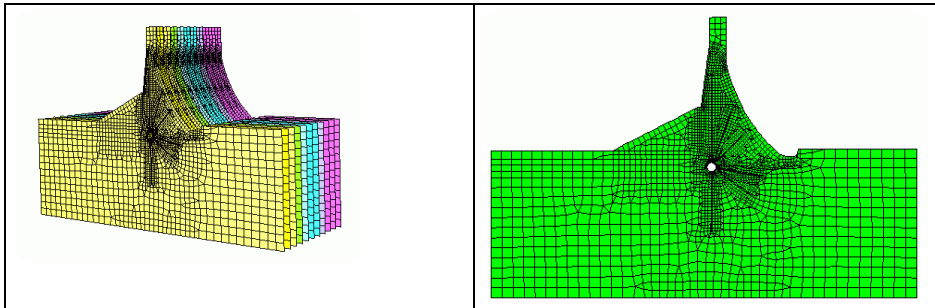
Fig. 87: Section parallel to the X-axis in SPRING

The specification of the layer number "4" in the *Plot generation → Top view / map presentation* (green layer in the left figure) results in the top view of the data of the 4.th layer in the X-Y coordinate plane (right figure).

2.4.3.2 View / cross section presentation in a 3D vertical model

If you want to create a section in the X-Z or Y-Z plane in a vertical 3D model, this corresponds to the view ("vertical section") in a horizontal 3D model. Which coordinate plane is shown reveals itself by the position of the intersection line defined by the user.

A section in the X-Z plane looks theoretically as follows:
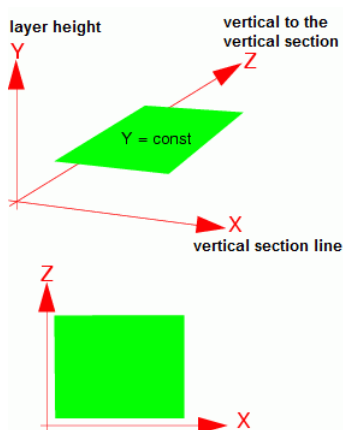
Fig. 6: Presentation in the X-Z plane

In SPRING it is implemented in the *Plot generation → View / cross section presentation* by choosing a section line that runs approximately parallel to the X-axis (green line in the upper figure). The result (lower figure) is a representation of the data (here: Z-coordinates of 0 to 2 m) in the X-Z plane ("view from above at the dam").
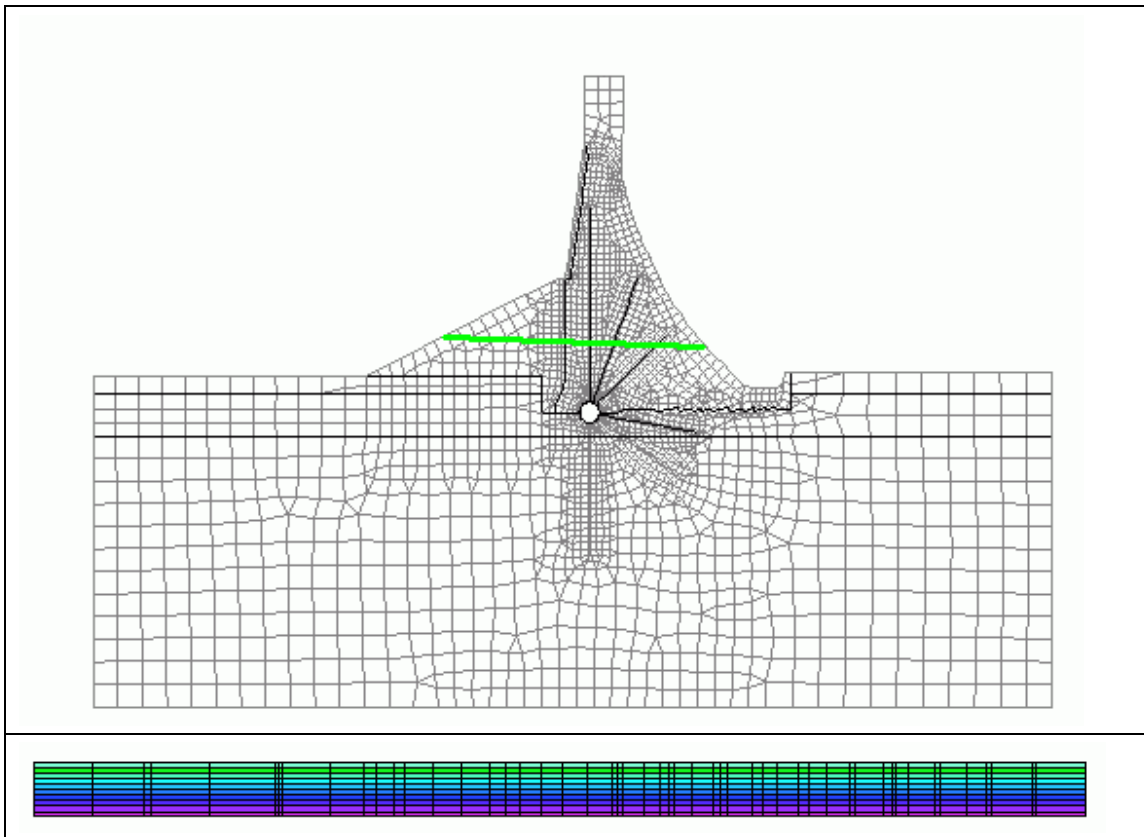
Fig. 88: Presentation in the X-Z plane in SPRING

A section in the Y-Z plane looks theoretically as follows:
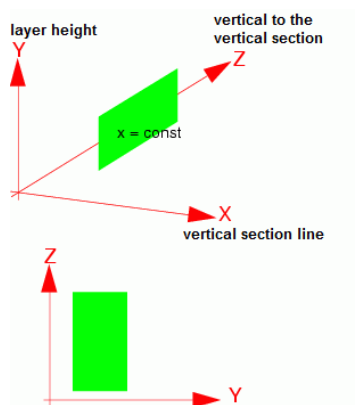
Fig. 7: Presentation in the Y-Z plane

In SPRING it is implemented in the *Plot generation → View / cross section presentation* by choosing a section line that runs approximately vertical to the X-axis (green line in the left figure). The result (right figure) is a representation of the data (here: K-values) in the Y-Z plane ("view from the right at the dam"). For a better understanding the resulting plot has been rotated 90 degrees to the left.
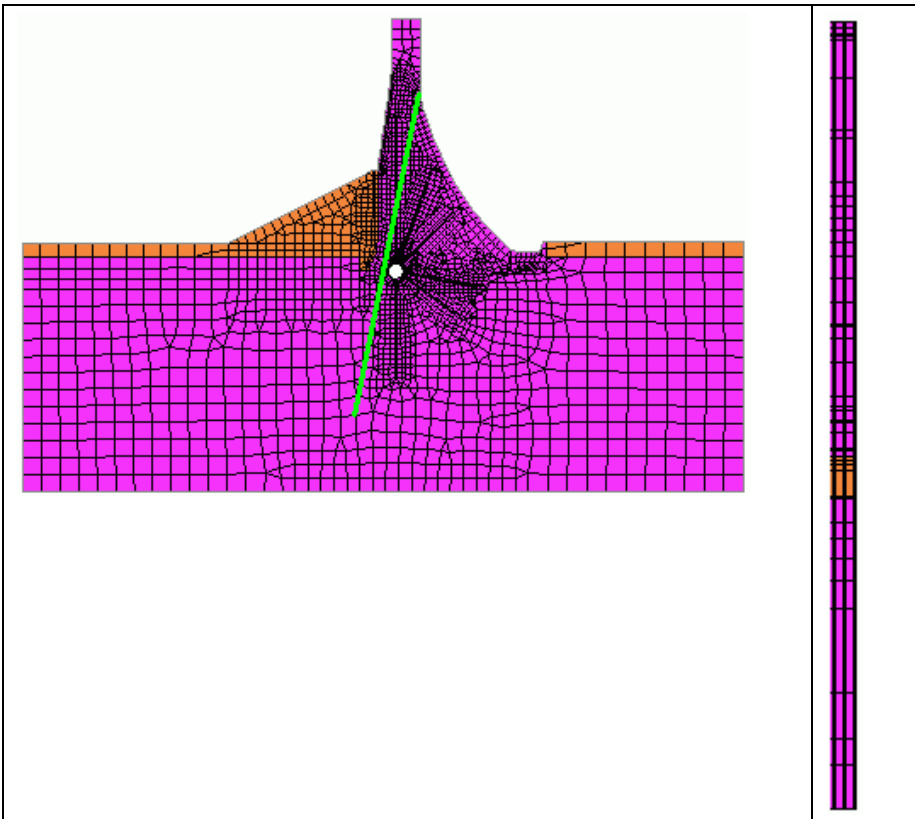
Fig. 89: Presentation in the Y-Z plane in SPRING

2.4.4 Special features in the plot generation

In the presentation of model and result data of a vertical 3D model the different allocation of the coordinate axes has to be considered. Moreover, not all sections are shown as you might intuitively be expected.

Since the usual presentation of the free surface does not make sense in a vertical 3D model, the presentation of the free surface in a vertical 3D model requires a detour on the following procedure:

- The Y-coordinates (= layer height) are saved as the attribute KKKK (*Attributes → Copy → Attribute-wise...*).
- Then the attribute KKKK is subtracted of the calculated potentials (*Attributes → Import model / result data...*) and stored on any node attribute (*Attributes → Computation → Intercalculation…*). The free surface is obtained as the contour lines of this attribute with value = 0.0.
- Hiding of the results above the free surface is not possible.

3 How To

In this chapter, special procedures are described that occur over and over during creation of groundwater models.

3.1 Determination of catchment areas

A catchment area is the whole area, from which the flow from the surface and subsurface flows at some point together. This point may be a withdrawal well or the mouth of a water in a receiving water course. The edge of the catchment area is the water divide.

The area is mainly determined by the topographic and geological conditions. The catchment area in a groundwater model can be determined by stream- or path lines or Schlieren or in a transport model by the inverse flow. The procedures are explained in the following.
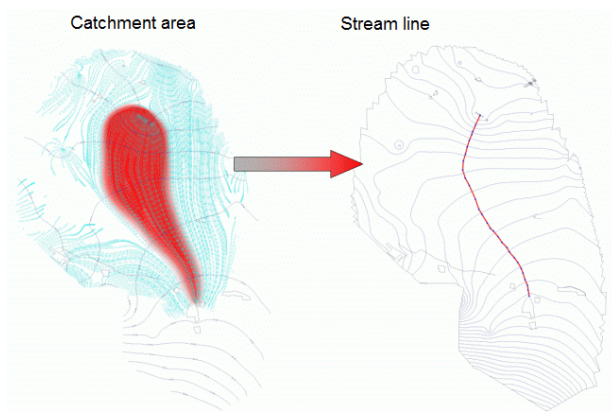
3.1.1 Theory of the path line calculation

A path line is the line resulting from following the movement of a single water particle over a certain flow time t. It is computed by following the flow path of an imaginary water particle through integration of the flow field. Path lines are calculated using the field velocity v_a, but not the Darcy velocity v_f. The field velocity is the actual interstitial velocity referring to the mesh flow area. It is obtained by:

$$v_a = v_f\, / \, n_d$$

with

v_a = field velocity

v_f = Darcy velocity

n_d = pore volume relevant to flow (effective porosity)

Two factors strongly influence the accuracy of path lines:

- First, since each element has a constant calculated velocity as a result of the linear shape function for potential heads, areas with changing potential gradients require a fine discretization.
- Second, the transition from one element to the next must be smooth.

To achieve this, the path line is first followed a certain distance (s) with the initial velocity; then in the next step, a new velocity is interpolated and the line is continued. The distances chosen depend on the homogeneity of the flow field.
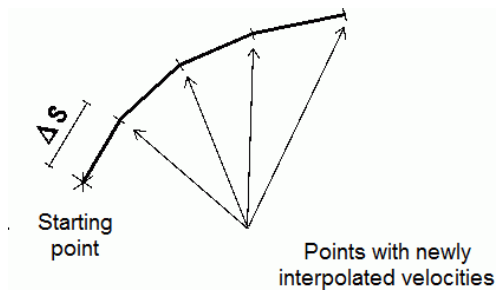
Fig. 90: Computing a path line

Exfiltration wells are difficult points for computing a path line. They must be considered as a secondary condition. Otherwise, the changing flow directions result in zigzag lines:
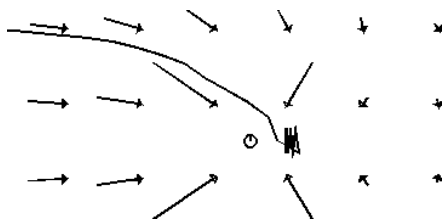
Fig. 91: Computed path lines at a well without any secondary condition

Better results are obtained by checking whether the computed path line has reached the "capture radius" of the well. Then the path line will end directly at the exfiltration node:
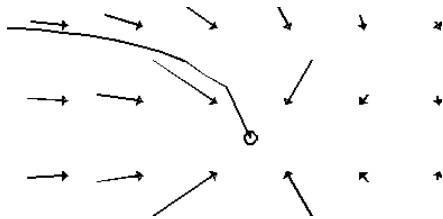
Fig. 92: Computed path lines at a well with "capture radius" depending on the exfiltration rate and the distance

If one of the following criteria is fulfilled, the computing of the path line ends:
- the defined time or distance is reached,
- the outer boundary of the model or a free water surface is reached,
- the path line is captured by an exfiltration well,
- the distance between two computed path line points is less than the defined limit.

Die Umsetzung bzw. die erforderlichen Eingaben in SPRING werden im Kapitel

The implementation and the necessary entries in SPRING are described in chapter "Calculating and plotting pathlines" (p. 63) .

The starting point of a path line can be a node, the centre of an element or any point with the coordinates x, y, (z). The direction for following the path line is either the flow direction or the direction opposite to the flow. The latter one is useful for determining the protection area around a well (e. g. 50-days-zone). The *plot generation* (p. **Fehler! Textmarke nicht definiert.**) can mark time intervals along the path lines. Finally, isochrones can be manually constructed with them.

The computation of transient path lines differs from the above procedure only in that the velocity field changes continually. Therefore, in addition to the distance s, the particle travel time t must be checked. It cannot be greater than the time-step length T in the transient flow calculation.

3.1.2 Calculating and plotting path lines

For the presentation of path lines the checkbox "Path lines" has to be enabled in the flow calculation with the module GEONEU (steady-state flow) or INSTAT (transient flow). The following window appears in GEONEU in which the criteria for the path lines are established:
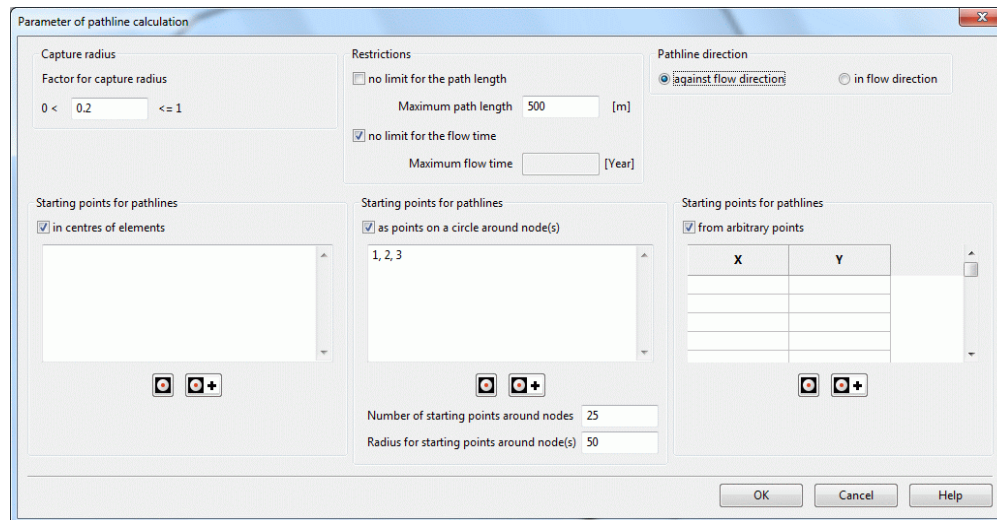
Fig. 93: Input for path lines

The input window of the module INSTAT differs by the lack of the input blocks "restrictions" and "pathline direction".

Capture radius

While tracking path lines on the basis of the calculated velocities near a well, the path line will run directly toward the well only in exceptional cases. To avoid oscillation of the path line around the well, a check is performed to determine if the path line passes within the area of influence of the well. If yes, the path line will be captured by the well. Only nodes lying on elements having a well as one of its corner nodes are considered. The "capture radius" is actually not a radius but a scale factor for the path line coordinates. The capture criterion is therefore dependent on the element size; in addition it is weighted according to the pumping/injection rate at the well. If necessary, the default value can be changed (always > 0).

Restrictions

To limit the path lines, two parameters can be used. You can define the maximum length (m) and/or the maximum flow time (unit of time defined in the *.net-file).

Pathline directions

The direction of the path line can be either in or against the flow direction.

Starting points for pathlines

When selecting the menu items "*Starting points for path lines* " open windows in which the starting points are established. Using the button 🔘 or 🔘➕ an interactive selection in the user interface is possible. A combination of all three types of starting points is allowed.

In the path line calculation is a porosity of 0.2 considered. It can be changed via the batch file of the flow computation.

A typical application of the path line representation is the issue of catchment areas of withdrawal wells, e.g. which area in the vicinity of the well needs to be shown as a water protection zone (e.g. 50-day protection zone). The representation contrary to the direction of flow provides this.

After the flow calculation the plot representing the path lines has to be created with the plot generation. This is done via *File → Plot generation →Top View/Map presentation* and then selecting *Result data → Path lines*. The following window opens:
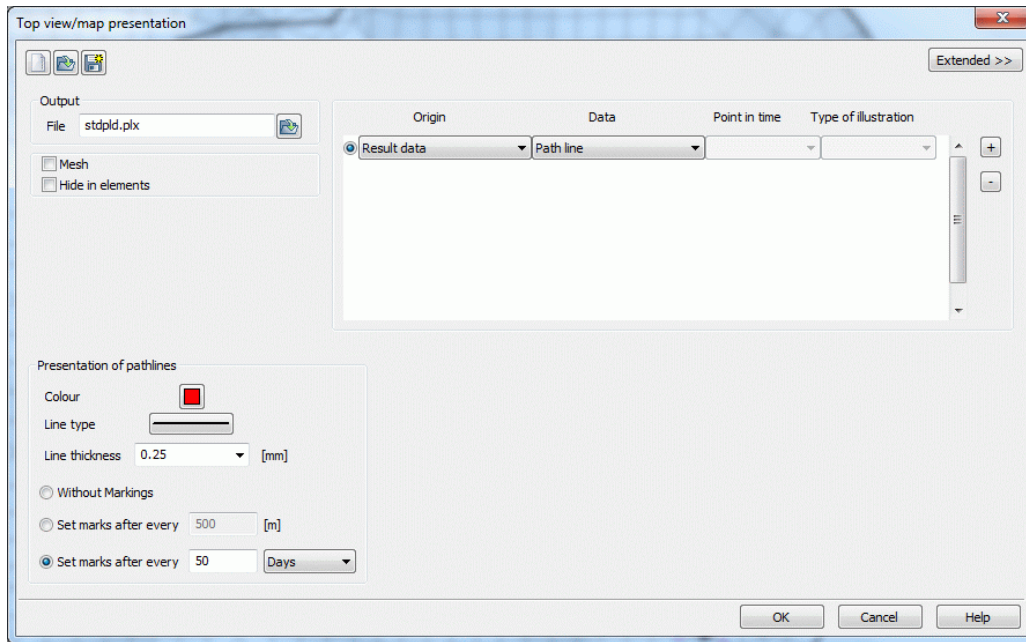
Fig. 94: Criteria of path lines in the plot generation

The statement "Set marks after every 50 Days" means that on the path line a marker after every 50 days is set. To identify the 50-day protection zone of a well by the time markers, the corresponding 50-day isochrone can be constructed manually. For the following image, the above parameters and the node number of the wells (1, 2, 3) were used:
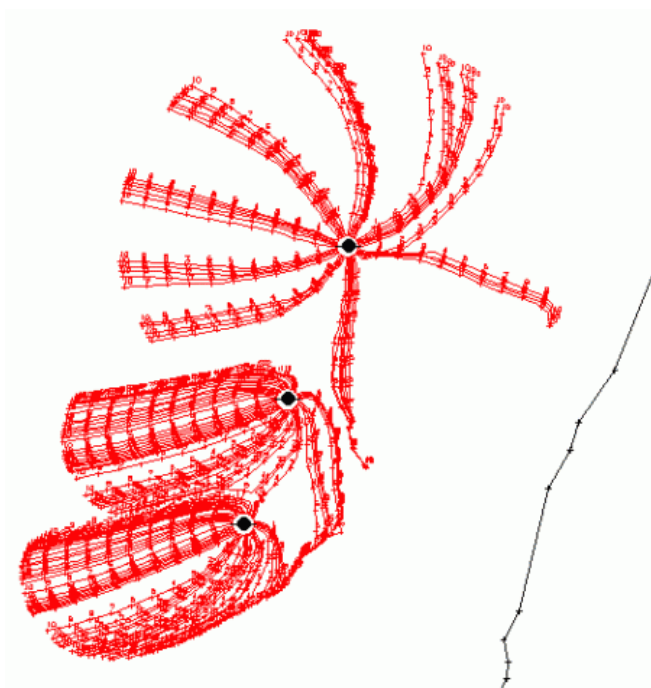
Fig. 95: Path lines with a length of 500 m and a marker after every 50 days

On this picture the parameters set in the flow calculation are clearly visible. The path line ends after reaching the 500-m limit, the distance corresponds to the well "radius for the starting points around node(s) = 25 m, and there are 50 path lines.

However, it requires a little practice, to obtain a meaningful picture of this problem.

3.1.3 Theory of the stream line calculation

In the flow field, a streamline represents a dividing line through which no water passes. Therefore, the mass flow rate of water between two streamlines is constant (assuming no infiltration from the top). These theoretical basics limit the calculation of streamlines related to mass flow rates in **2D horizontal models**.

An important aspect for streamlines, which is not taken into account for path lines, is the transition between two elements with different thicknesses. Mass balance requires closer streamlines in larger thicknesses than it does in smaller thicknesses. The streamline results from an equation system at each node. The mass balance defines that the inflow rate must always be equal to the outflow rate (plus possible infiltration rates).

The starting points are also determined by the mass balance for the nodes. First, a limit mass rate per streamline must be defined. The number of streamlines at each well node results from dividing the mass flow rates at the nodes by the limit rate. The streamlines are distributed to the adjacent element sides depending on the mass flow rates passing them. Starting at an arbitrary point, the rates are summed until the sum reaches the limit rate. At this point, the next streamline has its starting point. Starting at the computed points, the streamlines are followed through the next element and again over a side, and so on. In the same way, starting points along the outer boundary or receiving water courses are determined. The inflow and outflow rates are summed until the limit is reached.

The backwards-computed streamlines end at the side of an element adjacent to a node which has an inflow. This can be either a boundary node with a computed inflow rate or any node of the model with an infiltration rate (even from areal recharge).

In general, a horizontal model must take into account the areal recharge rate. It is assigned to all nodes as a small infiltration rate. Therefore, each node can be the endpoint of a backwards-computed streamline. This results in less fewer lines, e.g. along the drainage area of an exfiltration well, and represents the distribution of areal recharge. However, this effect does become apparent until a large number of streamlines per well has been generated.

The computed streamlines not only show the flow paths of the groundwater, but give an overview of the flow rates. Very close streamlines represent large mass flow rates, while widely separated ones represent small mass flow rates. In addition, the quality of the calibration can be checked with the help of streamlines. Individual elements with a much higher or lower permeability than the adjacent elements cause sharp bends of the stream lines.

The graphic representation of streamlines strongly depends on the quality of the finite element mesh, because large discontinuities at the element boundaries cause lateral displacements of the lines.

3.1.4 Calculating and plotting stream lines

The calculation of stream lines is possible only after a two-dimensional horizontal flow calculation (steady state or transient). To illustrate the stream lines (= quantity-based path lines) they have to be exported: *File →* *Export → Stream lines*. You will get this input window:
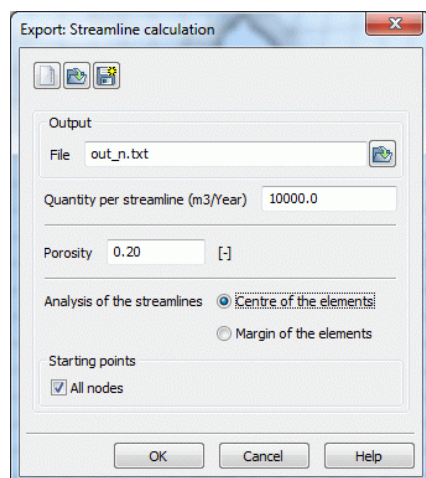
Fig. 96: Streamline calculation

The declaration of the "quantity per streamline (m³/year)" requires a bit of sensitivity for the available water quantities in the model. The example is calculated with 10,000 cubic meters per year, and there are stream lines run in all nodes (specifying individual node numbers is allowed only at withdrawal wells). After the data

export the result data "path lines" can be selected in the *plot generation*. The input window is the same as for the path lines and requires the same parameters. Here, "Set marks after every 100 m", the result is the following picture:
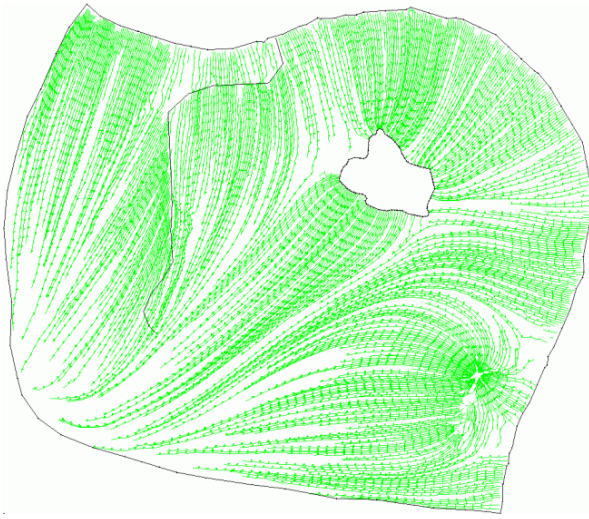
Fig. 97: Quantity of streamlines: 10.000 m³/year

Streamlines close together mean large quantities of water, in areas with sporadic stream lines flows only few water. Sharp bends in the flow curve are an indication of leaps in the K-values of the adjacent elements.

3.1.5 Determining a catchment area with Schlieren (FLIC)

Often the question of the catchment areas of water courses or wells arises, e.g. in the determination of drinking water protection zones or the influence area of a withdrawal well.

The program FLIC (Konrad-Zuse-Centre for Information Technology Berlin) can visualize the velocities computed in a flow calculation by the help of Schlieren. This visualization provides a much better image of flow paths than the velocity arrows used in the plot generation. This is especially important for transient and 3D models where a streamline calculation is not possible.

In the example model, there are two catchment areas of wells, which can be distinguished by the presentation of Schlieren clearly from each other. The procedure for presenting the Schlieren is this:

After a successful flow calculation, the data has to be exported via *File → Export→ Schlieren illustration*. The following input window appears:
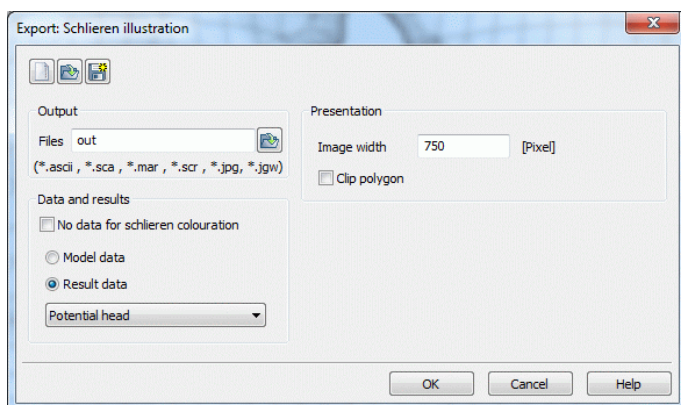
Fig. 98: Input for the Schlieren illustration

The output files are called "out.*" by default, the name can be changed by the user. Attached are the six output extensions .ascii, .sca, .mar, .scr, .jpg and .jwg.

The presentation of the Schlieren is done in the format. ".jpg" with the necessary geo-referencing file ".jwg", which allows a faithful representation of coordinates. The colour of the Schlieren can be changed by selecting

a type of data (model or result data). In the example, the potential heads were chosen as a colour sensor. The following picture results:
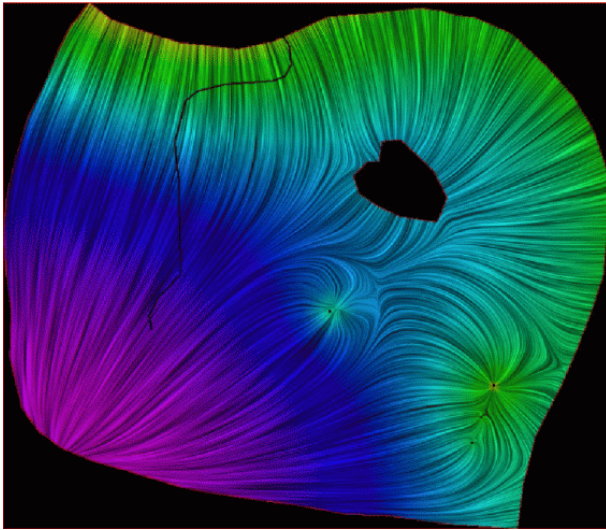
Fig. 99: Schlieren illustration of the potential heads

The image is called via *File → Import File → Overlay file: *.jpg*. You can see clearly that the course of the Schlieren corresponds to the course of the path and stream lines. The other files can also be used in other geographic information systems (e.g. ARC INFO).

By using the structure menu of SPRING the catchment areas of the (line of -) wells can be digitized along the course of the Schlieren and saved as area structure.
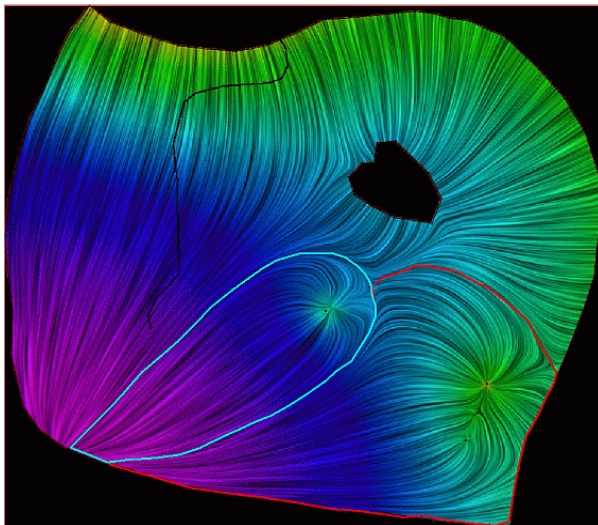
Fig. 100: Digitized edges of the catchment areas of the well (cyan) and the line of wells (red)

After deleting the Schlieren representation and overlaying the topographic map the catchment areas are referred to their geographical site. Eventually you have to change the display of the structures from a polygon into a filled area.
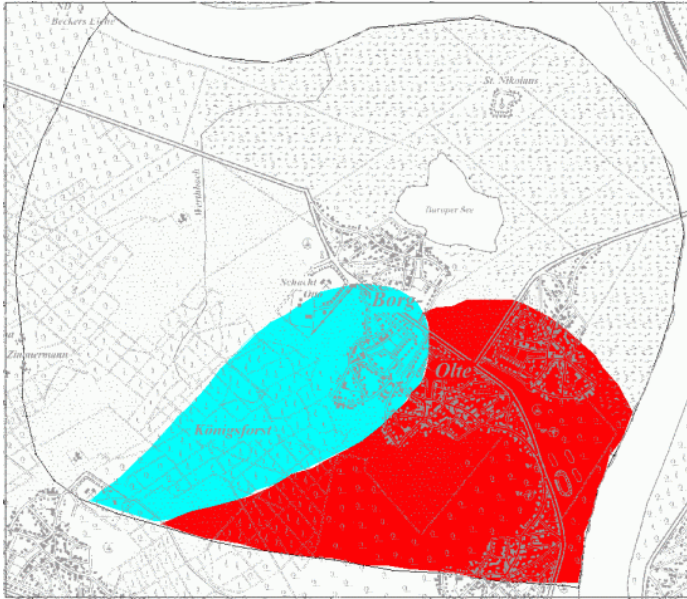
Fig. 101: Catchment areas of the wells

3.1.6 Geo referencing of Schlieren illustrations

For geo-referencing of files in *.jpg or *.jpeg format, a file of the type *.jgw has to be created. For this, the extreme coordinates of the model or the model boundary xmin/ymin and xmax/ymax in world coordinates (usually Gauss-Krüger coordinates) and the display size of the Schlieren illustration in pixels (sizeX/sizeY) is needed.

The coordinates are for example determined using the menu item *View → Show coordinates*. The extension of the Schlieren illustration can be seen in the output file *.scr, after the data was exported for the Schlieren illustration (*File → Export → Schlieren illustration*). The file can be opened with any editor. The line "size" refers to the pixel size of the plot.

If the required numbers are determined, an ASCII file is created with an editor. It contains the following numbers (upper suspension point) and scaling:

```
1st line: (xmax-xmin)/sizeX
2nd line: 0.0
3rd line: 0.0
4th line: (ymin-ymax)/sizeY
5th line: xmin
6th line: ymax
```

The file will be renamed by the name of the Schlieren image with the extension *.jwg. This geo referencing file is automatically evaluated when opening the Schlieren image with SPRING.

Tip: By increasing the pixel value in the Input window of the data export the resolution increases as well.

In the current SPRING-release, the Schlieren images are automatically geo referenced in horizontal and 3D models (only when representing defined layers).

3.1.7 Transport with inverse flow

It is possible to describe a catchment area in consideration of advection- and dispersion-/ diffusion processes. Thus also the local heterogeneities can be simulated with the dispersion term which are not or only limited ascertainable with conventional methods of velocity tracing.

The base of this approach is described in detail in *Uffink, F., Application of Kolmogorovs backward equation in random walk simulations of groundwater contaminant transport; Contaminat Transport in Groundwater, Kobus & Kinzelbach, 1986 and Wilson, J.L. and J. Liu, Backward particle tracking to find the source of pollution in waste management. Waste Management: From Risk to Reduction, 1995.*

If the concentration in the well is given with 1.0 the probabilities of the arriving particle at the well will be calculated as a concentration value. With the presentation of isoconcentration areas from 0.05 to 1.0 the 95%-propability of the arrival of a particle at the withdrawal well will be calculated.

The figures show a 3D example. Here the two aquifers are separated by a groundwater aquitard. The withdrawal wells deliver from the bottom aquifer. The 50 day area is shown in the vertical and horizontal section:
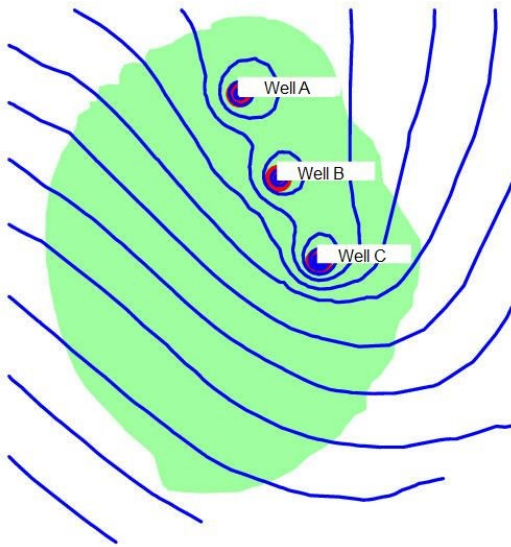
Fig. 102: Horizontal flow
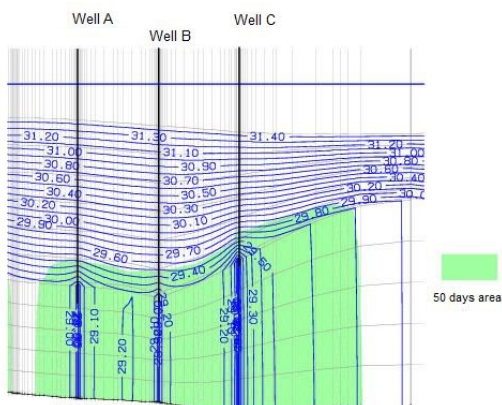
Fig. 103: Vertical flow

A detailed description of the procedure of inverted flow is beyond the scope of this manual. During a training session the calculation of transport processes with inverted flow can be discussed by request.

3.2 Mass balances

A mass balance is the volume-based coverage of the amount of water which flows during a certain period of time in and out of an area. Inflows are covered with a positive sign outflows are covered with a negative sign. Within the model area control lines define lines or surfaces on which the flow can be calculated.

3.2.1 Theory of control lines

Streamlines and path lines show where the water reaching the well comes from. With the help of control lines, one can estimate the mass flow rate coming from a certain direction. The rate is obtained by the flow velocity and the corresponding percolated area. Three-dimensional calculation determines the rates over the depth depending on the layers.
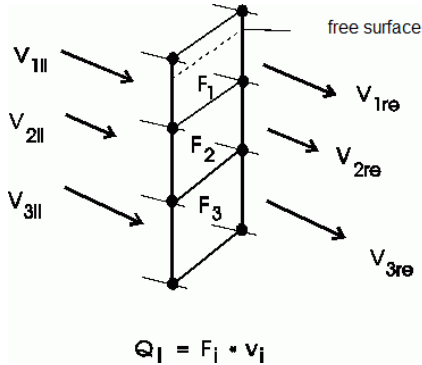
Fig. 104: Calculation of the quantities flowing through a vertical section

3.2.2 General hints for using control lines

The control lines are defined in the model file (*.net) by assigning the attribute KONT. The attribute KONT is independent of the model type (2D or 3D).

The lines can range from a 2D area into a 3D area however they must not run along the 2D/3D-boundary.

In the case of a fixed potential head boundary, mass flow rates crossing the boundary should be obtained by summing the nodal quantities, as these are more accurate (attribute BILK).

The control lines must always follow element edges. Therefore, velocities and flow-through areas of two elements are available. The adjoining areas are averaged, since they are generally of similar size. In the case of velocity, the user can choose between the average value, or the value of the left or right adjoining element. Obviously, the accuracy of the mass flow rate calculation strongly depends on the discretization and the discontinuity of the elements.

Using the attribute KONT the following items have to be considered:

- The control lines are calculated when the appropriate button is enabled in the main dialog box of the flow module.
- Results for each node and the whole line are written in the output file of the flow calculation (out.*). It needs a little bit attention to classify the control lines in the output file: The value of the control line does not match necessarily the number of the line in the output file! In the model file (*.net) the attribute values are sorted from the maximum (e.g. 12.0) to the minimum (e.g. 1.0). But in the output file the first line corresponds to the control line with the maximum value (12.0).
- If you define a closed control line, e.g. around a well, you must insert the last node (= first node) manually in the model file, otherwise the line is not closed.
- In the module GEONEU you can assign the attribute KONT to nodes along an element diagonal. But if later on a mesh refinement is done, the sequence of the nodes can be incorrect and a warning is given. Then a new assignment is required.
- In the module SITRA it is impossible to calculate a control line along an element diagonal (Warning nr. 34)!
- The sign of the calculated rates depends on the assignment direction:
- Flow over the control line from RIGHT to LEFT is defined positive in the direction of assignment flow over the control line from LEFT to RIGHT in assignment direction is defined negative.
- Three-dimensional calculation determines the rates over the depth from the layer where KONT was assigned down to the model base.

The following chapters describe concrete examples of control lines.

3.2.3 Determining the flow rate to a withdrawal well (KNOT)

To calculate the flow to a withdrawal well a closed control line is placed around the well. In the example, a well (KNOT) is defined in the centre of the model with an amount of withdrawal of -800000.0 m³ / year.

The control line (red) has a distance of about 400 m from the well. The area of the control volume amounts to 560505 m². This results in a groundwater recharge (FLAE = 0.200 m³/m²/a) of 112101 m³/a in this area. In the current state the FE-mesh around the well looks like this:

Fig. 105: Initial state of the well calculation

After running SITRA (partial saturation, 5 iterations) a flow into the control volume of 702533 m³/a results. From adding the groundwater recharge of 112101 m³/a you get a total amount of 814634 m³/a respectively 1.8 % deviation.

The next step is a local mesh refinement around the control volume. The following flow computation presents a total inflow of 112101 + 690024 = 802125 m³/a. So the refinement minimizes the deviation to 0.3%.

Fig. 106: Mesh refinement around the well

3.2.4 Determining the quantity of bank-filtered water of a main receiving stream (POTE)

If wells are positioned near a main receiving stream an undefined percentage of bank filtrate flows to the wells. To determine this part of the mass balance the attribute KONT is assigned to a node line parallel to the main receiving stream and a flow computation is started. After that the calculated control line quantities are visualized and you can read the flow rates directed to the wells, as shown in the following figure (section):
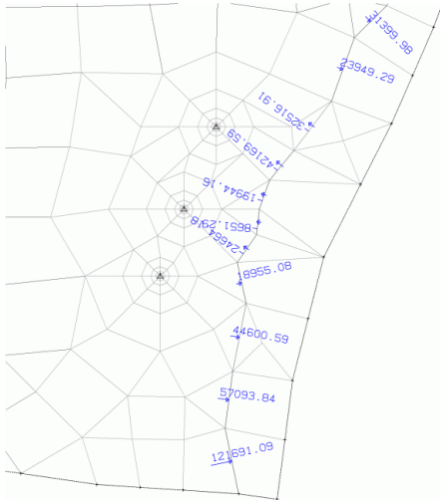
Fig. 107: Bank filtrate rates of the main receiving water course

An amount of 127946 m³/a results when adding the filtrate rates. This corresponds to 9% of the total withdrawal quantities (1450000 m³/a) of the wells.

3.2.5 Groundwater exfiltration into a receiving water course (VORF)

If a receiving water course exists in the investigation area which is defined by the attributes VORF and LERA (or LEKN) you can balance the in- and outflow to the water course with the help of control lines. In the known example file the local area around the receiving water course is refined one respectively two times. Then two control lines are created more or less parallel to the receiving water course. In this control volume, which is limited on the western and eastern side by the control lines, in the north by the main receiving water course and in the south by an imaginary line between the ends of the control lines, the in- and outflow rates are balanced. The mass balance in the steady state must have identical in-and outflows.

The following parameters are necessary:

- Inflow (+) and outflow (-) over the control lines in the control volume (KONT)
- Groundwater recharge in the control volume (FLAE)
- Leakage quantity of the water course (leakage coefficients LERA, LEKN)
- Outflow through the fixed potential heads at the embouchure (Mass flow rates)

Creating the control lines

The amount of water which is calculated by the control line computation corresponds to the groundwater runoff of the receiving water course. Looking at the Schlieren shows clearly that the water course has only a minor connection to the groundwater because the Schlieren run mainly parallel and not **to** the water course.
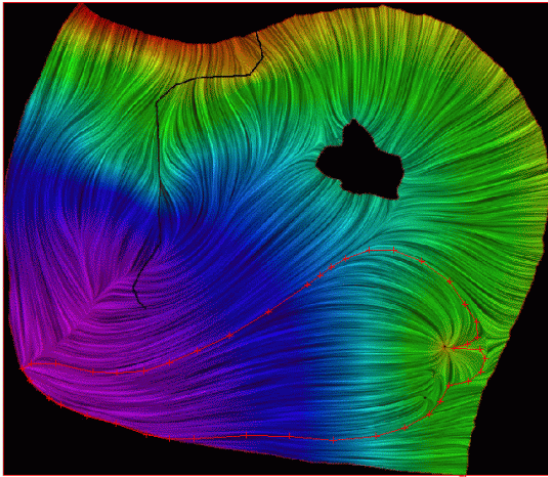
Fig. 108: Schlieren image of the model

The control lines are created in SPRING with *Attributes → Assign → Direct → KONT*. After running the flow computation the quantities flowing over each element side and of the whole control line can be read in the output file (out.* after GEONEU or SITRA).

The signs of the given quantities depend on the assignment direction of the nodes: Looking **IN** assignment direction, the flow from right to left has a positive sign! Otherwise the sign must be negative.
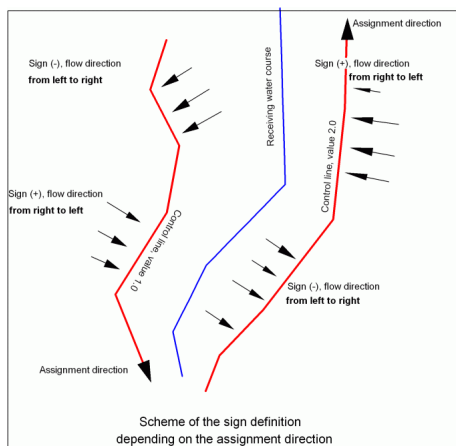
Fig. 109: Scheme of the flow direction depending on the assignment direction
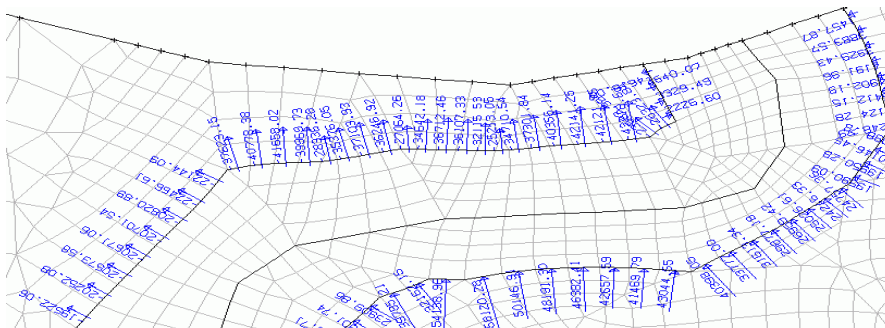
Fig. 110: Control lines and quantities (clipping)

An inflow (+) arises from the eastern side in the control volume and an outflow (-) results at the western side of the control volume.

Although the total sums of both control lines are positive, the value of the western line must be put negative because it is an outflow of the control volume. After the flow computation (SITRA, partial saturation, 5 iterations) the following amounts result: western line: (-) 740284 m³/a, eastern line: (+) 808193 m³/a.

Calculation of the groundwater recharge

After creating the control lines an area structure is created which covers the entire control volume (*Structure → New → Flat*). With *View → Calculate area* (or the corresponding button ◁?) and selecting the control volume the area of it is shown in the status bar of the main window. It amounts to 1975284 m². Multiplied with the groundwater recharge rate of 0.200 m³/m²/a a groundwater recharge of 395057 m³/a results.
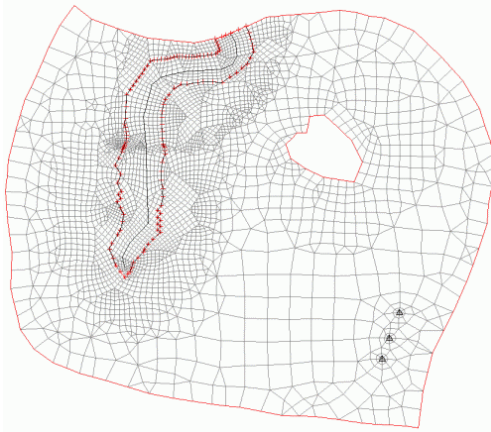
Fig. 111: Flat structure of the groundwater recharge calculation

Determining the leakage rates

For calculating the leakage rate the attribute BILK has to be assigned to the nodes of the water course. Then the total leakage quantity is given in the output file: it amounts to - 209405 m³/a.

Determining the mass flow rates

The water course flows into a main receiving stream, so the outflow through the fixed potential heads has to be considered as well. The quantities are read in SPRING with *Attributes → Import model data/computation results... → aaa → mass flow rates → KKKK*. With *Attributes → Computation → Sum attributes... → in polygon* you can sum up the mass flow rates inside the control volume. The mass flow rates of the left and right "boundary nodes" of the control volume are averaged and added up only one time. So the total mass flow quantity in the control volume amounts to - 260292 m³/a.

If all required values are determined you can balance them [m³/a]:

395057	+ 808193	- 740284	- 209405	- 260292	=	-6731 [m³/year]
Groundwater recharge	Inflow	Outflow	Leakage	Mass flow	=!	0

The deviation of - 6731 [m³/a] corresponds to 0.6% of the negative water quantity.

The mass balance is a useful tool to check the position of the control lines. The exactness of a control line depends considerably on the discretization and on the flow direction: the more upright the flow the more accurate the from the velocity derived quantity (Q = V * K * I).

3.2.6 Quantity of water flowing through a lake (GLEI)

When creating a groundwater model, some nodes have the same potential head, but the value is unknown. This is valid for:

- surface waters in horizontal and three-dimensional models,
- along the wetted boundary of a vertical well in a vertical model,
- as a boundary condition for the border line between a two-dimensional and a three-dimensional area,
- the nodes of a well below each other in a three-dimensional model.

For all nodes defined with GLEI, the calculation obtains the same value for potential head. Numerically, the equations for these nodes are summed to one equation. They are considered as one unknown value.

Therefore, in three-dimensional models, no different exfiltration rates over the depth of a vertical well are required. The complete exfiltration rate can be assigned to one node. This is not valid for the case of several pumps with different extraction rates over the depth. Only in case of a constant pumping rate can the total extraction rate be assigned to any one of the nodes.

3.2.6.1 Realization in SPRING

In a horizontal model the attribute GLEI (equal potential heads) is assigned to the nodes of a lake. If a lake is characterized like that, it is impossible to read the in- and outflow rates directly. First a flow (steady state or transient) calculation is done. Then you assign in SPRING the resulting potential heads to a user-defined attribute with *Attributes → Import model data/computation results... → aaa → result data: potential head → e.g. KKKK*. With *Attributes → Edit nodes* and picking a boundary node of the lake, you get the calculated potential head of the lake.

This value is assigned as attribute POTE to all boundary nodes of the lake, the attribute GLEI is deleted. At last the attribute BILK (integer value, identifies the balanced group) is assigned to these nodes to get the mass flow rates later on. After the flow computation you can read the mass flow rates of the boundary nodes in the output file. Or you display the resulting mass flow rate with the plot generation, where you can choose the quantity unit to be recalculated:
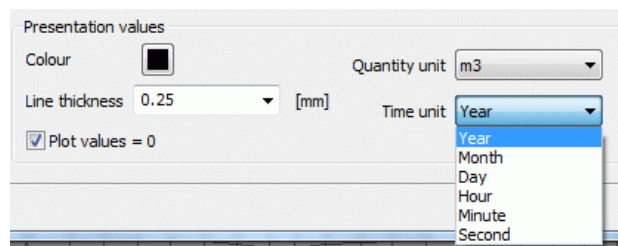
The unit m³/a is chosen. The following figure shows the lake with the calculated mass flow rates.
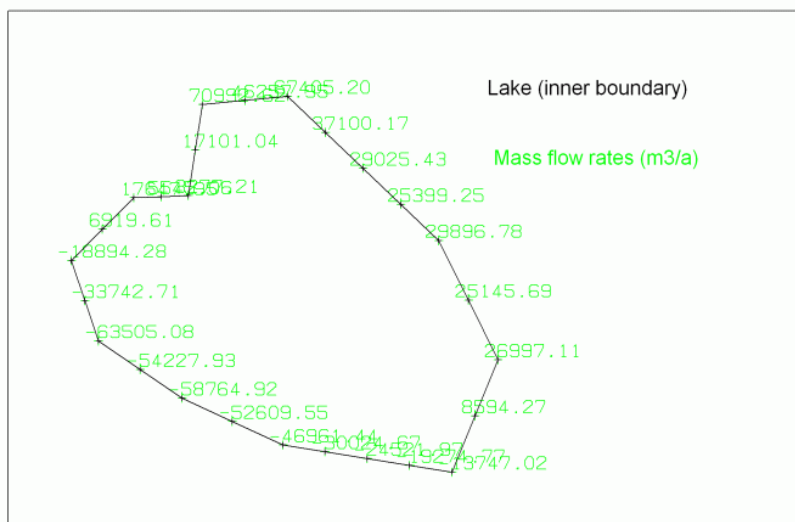
Fig. 112: In- and outflow of a lake

The quantity of water flowing through the lake $Q_{in} = Q_{out}$ is calculated by adding the negative and positive values. Depending on the discretization the values differ from each other in a marginal percentage:

From adding the inflow (= negative sign → it is an outflow of the model because of the inner boundary) results a mass flow rate of 430646 m³/a. From adding the outflow of the lake (= positive sign → inflow into the model) results a mass flow rate of 422458 m³/a. The difference between the two rates amounts to - 8188 m³/a, which means a deviation of 0.5% of the water quantity flowing through the lake. A mesh refinement minimizes the

deviation to -1032 m³/a respectively 0.2%, the water quantity flowing through the lake amounts to ca. 416790 m³/a.
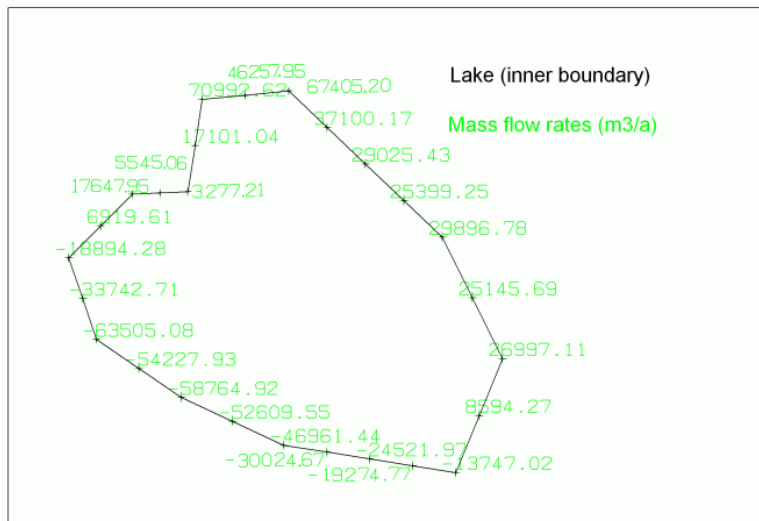
Fig. 113: In- and outflow of a lake after a mesh refinement

3.3 Calculating groundwater recharge rates

For calculating the average groundwater recharge rates in SPRING there are three, partially for Nordrhein-Westfalen developed, methods available:

- Recharge rate calculation by Schroeder and Wyrwich, originally developed for the region near Münster [*Schroeder und Wyrwich (1990): "Eine in Nordrheinwestfalen angewendete Methode zur flächendifferenzierten Ermittlung der Grundwasserneubildung" - Deutsche Gewässerkundliche Mitteilungen, 34, Koblenz 1990*.]
- Method of calculating the groundwater recharge by Meßer, originally developed for the Emscherregion [*Meßer (2008): „Ein vereinfachtes Verfahren zur Berechnung der flächendifferenzierten Grundwasserneubildung in Mitteleuropa"*]
- Method of the soil water balance

The element-wise calculation of the recharge rate takes place in SPRING via the menu *Attributes → Computation... → Groundwater recharge…*

3.3.1 Recharge rate calculation by Schroeder and Wyrwich

The method according to Schroeder and Wyrwich is based on the converted water balance equation:

Groundwater recharge = precipitation – total evaporation – direct runoff.

- **Groundwater recharge**: It will be calculated in SPRING from all input data and assigned to the attribute FLAE (column 16: "*") in the unit m³/m²/a. The time unit "year" is required in the model file.

- **Precipitation**: Precipitation in m/year for all elements or m³/m²/TU element-wise (column 16: „*") (attribute NIED).

- **Total evaporation**: The average total evaporation rate is taken into account depending on the soil type and land use class of the following table (values in mm/year, N means precipitation):

Land use / Soil type	Farmland, grass land	Deciduous forest	Mixed forest	Coniferous forest	Build-up areas	Water areas
Terrestrial sand soils	380	480	540	600	0.2*N	1.0*N
Terrestrial clay soils	440	540	600	660	0.2*N	1.0*N
Semi terrestrial soils	550	650	700	750	0.2*N	1.0*N

- **Direct runoff**: The still available water from the difference between precipitation and evaporation is then reduced by the direct runoff. Besides the parameters soil type and land use on farm- and grassland the relief energy or the slope is needed. The definitive slope is the maximum slope between two nodes of an element. The program computes this value automatically from the ground level (attribute GELA).

The direct runoff depends on the land use, for **farm- and grassland** it is given by:

Soil type	Relief energy [m/km²]	Slope [%]	Direct runoff [%]
Terrestrial sand soil	0 - 20	0 - 2	0
	21 - 90	2 -9	0 - 100
	> 90	> 9	100
Terrestrial clay soil	0 - 60	0 - 6	0 - 100
	> 60	> 6	100
Semi terrestrial soil	Without considering	Without considering	50

Other land uses are independent of the soil type and the slope:

Land use	Direct runoff [%]
Decidous, mixed, coniferous forest	0
Build-up areas	90*
Water areas	0

*: In contrast to Schroeder and Wyrwich the part of the direct runoff on build-up areas is set to 90% instead of 100%.

3.3.1.1 Input data

The following data must be available for all elements or nodes in the model file:

- Ground level (attribute GELA). To calculate the relief energy and the slope.

- Precipitation (attribute NIED).

- Soil type (attribute NSBT). The soil type must be specified in the model file for all elements according to the following classification:

Soil type	Classification (= NSBT)
Terrestrial sand soils	0
Terrestrial clay soils	10
Semi terrestrial soils	20, 21, 22

■ Land use (attribute NSFN): The land use must be specified in the model file for all elements according to the following classification:

Land use	Classification (= NSFN)
Farm-, grassland	0
Decidous forest	1
Mixed forest	2
Coniferous forest	3
Build-up areas	4
Water areas	5

For the determination of the attribute NSFN the digital land use map of the Regionalverband Ruhr (regional association Ruhr) can be used.

Depending on the format, the data can be imported as a structure and assigned in SPRING.

The transformation of the RVR-Code (State: 07/2002) into the classification of the method by Schroeder and Wyrwich is possible without additional model data via the menu item *Attributes → Computation → Groundwater recharge → According to Schroeder Wyrwich → Determine land use from RVR-Code (KVRN → NSFN)*.

With this method the groundwater recharge rate can be determined for each finite element according to the above-described water balance equation via: *SPRING → Attributes → Computation → Groundwater recharge → According to Schroeder Wyrwich → Calculate groundwater recharge (NSFN+NSBT+GELA+NIED → FLAE)*.

3.3.2 Recharge calculation by Meßer

The method according to Meßer is based on the converted water balance equation:

Groundwater recharge = precipitation – total evaporation – direct runoff.

■ **Groundwater recharge**: It will be calculated in SPRING from all input data and assigned to the attribute FLAE (column 16: "*") in the unit m³/m²/a. The time unit "year" is required in the model file.

■ **Precipitation**: Precipitation in m/year for all elements or m³/m²/TU element-wise (column 16: „*") (attribute NIED).

■ **Total evaporation**: The total evaporation rate is taken into account depending on the climate zone, the land use, the soil type and the class of the depth to water table.

■ **Direct runoff**: The still available water from the difference between precipitation and evaporation is then reduced by the direct runoff. Besides the parameters soil type and land use on farm- and grassland the relief energy or the slope is needed. The definitive slope is the maximum slope between two nodes of an element. The program computes this value automatically from the ground level (attribute GELA).

A detailed description of this method is given on this website:

Ein vereinfachtes Verfahren zur Berechnung der flächendifferenzierten Grundwasserneubildung in Mitteleuropa

3.3.2.1 Input data

The following data must be available for all elements or nodes in the model file:

- Ground level (attribute GELA). To calculate the relief energy and the slope.

- Depth to water table (attribute FLUR): To determine the evaporation and the runoff. The attribute FLUR can be calculated in SPRING as difference between GELA – EICH (*Attributes → Computation → Intercalculation → GELA – EICH*).

- Precipitation (attribute NIED).

- Soil type (attribute NSBT). The soil type must be specified in the model file for all elements according to the following classification (nFK = usable field capacity):

Soil type	Classification (= NMBT)
Sand, low nFK	1
loamy sand, middle nFK	2
Loamy clay, high nFK	3
Loess soils	4
Pseudo gley	5

- Land use (attribute NMFN): The land use must be specified in the model file for all elements according to the following classification:

Land use	Classification (= NMFN)
Agricultural use	11
Farmland	12
Grassland	13
Fallow land	14
Without vegetation	15
Decidous forest	21
Mixed forest	31
Coniferous forest	41
Mixed vegetation, share of sealing 0%	51
Mixed vegetation, share of sealing 1-20%	52
Mixed vegetation, share of sealing 21-40%	53
Mixed vegetation, share of sealing 41-60%	54
Mixed vegetation, share of sealing 61-80%	55
Mixed vegetation, share of sealing 81-100%	56
Water areas with discharge	61
Water areas without discharge	62

Dumps, indifferent deposit	71
Deposit in use (without vegetation), low nFK	72
Deposit recultivated (mixed vegetation), low nFK	73
Dump in use (without vegetation), low nFK	74
Dump recultivated (Mischvegetation), low nFK	75
Mud pool without vegetation, high nFK	76
Roof areas drained	81
Asphalt, concrete drained	82
Compund concrete (new), drained	83
Mosaic-/small paving stones, drained	84
Grass gid stones, drained	85
Compressed, drained	86
Sealed, not drained	91
Asphalt, concrete not drained	92
Compounf concrete (new), not drained	93
Mosaic-/small paiving stones, not drained	94

To determine the data type NMFN standardized land use codes as RVR-code (attribute KVNR), CORINE land use data (attribute NCLC) or ATKIS data (attribute NATK) can be used. Depending on the format, the data can be imported as a structure and assigned in SPRING.

■   Slope class (attribute NMGK): The definitive slope is the maximum slope between two nodes of an element. The program computes this value automatically from the ground level (attribute GELA). The slope class must be specified in the model file for all elements according to the following classification:

Slope class (NMGK)	Relief energy [m/km²] or slope [%]
1	0 - 20 or 0 - 2
2	21 - 40 or 2 - 4
3	41 – 100 or 4 -10
4	> 100 or > 10

Via the menu item *Attributes → Computation → Groundwater recharge → According to Meßer 2008 → Determine classes of incline of ground (GELA → NMGK)* the slope class can be determined automatically in SPRING.

■   Class of depth to water table (attribute NMFK): The class of the depth to water table must be specified in the model file for all elements according to the following classification:

Class of depth to water table [m]	Classification (= NMFK)
Depth to water table 0 -1.0	1
Depth to water table 1.0 – 2.0	2
Depth to water table 2.0 - 3.0	3
Depth to water table > 3.0	4

Via the menu item *Attributes → Computation → Groundwater recharge → According to Meßer 2008 → Determine depth to water table (FLUR → NMFK)* the classes of the depth to water table can be determined automatically in SPRING.

■ Climate zones (attribute NMKL): The climate zones must be specified in the model file for all elements according to the following classification:

Climate zones, depending on the pot. evapotranspiration (ETpot)	Classification (= NMKL)
ETpot 400 – 480 mm/a	1
ETpot 480 – 500 mm/a	2
ETpot 500 – 520 mm/a	3
ETpot 520 – 540 mm/a	4
ETpot 540 – 580 mm/a	5
ETpot 580 – 640 mm/a	6

Via the menu item *Attributes → Computation → Groundwater recharge → According to Meßer 2008 → Determine climate zones from ETpot (NETP → NMKL)* the climate zones can be determined automatically in SPRING if the potential evopotranspiration is assigned (attribute NETP) to all elements.

3.3.2.2 Determining the evaporation

If all input data for the determination of the real evaporation rate are available (attributes: NMKL, NMBT, NMFK, NMFN), the real evaporation (attribute NMET) is calculated automatically in SPRING via the menu item *Attributes → Computation → Groundwater recharge → According to Meßer 2008 → Calculate evaporation (…)*.

3.3.2.3 Determining the direct runoff

If all input data for the determination of the direct runoff are available (attributes: NMGK, NMBT, NMFK, NMFN), the direct runoff (attribute NMAD) is calculated automatically in SPRING via the menu item *Attributes → Computation → Groundwater recharge → According to Meßer 2008 → Calculate direct runoff (…)*.

With this method the groundwater recharge rate can be determined for each finite element according to the above-described water balance equation via: *SPRING → Attributes → Computation → Groundwater recharge → According to Meßer 2008 → Calculate groundwater recharge (...)*.

3.3.3 Land use classes from RVR-Code

For large parts of the Ruhr district detailed digital land use data of the Regionalverband Ruhr (RVR) are available as a so-called use catalogue of the land use map. This land use key can be transferred into the in SPRING required land use identification NSFN, for the recharge calculation by Schroeder and Wyrwich or NMFN for the recharge calculation by Meßer. For this the RVR key has to be provided about the element attribute KVRN.

It is emphasized that the transformation between the individual use classifications is based on experience. In individual cases this may differ. A review on plausible rates of groundwater recharge with the help of the RVR-code matching topographical map is recommended.

3.3.3.1 RVR-Code and land use by Schroeder and Wyrwich

■   RVR-Code 0 – 99, land use (NSFN) by Schroeder and Wyrwich

RVR-Code	RVR description	NSFN-Classification	NSFN-land use
10	built-up areas, serve to living, up to 3 floors	4	Development
20	built-up areas, serve to living, up to 5 floors	4	Development
30	built-up areas, serve to living, more than 5 floors	4	Development
40	Mixed build-up areas	4	Development
50	Trade areas	4	Development
51	Buildings/Plants	4	Development
52	Storage areas	4	Development
53	Empty industrial areas (possible spare areas)	4	Development
54	Parking areas in trade areas	4	Development
55	Urban building yard/ fleet of vehicles, public services, agency in charge of turnpike maintenance, Technical Inspection Authority (TÜV)	4	Development
56	Bus and tram depot	4	Development
57	Exhibition and fair grounds	4	Development
58	Telecommunications	4	Development
59	Other areas	4	Development
60	Industrial areas	4	Development
61	Buildings/Plants	4	Development
62	Storage areas	4	Development
63	Empty industrial areas (possible spare areas)	4	Development
64	Parking areas	4	Development
65	Storage areas for raw materials (coal, ore etc.)	4	Development
66	Other areas	4	Development
70	Build-up areas for sport and relaxation	4	Development
71	Indoor swimming pools	4	Development
72	Gyms, tennis-, ice sports, riding halls	4	Development
73	Multipurpose and event halls	4	Development
74	Stadium	4	Development
75	Structural plants for sports and leisure places	4	Development
76	Other areas	4	Development
80	Municipality need areas	4	Development
81	Public administrations, penal system	4	Development
82	Health service (hospitals, clinical complexes)	4	Development

83	Public, private educational institutions, education centers, libraries	4	Development
84	Play schools, after-school care club, youth and old-age homes/housing departments	4	Development
85	Churches and parish roomses, cloisters	4	Development
86	police, fire department, rescue centers, bunker plants	4	Development
87	Post office	4	Development
88	Culture places (museums, theaters)	4	Development
89	Other areas (i.e. animal homes)	4	Development
90	Agricultural yard and building areas	4	Development
91	Buildings/Plants	4	Development
92	Cultivation enterprises, poultry farms, hen batteries	4	Development
93	Other areas	4	Development

■ RVR-Code 100 – 199, land use (NSFN) by Schroeder und Wyrwich

RVR-Code	RVR description	NSFN-classification	NSFN-land use
100	Other edificial areas (military area)	4	Development
101	Buildings/Plants	4	Development
102	Open spaces within the area	4	Development
103	Other areas	4	Development
110	Freeway and freeway similar streets	4	Development
140	Primary streets and main streets	4	Development
150	Life estate and opening up streets	4	Development
151	Life estate and opening up streets	4	Development
152	Other ways/streets	4	Development
160	Pedestrian zones	4	Development
170	Parking places	4	Development
171	Parking places	4	Development
172	Parking garage	4	Development
173	Bus stations	4	Development
174	Other areas (e.g. garage court not belong to 10-40)	4	Development
180	Railroad traffic areas and side plants	4	Development
181	Stations	4	Development
182	Premises (goods stations, repair shops, engine shacks)	4	Development
183	Track system (city railway, tram)	0	Farm-, grassland

RVR-Code	RVR description	NSFN-classification	NSFN-land use
184	Other areas	4	Development
190	Flight and landing places	4	Development
191	Buildings/Plants	4	Development
192	Landing places	4	Development
193	Open spaces	0	Farm-, grassland
194	Other areas	4	Development

■ RVR-Code 200 – 299, land use (NSFN) by Schroeder und Wyrwich

RVR-Code	RVR description	NSFN-classification	NSFN-land use
200	Other public places	4	Development
210	Energy supply	4	Development
211	Buildings/Plants	4	Development
212	Storage areas	4	Development
213	Transformation plants	4	Development
214	Parking places	4	Development
215	Other areas, i.e.open spaces	0	Farm-, grassland
220	Water supply	4	Development
221	Buildings/Plants, i.e. pumping stations, water-towers	4	Development
222	Water extraction plants (wells, enrichment basins, well galleries)	5	Water areas
223	Other areas, i.e.open spaces	0	Farm-, grassland
230	Sewage disposal	4	Development
231	Buildings/Plants	4	Development
232	Sewage ponds and basins	5	Water areas
233	Rain storage reservoir	5	Water areas
234	Other areas, i.e.open spaces	0	Farm-, grassland
240	Waste disposal	4	Development
241	Waste incinerating plant, recovery plants	4	Development
242	Buildings/Plants	4	Development
243	Dumpsite areas	4	Development
244	Recultivated dumpsite areas	4	Development
245	Mud deposition	4	Development
246	Gathering points for recycling/biological degradation areas	4	Development
247	Other areas	0	Farm-, grassland
250	Embankment areas for soil, rubble	0	Farm-, grassland

260	Exploitation areas	0	Farm-, grassland
261	Buildings/Plants (conveyers, transshipping station)	4	Development
262	Excavation areas including the safety stripe and edge green	0	Farm-, grassland
263	Exploited, at present resting, not filled areas	0	Farm-, grassland
264	Exploited, filled areas, without any use	0	Farm-, grassland
270	Public and private public parks and parkways	0	Farm-, grassland
271	Arranged green spaces near the house area (criteria 10, 20, 30, 40)	0	Farm-, grassland
272	Parks (parks, botanic gardens, zoo)	0	Farm-, grassland
273	Other green areas	0	Farm-, grassland
280	Cemetry	0	Farm-, grassland
281	Structural plants	0	Farm-, grassland
282	Occupancy and green	0	Farm-, grassland
283	Expansion areas	0	Farm-, grassland
284	Other areas	0	Farm-, grassland
290	Allotments	0	Farm-, grassland
291	Open spaces near the house area (criteria 10, 20, 30, 40, 91)	0	Farm-, grassland
292	Duration allotments, little gardens, allotments	0	Farm-, grassland
293	Digging land	0	Farm-, grassland
294	Other areas	0	Farm-, grassland

■ RVR-Code 300 – 399, land use (NSFN) by Schroeder und Wyrwich

RVR-Code	RVR description	NSFN-Classification	NSFN-land use
300	Game and sports facilities	0	Farm-, grassland
301	Sports fields	0	Farm-, grassland
302	Free and beach baths	4	Development
303	Tennis courts	4	Development
304	Plants for water sport (sport boat anchorage, footbridge)	4	Development
305	Dog training places	0	Farm-, grassland
306	Ride and racecourses	0	Farm-, grassland
307	Golf courses	0	Farm-, grassland
308	Game and kicking places	0	Farm-, grassland
309	Other leisure plants (crazy golf, shooting range, amusement park, drive-in movies, motocross, traffic drill ground, model airfield)	4	Development

310	Camping sites	0	Farm-, grassland
311	Regular camping, trailer parking spaces	0	Farm-, grassland
312	Tent pitch	0	Farm-, grassland
313	Other areas	0	Farm-, grassland
320	Accompanying green	1	Decidous forest
321	Copses	1	Decidous forest
322	Lawn, small copses	1	Decidous forest
323	Other areas	0	Farm-, grassland
330	Running waters	5	Water areas
331	Flowing waters	5	Water areas
332	Fully developed waters	5	Water areas
333	Other waters	5	Water areas
340	Channels and ports	5	Water areas
341	Sluice plants	5	Water areas
342	Channels	5	Water areas
343	Port waters	5	Water areas
344	Other areas	5	Water areas
350	Lakes and ponds	5	Water areas
351	Stationary waters close to nature	5	Water areas
352	Fully developed, stationary waters (bath, boat and surfing waters)	5	Water areas
353	Exploitation waters	5	Water areas
354	Ponds in parks	5	Water areas
355	Fish farming waters	5	Water areas
356	Sports ports	5	Water areas
357	Other areas	5	Water areas
360	Duration meadows and pastures	0	Farm-, grassland
361	Meadows and pastures	0	Farm-, grassland
362	Fruit meadows and fruit pastures	0	Farm-, grassland
363	Other areas	0	Farm-, grassland
370	Farm land	0	Farm-, grassland
380	Acquisition horticulture	0	Farm-, grassland
381	Structural plants (hothouses)	4	Development
382	Acreages, special cultures, tree nurseries	0	Farm-, grassland
383	Other areas	0	Farm-, grassland

■ RVR-Code 400 – 499, land use (NSFN) by Schroeder and Wyrwich

RVR-Code	RVR description	NSFN-Classification	NSFN-Land use
400	Decidous forest	1	Decidous forest
410	Coniferous forest	3	Coniferous forest
420	Mixed forest	2	Mixed forest
430	Copse inventory	1	Decidous forest
431	Ccopse inventory	1	Decidous forest
432	Clusters of trees and tree rows	1	Decidous forest
433	Other areas	1	Decidous forest
440	Reafforestation areas	2	Mixed forest
441	Reafforestations and cultivated areas	2	Mixed forest
442	Deforestation	0	Farm-, grassland
450	Living fallow	0	Farm-, grassland
451	Unused areas within living quarters (empty sites)	0	Farm-, grassland
452	At present unused areas with recognizable opening up measures	0	Farm-, grassland
453	At present unused areas which are provided for living buildings in the land development plan	0	Farm-, grassland
454	Other areas, which are provided e.g. for other uses in the land development plan	0	Farm-, grassland
460	Ccommercial and industrial fallow areas	0	Farm-, grassland
461	At present unused areas with recognizable opening up measures	0	Farm-, grassland
462	At present unused areas which are provided for trade and industry settlements in the land development plan	0	Farm-, grassland
463	Other areas which are provided e.g. for other uses in the land development plan	0	Farm-, grassland
470	Unused areas of farming and forestry	0	Farm-, grassland
471	Agricultural fallows	0	Farm-, grassland
472	Scrubed fallows	2	Mixed forest
473	Other areas which are provided e.g. for green spaces in the land development plan	0	Farm-, grassland
480	Coalmine fallows	0	Farm-, grassland
481	Buildings and plants	4	Development
482	Vacated, unused business areas	0	Farm-, grassland
483	Other areas	0	Farm-, grassland
490	Unused traffic areas	4	Development

- RVR-Code 500 – 510, land use (NSFN) by Schroeder and Wyrwich

RVR-Code	RVR description	NSFN-classification	NSFN-land use
500	Dumps	4	Development
501	Dumps situated in embankments or material losses	4	Development
502	Recultivated dumps, also parts of a dump	4	Development
503	Other areas	4	Development

Literature: [*Regionalverband Ruhr, Abt. Karten-/Luftbildwesen und Stadtklimatologie (Hrsgb., 2002): " Nutzungsartenkatalog der Flächennutzungskartierung / FNK", Stand der Daten: 10.07.2002, RVR, Essen*]

3.3.3.2 RVR-Code with land use by Meßer

For the method by Meßer, version 2008 a sealing component for the conversion of the RVR-code (as of 07/2002) into the land use by Meßer is not required anymore.

For the transformation of land use data from the RVR-code or from other area codes such as CORINE land use data or ATKIS data into the attribute NMFN Meßer has developed a special calculation routine.

This conversion is based on dbase files which are stored during the installation of SPRING in the directory "C:\Users\Public\Documents\SPRING\Konfig" (Windows 7 or Windows Vista). This path varies depending on the operating system or the hard drive partitioning (e.g. on Windows XP: "C:\Documents and Settings\All Users\Documents\SPRING\Konfig").

Land use data in the format of RVR code are imported as structure data and assigned to the attribute KVRN.

Then, using *Attributes → Computation → Groundwater recharge → According to Meßer 2008 → Determine land use from RVR code (KVRN → NMFN)* with the help of the dbase file the attribute NMFN, that is necessary for the calculation of the evapotranspiration and the direct runoff, is created.

If there are land use data in the ATKIS format, the proceeding is the same as for the RVR code, only that the data must be assigned to the attribute NATK.

3.3.4 Recharge rate calculation on soil water balance

The process of the (transient) calculation of recharge rates to land on the soil water balance is based on the theoretical foundations that are described in DVWK 238/1996 "Determination of evaporation from land and water areas" in detail.

For the calculation of the recharge rate on the soil water balance is the crucial role of the soil as a water reservoir by including the following components considered:

- Potential evapotranspiration (ET_P): evaporation from surfaces under given meteorological conditions and an unlimited mass of water available. The potential evaporation is an operand, which is determined from measured meteorological values (from: DVWK 238/1996)
- Field capacity (FK): The field capacity is the water-holding capacity of the soil against the force of gravity. The field capacity is the maximum water content, that a soil can hold under natural conditions.
- Permanent wilting point (PWP): The permanent wilting point describes the water content of the soil (depending on the plant), below which available water for plants no longer exists (see: Müller, Tibor: "Wörterbuch und Lexikon der Hydrogeologie"). It is therefore the lowest water content, which reached a soil under natural conditions. It depends on the soil and plant.
- Usable field capacity (nFK): The part of the field capacity, which can be absorbed by plants through the roots is called the usable field capacity (see: Müller, Tibor: "Wörterbuch und Lexikon der Hydrogeologie").

The usable field capacity is thus obtained from the difference between field capacity - permanent wilting point (nFK = FK - PWP).

- Effective root depth (W_e): The effective root depth is about 50 - 60% of the maximum root depth. It features the soil volume in which the soil water balance is affected by intensive plant water consumption.

The recharge rate calculation on to the soil water balance applies in the first place to natural undeveloped areas, but a groundwater model usually also includes built-up areas, these areas are in SPRING still considered by the method of recharge calculation by Schroeder and Wyrwich.

The recharge rate calculation on soil water balance, as it is currently implemented in SPRING, is in a development phase. Therefore, in future, certainly changes in required data or input files are possible. The current program description is always available on our hompage under the menu item "Download & Support: Manual SPRING 4.1".

3.3.4.1 Input data in the model file

The following data must be available for all elements and nodes for the top element or node layer in the model file:

- Ground level (attribute GELA). To calculate the relief energy or the slope to take into account the surface runoff based on Schroeder and Wyrwich.

- Initial potential heads: To calculate the distance to water level. The conversion of node in element values is done automatically within the recharge calculation.

- Soil type (attribute NSBT): The soil type must be specified in the model file according to the following classification as an element attribute:

Soil type	Classification (= NSBT)
Terrestrial sand soils	0
Terrestrial clay soils	10
Semi terrestrial soils	20, 21, 22

- Degree of sealing (attribute VERS): The degree of sealing is specified for each element in the unit [%]. It is necessary to calculate the conditional sealing surface runoff.

- Land use class (attribute NSFN): The land use class must be specified according to the following classification for all elements in the model file:

Land use	Classification (= NSFN)
Farm-, grassland	0
Decidous forest	1
Mixed forest	2
Coniferous forest	3
Build-up areas	4
Water areas	5

3.3.4.2 Transient climate data

First, a file containing daily values of precipitation, potential evaporation (unit [mm]) and temperature must be provided. The column with the temperature is not currently used, but is expected by the program. The file with the name "niedevap.prn" must be provided in the directory of the model file in the format "formatted text (space delimited)". Each column of the file must be 10 characters wide.

Example of a file "niedevap.prn":

```
          1         2         3         4
1234567890123456789012345678901234567890
----------------------------------------
Datum     N [mm]    ETp(Haude)T [14:30h]
01.11.2004       0.0   0.67170      11.2
02.11.2004       0.0   0.36714       9.8
03.11.2004       1.4   0.66568      13.5
04.11.2004       3.3   0.18652      13.2
05.11.2004       1.2   0.81364      10.7
06.11.2004       2.2   0.62583      10.6
07.11.2004       0.0   0.81942       9.8
08.11.2004       0.0   1.03164       9.6
----------------------------------------
          1         2         3         4
1234567890123456789012345678901234567890
```

3.3.4.3 Global soil parameters

Global soil parameters are defined in the file "bopa.txt". The file must be provided in the directory of the model file. If such a file exists, the default parameters are replaced by this. If there exists no file "bopa.txt", the following default values are used.

Structure of the soil parameter file "bopa.txt" with default values:

Column	Meaning	Standardwerte
1	Medium field capacity in [%]: 1st value: sand soils, 2nd value: clay soils	10., 15.
2	Effective root depth in [dm]: 1st value: farm/grass land, build up areas, 2nd value: forest	3., 6.
3	Permanent wilting point in[%]: 1st value: sand soils, 2nd value: clay soils	3., 4.
4	Correction factor of possible over-saturated field capacity	1., 1.

Example of a soil parameter file "bopa.txt":

```
20.,40
3.,6
4.,10
1.0,1.2
```

The correction factor takes an over-saturation of the soil by the field capacity into account, which is reduced in heavy soils after 1 or 2 days. In sand, the correction factor is 1, in heavy soils it is up to 1.2 (from: DVWK 238/1996).
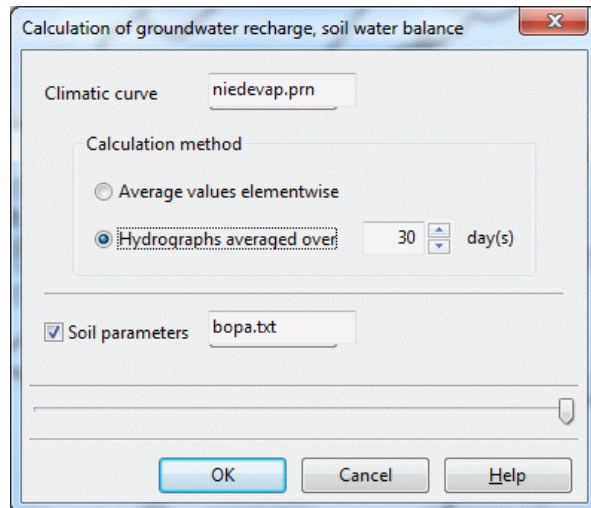
The rate of capillary rise is not considered.

3.3.4.4 Input dialog in SPRING

In contrast to the calculation according to Schroeder and Wyrwich or Meßer 2008 after clicking on the menu item "recharge rate calculation":

> Transform land use (NMFN->NSFN)
>
> Compute groundwater recharge (NSFN+NSBT+VERS+GELA+EICH->FLAE)
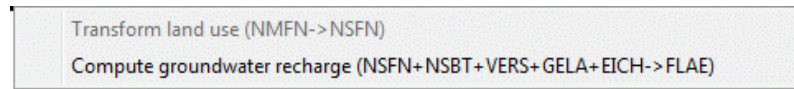
a further input window opens:

Calculation of groundwater recharge, soil water balance

Climatic curve   niedevap.prn

Calculation method

○ Average values elementwise

● Hydrographs averaged over   30 ▴▾   day(s)

☑ Soil parameters   bopa.txt

[OK]   [Cancel]   [Help]

Climatic curve

In the input window the name of the file with the transient climate data has to be entered.

Calculation method

When activating the button "Average values, elementwise" the middle ground water recharge (FLAE) is calculated for each element.

When activating the button "hydrographs, averaged over" the transient data for every x days is generated, averaged over the interval groundwater recharge rate for each element.

The transient values are stored internally during the computation process, and can be exported to a transient input file via *Special Features → Transient → Export transient input file....*

Soil parameters

When activating the check box, a file with user-specific soil parameters can be specified. Otherwise, the file "bopa.txt" with the default values is considered in the calculation.

3.4 Calculating a transient groundwater recharge rate

(The application described below is a preliminary test version!)

As part of the research paper "Development of the groundwater level as a result of climate change on the city of Dusseldorf" (delta h Ingenieurgesellschaft mbH / Institute of Applied Physical Geography RUB, 2014) a method for calculating transient space-differentiated rates of groundwater recharge has been developed which is based on the basis of a soil water balance model (Internet link later ....). The method is integrated in SPRING via the menu RUBINFLUX.

The following figure shows the relevant balance components to determine the daily leachate quantity depending on the precipitation, which is recognized as "groundwater recharge" in the groundwater model.
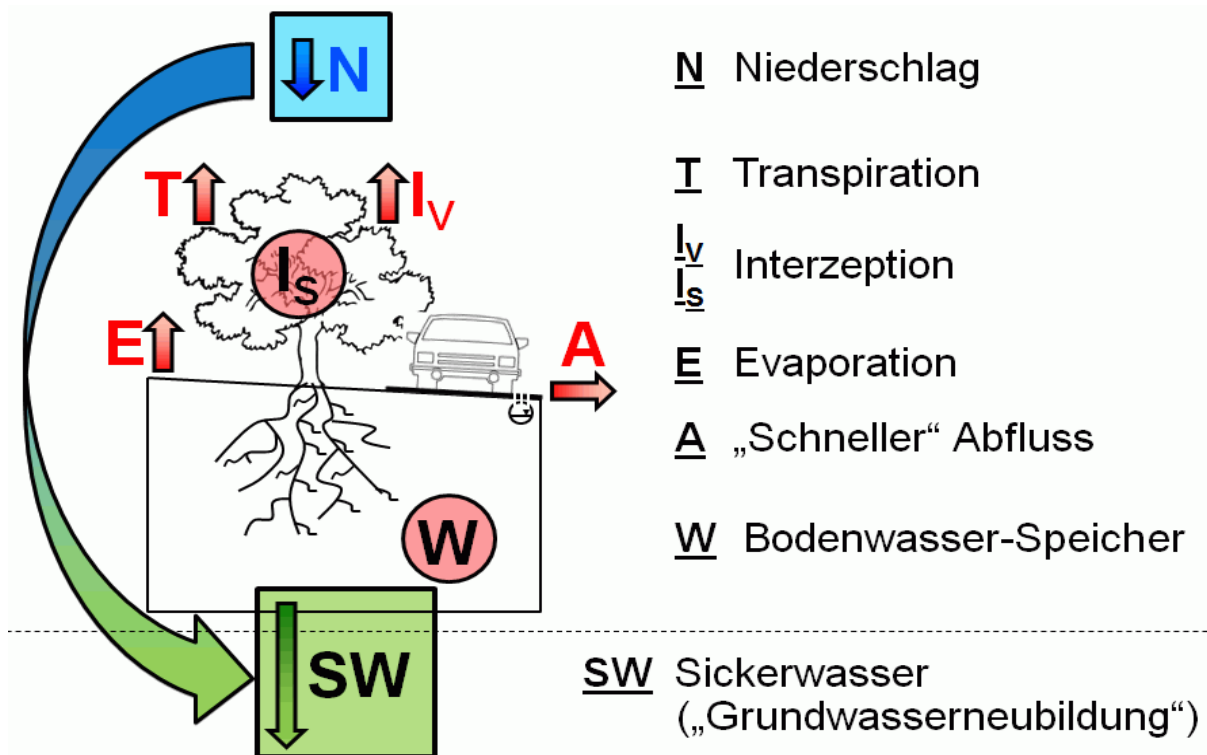
Fig. 114: Balance items for the determination of groundwater recharge rates

■ Transpiration, interception and evaporation are the components of the actual ("current") evaporation. They are limited by physical (e.g. heat radiation), astronomical (e.g. the sun) and climatic (e.g. current temperature) constraints and site-related restrictions (e.g. use, vegetation, soil).

■ "Rapid" runoff components are determined by their slope, land use, soil class and humidity (part A_{SCS}) and the sealing degree at the observed location (part A_{VERS}). The precipitation N must be reduced by the amount of rapid runoff from soil-dependent parameters and slope (A_{SCS}) if the attribute GGRD is assigned.

■ Soil water storage: Its size is defined by the shares of (useful) field capacity and permanent wilting point of soil.
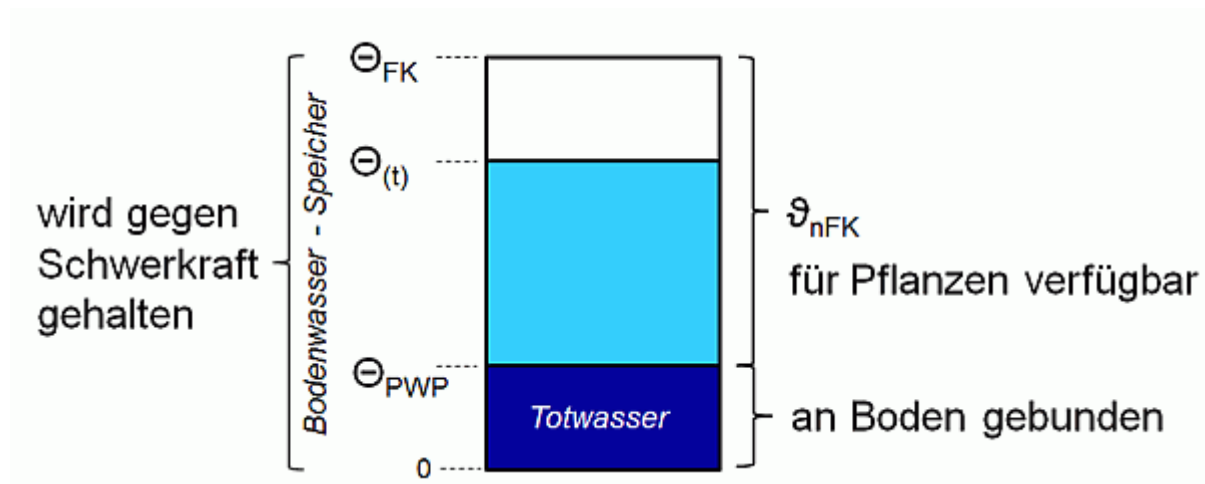
Fig. 115: Water shares of the soil water storage

3.4.1 Input data in the model file

The following data must be available for all elements in the model file.

- Surface sealing (attribute VERS, optional): The degree of sealing is specified in the unit [%]. It is necessary to calculate the sealing dependent "rapid" runoff.

- Slope (attribute GGRD): For elements to which the attribute GGRD is not assigned the rapid runoff component A_{SCS} is not disconnected from the precipitation.

- Layer height (attribute NKLH): NKLH is required for the calculation of the reference grass evaporation and refers to the sea level (e.g. m NN, m NHN, mamsl).

- Latitude (attribute NKBR): NKBR is required for the calculation of the reference grass evaporation. The unit is [°].

- Land use (attribute NKFN) [nr.]: The specification of the land use is made via the ID assignment according to the definitions in the files for the global location parameters (see below). Unless own definitions are made the IDs correspond to the following table:

ID	Land use
1	Grass land
2	Farm land
3	Garden
4	Tree nursery / orchard
5	Water course
6	Standing water
7	Residential, industrial, commercial, maritime transport, airport, public buildings, other land with buildings
8	Mixed use (Sealed + associated open space, for example, farm), fallow land
9	Landfill
10	Mining, mine, quarry
11	Sports, leisure and recreation area, cemetery, playgrounds,
12	Traffic areas
13	Coniferous forest, trees / tree row of softwood
14	Deciduous forest, trees / tree number from hardwood
15	Hedges, bushes, orchard
16	Mixed forest, trees / tree row of deciduous and coniferous or without differentiation
17	Reeds
18	Unknown use
19	vegetation-free area
20	heath

- Soil class (attribute NKBT) [nr.]: The specification of the soil class is made via the ID assignment according to the definitions in the files for the global location parameters (see below). Unless own definitions are made, the encryption is based on the BK50.

Bodenartengruppe der
obersten Bodenartenschicht

0 = Torf, Feinhumus,
        künstliches Material
1 = lehmig-tonig
2 = tonig-lehmig
3 = tonig-schluffig
4 = sandig-lehmig
5 = stark lehmig-sandig
6 = sandig-schluffig
7 = lehmig-sandig
8 = sandig
9 = feinbodenarm

Fig. 116: Soil classes by BK50

■ Climate data ID (attribute NKID) [nr.]: For the classification of climate data time series a climate data ID is assigned to the elements. The climate data ID is part of the file name that contains the associated climate data for the corresponding element. Example: Climate zone 1008 → NKID = 1008 → file name = input_id1008.csv. Example of a file:

```
Datum;P;T;S;ea;u2
01.01.2004;0.00;-0.10;0.00;4.50;3.15
02.01.2004;0.00;-1.40;6.30;3.90;3.07
03.01.2004;0.00;-2.70;0.00;3.80;1.61
04.01.2004;2.80;0.00;0.00;5.30;3.39
05.01.2004;5.86;3.40;0.00;6.90;2.58
06.01.2004;7.76;5.30;0.00;8.00;4.28
```
with:

P = precipitation [mm/d]

T = Temperature [C°]

S = Sunshine duration [h/d]

ea = Vapor pressure [hPa]

u2 = Wind speed at 2 m height above ground level [m/s]

These time series must be provided by the user. All climate time series must begin with the same date and be dissolved in daily steps. The file directory is defined within the input menu.

■ Water content at field capacity (attribute NKFK): The water content at field capacity can, for example, be determined by the BK50. The unit of the attribute NKFK is [vol.-%].

■ Water content at permanent wilting point (attribute NKWP): The water content at permanent wilting point can, for example, be determined by the BK50. The unit of the attribute NKWP is [vol.-%].

3.4.2 Input dialog in SPRING

The calculation is performed in SPRING currently via the menu Special Features → *RUBINFLUX*. (später *Attribute → Berechnen → Neubildung → RUBINFLUX*(später!!!!).

The following input window appears ("Main settings"):
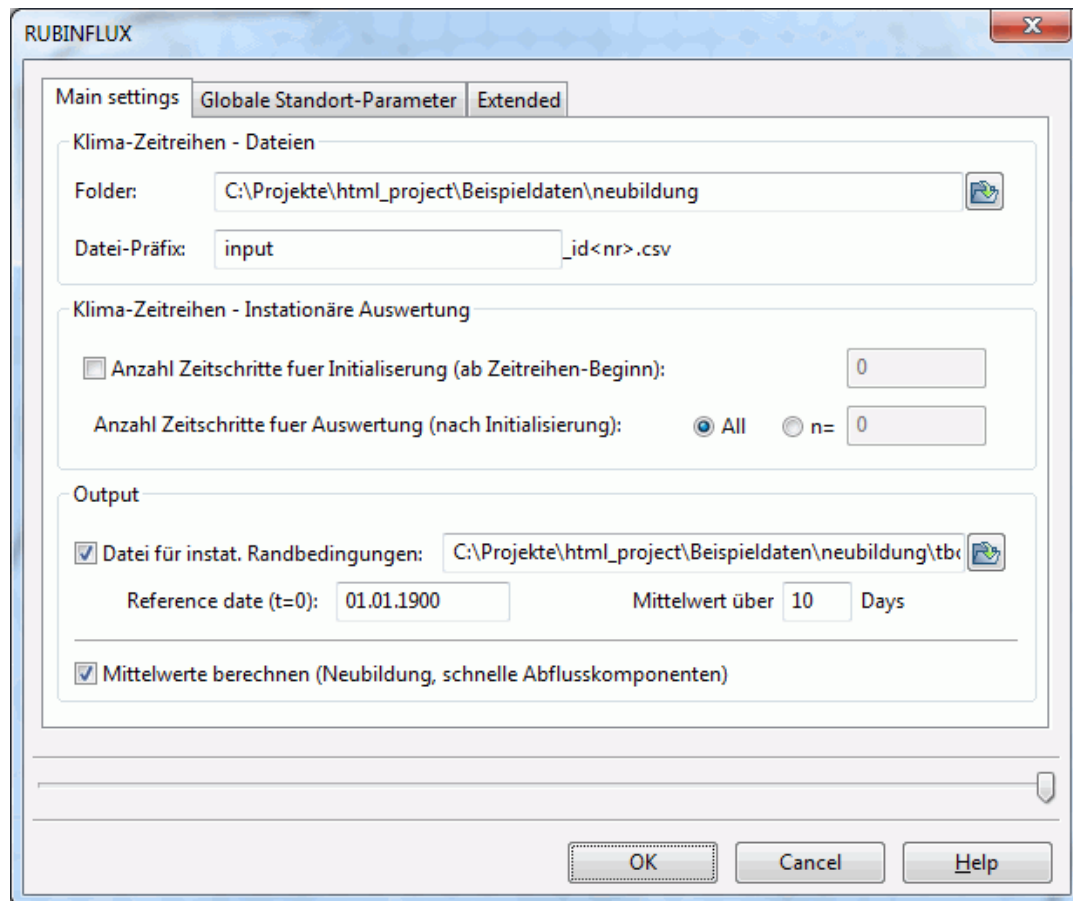
Climate time series data

This determines in which local folder the climate time series files are stored. The file prefix must be the same for all climate files. Example:

Prefix = klima, then the file names are: klima_id10001.csv, klima_id10001.csv (for elements with NKID=10001), klima_id50.csv (for elements with NKID=50) etc. All climate time series have to begin with the same date.

Climate time series - transient analysis

Here, the portion of a time series is set for which a recharge calculation is to be made. If the time series runs for several years, the number of time steps determines the initialization date from which the data is to be exported as transient boundary condition. With the given time steps after the initialization is determined until the data should be calculated and exported.

Output

The name of the output file with the transient boundary conditions as well as the reference date in the transient input file are defined. The user can specify on the indication of averaging whether the transient recharge rate is written for each day (average = 1) or, for example, only for every 10th day (average = 10, corresponding to the average over 10 days) in the file with the transient boundary conditions.

By activating the checkbox the element averages of the groundwater recharge (attribute FLAE) and the rapid runoff components are calculated. The rapid runoff component of soil-dependent parameters A_{SCS} is assigned to the attribute NKAG and the sealing-dependent run off component A_{VERS} is assigned to the attribute NKAV.

3.4.2.1 Global location parameters

After selecting the tab "Global location parameters" the following input window appears:
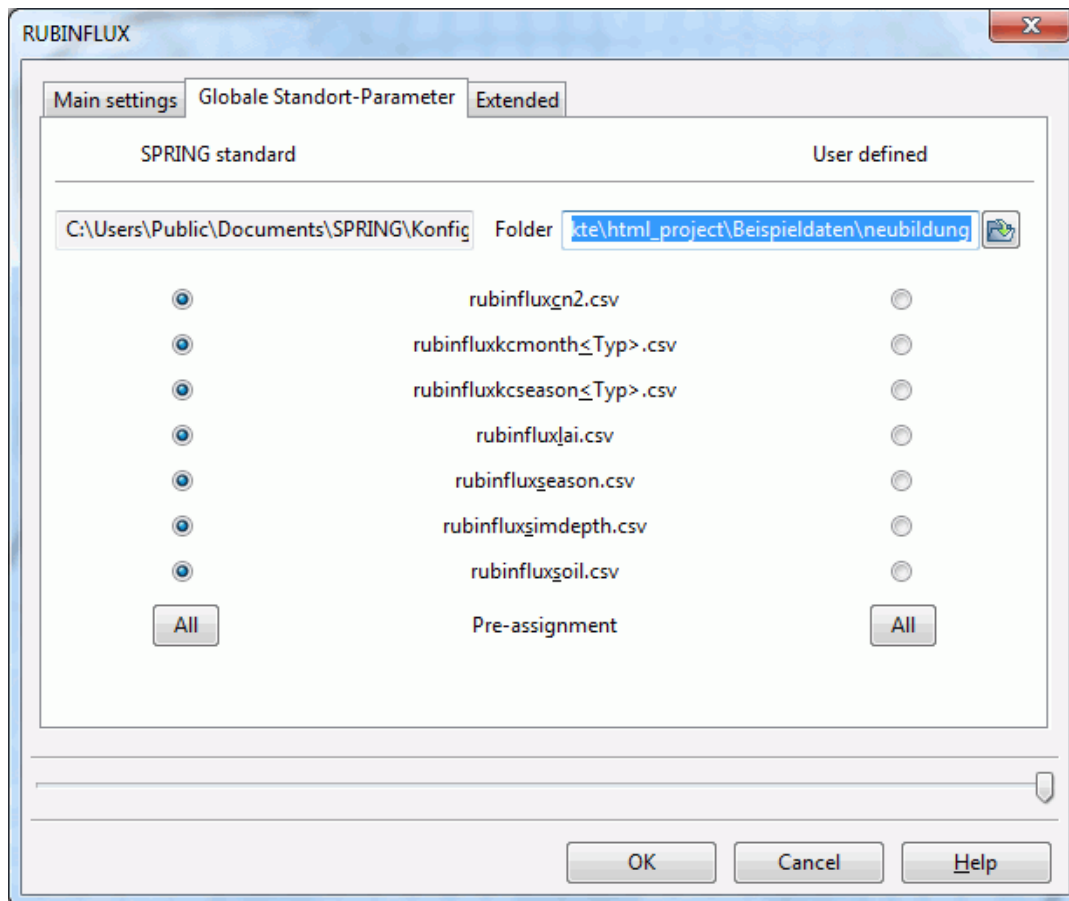
The files with the default parameters are stored in the directory "C:\Users\Public\Documents\SPRING\Konfig" (Windows 7). They contain data related to soil type (_soil), vegetation period (_season), leaf area index (_lai), SCS parameters (_cn2), depth of simulation (_simdepth) and inventory coefficients (_kc_season, _kc_month). These files can also be custom made available and stored in a different directory.

For the format of the *.csv files applies: Column separator = semicolon, decimal separator = point or comma.

During the calculation of the transient recharge an "on-the-fly" -linking of the basic tables happens based on the existing data of the model elements. Thus, different parameter combinations can be tested without interrupting the SPRING session.

3.4.2.2 Extended
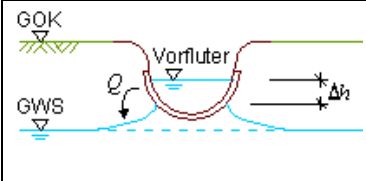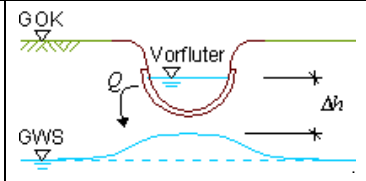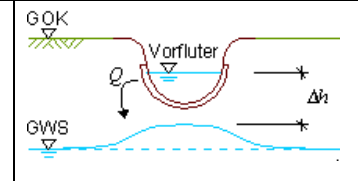
Program control by command line parameters is currently used exclusively for further development purposes. Usage is only for the developers.

3.5 Leakage relations and receiving water interactions

If a large receiving water course exists in the model area or at the field edge, it is in most cases directly connected to the groundwater aquifer. The water level can be assigned as a fixed potential head (POTE) to the corresponding nodes. The resulting inflow or rather outflow rates are represented by mass flow rates. The process with water flowing from the aquifer into the surface water is called exfiltration. The opposite process, with water flowing from the surface water into the aquifer, is known as infiltration. In the case of smaller water courses, there is often no direct connection with the aquifer.

Because of colmation or the formation of an unsaturated zone between the surface water and the groundwater the exchange of flow is disturbed.

The following figures illustrate the various terms related to the aquifer:

Exfiltration of the groundwater into the water course	Infiltration of the water course into the groundwater	Infiltration with tear-off height

The direction of the flow and the flow rate is determined from the mostly linear relation between the water level in the surface water and the groundwater level. The corresponding function is called leakage function. The leakage coefficient [1/s] represents the quotient between the inflow / outflow rate Q (referring to the width of the surface water and the proportional length) and the average potential difference. Especially for infiltration, there exists a maximum rate. Independent of further increase of the potential difference, the current rate will never exceed this maximum. For certain situations, limiting the exfiltration rate can also be useful.
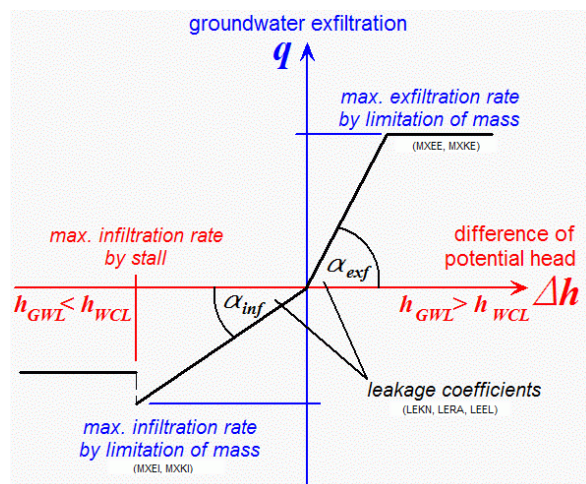
Fig. 117: Schematic representation of the leakage function

Predominantly, leakage functions are defined along lines. But surface leakage relationships are useful as well. SPRING provides the following input data for the definition of leakage processes

- VORF: node wise height of the water level in the surface water
- ABRI: Potential difference between water course and groundwater level from which the infiltration rate does not increase any more (tear-off height)
- LEKN: Leakage coefficient of a node
- LERA: polyline wise leakage coefficient
- LEEL: Leakage coefficient for elements
- LKFA: Ratio between exfiltration und infiltration at nodes (LEKN) or polylines (LERA)
- LEFA: Ratio of exfiltration and infiltration in elements (LEEL)
- MXKE: max. exfiltration rate of a polyline
- MXEE: max. exfiltration rate for elements
- MXKI: max. infiltration rate for a polyline
- MXEI: max. infiltration rate for elements

Absolutely necessary data for a leakage calculation are VORF and values for one of the data sets LEKN, LERA or LEEL.

Leakage in case of impermeable layers
The water exchange through an 'impermeable' layer is a special case for surface leakage. The relatively tight layer slows down the rising of the groundwater level (see max. thickness of the groundwater aquifer), but enables water exchange processes, especially for steady state.

Leakage through an 'impermeable layer' requires the following data:

- UNDU: Thickness of the impermeable layer (requires the input of the ground level GELA)
- LEEL: The meaning of this value depends on the data type UNDU: UNDU>0: LEEL = vertical K-value of the impermeable layer in m/s; UNDU=0 or no input: LEEL = direct leakage coefficient in 1/time unit.
- VORF: Water level of the surface water (if no surface water is existing, VORF is equal to the ground level)

In the following chapters, the practical implementation of individual water course interactions in the ground water model is explained using typical examples.

3.5.1 Controlling the depth to water table with a withdrawal well

The mining industry has caused an increase of the groundwater level in many cultivated areas, so that the groundwater level is often too near to the surface. To avoid damage at existing buildings controlling the groundwater level with withdrawal wells is necessary. In contrast to wells, used for groundwater winning, the pumping rate of local drain wells is often unknown. So they cannot be considered as boundary condition KNOT. More important than the pumping rate is the altitude of the groundwater subsidence. The following describes the modelling of such a situation with SPRING.

The figure shows schematically the problem:
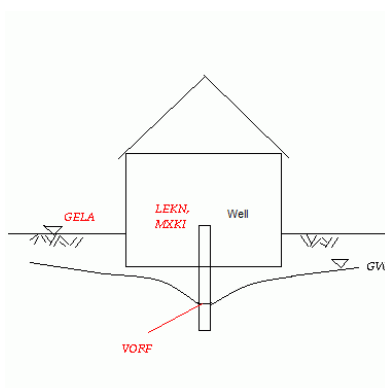
Fig. 118: Schematic representation of a polder measurement in the basement of a building

- VORF

The FE node of the drain well gets the attribute VORF, its value corresponds to the maximum admissible altitude of the groundwater level (e.g. basement foundations - 0.8 m). In a 3D model it is sometimes necessary that the FE nodes in the filter area of the drain well get the attribute GLEI. This depends on the vertical discretization of the FE net. In this case the attribute VORF has to be assigned to one of these nodes.

- LEKN

Additionally to the attribute "VORF" the leakage coefficient LEKN must be assigned to the FE node of the drain well. A very great value of the attribute "LEKN" acts like the fixed potential head "POTE". This guarantees keeping the requested altitude of the groundwater level. If pumping rates of the drain well exist the leakage coefficient can be calibrated by comparing the measured and calculated rates.

In SPRING you can assign the calculated pumping rates to any attribute (e.g. KKKK) with *Attributes → Import result data*, selection of the background file "aaa" and selection of "Leakage rate (m³/node/TU)". With *Attributes → Edit nodes → Capture* all attributes of the node are listed, including the pumping rate.

- MXKI

If the groundwater level is below the defined value of "VORF", the drain well infiltrates into the groundwater. To prevent that the attribute "MXKI = 0.0" has to be assigned to the FE node of the well.

- MXKE

Sometimes it is necessary to limit the pumping rate to a maximum, e.g. to consider the capacity of the drain well. In this case the attribute "MXKE" [m³/TU/m] defines the maximum possible pumping rate. Attention: Taking water out of the model requires a negative value!

3.5.2 Modelling of a drainage

In areas with a groundwater level near the surface this level has to be controlled. Especially in cultivated regions you can do this with drain pipes. The groundwater exfiltrates in these drainages and they drain the water to the next receiving water course.

The following figures show the problem:
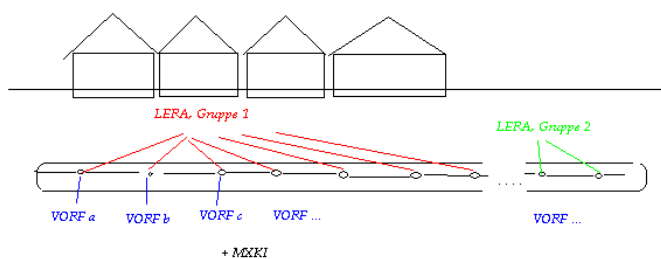
Fig. 119: Longitudinal section through a drain pipe in the street
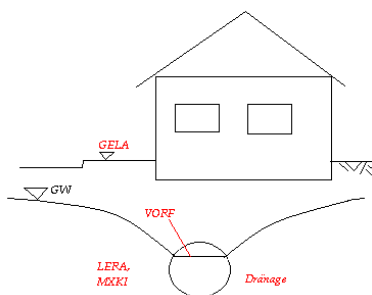
Fig. 120: Cross section through a drain pipe in the street

The realization in SPRING looks like this:

■ VORF

The attribute VORF is assigned to the nodes which represent the drain pipe. The values correspond to the requested level of the groundwater surface (e.g. basement foundations - 0.8 m).

■ LERA

When modelling a drain pipe the attribute LERA ([m/TU]) has to be assigned to the nodes. It is very important to keep the nodes in order and to separate each drain pipe in different node groups. If discharge rates of the drain pipes exist the leakage coefficient of the drain pipe can be calibrated while comparing the calculated leakage rates with the measured leakage rates. In SPRING you can assign the calculated leakage rates to any attribute (e.g. KKKK) with *Attributes → Import result data*, selecting the background file "aaa" and selecting "Leakage rate (m³/node/TU)". With *Attributes → Edit nodes → capture* all node attributes are listed, including the leakage rate "KKKK". The summation of the single node rates gets you the total quantity of the calculated runoff of the drain pipe.

■ MXKI

If the groundwater level is below the defined value of "VORF", the (empty) drain pipe infiltrates into the groundwater. To prevent that the attribute "MXKI = 0.0" has to be assigned to the nodes of the drain pipe.

■   MXKE

Sometimes it is necessary to limit the runoff rate to a maximum, e.g. to consider the capacity of the drain pipe. In this case the attribute "MXKE" defines the maximum possible runoff rate. Attention: Taking water out of the model requires a negative value! "MXKE" is defined as m³/TU/m, so a total quantity is given to each drain pipe. The sequence of the nodes must be kept! The maximum value for each node is calculated through the multiplication with the proportionate section between the left and right node.

3.5.3 Seepage surface in a dam

A calculation of the seepage surface in a dam needs precise discretization. Inside the dam is a permeability which is about tenfold lower than the permeability of the outer material. This leads to oscillation problems in the flow calculation, if the element size is not adjusted stepwise.

The following figure shows the essential model data for a dam:
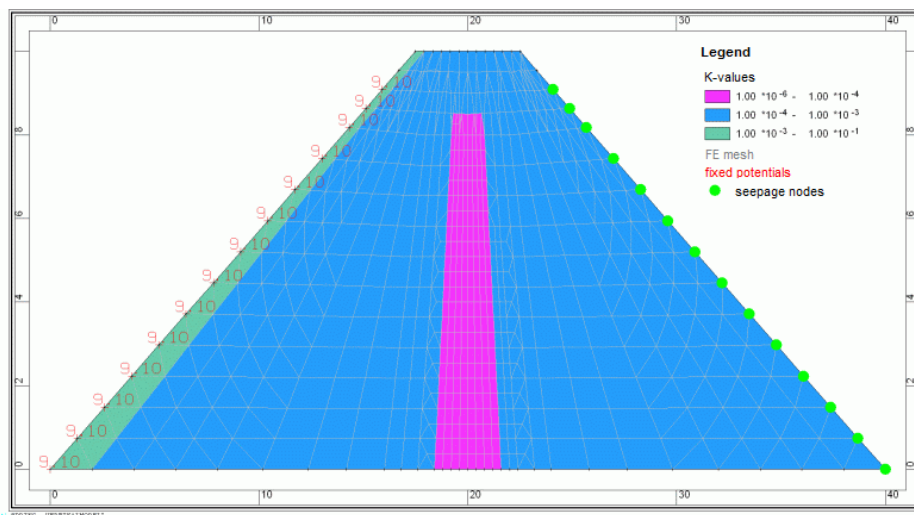
Fig. 121: Model data of a dam (double super elevation in y-direction)

These are the necessary parameters for a flow calculation:

■   POTE: fixed potential heads at the boundary nodes of the water side in m NN (altitude of the water level)
■   KWER: green = 0.001 m/s, blue = 0.0001 m/s, pink = 0.000001 m/s
■   MAEC: horizontal expansion (user defined)
■   SICK: at the land side boundary nodes up to the altitude of the water level (value = 0.0)

After a steady state flow computation (10 iterations) you get the following figure:
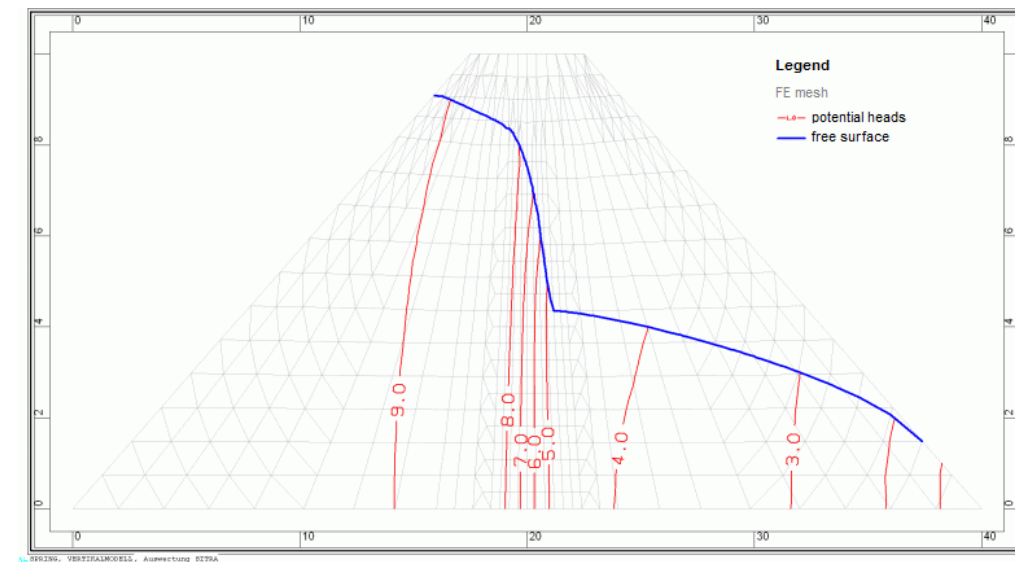
Fig. 122: Result data: potential heads and seepage surface of the dam

The location of the seepage surface (blue) corresponds to the calculated zone of the free surface.

Where the seepage line intersects the land side of the dam water percolates to the surface. In figure 6 you can see it at three nodes at the right bottom of the dam. In SPRING you can assign the percolation rate to any attribute with *Attributes → Import result data*, selecting the background file "aaa" and selecting "Seepage rate [m³/no./TU]", e.g. to "KKKK". With *View → Show attributes* you can then visualize the chosen attribute and with *Attributes → Edit nodes* all node attributes are listed, including the seepage rate. The total quantity of the seepage rate has to be calculated per running dam-meter.

3.6 Simulation of flooding processes

Agreements

A mine is at the beginning a total unsaturated 3D area which is totally or in parts under the compound surface. In addition:

- The nodes which belong to the mine are indicated by explicit zone numbers (data type GRUB, numbers > 0).
- For all FE-mesh-nodes of the considered mine where an inflow is possible the boundary conditions must be defined as "free off seepage" (data type SICK). If there are drainages for the mine pumping facilities which drain into the mine the affected nodes can get the boundary condition "water course level and leakage", additional the node can be supplied with the attribute for the zone number of the mine (data type GRUB, see above).
- For the time $t=t_0$ is no water in the mine located (no initial volume).
- For a transient flow calculation a steady ascending curve is available for the description of the ratio between water spreading volume (m³/s) and the water spreading height (m NN).

Temporal course of the water spreading in a mine

After each time step $t = t_i$ the flow rate is balanced which inflow happens from time $t = t_0$ to time $t = t_i$ into the mine. The volume is composed of the inflow considered at the point in time $t = t_i - t_{i-1}$ on the mine nodes and the spreading volume until to the point in time $t = t_{i-1}$. If there are drainages in the mine and marked in the way which is described above the inflow of the drainage will be added to the volume in the cavity.

With the relationship between the water spreading volume and the water spreading height the resulting water spreading height $h(V_i)$ can be calculated at the considered point in time from the volume $V(t_i)$.
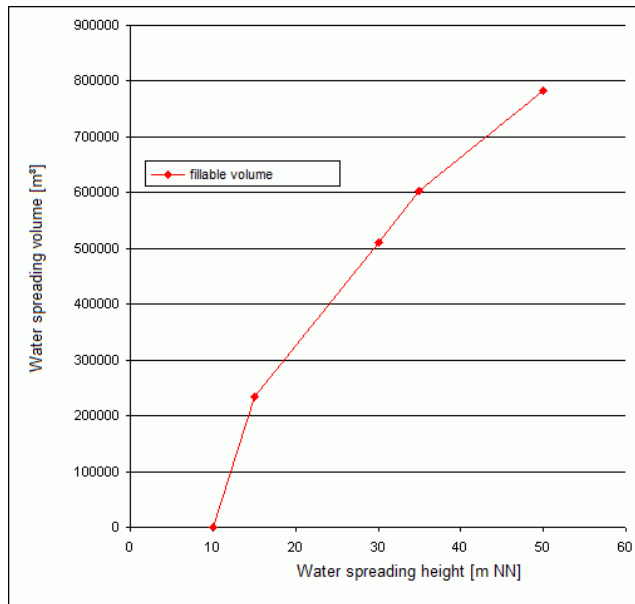
For each node with the boundary condition "free off seepage" the vertical location z estimates if the node lies over the free surface $h(V_i)$. For a node lying under the free surface the boundary condition "free infiltration boundary" is changed into "fixed potential" or alternatively in "water course with leakage" with the height $h(V_i)$. For all other nodes with the boundary condition "fixed potential" or "water course with leakage" the height $h(V_i)$ will be updated.

This procedure guarantees that the mass which is removed out of the model by the boundary conditions "free off seepage" will be completely added back through the definition of fixed potentials or water course nodes. So the mass balance in the model is ensured.

If the "off seepage nodes" change in "potential nodes" or in the damped boundary condition "water course node" is defined through flow parameters.

When choosing the "water course node" entering a leakage value for the nodes (LEKN) is required.

Pile-up of a mine

If all nodes of a mine have at one point in time the boundary condition "confirmed potential" the mine considered to be overflown. In this case all (potential-) boundary conditions are "deactivated" so that the mine will be considered in the flow calculation as a totally saturated subarea of the aquifer.

Lowering device of the water level in a mine

If the water level drops in the mine the nodes now lying over the free surface will change (inverted equal to the water spreading procedure) from the boundary condition "confirms potential" back to the boundary condition "free off seepage".

If the water level drops down after a pile-up in the mine all mine nodes get back their originally boundary conditions. That means, when the water volume in the mine is no more filling it up the mine nodes which lie below the surface get the boundary condition "confirmed potential" and the nodes lying above the surface get the boundary condition "free off seepage", always considering the relationship between water spreading volume and water spreading height.

Limitation of the water level

If for a mine a maximum water level is defined and achieved the volume which guarantees keeping the water on that level in each time step is removed from the model. The calculation of the shrinking volume is based on

the water spreading volume-water spreading height-relationship. Thus the water level must be located in the codomain of the function.

A temporal changing of the water level is always possible using the transient input file. The data type is described through HMAX. The values correspond to the zone numbers of the mine.

### Addition of water from outside the model

When water is added to the mine from outside the model during the flooding phase this can be defined as a flow rate. If such a volume is defined for a time step it is considered in the calculation of the water spreading height using the water spreading volume-water spreading height-relationship. A mathematically positive volume defines a water supply to the mine and a negative volume defines a water removal from the mine.

A temporal changing of the water supply is always possible using the transient input file. The type of data is described with MENG. The values correspond to the zone numbers of the mine (see "Limitation of the water level").

Definition of the flooding parameters in SPRING

First of all each FE-mesh-nodes which describe a mine get an explicit number at the data type GRUB before a flooding simulation with the above described method can be executed. In the following procedure this number is used as the zone number (X) of the mine.

If nodes indicate a mine the further flooding parameters "water level" and "addition volume" can be defined with the menu *Attributes → Special features → Flooding parameters* (GRUB). Should a set parameter be changed the entry in the relevant field has to be deleted (field remained blank).
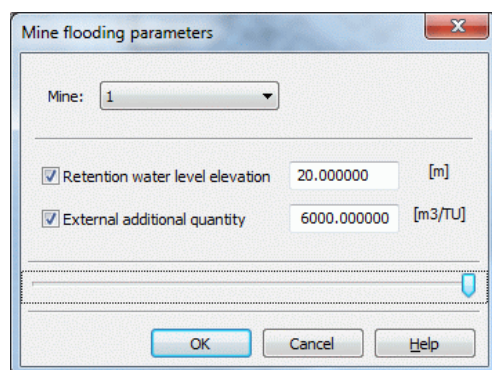
Definition of the water spreading volume - water spreading height-relationship

For each mine a continuous function must be defined which describes the dependence on the spreading volume and the water level in the mine:
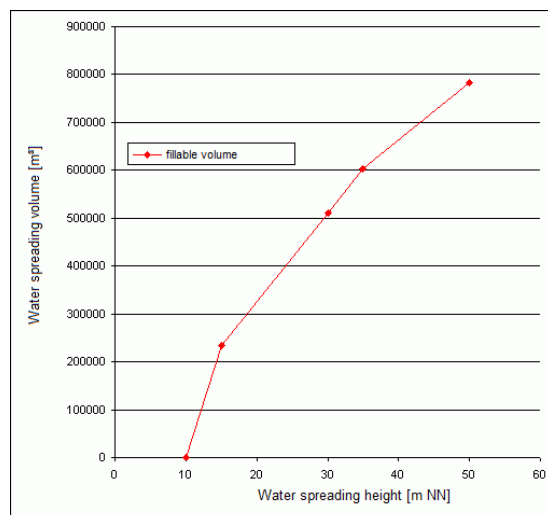
The function should be defined over the total interval of the water spreading procedure, that means, for all appearing values of the volume and the height during the simulation.

For the flooding calculation the water spreading volume-water spreading height-relationship for each mine must be prepared in the format CSV (separators disconnected) and placed in the working directory. The files have to be named after "grubeX.hv" where "X" is the corresponding zone number. This file is read automatically by the calculation module.

```
10.0,0.0
15.0,234900.0
30.0,511600.0
35.0,602800.0
50.0,783000.0
```

Results

Beside the known results (potentials, free surface, hydrograph, etc.) from the transient flow calculation SPRING creates for each cavity during the flooding simulation a volume hydrograph which describes for each time step the actual water spreading volume and by definition of the water level a volume hydrograph of the shrinking volume. Additionally the volumes for each boundary condition for each time step are saved. The file is created with the name "gruben.csv" (separators disconnected) in the working directory.

3.6.1 Example of a realization in SPRING

The software SPRING affords the simulation of flooding processes for mines and barrages. The following tutorial shows how to handle flooding processes of a mine.

The model files and the files needed for the computation are ready for download on our under "Tutorial (EN)" (

). The archive file "Tutorial_bsp_files.zip" includes a directory "HowTo_Grube" in which the files "fl0.zip", "fl1.zip" and "fl2.zip" are to be found.

The tutorial model is shown in the figure below. It contains three geological layers which describe two aquifers and one aquitard. In the second aquifer the mine is provided. The system is discretised up to nine node layers.
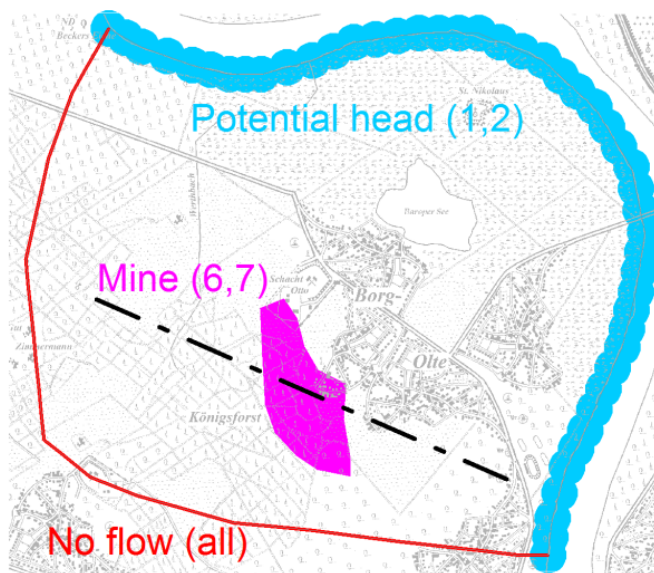
Fig. 123: Top view of the model with the boundary conditions of the individual layers
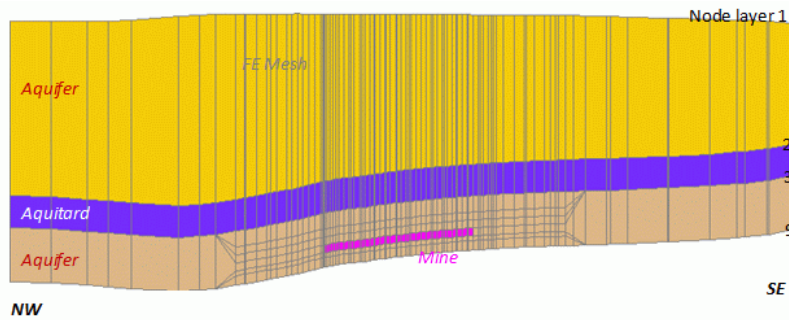
Fig. 124: Model basics (vertical cross section with 20x vertical exaggeration)

3.6.1.1 Initial conditions

Because flooding is always a transient simulation type you have to prepare the initial conditions first. Therefore it is necessary to define a seepage boundary condition at the nodes which describe the mine.

The example "fl0.zip" includes a structure (attribute type SICK) to set the boundary condition. Assign this structure to node layer 6 and 7.

Save the mesh and start the model checks with the item *Calculation → Model checking*. Continue with running a steady state flow simulation with the item *Calculation → Steady state flow.*
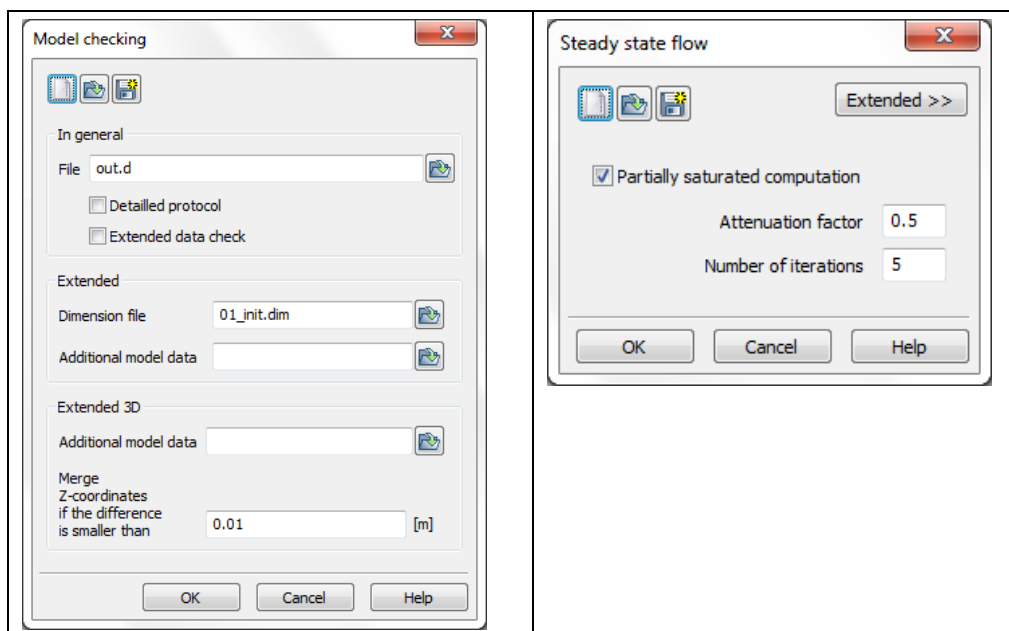
Fig. 125: Model checks (step 1) and steady state flow parameters (step 2)

The results of the computation are shown in the figure below. You can create this vertical cross section using the plot batch file "cs.bpl" (archive "fl0.zip").
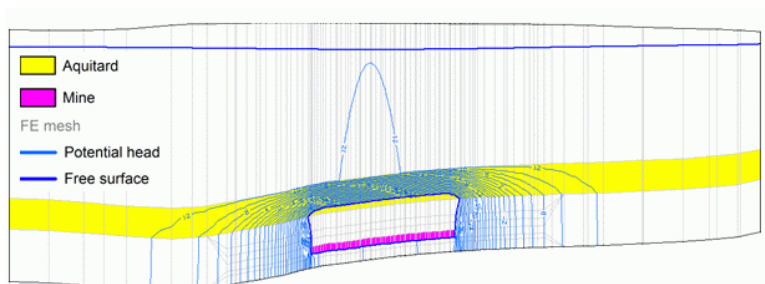
Fig. 126: Vertical cross section with results of steady state simulation

3.6.1.2 Simulation 1: Uncontrolled flooding

(Archive fl1.zip)

For a simulation of uncontrolled flooding processes it is necessary to mark the mesh nodes which represent the mine and to describe the correlation between mine water level and the water volume inside the mine. This correlation is called "h-V relation". Uncontrolled flooding means that the draining of the mine will be stopped and the mine will begin to fill with groundwater until reaching the equilibrium.

The attribute for defining the nodes of the mine is called GRUB. You have to assign this boundary condition at node layer 6 and 7 with the help of the structure with the attribute GRUB.

The attribute GRUB contains the zone number of the mine. In this example the number is "1". The corresponding h-V relation has to be defined in an ascii file which is called "grube1.hv" (file name = "grube"+zone number+".hv", see "fl1.zip"). The figure below shows the h-V relation of this example. For a better understanding during later simulation steps the gradient of the curve is displayed too.
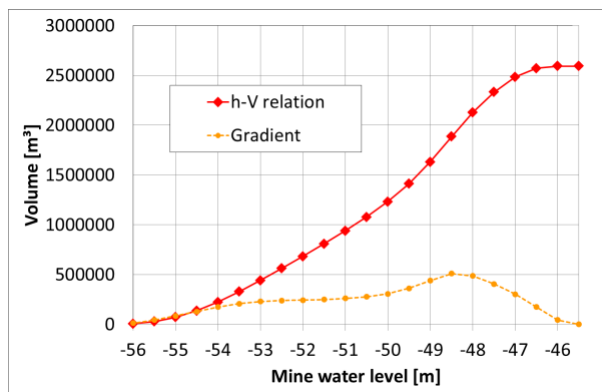
Fig. 127: Relation between mine water level and flooded volume of the mine

For transient simulations in SPRING a file including the transient boundary conditions is always needed. In this example no transient boundary conditions were changed. Therefore the file is "empty" (see also trans.txt in "fl1.zip").

File with transient boundary conditions (Flooding 1):

```
# SPRING tutorial
# Flooding
# Simulation 1

ZEITEINHEIT MENG JAHR
BEZUGSDATUM 01.01.2012

DATUM
02.01.2012
```

Before starting the simulation the definition of the initial values for the potential head of each node is needed. The necessary attribute is called EICH. Therefore import the results of the steady state flow computation (from directory fl0) using the item *Attributes → Import Data/Computation Results*:
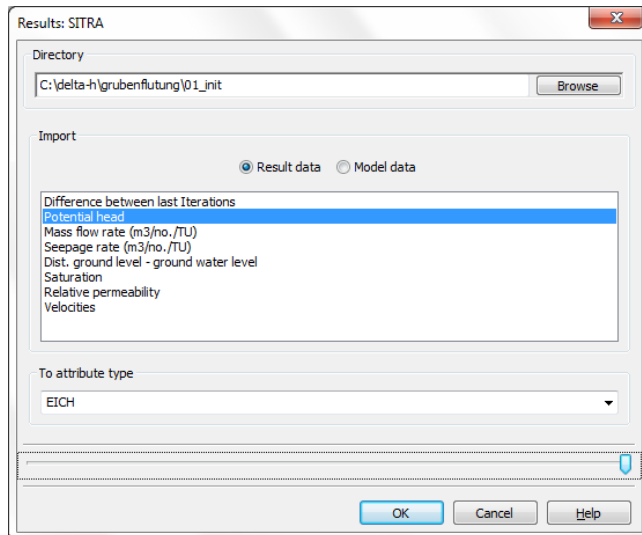
Fig. 5: Setting initial potential heads

After setting the initial potential heads save the mesh and start the model checks with the item *Calculation → Model checking*. Continue with running the transient flow simulation with the item *Calculation → Transient flow*:
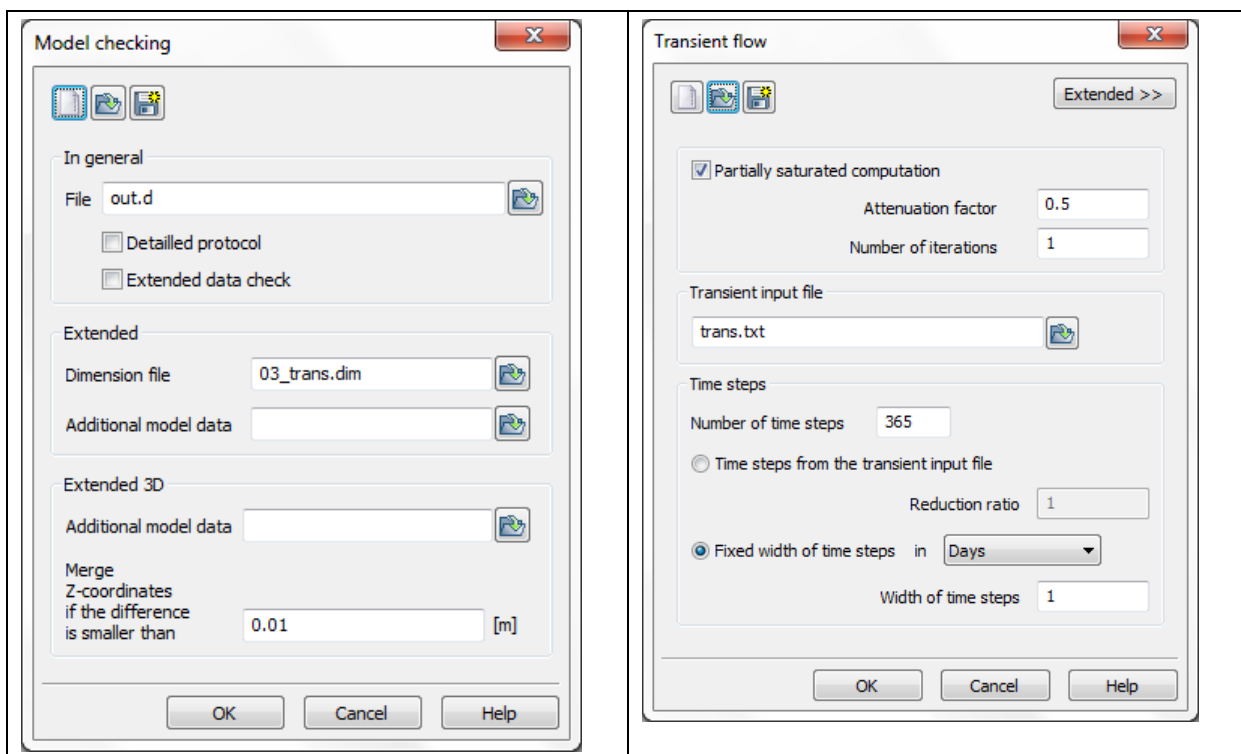

Fig. 6: Model checks (step 1) and transient flow parameters (step 2)

The resultant potential heads can be displayed by using the plot batch file "cs.bpl" for a vertical cross section and "pot56-7.bpl" for a horizontal cross section through layer 7 with the values of the 56th time step (see also figure below). The right figure also includes hydrographs of the resultant potential heads at selected nodes at the bottom of the mine. The dip in the curves is caused by reaching the end of the flooding process when the mine is filled completely with water. After this point the potential heads are rising up until reaching steady state conditions.
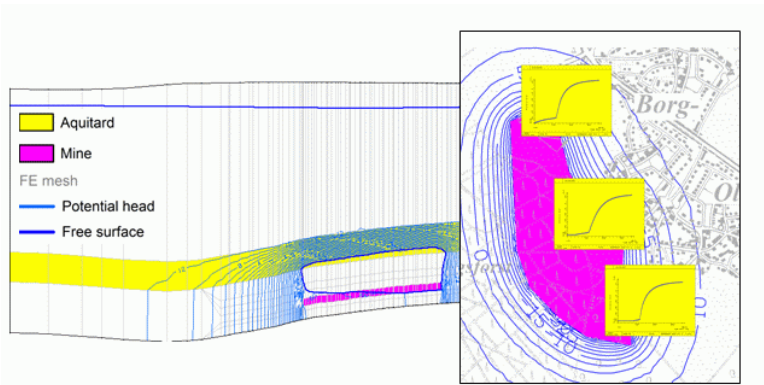
Fig. 128: Vertical cross section (left) and horizontal cross section through layer 7 (right) with results 56 days after begin of flooding (Flooding 1)

In addition to a standard transient computation simulating mine flooding creates a file called "gruben.csv". This file contains a table with a complete balance for the flooding box for each time step. This allows you to display the resultant balance curves with the help of a spreadsheet. For this example the evaluation was done with MS Excel (see file "gruben1.xls" in "fl1.zip") and is shown in the figures below.

The first figure presents the resultant mine water level and the ratios of the water balance depending on the several boundary conditions. For each time step the nodes which describe the mine were set to the boundary condition "seepage" or "potential head" depending on their current vertical position:

■   "Seepage": The vertical position of the node (z-coordinate) is higher than the mine water level (unsaturated).

■   "Potential head": The vertical position of the node (z-coordinate) is lower than the mine water level (saturated).

Both boundary conditions cause a mass flow rate respectively a mass flow which is exchanged between the mine and the aquifer. In the present example the sum of the mass flow from both boundary conditions is equivalent with the ground water inflow (see below).

The dip in the mass flow curves is caused by the gradient in the h-V relation. The mine water level reaches the top of the mine after 124 days.
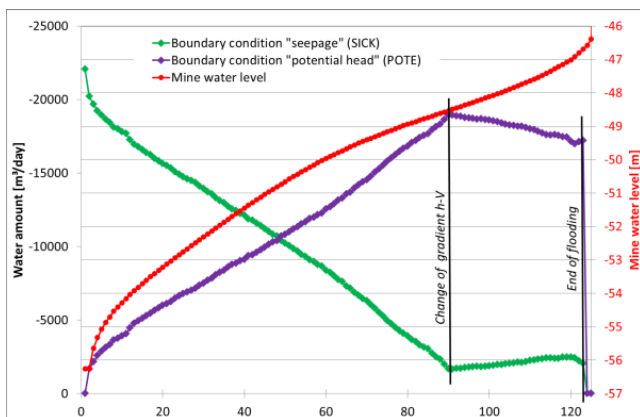
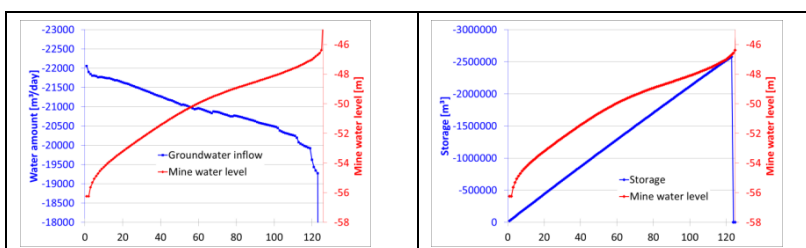Fig. 129: Water level and water amount until reaching end of flooding (Flooding 1)

Fig. 130: Groundwater inflow (left) and storage (right) until reaching end of flooding (Flooding 1)

3.6.1.3 Simulation 2: Controlled flooding

(Archive fl2.zip)

In the second example the use of additional mine-specific transient boundary condition will be shown.

It is possible to define a retention water level elevation and external water supply for each mine. Both parameters can be changed in the transient input file. For external water supply a positive value has to be set if water should be pumped into the mine to make flooding faster. A negative value implies pumping water out of the mine. Both parameters can be initialized with dialog in *Attributes → Special features → Flooding parameters*. Note: If you want to vary one of these parameters during a simulation an initialisation is needed here.
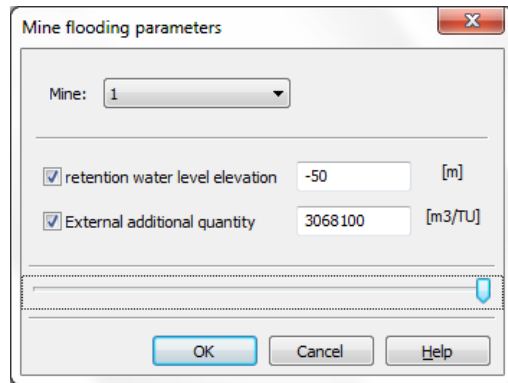
Fig. 131: SPRING dialog for managing flooding parameters

To show the working of these flooding parameters the following transient boundary conditions were set:

Table: Transient boundary conditions (Flooding 2)

Date	Time step	Boundary condition	Description
01.01.2012	0	HMAX = -50 m MENG = 3068100 m³/year	Begin of flooding up to mine water level -50 m with water supply of about 8400 m³/day
01.03.2012	60	MENG = -13149000 m³/year	Empty the mine by pumping 36000 m³/day (= 1500 m³/h)
01.07.2012	182	HMAX = --46.5 m MENG = 5259600 m³/year	Continue flooding up to mine water level -46.5 m with water supply of about 14400 m³/day
01.09.2012	244	MENG = 0 m³/year	Continue flooding without external water supply
31.12.2012	365		End of simulation

The file with the transient boundary conditions can be found in "fl2.zip" and looks as follows.

File with transient boundary conditions (Flooding 2):

```
# SPRING tutorial
# Flooding
# Simulation 2

ZEITEINHEIT MENG JAHR
BEZUGSDATUM 01.01.2012

DATUM
01.03.2012

MENG #empty mine with pumping rate 1500 m3/hour
     1-13149000.
```

```
DATUM
01.07.2012

HMAX #restart flooding upto mine water level -46.5m
    1    -46.5

MENG #add about 600 m3/hour
    1  5259600.

DATUM
01.09.2012

MENG #continue flooding without water supply
    1     0.5
```

The control of the simulation follows the steps of the example above.

The results of the present simulation are shown in the figures below. They base on the evaluation of the file "gruben.csv" and are included in the file "gruben2.xls" ("fl2.zip").

The given transient boundary condition for the water supply is shown by the curve "potential external water supply" (identical to definition of attribute MENG). The curve named "actual external water supply" shows the water volume which can be pumped in or out of the mine in a compatible way with the physics of the given system. E.g. the actual and potential water supply differs between reaching the end of emptying of the mine and continuing of the flooding. This is caused by the defined water supply (MENG = 36000 m³/day) which is not available after emptying the mine. Therefore the actual pumping rate after emptying the mine will be lower (about 22.000 m³/day).
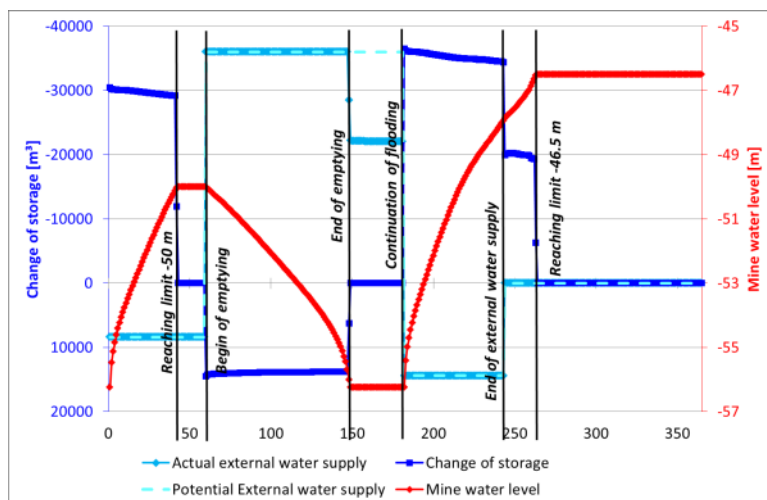
Fig. 132: Mine water level, change of storage and external water supply (Flooding 2)
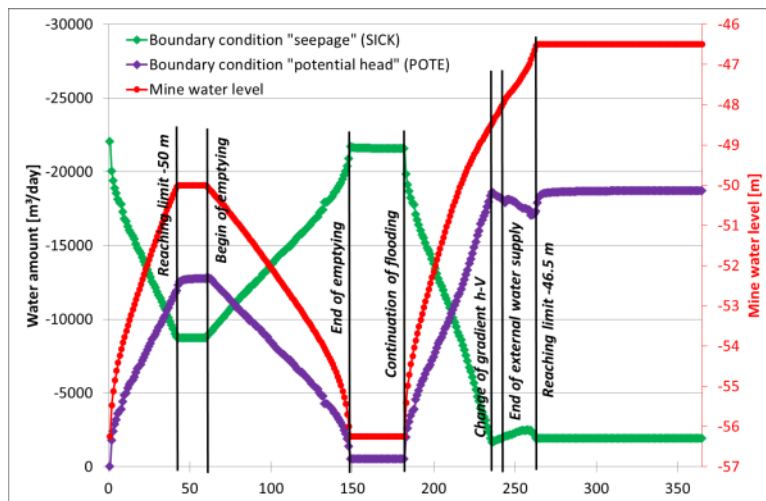
Fig. 133: Boundary condition dependent mass flow rates (Flooding 2)

4 Index