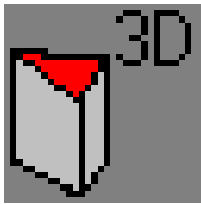


Modelling of groundwater flow in three-dimensional
groundwater systems using finite element methods

GGU-3D-SSFLOW

VERSION 6



Last revision:	October 2017
Copyright:	Prof. Dr. Johann Buß
Technical implementation and sales:	Civilserve GmbH, Steinfeld

Contents:

1	Preface	7
2	Licence protection and installation	7
3	Language selection.....	8
4	Starting the program	8
5	Program concept	9
6	Short description.....	10
7	Theoretical principles	12
7.1	General.....	12
7.2	Differential equation	13
8	Description of menus items	15
8.1	File menu.....	15
8.1.1	"New" menu item.....	15
8.1.2	"Load" menu item.....	15
8.1.3	"Save" menu item	15
8.1.4	"Save as" menu item	16
8.1.5	"Import ASCII file" menu item.....	16
8.1.6	"Export to ASCII file" menu item.....	17
8.1.7	"Print output table" menu item.....	17
8.1.7.1	Selecting the output format	17
8.1.7.2	Button "Output as graphics".....	18
8.1.7.3	Button "Output as ASCII"	20
8.1.8	"Printer preferences" menu item.....	21
8.1.9	"Print and export" menu item	21
8.1.10	"Batch print" menu item	23
8.1.11	"Export to GGU-3D-TRANSIENT" menu item.....	23
8.1.12	"Exit" menu item.....	24
8.1.13	"1, 2, 3, 4" menu items.....	24
8.2	Mesh(x/y) menu	24
8.2.1	"Preferences" menu item.....	24
8.2.2	"Mesh" menu item	24
8.2.3	"Outline" menu item	24
8.2.4	"Define nodes" menu item	25
8.2.5	"Change" menu item	25
8.2.6	"Move" menu item.....	26
8.2.7	"Edit" menu item.....	27
8.2.8	"Array" menu item.....	27
8.2.8.1	Select type of array.....	27
8.2.8.2	Button "Regular"	28
8.2.8.3	Button "Irregular".....	29
8.2.9	"Common systems" menu item.....	30
8.2.10	"Manual mesh" menu item.....	31
8.2.11	"Automatic" menu item	32
8.2.12	"Round off" menu item.....	32

8.2.13 "Delete" menu item.....	32
8.2.14 "Optimise" menu item.....	33
8.2.15 "Refine individually" menu item	34
8.2.16 "Section" menu item	34
8.2.17 "All" menu item	34
8.2.18 Refinement methods	35
8.3 z menu	36
8.3.1 "Default depths" menu item.....	36
8.3.2 "Default soil column" menu item.....	37
8.3.3 "Individual depths" menu item	37
8.3.4 "Modify depths" menu item.....	38
8.3.5 "Depths in section" menu item.....	38
8.3.6 "Contours (planes)" menu item.....	39
8.3.7 "Contours (layer thickness)" menu item	39
8.3.8 "Define section" menu item.....	39
8.3.9 "Save section" menu item	40
8.3.10 "Mesh (section)" menu item	40
8.3.11 "Outline (section)" menu item	40
8.3.12 "Refine" menu item.....	40
8.3.13 "Unrefine" menu item	40
8.4 I mesh menu	41
8.4.1 "Interpolation mesh" menu item	41
8.4.2 "Nodes" menu item	41
8.4.3 "Mesh" menu item	42
8.4.4 "Modify" menu item	42
8.4.5 "Contours" menu item.....	42
8.4.6 "Import/export" menu item	43
8.4.7 "Assign" menu item	43
8.5 Boundary menu	44
8.5.1 "Preferences" menu item.....	44
8.5.2 "Individual potentials" menu item	44
8.5.3 "(Potentials) In section" menu item	45
8.5.4 "(Potentials) Linear" menu item	45
8.5.5 "Point sources" menu item.....	45
8.5.6 "Linear sources" menu item.....	46
8.5.7 "Diffuse sources" menu item	46
8.5.8 "Individual soils" menu item	46
8.5.9 "(Soils) In section" menu item	47
8.6 System menu	47
8.6.1 "Info" menu item	47
8.6.2 "Project identification" menu item.....	47
8.6.3 "Test" menu item	47
8.6.4 "Soil properties" menu item.....	47
8.6.5 "Model" menu item.....	49
8.6.6 "Undo" menu item	50

8.6.7	"Restore" menu item	50
8.6.8	"Preferences" menu item.....	50
8.7	Graphics preferences menu	51
8.7.1	"Refresh and zoom" menu item	51
8.7.2	"Zoom info" menu item	51
8.7.3	"Pen colour and width" menu item	51
8.7.4	"Legend font selection" menu item.....	51
8.7.5	"Draw Mini-CAD objects first" menu item	52
8.7.6	"Mini-CAD toolbar" and "Header toolbar" menu items	52
8.7.7	"Toolbar preferences" menu item	52
8.7.8	"3D toolbar" menu item	53
8.7.9	"Layer/plane legend" menu item.....	54
8.7.10	"General legend" menu item.....	55
8.7.11	"Soil properties legend" menu item	56
8.7.12	"Section legend" menu item.....	57
8.7.13	" $k_r = f(u)$ function legend" menu item.....	57
8.7.14	"Move objects" menu item.....	58
8.7.15	"Save graphics preferences" menu item.....	58
8.7.16	"Load graphics preferences" menu item	58
8.8	Page size + margins menu	59
8.8.1	"Auto-resize" menu item.....	59
8.8.2	"Manual resize (mouse)" menu item.....	59
8.8.3	"Manual resize (editor)" menu item.....	59
8.8.4	"Page size and margins" menu item.....	60
8.8.5	"Font size selection" menu item.....	60
8.8.6	"Margins and borders" menu item	61
8.8.7	"Designation of potential" menu item.....	61
8.9	Evaluation menu.....	62
8.9.1	General notes on the "Evaluation" menu	62
8.9.2	"Normal contours" menu item	62
8.9.3	"Coloured" menu item	63
8.9.4	"3D" menu item	64
8.9.5	"3D array" menu item	66
8.9.6	"Discharges" menu item	66
8.9.7	"Discharge in vertical section" menu item.....	67
8.9.8	"Values along line" menu item	68
8.9.9	"Values in section" menu item.....	69
8.9.10	"Velocities + gradients" menu item	70
8.9.11	"Potentials + velocities" menu item.....	71
8.9.12	"Maximum gradients" menu item	71
8.9.13	"Individual values" menu item.....	71
8.9.14	"Groundwater-surface distance" menu item	71
8.9.15	"Groundwater thickness" menu item	71
8.9.16	"Fence diagram" menu item.....	72
8.9.17	"Difference" menu item	72

8.10 3D menu.....	73
8.10.1 "Wire model" menu item	73
8.10.2 "External model" menu item.....	73
8.10.3 "Potential external model" menu item	74
8.10.4 "Potential surface" menu item.....	75
8.10.5 "Seepage surface" menu item	75
8.10.6 "Flow lines preferences" menu item	75
8.10.6.1 Principles.....	75
8.10.6.2 Preferences	76
8.10.7 "Distance increment" menu item	77
8.10.8 "Time increment" menu item.....	78
8.11 ? menu	79
8.11.1 "Copyright" menu item.....	79
8.11.2 "Maxima" menu item.....	79
8.11.3 "Help" menu item	79
8.11.4 "GGU on the web" menu item.....	79
8.11.5 "GGU support" menu item.....	79
8.11.6 "What's new?" menu item.....	79
8.11.7 "Language preferences" menu item.....	79
9 Tips and tricks.....	80
9.1 Keyboard and mouse.....	80
9.2 Function keys	81
9.3 "Copy/print area" icon.....	82
10 Worked example for a flood protection embankment.....	83
10.1 Step 1: Define the system.....	83
10.2 Step 2: Create record.....	85
10.3 Step 3: Generate FEM nodes and mesh.....	85
10.4 Step 4: Assign z ordinates for the embankment foreland and hinterland.....	86
10.5 Step 5: Assign z ordinates for the top of the embankment.....	87
10.6 Step 6: Refine and optimise mesh in x-/y-direction	88
10.7 Step 7: Assign soil numbers	89
10.8 Step 8: Define boundary conditions	90
10.9 Step 9: Refine mesh in z-direction	91
10.10 Step 10: Assign soil properties	91
10.11 Step 11: Model system	92
10.12 Step 12: Evaluate potentials in "horizontal" planes.....	93
10.13 Step 13: Evaluate potentials in section.....	94
10.14 Step 14: Represent a cut-away system	94
10.15 Step 15: Evaluate discharges	96
10.16 Step 16: Remove the weak spot.....	96
11 Index.....	97

Index of figures:

<i>Figure 1 Definition of planes and layers</i>	<i>9</i>
<i>Figure 2 $kr = f(u)$ function</i>	<i>13</i>
<i>Figure 3 Unsaturated zone</i>	<i>14</i>
<i>Figure 4 Graphical representation of layers/planes in legend</i>	<i>15</i>
<i>Figure 5 Optimisation of diagonals</i>	<i>33</i>
<i>Figure 6 FEM refinement demonstration mesh</i>	<i>35</i>
<i>Figure 7 FEM mesh refinement using Method 1.....</i>	<i>35</i>
<i>Figure 8 FEM mesh refinement using Method 2.....</i>	<i>35</i>
<i>Figure 9 FEM mesh refinement using Method 3.....</i>	<i>36</i>
<i>Figure 10 Worked example in 3D</i>	<i>83</i>
<i>Figure 11 Worked example in section.....</i>	<i>84</i>

1 Preface

The **GGU-3D-SSFLOW** program allows analysis of steady-state groundwater flow in three-dimensional groundwater systems using finite element methods. The program includes a powerful mesh generator and routines (contours, 3D graphics, etc.) for comfortable evaluation of the analysis results.

Data input is in accordance with conventional WINDOWS operations and can therefore be learned almost entirely without the use of a manual. Graphics output supports the true-type fonts supplied with WINDOWS, so that excellent layout is guaranteed. Colour output and any graphics (e.g. files in formats BMP, JPG, PSP, TIF, etc.) are supported. PDF and DXF files can also be imported by means of the integrated Mini-CAD module (see the "**Mini-CAD**" manual).

It is not the aim of this manual to offer an introduction to the finite-element method. For details of the finite-element method, please see O. C. Zienkiewicz, "*Methode der Finiten Elemente*" ("Finite-Element Methods"), Carl Hanser Verlag Munich Vienna, 1984. The hydraulic basics used in the FEM program can be taken from, e.g., J. Buß "*Unterströmung von Deichen*" (Announcement 92, Leichtweiß-Institut, TU Braunschweig).

The program system has been used in a large number of projects and has been thoroughly tested. No faults have been found. Nevertheless, liability for completeness and correctness of the program and the manual, and for any damage resulting from incompleteness or incorrectness, cannot be accepted.

2 Licence protection and installation

In order to guarantee a high degree of quality, a hardware-based copy protection system is used for the **GGU-3D-SSFLOW** program.

The GGU software protected by the **CodeMeter** copy protection system is only available in conjunction with the **CodeMeter stick** copy protection component (hardware for connection to the PC, "**CM stick**"). Because of the way the system is configured, the protected software can only be operated with the corresponding CM stick. This creates a fixed link between the software licence and the CM stick copy protection hardware; the licence as such is thus represented by the CM stick. The correct Runtime Kit for the CodeMeter stick must be installed on your PC.

Upon start-up and during running, the **GGU-3D-SSFLOW** program checks that a CM stick is connected. If it has been removed, the program can no longer be executed.

For installation of GGU software and the CodeMeter software please refer to the information in the **Installation notes for GGU Software International**, which are supplied with the program.

3 Language selection

GGU-3D-SSFLOW is a bilingual program. The program always starts with the language setting applicable when it was last ended.

The language preferences can be changed at any time in the "**Info**" menu, using the menu item "**Spracheinstellung**" (for German) or "**Language preferences**" (for English).

4 Starting the program

After starting the program, you will see two menus at the top of the window:

- File
- Info

After clicking the "**File**" menu, either an existing file can be loaded by means of the "**Load**" menu item, or a new system can be created using "**New**". The program allows simple system input by moving directly to the "**Mesh(x/y)/Common systems**" menu item after "**New**" is clicked (see Section 8.2.9). If you do not want to work with the "**Common systems**" dialog box, click "**No**" in the query box or "**Cancel**" in the subsequent boxes. Then, to start the program you have to enter the number of element layers in a dialog box. You will then return to the home screen. Depending on the type of system selected, you will now see eleven menus along the menu bar:

- File
- Mesh(x/y)
- z
- I mesh
- Boundary
- System
- Graphic preferences
- Page size + margins
- Evaluation
- 3D
- Info

After clicking one of these menus, the so-called menu items roll down, allowing access to all program functions.

The program works on the principle of ***What you see is what you get***. This means that the screen presentation represents, overall, what you will see on your printer. In the last consequence, this would mean that the screen presentation would have to be refreshed after every alteration you make. For reasons of efficiency and as this can take several seconds for complex screen contents, the **GGU-3D-SSFLOW** screen is not refreshed after every alteration.

If you would like to refresh the screen contents, press either [F2] or [Esc]. The [Esc] key additionally sets the screen presentation back to your current zoom, which has the default value 1.0, corresponding to an A3 format sheet.

5 Program concept

The program uses the finite element method. The real solution is a linear approximation for each element. Triangular prisms are used as finite elements. To simplify data input, the mesh is initially generated in plan. The projection of the triangular prisms onto the plan surface results in triangles, so you must first generate a triangular mesh. Depending on the complexity of the system, a number of height ordinates are associated with each node of this basic mesh, which describe the system in the third dimension. The height ordinates can possess different values at each node. The only condition is that every node has the same number of height ordinates. This allows complex systems to be generated.

Using this type of mesh generation, successive planes of nodes are located above each other, whereby *plane* can be somewhat misleading because the nodes of a *node plane* are not necessarily on one plane. Instead of the term *node plane* we will simply use *plane*.

The regions between these planes are called (element) *layers*. They are filled with triangular prisms.

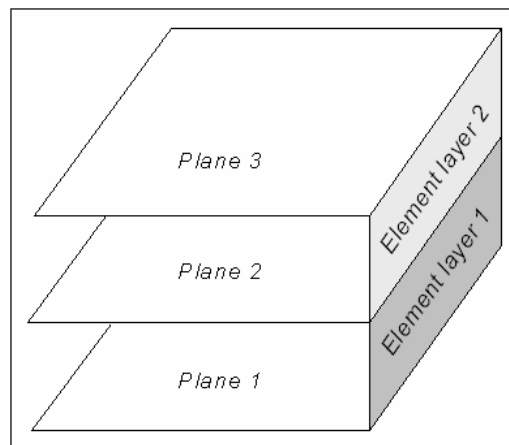


Figure 1 Definition of planes and layers





You can assign a boundary condition to each node (source or potential). You can also assign different soil properties to each triangular prism.

If you have already worked with the GGU program **GGU-SS FLOW2D**, you will hardly encounter problems with the **GGU-3D-SSFLOW** mesh generator. The additional effort involved consists of having to manage a *battery* of z ordinates for each node.

6 Short description

Since I know from personal experience that reading user-manuals is quite a chore, there will now follow a short description of the main program functions. After studying this section, you will be in a position to model steady-state three-dimensional groundwater flow. More detailed information about the program can be obtained from the subsequent chapters as you require it.

- Design the system to be modelled.
- Start **GGU-3D-SSFLOW** and go to the "**File/New**" menu item. If you do not wish to proceed via "**Common systems**" answer no to the question and enter the number of element layers. This can be increased or reduced later.
- If necessary, adjust the sheet coordinates to fit your system. Use "**Page size + margins/Manual resize (editor)**" to do this.
- Then go to "**Mesh(x/y)/Define nodes**".
- Using the mouse, click the governing nodes (points) in your system. The points will be numbered. Alternatively, you can also enter the nodes in tabular form using "**Mesh(x/y)/Change**". Each newly generated node is awarded the values entered in "**z/Default depths**" as z ordinates. During and after mesh generation, you can edit the z ordinates associated with a node at will (see below).
- If the nodes lie outside of the visible sheet coordinates, go to "**Page size + margins/Auto-resize**" or press [F9].
- Now go to "**Mesh(x/y)/Manual mesh**" and unite groups of three nodes to single triangular elements. In this manner you generate a rough structure for your system. Alternatively, by using "**Mesh(x/y)/Automatic**", you can have the program do this work automatically for you. During automatic mesh generation the triangular prisms are assigned the default soil column for all new triangular elements. The default soil column defines the soil numbers of the individual triangular prisms. The soil numbers can also be subsequently edited at any time.
- If you want to edit the positions of mesh nodes, select one of the menu items "**Mesh(x/y)/Edit**", "**Mesh(x/y)/Change**" or "**Mesh(x/y)/Move**".
- To define the depths of the planes, enter the plane surfaces in the "**z/Default depths**" menu item.
- If you want to delete a triangular element, select the "**Mesh(x/y)/Manual mesh**" menu item once again and click the three corner nodes of the corresponding element. After selecting this menu item, try a double-click in a triangular element.
- You can refresh the screen at any time using [Esc] or [F2].
- Turn your rough structure into a fine structure by using the menu items "**Mesh(x/y)/Individually refine**" or "**Mesh(x/y)/Section**" or "**Mesh(x/y)/All**".
- After mesh refinement, you can still edit your system at will using "**Mesh(x/y)/Define nodes**", "**Mesh(x/y)/Manual mesh**", etc.
- For demonstration purposes, use the menu item "**Mesh(x/y)/Move**" to generate one or more acute, and thus numerically unfavourable, triangular elements. Then select the "**Mesh(x/y)/Optimise**" menu item and follow the effects on the screen.
- Previously, each node was assigned the default depths during mesh generation. If you want to see your mesh in a vertical section, use "**z/Define section**" to define a section through the system.
- Use "**Boundary/Individual potentials**", for example, to define the boundary conditions governing your system. A potential can be assigned to the node in the current plane or to all nodes associated with this point.

- You can move through the planes using the  and  tools, which you can find in the **3D toolbar** at the lower left of the program window. The active plane and the active layer, in which you now find yourself, is shown in the *Layer/plane legend*.
- If you wish, you can edit the soil number using the "**Boundary/Individual soils**" or "**Boundary/In section**" menu items. Every soil number can subsequently be assigned different soil properties. The soil numbers defined refer to the current layer.
- If necessary, edit the soil properties using the menu item "**System/Soil properties**".
- Once you have completed mesh generation, go to "**System/Model**" to commence modelling. Where required, a bandwidth optimisation will be carried out before modelling commences, in order to achieve a numerically favourable equation system.
- If wished for, you can send the results to a printer as an output table or write them to a file once modelling is finished. However, this method of result presentation is generally less than satisfactory to the client.
- So, proceed immediately to the "**Evaluation**" or "**3D**" menus. You can almost always leave the subsequent dialog boxes using the "**OK**" button, without having to make further changes. The program will usually suggest sensible input. The "**Determine extreme values...**" button, which appears in a number of dialog boxes, should be clicked once however, otherwise an error message will appear.
- Define a section (menu item "**z/Define section**") and then press [**Return**]. Now select the  tool from the toolbar at the lower left of the program window. You will now see a *cut-away* of the system. You may rotate the system in steps around the z axis using the  tool. Try pressing the [**F9**] function key (or use "**Page size + margins/Auto-resize**"). Change the current plane from the one you are now in.
- If you find yourself in a section and wish to return to a representation or evaluation in plan, go to "**Mesh(x/y)/Mesh**".
- If you have a colour printer correctly installed in WINDOWS, you can create colour output by going to "**File/Print and export**" and then pressing the "**Printer**" button in the subsequent dialog box. For *monochrome* printers, grey scales will be used.

At the end of this manual, the program is described using a comprehensive example. This short description demonstrates that only a few items need be selected to model a groundwater system. All further menu items deal principally with saving data, layout and further evaluation of the model, where required. The description is given in the following chapters.

7 Theoretical principles

7.1 General

An analytical solution is only possible for simple groundwater systems. When modelling complicated systems, numerical solutions are required. In the main

- finite-difference-methods (FDM) and
- finite-element-methods (FEM)

are used. Using finite methods, the whole area is subdivided into a large amount of smaller (finite) areas (elements). Generally, using FEM, triangles are selected for these areas. Within these triangles simple, generally linear, approximation functions are used. The actual, complicated, whole solution is pieced together like a mosaic from the many simple partial solutions. Equation systems result, in which the number of variables corresponds to the number of system nodes. With the finite-difference-method one generally has only the possibility of defining a discrete whole using rectangular partial-areas. In contrast to FEM therefore, FDM is a lot less flexible when it comes to adjusting to complicated boundary structures. Also, in some areas, mesh refinement is not as easy. Further, the resulting equation systems are numerically more stable for FEM. The main advantage in FDM is in the theoretically simpler mathematical relationships, which generally will be of little interest to the program user. The **GGU-3D-SSFLOW** program uses the finite-element-method.

When using the application please remember that all finite-element or finite-difference-methods are approximation methods. The quality of the approximation compared to the actual solution increases with the fineness of the mesh. You should pay attention to the fact that in areas where the main subsurface hydraulic action takes place (e.g. wells, seepage elements), the mesh subdivisions should be finer. The type of triangle used also exerts a certain influence. Optimum conditions are achieved with equilateral triangles. You can get an overview of the solution quality by calculating the same system with a coarser and a finer mesh subdivision, and comparing the deviation of the two results.

The following general notes are important:

- triangular elements are used,
- Darcy's Law is valid,
- the piezometric heads are calculated in a linear manner for each element,
- from the linear approximation of the piezometric heads, a constant approximation of velocities results. In order to achieve a better quality of velocity approximation, the velocities for calculation of flow lines are averaged to node values in a follow-up calculation.

The procedure is:

For each element node the velocities in the neighbouring elements are added and then divided by the number of neighbouring elements. By doing this, the velocity course can be better presented. At the boundary nodes, the results are naturally not quite as exact. Further, the approximation of velocities in the region of element nodes with differing material types can be worsened in this way. If the velocities at such **boundary nodes** are of great interest, help is available by refining the element mesh in these areas.

7.2 Differential equation

The application solves the differential equation:

$$k_r \cdot k_x \cdot \partial^2 h / \partial x^2 + k_r \cdot k_y \cdot \partial^2 h / \partial y^2 + k_r \cdot k_z \cdot \partial^2 h / \partial z^2 + Q = 0$$

Where

- k_x, k_y, k_z = permeability in m/s, for example, for the x , y and z directions
- h = piezometric head in m, for example
- k_r = coefficient for determining the permeability in the unsaturated zone (dimensionless)
- Q = discharge in m³/s, for example
- x, y, z = coordinates in m, for example

The value k_r records the change in permeability in the unsaturated zone r , above groundwater, and can take on values between 0.0 and 1.0. In the saturated area of the system k_r is 1.0. The value k_r is a function of the pore water pressure u . The pore water pressure u is calculated from the piezometric head h , the elevation head z and the unit weight of water γ_w .

$$u = (h - z) \cdot \gamma_w$$

In Figure 2, the curves are presented for three typical soils. Using this value in the differential equation has the distinct advantage, for the following finite-element calculation, that phreatic lines can be very easily calculated.

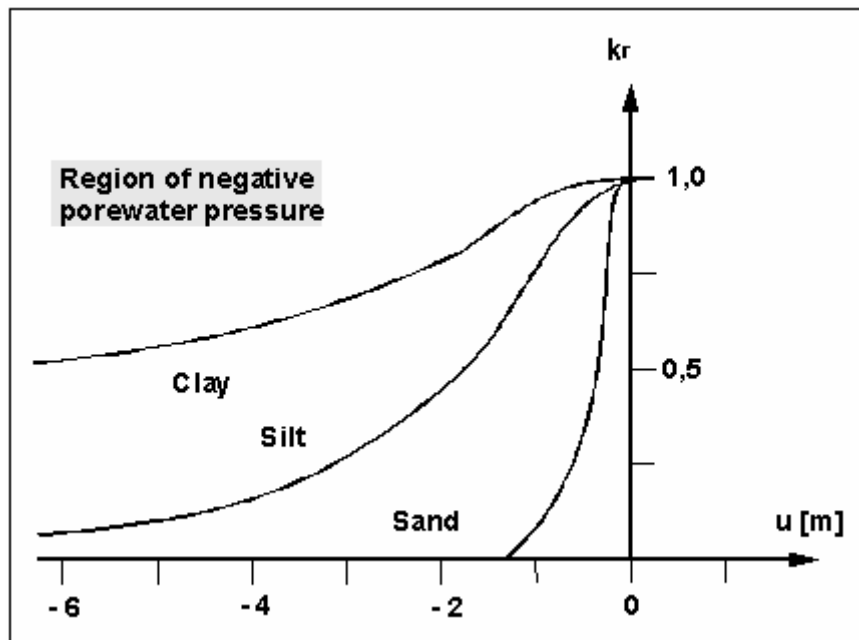


Figure 2 $k_r = f(u)$ function

Figure 3 serves as a further explanation. For the case of a horizontal groundwater table, all four variables h , y , u and kr are given as functions.

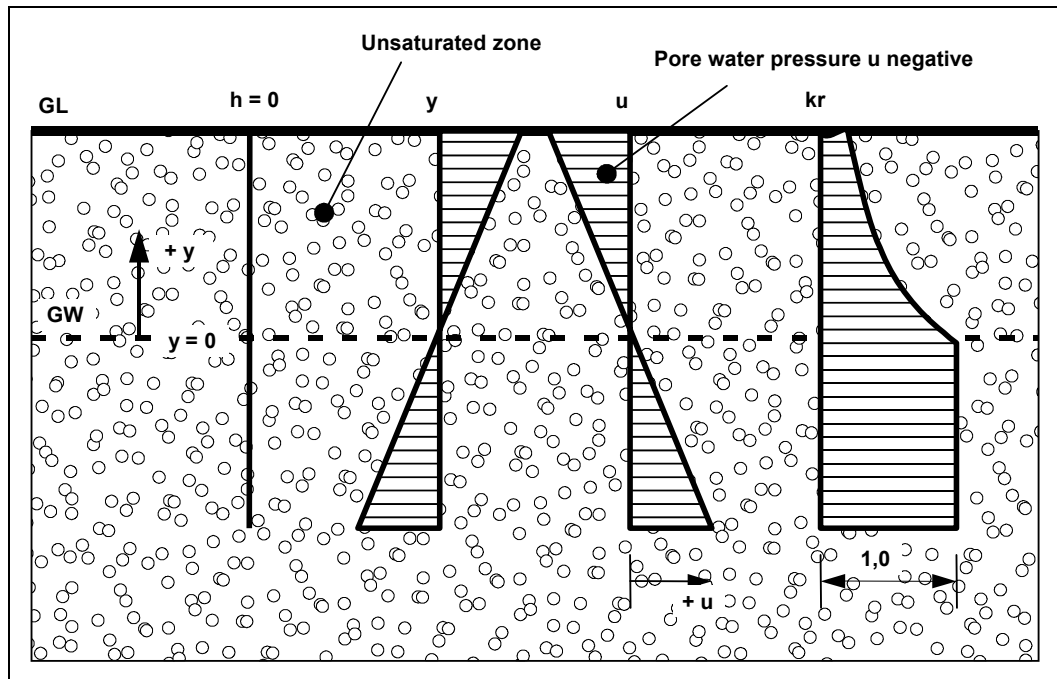


Figure 3 Unsaturated zone

If unsaturated zones are present in the system, and a phreatic line analysis is to be carried out, an iteration process is required, as the position of the phreatic line is not known from the outset. In the first iteration step, the program assumes that all areas of the system are saturated:

$$kr = 1.0$$

After the initial calculation of the piezometric heads at the system nodes, the pore water pressures u are calculated and kr determined. With the altered permeabilities, a new calculation of the system follows, with new piezometric heads and correspondingly new pore water pressures and kr values. The iteration process is carried on until the difference in piezometric head in iteration step (i) and (i - 1) falls below the user-defined limit value (iteration divergence).

Note on boundary conditions:

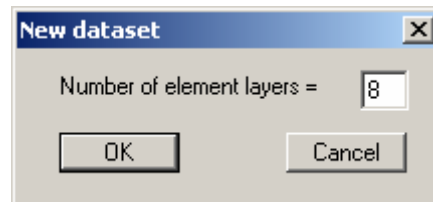
The case of an impermeable boundary is automatically taken in to account by the finite-element-method. Valid is, that all system boundaries or partial system boundaries which have no water level or source boundary conditions, are automatically impermeable.

8 Description of menus items

8.1 File menu

8.1.1 "New" menu item

After going to the "New" menu item, define the number of element layers for your system in the dialog box.



Element layer numbering begins at the bottom with *1*, the planes that bound on an element layer are also continuously numbered from top to bottom. Because the system ends at the bottom, the number of element layers is one less than the number of planes. To aid orientation, the active layer and plane are labelled and coloured in the *Layer/plane legend* (see Figure 4).

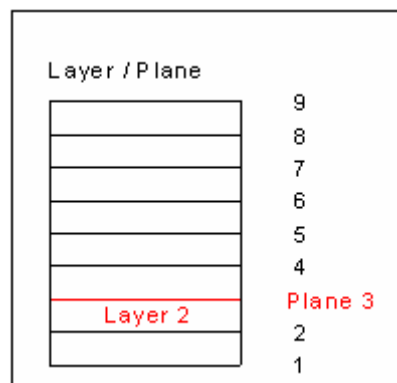


Figure 4 Graphical representation of layers/planes in legend

8.1.2 "Load" menu item

You can load a file with system data, which was created and saved at a previous sitting, and then edit the system.

8.1.3 "Save" menu item

You can save data entered or edited during program use to a file, in order to have them available at a later date, or to archive them. The data is saved without prompting with the name of the current file. Loading again later creates exactly the same presentation as was present at the time of saving.

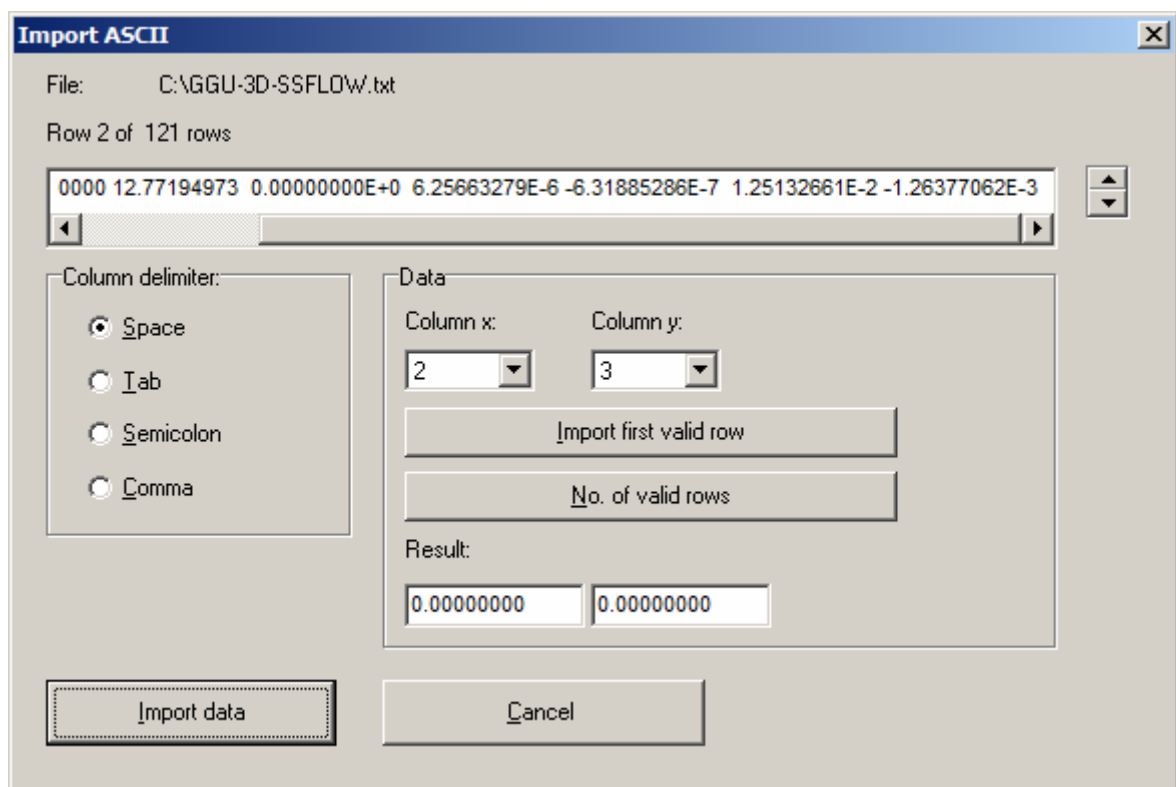
8.1.4 "Save as" menu item

You can save data entered during program use to an existing file or to a new file, i.e. using a new file name. For reasons of clarity, it makes sense to use ".f3d" as file suffix, as this is the suffix used in the file requester box for the menu item "File/Load". If you choose not to enter an extension when saving, ".f3d" will be used automatically.

If, at the time of saving, the current system has been analysed, you can decide in a dialog box whether to save the analysis results with the document. If you select "No" here you must reanalyse the file if it is opened again later.

8.1.5 "Import ASCII file" menu item

If the coordinates of the FEM mesh nodes are available in ASCII file format, they can be imported into the program. Each row of the file must contain the x- and y-value of a node. Decimal fractions must use a point, not a comma. When importing the ASCII file you must specify the columns containing the x- and the y-values. New x and y coordinates are assigned the default depths.



The current row of the ASCII file is shown at the top. You can navigate through the file using the arrow buttons on the right. If all the information is correct, the result for the row is shown in the box below the column. Otherwise, an error message appears. You may need to change the column delimiter. If the file contains invalid as well as valid rows, these will simply be skipped during the subsequent import. Finally, select the "Import data" button. The imported coordinates can then be processed to form an FEM mesh.

8.1.6 "Export to ASCII file" menu item

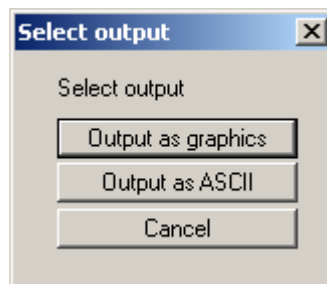
If an FEM mesh has been generated, the node coordinates can be saved to an ASCII file. The x and y coordinates of the underlying mesh (2D) are saved and, in addition, if the system has been modelled, the potential of the active plane.

8.1.7 "Print output table" menu item

8.1.7.1 *Selecting the output format*

You can have a table printed containing the current analysis results. The results can be sent to the printer or to a file (e.g. for further editing in a word processor). The output contains all information on the current state of analysis, including the system data.

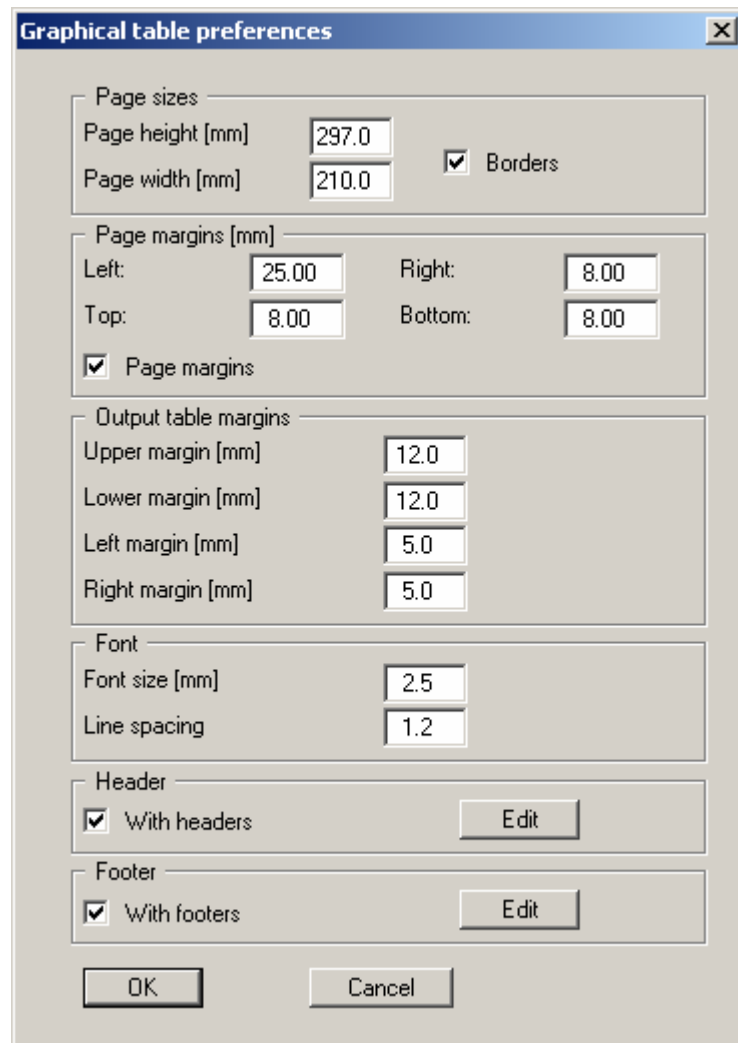
You have the option of designing and printing the output table as an annex to your report within the **GGU-3D-SSFLOW** program. To do this, select "**Output as graphics**" from the following options.



If you prefer to easily print or process the data in a different application, you can send them directly to the printer or save them to a file using the "**Output as ASCII**" command button.

8.1.7.2 Button "Output as graphics"

If you selected the "Output as graphics" button in the previous dialog box a further dialog box opens, in which you can define further preferences for result visualisation.



The dialog box titled "Graphical table preferences" contains several sections for configuring the output table. The "Page sizes" section includes input fields for "Page height [mm]" (297.0) and "Page width [mm]" (210.0), along with a checked "Borders" checkbox. The "Page margins [mm]" section has input fields for "Left" (25.00), "Right" (8.00), "Top" (8.00), and "Bottom" (8.00), with a checked "Page margins" checkbox. The "Output table margins" section includes input fields for "Upper margin [mm]" (12.0), "Lower margin [mm]" (12.0), "Left margin [mm]" (5.0), and "Right margin [mm]" (5.0). The "Font" section has input fields for "Font size [mm]" (2.5) and "Line spacing" (1.2). The "Header" section has a checked "With headers" checkbox and an "Edit" button. The "Footer" section has a checked "With footers" checkbox and an "Edit" button. At the bottom are "OK" and "Cancel" buttons.

Page sizes	
Page height [mm]	297.0
Page width [mm]	210.0
<input checked="" type="checkbox"/> Borders	

Page margins [mm]	
Left:	25.00
Right:	8.00
Top:	8.00
Bottom:	8.00
<input checked="" type="checkbox"/> Page margins	

Output table margins	
Upper margin [mm]	12.0
Lower margin [mm]	12.0
Left margin [mm]	5.0
Right margin [mm]	5.0

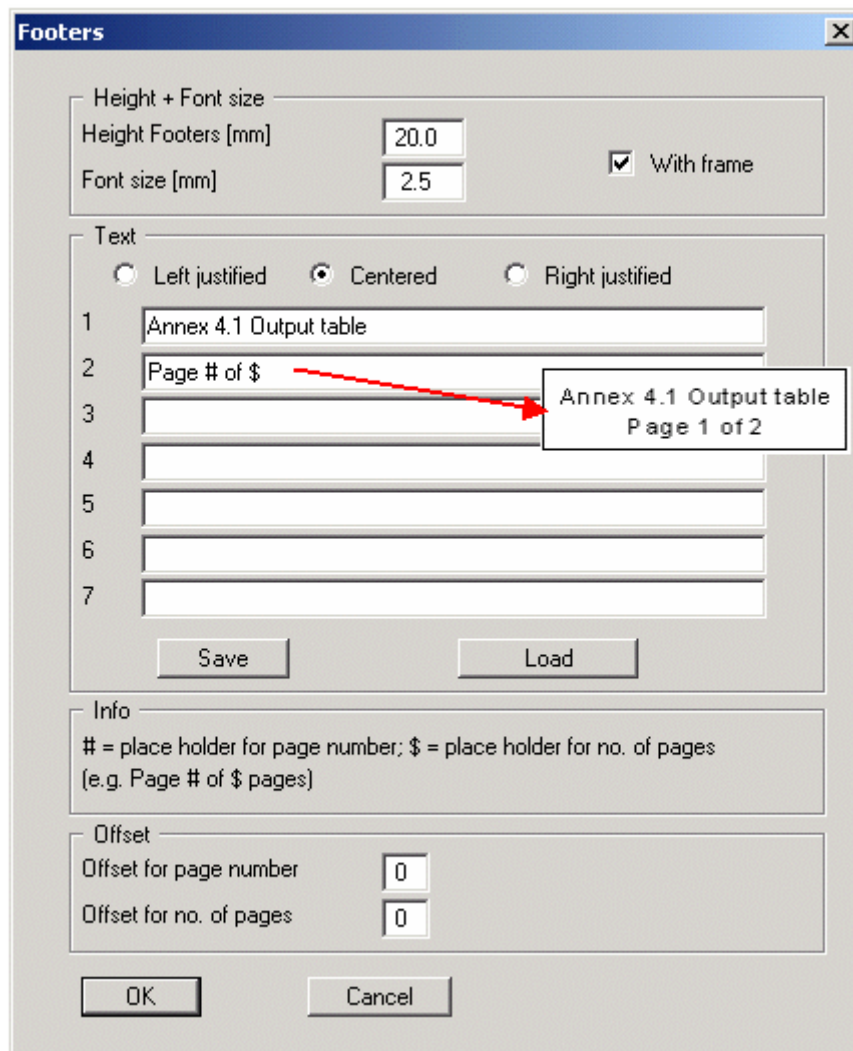
Font	
Font size [mm]	2.5
Line spacing	1.2

Header	
<input checked="" type="checkbox"/> With headers	Edit

Footer	
<input checked="" type="checkbox"/> With footers	Edit

OK Cancel

You can define the desired layout for the output table in various areas of the dialog box. If you need to add a header or footer (e.g. for page numbering), activate the appropriate check boxes "With headers" and/or "With footers" and click on the "Edit" button. You can then edit as required in a further dialog box.



Footers

Height + Font size

Height Footers [mm]

Font size [mm]

☒ With frame

Text

☐ Left justified ☒ Centered ☐ Right justified

1

2

3

4

5

6

7

Info

= place holder for page number; \$ = place holder for no. of pages
(e.g. Page # of \$ pages)


Offset

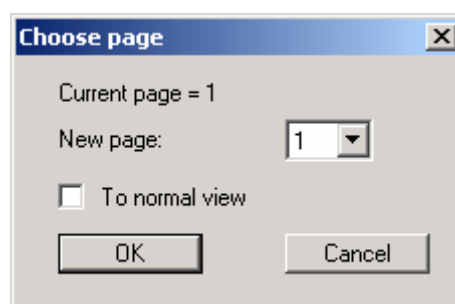
Offset for page number

Offset for no. of pages

Automatic pagination can also be employed here if you work with the placeholders as described. After exiting the dialog boxes using "OK" you will see a further dialog box in which you can select the parameters to be used in the output table. If you click the "Start" button the output table is presented on the screen page by page. To navigate between the pages, use the arrow tools



in the toolbar. If you need to jump to a given page or back to the graphical representation, click on the  tool. You will then see the following box:



Choose page

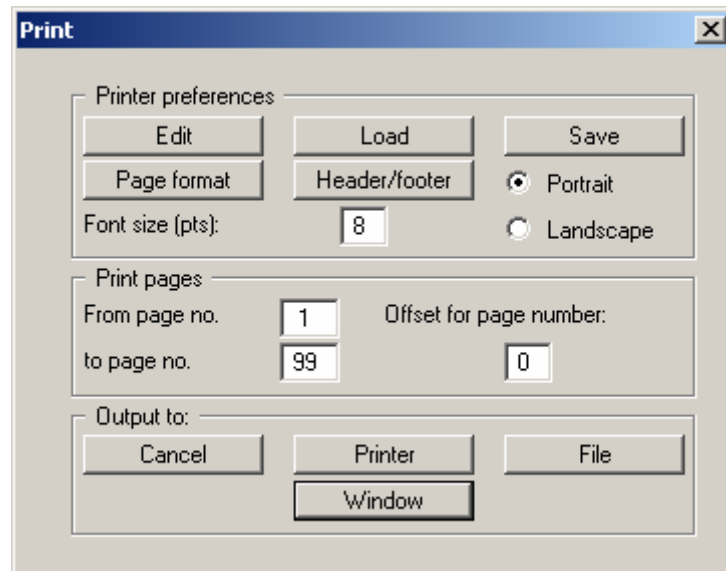
Current page = 1

New page:

☐ To normal view

8.1.7.3 Button "Output as ASCII"

You can have your analysis data sent to the printer, without further work on the layout, or save it to a file for further processing using a different program, e.g. a word processing application. After selecting the button **"Output as ASCII"** you will see a further dialog box in which you can select the parameters to be used in the output table. If you click the **"Start"** button, the following dialog box appears in which you can define output preferences.



In the dialog box you can define output preferences:

- **"Printer preferences"** group box
Using the **"Edit"** button the current printer preferences can be changed or a different printer selected. Using the **"Save"** button, all preferences from this dialog box can be saved to a file in order to have them available for a later session. If you select **"GGU-3D-SSFLOW.drk"** as file name and save the file in the program folder (default), the file will be automatically loaded the next time you start the program.

Using the **"Page format"** button you can define, amongst other things, the size of the left margin and the number of rows per page. The **"Header/footer"** button allows you to enter a header and footer text for each page. If the **"#"** symbol appears within the text, the current page number will be entered during printing (e.g. **"Page #"**). The text size is given in **"Pts"**. You can also change between **"Portrait"** and **"Landscape"** formats.
- **"Print pages"** group box
If you do not wish pagination to begin with **"1"** you can add an *offset number* to the check box. This offset will be added to the current page number. The output range is defined using **"From page no."** **"to page no."**.
- **"Output to:"** group box
Start output by clicking on **"Printer"** or **"File"**. The file name can then be selected from or entered into the box. If you select the **"Window"** button the results are sent to a separate window. Further text editing options are available in this window, as well as loading, saving and printing.

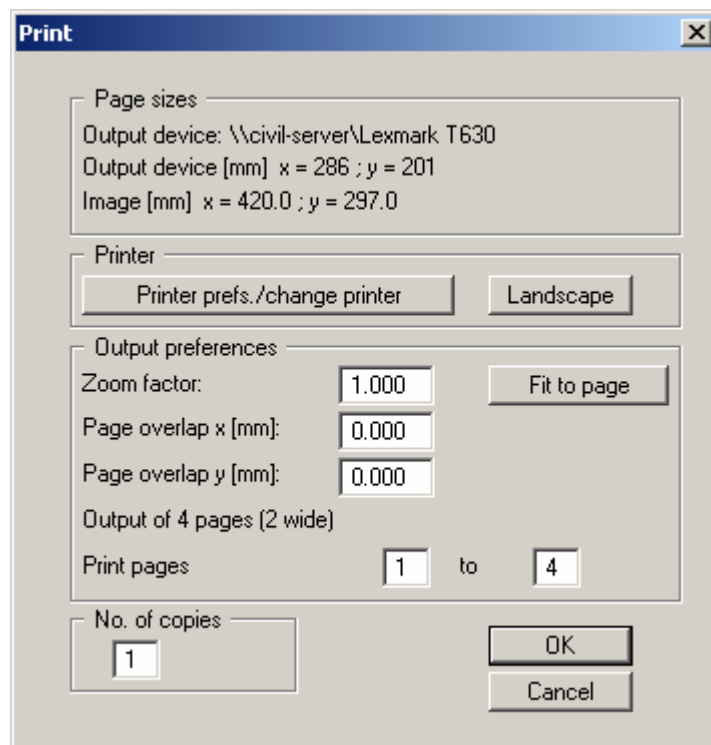
8.1.8 "Printer preferences" menu item

You can edit printer preferences (e.g. swap between portrait and landscape) or change the printer in accordance with WINDOWS conventions.

8.1.9 "Print and export" menu item

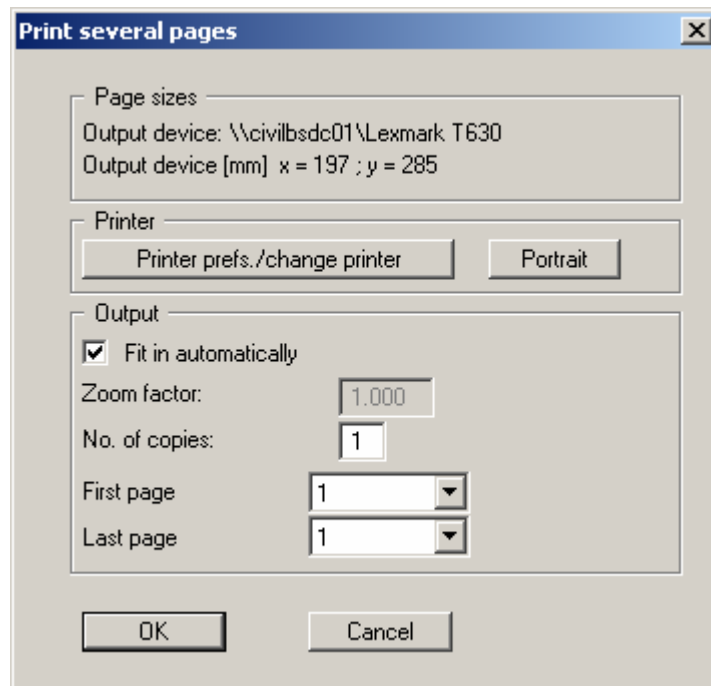
You can select your output format in a dialog box. You have the following possibilities:

- **"Printer"**
allows graphic output of the current screen contents (*graphical representation*) to the WINDOWS standard printer or to any other printer selected using the menu item **"File/Printer preferences"**. But you may also select a different printer in the following dialog box by pressing the **"Printer prefs./change printer"** button.




In the upper group box, the maximum dimensions which the printer can accept are given. Below this, the dimensions of the image to be printed are given. If the image is larger than the output format of the printer, the image will be printed to several pages (in the above example, 4). In order to facilitate better re-connection of the images, the possibility of entering an overlap for each page, in x and y direction, is given. Alternatively, you also have the possibility of selecting a smaller zoom factor, ensuring output to one page ("**Fit to page**" button). Following this, you can enlarge to the original format on a copying machine, to ensure true scaling. Furthermore, you may enter the number of copies to be printed.

If you have activated the **table representation** on the screen, you will see a different dialog box for output by means of the "File/Print and export" menu item button "Printer".



Here, you can select the table pages to be printed. In order to achieve output with a zoom factor of 1 (button "Fit in automatically" is deactivated), you must adjust the page format to suit the size format of the output device. To do this, use the dialog box in "File/Print output table" button "Output as graphics".

- **"DXF file"**
allows output of the graphics to a DXF file. DXF is a common file format for transferring graphics between a variety of applications.
- **"GGUCAD file"**
allows output of the graphics to a file, in order to enable further processing with the **GGUCAD** program. Compared to output as a DXF file this has the advantage that no loss of colour quality occurs during export.
- **"Clipboard"**
The graphics are copied to the WINDOWS clipboard. From there, they can be imported into other WINDOWS programs for further processing, e.g. into a word processor. In order to import into any other WINDOWS program you must generally use the "Edit/Paste" function of the respective application.
- **"Metafile"**
allows output of the graphics to a file in order to be further processed with third party software. Output is in the standardised EMF format (Enhanced Metafile format). Use of the Metafile format guarantees the best possible quality when transferring graphics.

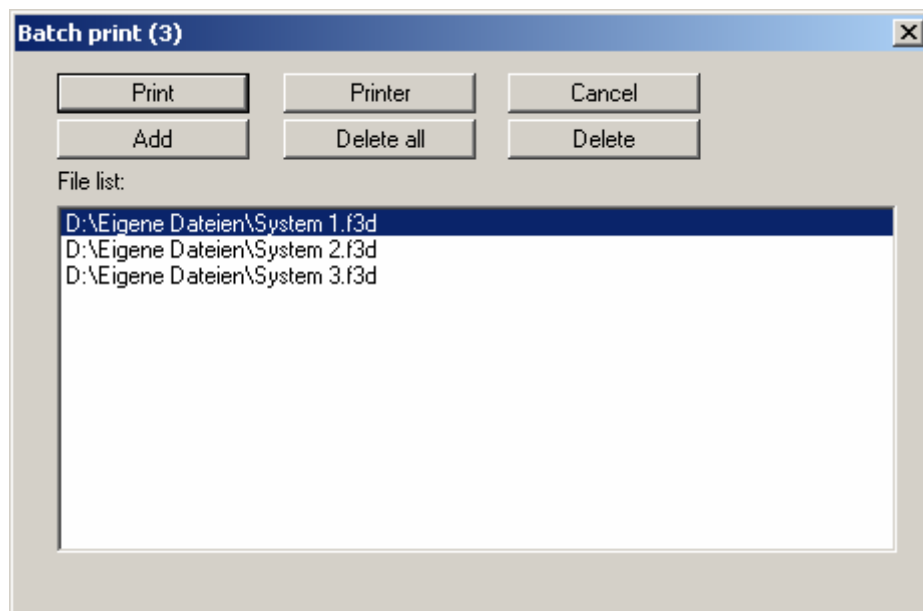
If you select the "Copy/print area" tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see "Tips and tricks", Section 9.3).

Using the "Mini-CAD" program module you can also import EMF files generated using other GGU applications into your graphics.

- **"MiniCAD"**
allows export of the graphics to a file in order to enable importing to different GGU applications with the **Mini-CAD** module.
- **"GGUMiniCAD"**
allows export of the graphics to a file in order to enable processing in the **GGUMiniCAD** program.
- **"Cancel"**
Printing is cancelled.

8.1.10 "Batch print" menu item

If you would like to print several appendices at once, select this menu item. You will see the following dialog box:



Create a list of files for printing using **"Add"** and selecting the desired files. The number of files is displayed in the dialog box header. Using **"Delete"** you can mark and delete selected individual files from the list. After selecting the **"Delete all"** button, you can compile a new list. Selection of the desired printer and printer preferences is achieved by pressing the **"Printer"** button.

You then start printing by using the **"Print"** button. In the dialog box which then appears you can select further preferences for printer output such as, e.g., the number of copies. These preferences will be applied to all files in the list.

8.1.11 "Export to GGU-3D-TRANSIENT" menu item

You are offered the opportunity of subjecting your steady-state, three-dimensional groundwater model to a transient analysis using the **GGU-3D-TRANSIENT** program. The necessary transient data can be generated using this menu item.

8.1.12 "Exit" menu item

After a confirmation prompt, you can quit the program.

8.1.13 "1, 2, 3, 4" menu items

The "1, 2, 3, 4" menu items show the last four files worked on. By selecting one of these menu items the listed file will be loaded. If you have saved files in any other folder than the program folder, you can save yourself the occasionally onerous *rummaging* through various sub-folders.

8.2 *Mesh(x/y) menu*

8.2.1 "Preferences" menu item

In the dialog box for this menu item you specify how the FEM mesh for your system or the outline of the FEM mesh is displayed on the screen. The element no. and the soil no. cannot be displayed simultaneously. The preferences are accepted after closing the box using "OK".


8.2.2 "Mesh" menu item

If you go to this menu item or click the  tool in the toolbar, the FEM mesh is displayed as defined in "**Mesh(x/y)/Preferences**".

The active element layer and plane are shown in the *Layer/plane legend* in the top right corner of the screen. To select a different plane click the arrow buttons in the *3D toolbar* (see Section 8.7.8).

If the mesh representation does not fill the screen, point to the "Auto-resize" menu item in the "Page size + margins" menu or press [F9].

8.2.3 "Outline" menu item

If you go to this menu item or click the  tool in the toolbar, the outline of the various soils in the FEM system is displayed as defined in "**Mesh(x/y)/Preferences**".

The active element layer and plane are shown in the *Layer/plane legend* in the top right corner of the screen. To select a different plane click the arrow buttons in the *3D toolbar* (see Section 8.7.8).

8.2.4 "Define nodes" menu item

You can use the left mouse button to define a new node or the right mouse button to delete a previously defined node. This menu item can also be reached using [F3]. Each new point created with this menu item will be assigned the default depths z values (see "**z/Default depths**" menu item, Section 8.3.1).

The current x- and y-coordinates of the mouse are shown in the status bar. If you have access to a scanner you can scan in the system to be processed and save it as a bitmap file (extension: .BMP). This bitmap can be displayed on the screen using the Mini-CAD program module (see the "**Mini-CAD**" manual). This greatly simplifies input of the principal system nodes. The size of the bitmap can be adjusted to scale. However, this is not a necessity for mesh generation. If you do not model the system to scale using a bitmap, you can perform a scale correction after coordinate input is complete by going to "**Mesh(x/y)/Change**" and selecting the "**Using formula**" button.

Alternatively, you can also import a DXF file using Mini-CAD (see "**Mini-CAD**" manual). This can contain the system outline, for example. If Mini-CAD data are already present, the initial dialog box provides the check box "**Lock onto Mini-CAD lines**". If you activate this check box the mouse cursor appears as a rectangle with cross-hairs. If the end point of a Mini-CAD line is located within this rectangle the program will lock onto this point precisely; if a number of points are located within the rectangle, it will lock onto the one nearest the centre of the cross-hairs.

8.2.5 "Change" menu item

The coordinates of existing nodes can be edited. Three options are available for this:

- "**Via a table**"

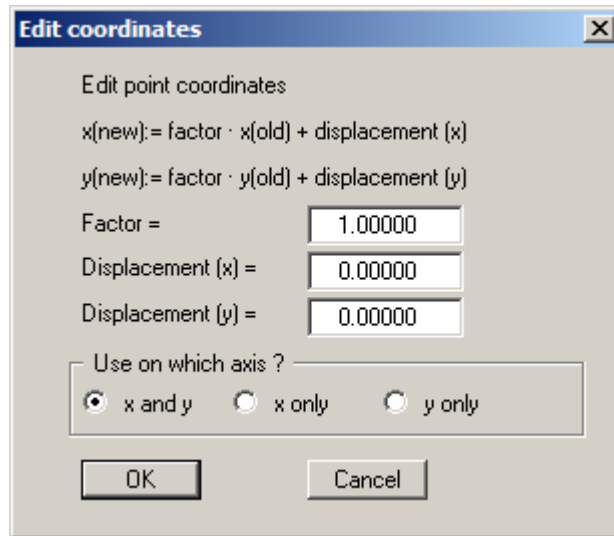
You can edit the coordinates of existing nodes or, alternatively, enter the coordinates of new nodes.

If you need to edit the current number of nodes click the "**x FEM nodes to edit**" button and enter the new number of nodes. You can navigate through the table using "**Forw.**" and "**Back**". The new nodes will be assigned the default depths as z values (see "**z/Default depths**" menu item, Section 8.3.1). If you set the number of nodes to **0** the FEM incidence table is deleted..

It is even easier to import node coordinates via the Windows clipboard. For example, if the x-/y-coordinates of the FEM mesh nodes are available in an Excel table, it is possible to copy the two columns containing the data into the clipboard ("**Edit/Copy**") and then to paste them into the dialog box "**Via a table**" by pressing "**Import clipboard**".

- **"Via equation"**

If you have entered the coordinates using the wrong scale, for instance, you can correct this using this menu item.



You have the option of applying the factor for one direction only. Activate the required option buttons.

- **"In section"**

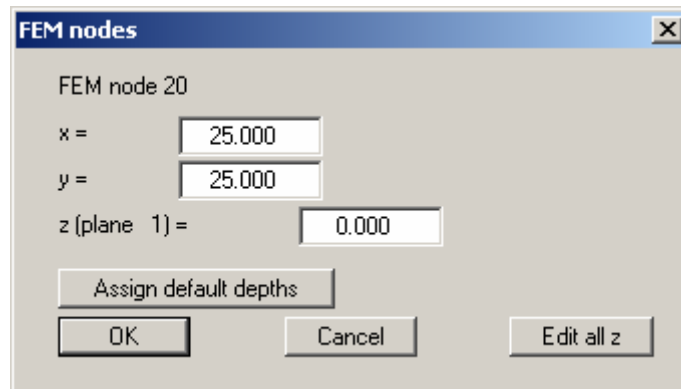
If the coordinates are to be altered for nodes in a specific area only, the area can be specified after selecting this button and the displacement of the nodes in the x- and/or y-directions then be entered in a dialog box. By activating the **"Do not displace nodes with boundary conditions"** check box the nodes concerned can be excluded from any displacement.

8.2.6 "Move" menu item

After selecting this function, the defined FEM system, complete with FEM elements appears on the screen. The nodes can now be moved with the left mouse button pressed. The current node coordinates are shown in the status bar at the bottom of the window. The last node displacement can be undone using backspace.

8.2.7 "Edit" menu item

By double-clicking a node using the left mouse button a dialog box appears allowing the coordinates to be edited via the keyboard.

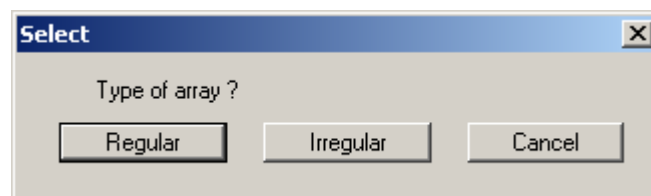


Besides altering the horizontal position of the node, the vertical position in the current plane can also be modified. After clicking "**Edit all z**" the depth of all nodes at the selected x/y location can be altered. Alternatively, the default depths (see Section 8.3.1) can be assigned.

8.2.8 "Array" menu item

8.2.8.1 *Select type of array*

After selecting this menu item you can decide whether new nodes are generated using a regular or an irregular array.



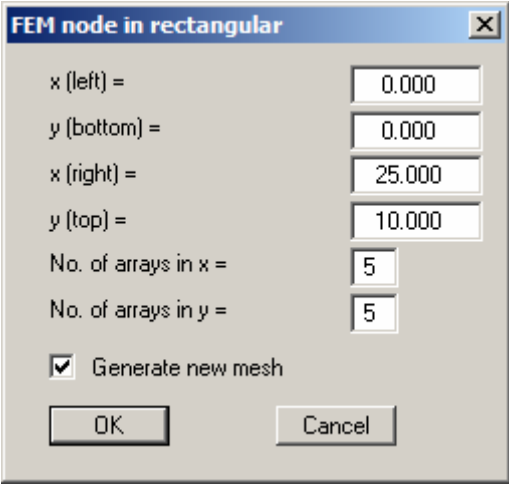
Each new point created with this menu item will be assigned the default depths (see "**z/Default depths**" menu item, Section 8.3.1).

8.2.8.2 Button "Regular"

The regular array allows the nodes to be defined in a number of ways:

- **"Line"** - along one or more lines,
- **"Rectangle"** - in one or more rectangles,
- **"Quadrilateral"** - in one or more quadrilaterals.

The procedure is similar for all three cases. Therefore, only the rectangles will be described.



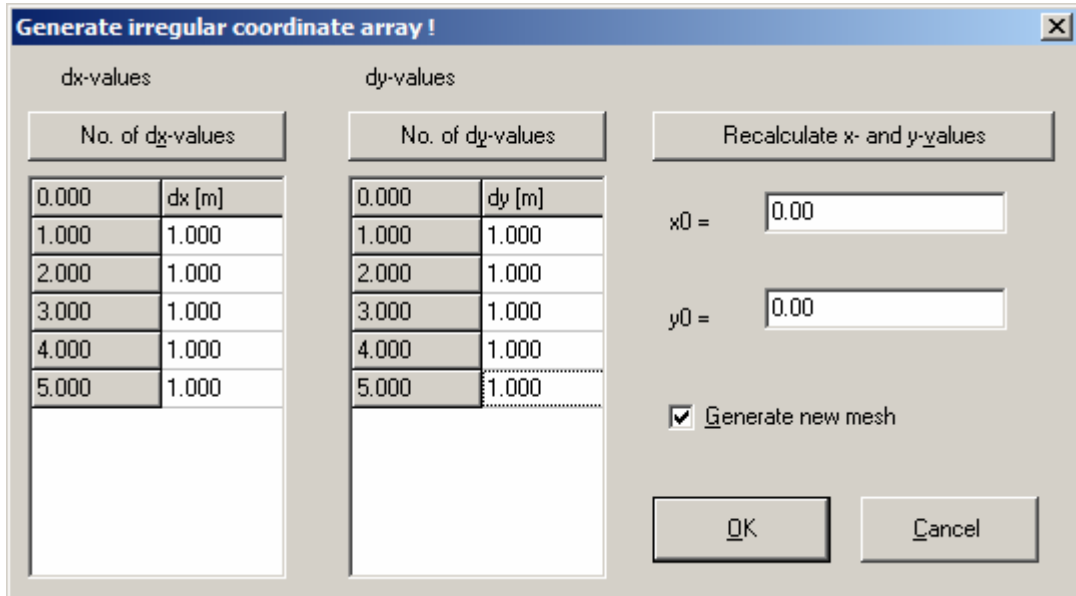
The screenshot shows a dialog box titled "FEM node in rectangular". It contains the following fields and controls:

- x (left) = 0.000
- y (bottom) = 0.000
- x (right) = 25.000
- y (top) = 10.000
- No. of arrays in x = 5
- No. of arrays in y = 5
- ☒ Generate new mesh
- OK button
- Cancel button

Enter the corner points of the array and the number of subdivisions. If the **"Generate new mesh"** check box is activated, all existing nodes are deleted and then the user-defined FEM nodes in the dialog box are generated with the appropriate FEM mesh.

8.2.8.3 Button "Irregular"

In contrast to the regular array procedure, where the generated nodes are evenly spaced within the generated rows, the spacing can be varied using the irregular array. This can be done in the following dialog box.



The dialog box titled "Generate irregular coordinate array !" contains two main sections for defining the array: "dx-values" and "dy-values". Each section has a button "No. of dx-values" and "No. of dy-values" respectively. Below these buttons are two tables. The "dx-values" table has two columns: the first column contains values from 0.000 to 5.000, and the second column, labeled "dx [m]", contains the value 1.000 for each row. The "dy-values" table has two columns: the first column contains values from 0.000 to 5.000, and the second column, labeled "dy [m]", contains the value 1.000 for each row. To the right of these tables is a button "Recalculate x- and y-values". Below this button are two input fields: "x0 =" and "y0 =", both containing the value 0.00. Below these input fields is a checked checkbox labeled "Generate new mesh". At the bottom right are two buttons: "OK" and "Cancel".

dx-values	
No. of dx-values	
0.000	dx [m]
1.000	1.000
2.000	1.000
3.000	1.000
4.000	1.000
5.000	1.000

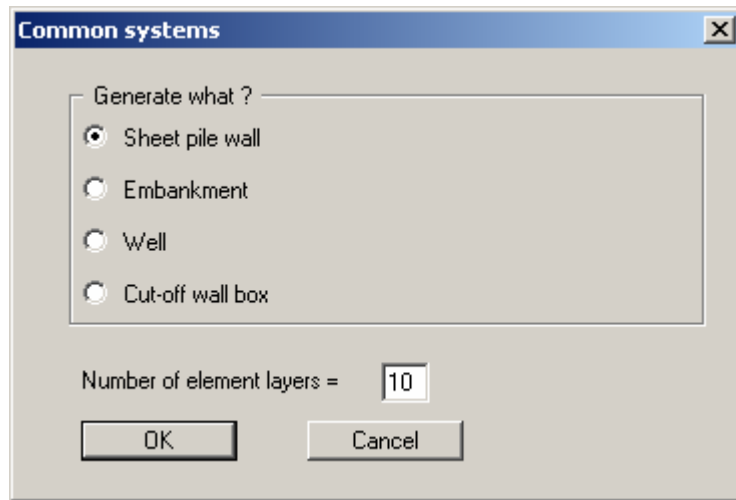
dy-values	
No. of dy-values	
0.000	dy [m]
1.000	1.000
2.000	1.000
3.000	1.000
4.000	1.000
5.000	1.000

The array numbers can be defined using the "No. of dx values" and "No. of dy values" buttons. Enter the array spacing in "dx" and "dy". If you press the "Recalculate x and y values" button the first column to the left of "dx" and "dy" are recalculated. These are absolute values for the array. The array origin is entered in "x0" and "y0".

If the "Generate new mesh" check box is activated, all existing nodes are deleted and then the user-defined FEM nodes in the dialog box are generated with the appropriate FEM mesh. If you want to add further nodes, deactivate the "Generate new mesh" check box.

8.2.9 "Common systems" menu item

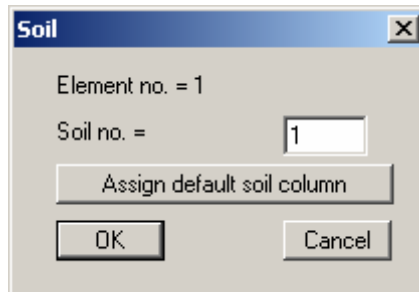
You can generate a mesh for a variety of common systems using this menu item. The following systems are available:



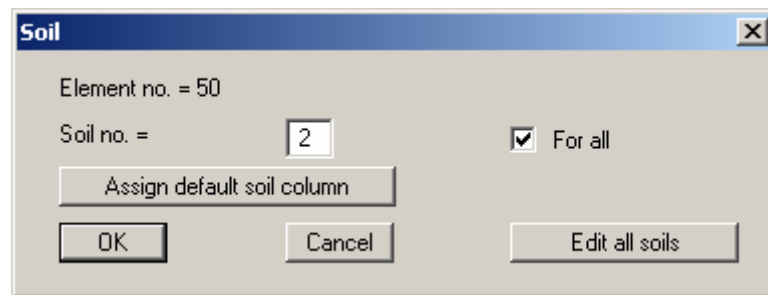
A dialog box opens when you press the required button. Enter the dimensions and the necessary boundary conditions data for the selected system. Once the data is entered the system is displayed on the screen. Of course, you may also use these generated systems as the basis for further refinements.

8.2.10 "Manual mesh" menu item

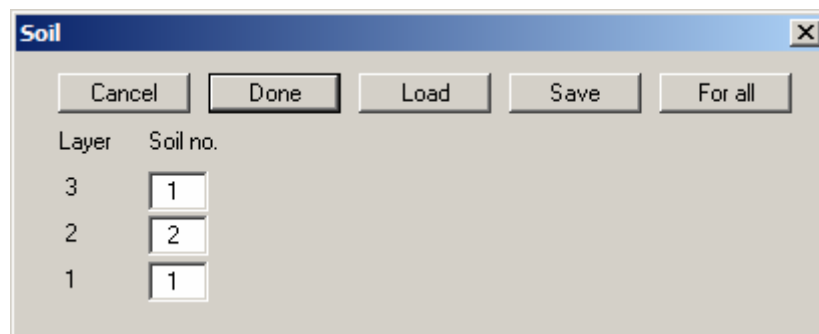
After input of the mesh nodes this menu item is used to define the FEM mesh. Alternatively this menu item can also be reached using [F4]. Three nodes must be clicked using the left mouse button. Once the three nodes have been selected a box appears for defining the soil no. of the FEM element. The soil number entered is assigned to all layers in this FEM element. Alternatively, the default soil column (see Section 8.3.2) can be assigned.



If FEM elements are already defined, you will see a prompt after clicking 3 nodes. By clicking the "Yes" button an existing FEM element can be deleted. After clicking "Edit" you will see the following dialog box:



Here, you can assign the active layer a different soil number. If the "For all" check box is activated this soil number will be used for all layers of this FEM element. You can assign the respective layers different soil numbers using the "Edit all soils" button.



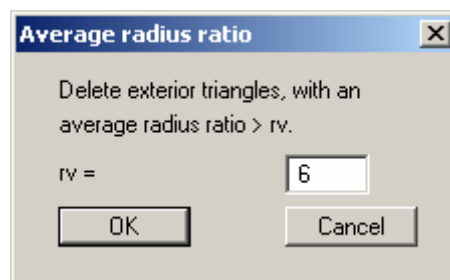
This distribution (soil column) can be saved in a file with the ".bos" extension and loaded for use with a different FEM element. If you click the "For all" button in this dialog box the defined soil column can be applied to all existing elements after confirming in a prompt.

8.2.11 "Automatic" menu item

After entering the mesh nodes automatic mesh generation can be carried out using this menu item (Delauney triangulation). If an FEM mesh already exists it can be either deleted or supplemented. Under certain circumstances, *air holes* (incompletely filled FEM mesh) may occur in an existing FEM mesh being supplemented if this mesh was not generated by means of Delauney triangulation. These regions will require manual post-processing or re-triangulation of the complete FEM mesh. All newly generated triangles will be assigned the default soil column.

8.2.12 "Round off" menu item

During Delauney triangulation a triangular mesh is generated that envelops all nodes. This can lead to acute-angled triangle elements in the boundary regions. These triangles can be removed from the FEM mesh using this menu item.



The radius ratio describes the relationship between external radius and internal radius of a triangle. For an equilateral triangle, this ratio equals 2.0 (optimum). Before you arrive at the dialog box above you will be shown the maximum and minimum radius ratios. In the example above, all external triangles with a radius ratio greater than 6 will be removed. To avoid chaos, only exterior triangles will be removed.

8.2.13 "Delete" menu item

Using this menu item you can delete selected system triangles. You must first click four points in anti-clockwise direction. All triangles with their centroid within this quadrilateral will be deleted.

8.2.14 "Optimise" menu item

You first select in a dialog box whether the diagonals or the topology should be optimised.

"Diagonals" button

Optimisation of diagonals is implemented in order to create a numerically favourable FEM mesh, i.e. where possible, equilateral triangles. The effect of the optimisation of diagonals can best be seen using an example:

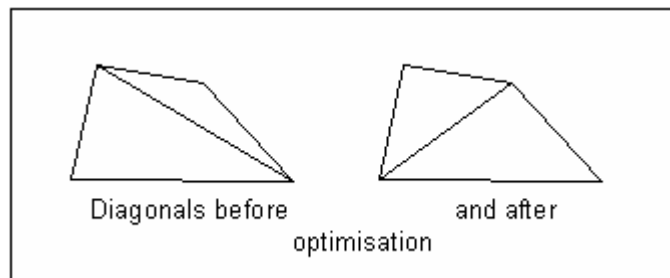


Figure 5 Optimisation of diagonals

If an existing *unfavourable* diagonal cuts two different soil areas, no optimisation takes place, because this would alter the system.

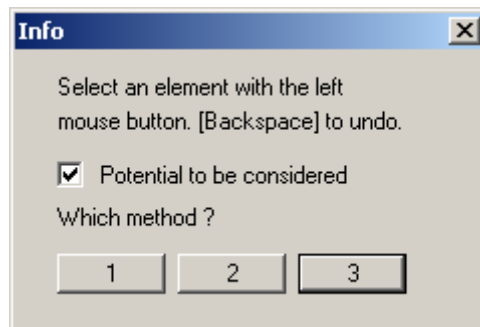
"Topology" button

This program routine displaces the triangular element nodes so that, where possible, equilateral triangles are created. Equilateral triangles have especially favourable numerical properties. Because the displacement of system boundaries and element boundaries with neighbouring elements consisting of different soils does not make much sense, these element boundaries are fixed from the outset. Nodes with defined water level boundary conditions also remain unaltered. Optimisation of the FEM mesh can be followed on the screen by setting the "**With graphics**" check box. The optimisation routine can be aborted at any time by pressing the right mouse button.

In principle, it is always a good idea to optimise the element topology before commencing modelling. However, this leads to a problem; after individual nodes are displaced, the z-values cannot be adapted to the new position. The program therefore displays a warning message beforehand. It is possible to assign the precise values via an interpolation mesh following the optimisation (see "**I mesh**" menu, Section 8.4.1 pp.).

8.2.15 "Refine individually" menu item

Mesh elements can be selected for refinement using the following menu item.



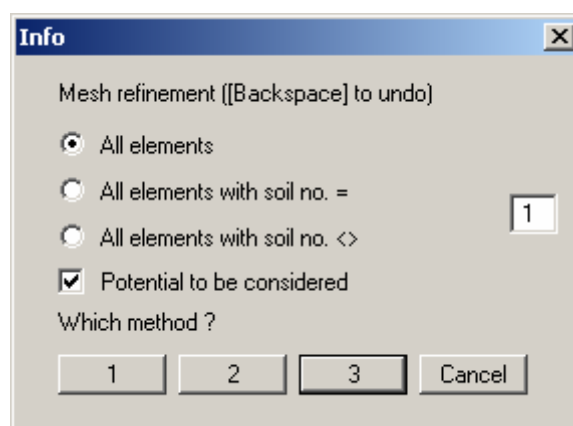
Upon activating the "**Potential to be considered**" check box, new nodes located immediately between two nodes with potential boundary conditions will be assigned the average of the two values. This procedure is not unequivocal when applied to *source boundary conditions* and can lead to misunderstandings. Source boundary conditions are therefore not refined in the course of mesh refinement. During mesh refinement the z ordinates of new nodes are arrived at by linear interpolation from the neighbouring nodes. This ensures that the topology of the original mesh is retained on all planes. A description of the 3 refinement methods can be found in Section 8.2.18.

8.2.16 "Section" menu item

A number of elements previously enveloped in a polygon can be refined using this menu item. Potential boundary conditions can be taken into consideration (see "**Mesh(x/y)/Refine individually**" menu item, Section 8.2.15). A description of the 3 refinement methods can be found in Section 8.2.18.

8.2.17 "All" menu item

The following dialog box appears after selecting this menu item or, alternatively, pressing [F7]:



Either all elements or only element with certain material numbers can be refined. Here, too, potential boundary conditions can be taken into consideration (see "**Mesh(x/y)/Refine individually**" menu item, Section 8.2.15). A description of the 3 refinement methods can be found in Section 8.2.18.

8.2.18 Refinement methods

Three different refinement methods can be applied for element refinement. Refinement will be demonstrated on the following mesh using element 23 as an example.

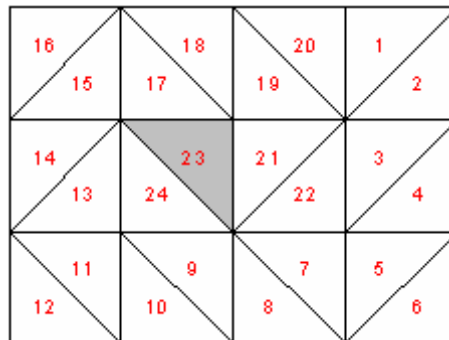


Figure 6 FEM refinement demonstration mesh

- **"Method 1"**

An additional node is generated in the centroid of the selected triangle.

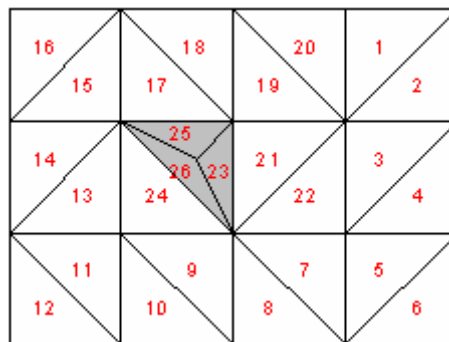


Figure 7 FEM mesh refinement using Method 1

- **"Method 2"**

The selected triangle element and the neighbouring triangle element are halved.

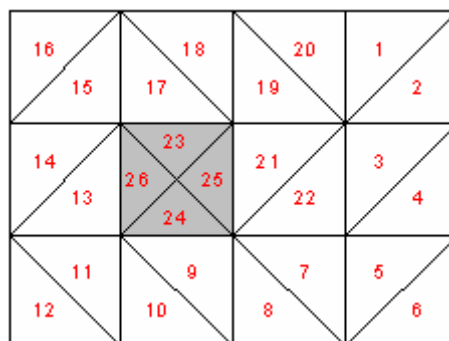


Figure 8 FEM mesh refinement using Method 2

- **"Method 3"**

A new triangle element is inserted at the median of the clicked triangle element. The neighbouring triangle elements are halved.

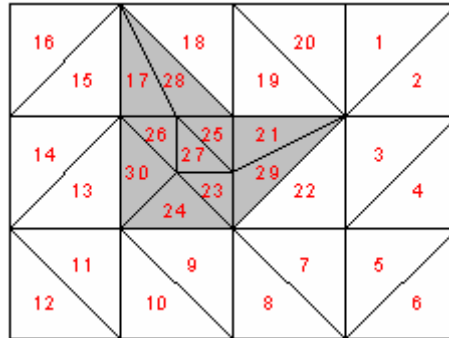


Figure 9 FEM mesh refinement using Method 3

8.3 z menu

8.3.1 "Default depths" menu item

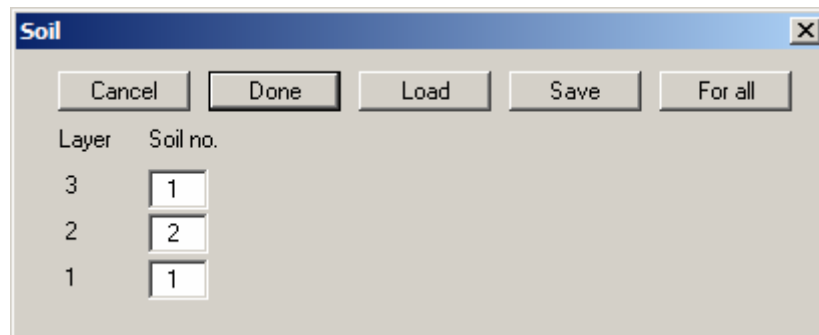
The default depths defined in this dialog box are assigned to all new FEM nodes defined using the "Mesh(x/y)/Define nodes" menu item.

Plane	z [m]
4	3.000
3	2.000
2	1.000
1	0.000

If you edit the given z values and leave the dialog box using **"Done"** the newly defined depths will be adopted as of this moment for newly defined nodes. If nodes are already defined, they can also be retroactively assigned the new depths using the **"For all"** button.

8.3.2 "Default soil column" menu item

The default soil column defined in this dialog box is assigned to all new FEM elements defined using the "**Mesh(x/y)/Automatic**" menu item.



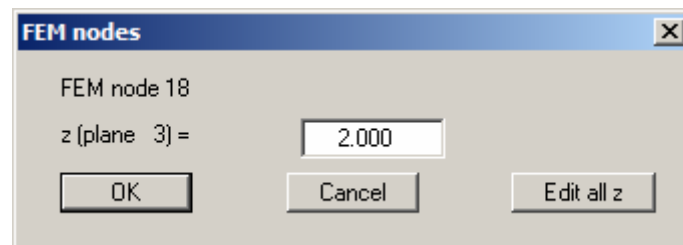
The "Soil" dialog box features a title bar with a close button. Below the title bar are five buttons: "Cancel", "Done", "Load", "Save", and "For all". The main area contains a table with two columns: "Layer" and "Soil no.". The table has three rows with the following values:

Layer	Soil no.
3	1
2	2
1	1

If you edit the given soil numbers and leave the dialog box using "**Done**" the newly defined soil column will be adopted as of this moment for newly defined elements. If elements are already defined, they can also be retroactively assigned this soil column using the "**For all**" button.

8.3.3 "Individual depths" menu item

You can edit the depths of a node by double-clicking. After selecting a node the depth of the current depth of the active plane is shown in the dialog box.

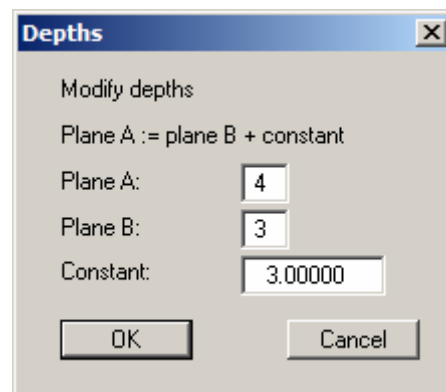


The "FEM nodes" dialog box has a title bar with a close button. The main area displays "FEM node 18" and "z (plane 3) =". To the right of the text is a text input field containing the value "2.000". At the bottom are three buttons: "OK", "Cancel", and "Edit all z".

By entering a different value for z directly you change only the depth of the selected node in the active plane. After clicking "**Edit all z**" the depth of the nodes in all planes at the selected x/y position can be altered.

8.3.4 "Modify depths" menu item

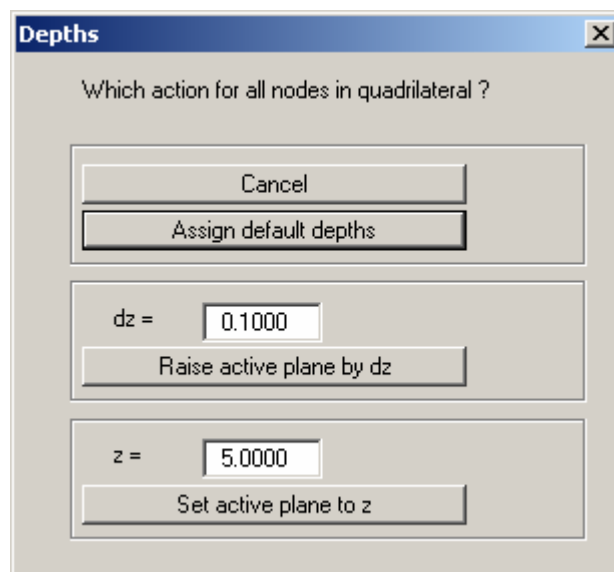
All depths in a given plane can be altered as a function of the existing depths in a different plane.



In the example shown in the dialog box above, all nodes of plane 4 (= top of layer 3) were moved 3 length units above the existing level of plane 3 (= top of layer 2). This menu item can be used to generate parallel layers, especially for complex layer surfaces.

8.3.5 "Depths in section" menu item

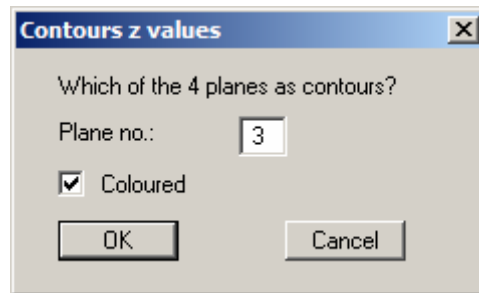
After defining a quadrilateral, all nodes within it can be assigned new depths. Select the action required from the following dialog box:



- **"Assign default depths"**
The default depths are assigned. This alters the depths at the selected node for all planes.
- **"Raise active plane by dz"**
The depths of the selected nodes are raised by the value dz given for the active plane.
- **"Set active plane to z"**
The depths of the selected nodes are set to the depth z given for the active plane.

8.3.6 "Contours (planes)" menu item

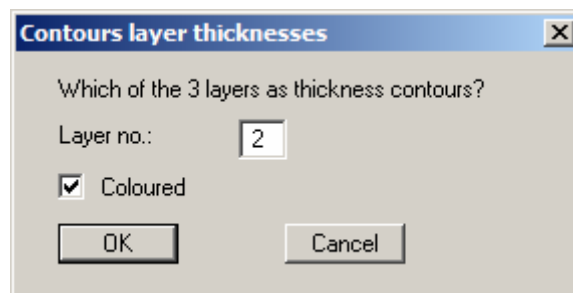
In order to clearly present the layer situation it is possible to generate a contour diagram of the depths (z values) in the selected plane.



In the dialog box you select the desired plane and type of contours. If the "**Coloured**" check box is activated, colour filled contours are created. Details are described in Sections 8.9.2 ("**Evaluation/Normal contours**" menu item) and 8.9.3 ("**Evaluation/Coloured**" menu item).

8.3.7 "Contours (layer thickness)" menu item

You can create a contour diagram of the layer thicknesses.



In the dialog box you select the desired layer and the type of contours. If the "**Coloured**" check box is activated, colour filled contours are created. Details are described in Sections 8.9.2 ("**Evaluation/Normal contours**" menu item) and 8.9.3 ("**Evaluation/Coloured**" menu item).

8.3.8 "Define section" menu item

Using this menu item you can generate vertical sections through the system by clicking the appropriate nodes. Intermediate points are automatically generated by the program. The last point defined is deleted from the list by clicking the right mouse button. You can also define a **circular section**, ending back at the start point. If the circular section is defined anti-clockwise, it protrudes similar to a tower. If the circular section is defined clockwise, you will see a **depression**; you can "see inside" this "hole". When the section is how you want it press [Return].

You will then see the FEM mesh for your defined section on the screen. In addition, a site plan is automatically displayed on the screen, showing the system outline and the defined section. This site plan can be switched off using the menu item "**Graphics preferences/Section legend**" by deactivating the "**Show section line**" check box (see Section 8.7.12).

You can use one of the numerous options for graphical visualisation of the defined section from the "**Evaluation**" menu, for example as potential contour lines or discharges. If you use the evaluation options from the "**3D**" menu, a cut-away, three-dimensional view of the system is generated.

In order to move from the section view to the normal system view for evaluation, go to the menu item "**Mesh(x/y)/Mesh**" (see Section 8.2.2).

If you have previously saved a section it can be loaded again using "**Load section**". The "**Prev. section**" button is only visible if a section has already been defined and loads the section defined immediately prior to this action.

8.3.9 "Save section" menu item

The current section can be saved in a file with the ".fes" extension. It can be reloaded later for evaluations using the "**z/Define section**" menu item (see Section 8.3.8).

8.3.10 "Mesh (section)" menu item

When you select this menu item the FEM mesh in the defined section is displayed.

8.3.11 "Outline (section)" menu item

When you select this menu item the system outline in the defined section is displayed. Soil boundaries are also represented.

8.3.12 "Refine" menu item

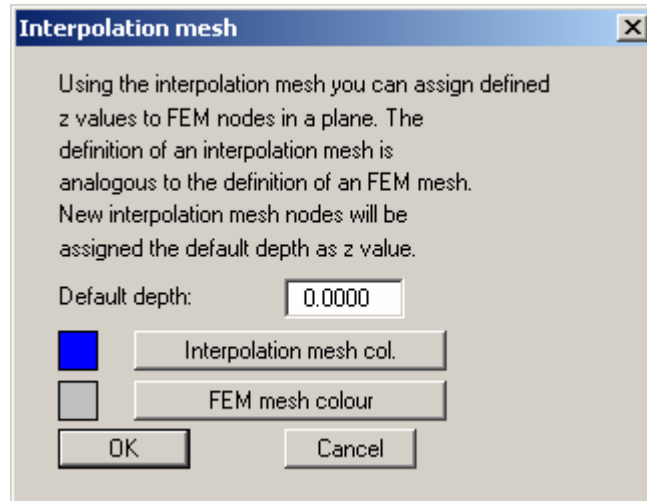
You can refine the FEM mesh within a layer for a defined section by clicking in the layer with the left mouse button. The program generates a further node layer in the centre of the selected layer, including the corresponding element; you have pasted a new element layer. The last refinement can be undone using [**Backspace**].

8.3.13 "Unrefine" menu item

You can "*unrefine*" the FEM mesh in a defined section, i.e. you can remove an element layer. If you click in a layer with the left mouse button, the node plane below it is deleted, together with the corresponding elements, i.e. the clicked layer is amalgamated with the layer below. The previous step can be undone using [**Backspace**].

8.4.1 "Interpolation mesh" menu item

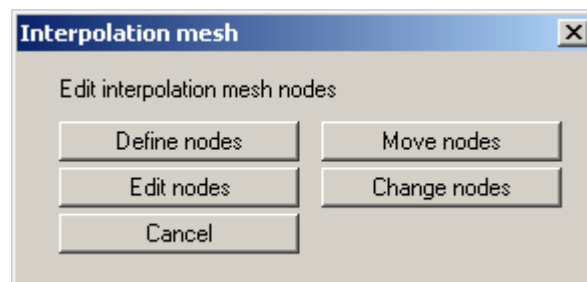
In principle, the z values can be completely defined using the previous menu items. A further defining simplification is offered by the interpolation mesh. Similarly to the FEM mesh, this interpolation mesh consists of nodes and triangles. A certain z value can be defined at the nodes. After defining an interpolation mesh you can assign the z values to the user-defined plane of the FEM mesh. These are assigned by means of linear interpolation. The interpolation mesh should (but not must) completely blanket the FEM mesh. In the simplest case possible, this may consist of one triangle.



Different colours can be defined for the interpolation mesh and the FEM mesh to improve the clarity of the screen visualisation.

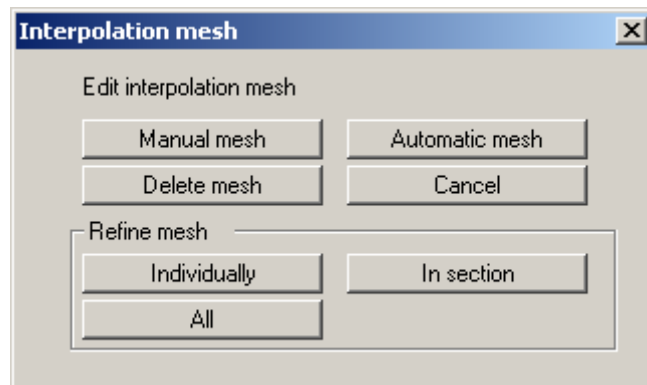
8.4.2 "Nodes" menu item

In complete analogy to the FEM mesh you can define, move, edit or change the interpolation mesh nodes.



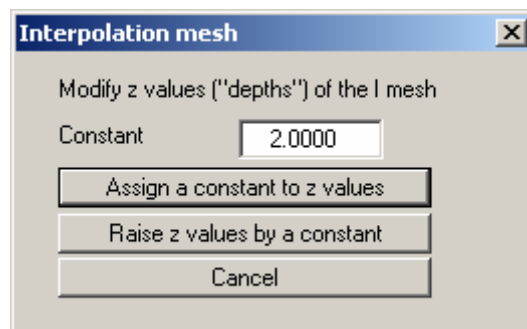
8.4.3 "Mesh" menu item

In complete analogy to the FEM mesh you can edit the interpolation mesh.



8.4.4 "Modify" menu item

The z values of given interpolation mesh nodes can be modified. After clockwise definition of a quadrilateral the following dialog box appears:



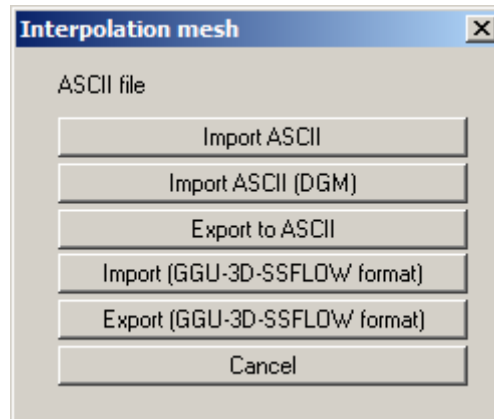
Once a constant has been entered the two options shown in the dialog box are available for modifying the z-values defined by the quadrilateral.

8.4.5 "Contours" menu item

You can create a contour diagram of the interpolation mesh z values. Either coloured contours or a 3D representation can be selected. You can find more details on contour line visualisation in the "Evaluation" menu in Sections 8.9.3 and 8.9.4.

8.4.6 "Import/export" menu item

The x- and y-coordinates of the interpolation mesh can be imported from an ASCII file or exported for other applications. A dialog box as described in Section 8.1.5 opens for the "**Import ASCII**" option.



If the coordinates are exported using the **GGU-3D-SSFLOW** format, the information regarding triangles (incidence table) and z values will also be exported and can be imported again.

8.4.7 "Assign" menu item

When you are completely finished with the interpolation mesh you can decide here to which plane the z values of the interpolation mesh will be assigned as depths.

Because no undo is possible after assigning the z values, it is expedient to save the file beforehand.

8.5 Boundary menu

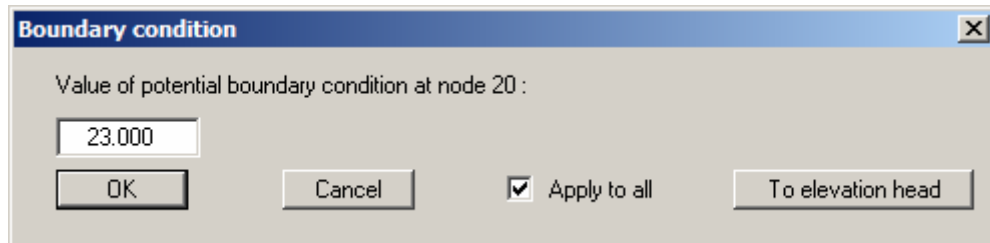
8.5.1 "Preferences" menu item

In the dialog box for this menu item preferences for the presentation of the boundary conditions on the screen can be specified.

8.5.2 "Individual potentials" menu item

With the assistance of this menu item a potential boundary condition can be defined at a node using the left mouse button, or deleted with the right.

After selecting this menu item a descriptive dialog box opens, followed by the screen presentation of the system mesh with the previously defined potential boundary conditions of the current plane. By clicking with the *right* mouse button you can delete existing potentials. By clicking with the *left* mouse button you can define new potentials or edit existing potential boundary conditions.



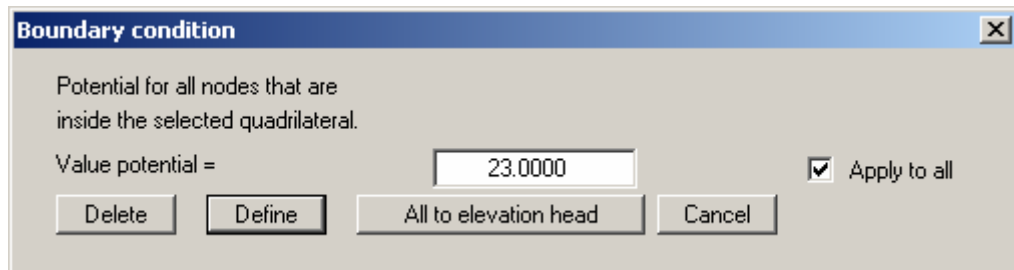
In the dialog box you define the value of boundary condition. By pressing "**To elevation head**", the potential is defined as being at the elevation head entered for the system. If the "**Apply to all**" check box is activated the boundary condition at this point is adopted for all planes. The current element layer and plane are shown in the *Layer/plane legend* in the top right corner of the screen. To select a different plane click the arrow buttons in the *3D toolbar* (see Section 8.7.8).

Note:

The piezometric head must be given when handling problems involving groundwater hydraulics. The piezometric head h results from adding the pore water pressure u ($= \gamma_w \cdot h$) and the elevation head y . The piezometric head is also known simply as the water level. Further common designations include potential or hydraulic head.

8.5.3 "(Potentials) In section" menu item

After selecting this menu item several FEM nodes can be simultaneously assigned a potential boundary condition in a defined section, or an existing boundary condition be deleted. For this click four points in an anti-clockwise direction using the left mouse button. The right mouse button will undo. The definition refers to all nodes within the quadrilateral.



You can now define the potentials or delete all marked potentials. The definition applies to the nodes located within the section. In addition, by pressing "**All to elevation head**", it is possible to define the potentials in the enclosing polygon as being at the elevation head entered for the system. If the "**Apply to all**" check box is activated the boundary condition at the marked points are adopted for all planes.

8.5.4 "(Potentials) Linear" menu item

After going to this menu item you click the nodes of the FEM mesh to which a potential boundary condition is to be assigned. Press [**Return**] to then enter a start and an end potential and, if necessary, activate the "**Apply to all**" check box in order to apply the boundary conditions at the marked points for all planes

The selected node is assigned a potential which increases linearly from the start potential to the end potential, proportional to the separation of the nodes to each other. This menu item is helpful if you need to consider a water course, for example, the water level of which changes linearly within the system.

8.5.5 "Point sources" menu item

Using this menu items source boundary conditions are defined within the FEM system in analogy to the potential boundary conditions (see Section 8.5.2). A source is positive if it brings water into the system.

Note on impermeable boundaries:

The case of an impermeable boundary is automatically considered by the finite-element-method. Valid is, that all system boundaries or partial boundaries which have no water level or source boundary conditions, are automatically impermeable.

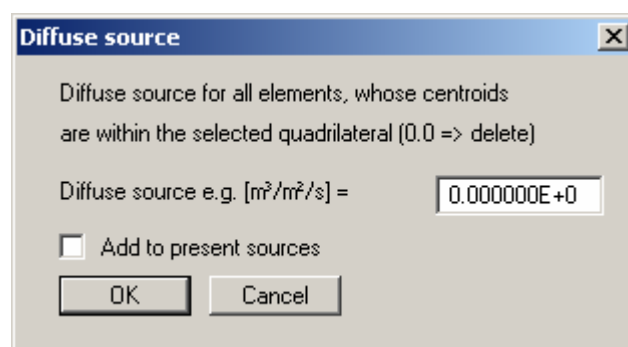
8.5.6 "Linear sources" menu item

After going to this menu item you define a polygon by clicking 4 points anti-clockwise. The element nodes with their centroid within the thus marked quadrilateral are assigned a linear source with the user-defined value. If an element lies entirely within the quadrilateral, the corresponding nodes are not assigned a linear source.

If you need to delete existing linear sources, define the polygon again and enter the value **0**.

8.5.7 "Diffuse sources" menu item

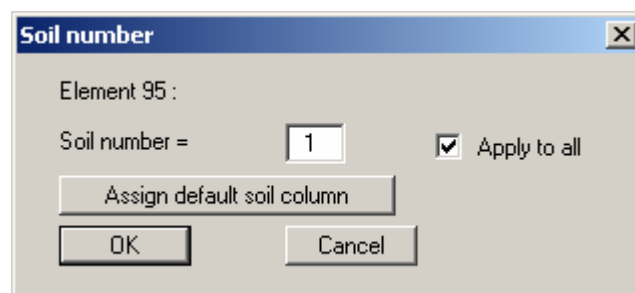
You can assign diffuse sources to certain elements, e.g. for taking precipitation events in a given area into consideration. After defining the area by clicking 4 points anti-clockwise to form a polygon, you will see the following dialog box:



If you activate the **"Add to present sources"** check box, the value entered will be added to any diffuse sources already present. If negative entries are made the diffuse source is reduced correspondingly. Existing diffuse sources can be deleted by entering a value **0**.

8.5.8 "Individual soils" menu item

After selecting the type of FEM system screen representation the soil number assignments can be edited for individual elements of the FEM mesh. This assigns these elements the soil properties that were defined in the **"System/Soil properties"** menu item (see Section 8.6.4).



If the **"Apply to all"** check box is activated, all triangular prisms will be assigned this soil number. Otherwise, this only applies to the triangular prism in the current layer. If you click the **"Assign default soil column"** button, all selected elements will be assigned the default soil column (see Section 8.3.2).

8.5.9 "(Soils) In section" menu item

After selecting the type of FEM system screen representation you define a polygon by clicking 4 points in counter-clockwise direction. All elements with their centroid within this polygon can be assigned a different soil number. The associated soil properties are defined in the "**System/Soil properties**" menu item (see Section 8.6.4).

8.6 System menu

8.6.1 "Info" menu item

You will see information on the current system with the number of elements, nodes and boundary conditions in a dialog box.

8.6.2 "Project identification" menu item

You can enter a description of the current system; this will then be used in the general legend (see Section 8.7.10).

8.6.3 "Test" menu item

The system is tested for the minimum node spacing and overlapping of triangle elements. The test can be aborted at any time by pressing the right mouse button.

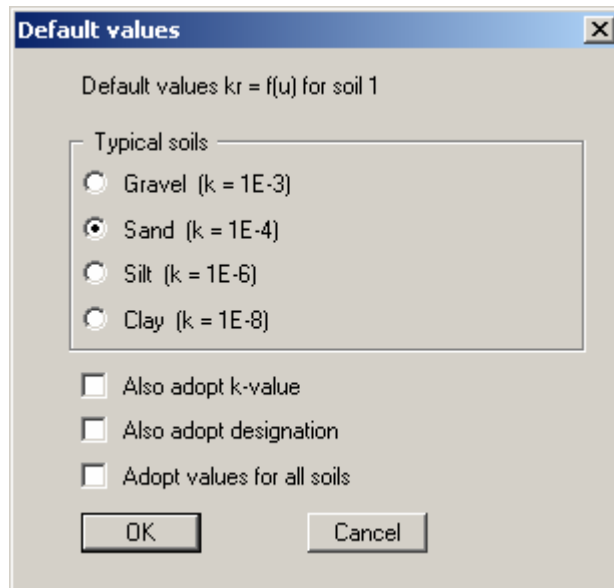
8.6.4 "Soil properties" menu item

You can edit the soil properties in a dialog box. The number of input rows corresponds to the soil numbers assigned either whilst defining the mesh or the soil boundary conditions.

No.	kx	ky	kz	neff	Designation
1	1.000E-4	1.000E-4	1.000E-4	0.20	
2	1.000E-7	1.000E-7	1.000E-7	0.05	

The Darcy permeabilities in x -, y - and z -direction (" **k_x** ", " **k_y** " and " **k_z** ") and the effective pore space (" **$neff$** ") must be entered. The effective pore space is not incorporated in the basic differential equation. These values are only required for the subsequent evaluation of the data computed for flow line generation. A designation entry is not required.

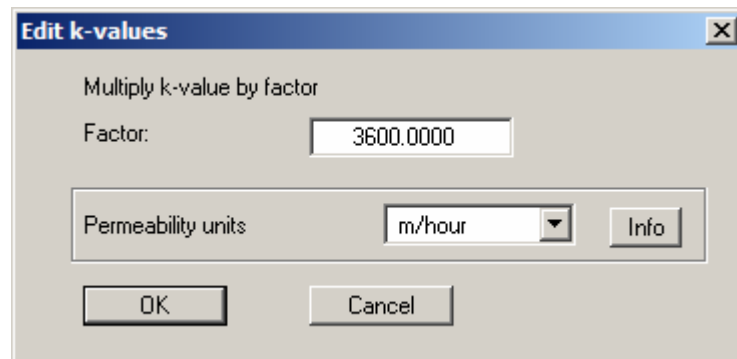
The variable kr records the change in permeability in the unsaturated zone above the water table and can assume values between 0.0 and 1.0. In saturated system zones kr equals 1.0. The kr function curve must be entered as a polygon course with increasing pore water pressures u for all soils. A polygon course need only be specified for negative pore water pressures, since positive pore water pressures represent saturated conditions. Press the " **$kr = f(u)$** " button in order to edit the values in a dialog box. Alternatively, the default values for the various soil types (gravel, sand, silt, clay) can be automatically adopted by the program by pressing the "**Std**" button. The following dialog box opens:



By activating the check boxes at the bottom of the dialog box the permeability k and the designation can be additionally adopted for the selected row (= soil layer). The selected values are adopted for all soil layers after an additional confirmation if the "**Adopt values for all soils**" check box is activated.

If you would like to arrange the soils entered in a different sequence, you can do this by pressing the "**Swap soils**" button. You can choose to also adopt the colours.

It often makes more sense in a transient analysis to enter the permeability in "**m/hour**" or "**m/day**". Using the "**Edit k-values**" button it is possible to convert the given k -value to a different unit using the appropriate factor (e.g. factor 3600 to convert from m/s to m/h).



To avoid unnecessary explanations when submitting your analyses, the k -value is shown in the soil properties legend in "**m/second**". To achieve this, the program internally converts the entered coefficient of permeability from your selected unit back to m/s (see "**Info**" button).

8.6.5 "Model" menu item

After pointing to the "System/Model" or, alternatively, pressing [F5], the equation system bandwidth is examined and a message box displayed where necessary. If you do not agree with the bandwidth optimisation the analysis is cancelled.

NOTE ON BANDWIDTH:

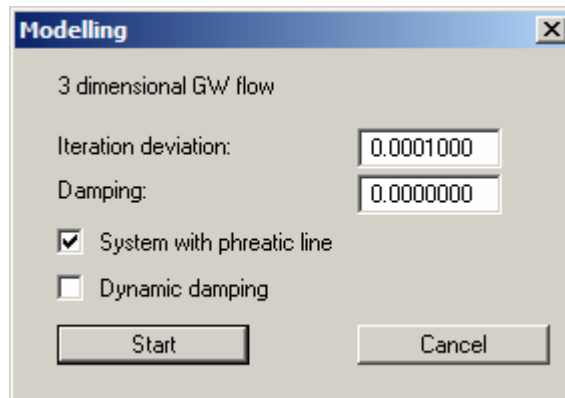
Finite-element methods give rise to equation systems which, depending on the system, only have values different to "0" in the region of the principle diagonals. The iterative equation solver contained in the FEM applications uses this in order to save memory and thus shorten computation time. The maximum distance between the principle diagonal and the system's *non-zero element most distant* from the principle diagonal is known as the bandwidth. In principle, iterative equation solvers do not require bandwidth optimisation. However, the results are much more stable and the computation time shorter if the bandwidth is small.

NOTE ON UNITS:

The program works true to your units. If you want to work with common length units (metres), it follows that all length input must then be made in metres. Beside the permeability and discharge, time is employed as an additional unit. Here too, only one unit of time T , e.g. seconds [s], may be used. *The results are provided in accordance with the input units used.*

In transient analyses it is often useful to define permeability not in m/s, but in m/hour or m/day, for example. If you have entered appropriately modified permeability values the program converts these permeability values back to m/s for display in the legend only. This saves unnecessary explanations when submitting analyses.

The following dialog box is displayed subsequent to bandwidth optimisation:



The following check boxes and information can be provided in the dialog boxes:

- **Iteration deviation**

If unsaturated zones are present within the system and calculation of the phreatic line is required an iteration process will be necessary. The required iteration deviation for computation of the phreatic line is defined here. The program aborts the iteration if the deviation of the potentials at all nodes is below this limit value for two consecutive iterations. It avoids a negative iteration values in horizontal-plane systems.

- **Damping**
Oscillation around the actual solution can occur during computation of the phreatic line in complicated groundwater systems. This oscillation is damped by the damping factor. Values between 0.0 and 0.95 make the most sense. A value of 1.0 (complete damping) makes no sense and will therefore not be accepted by the program. Oscillations occur primarily when the curve of the $kr(u)$ function is very steep (e.g. as for sand).
- **System with phreatic line**
Specify whether a phreatic line should be computed for the system.
- **"Dynamic damping"**
In extreme groundwater systems it may not be possible to reach a solution using normal damping. An analysis using dynamic damping may then be carried out. After every five iteration steps the damping value is increased by 0.05 up to a maximum of 0.99.

You will be informed on progress during the computation. After computation is complete you will see a message box showing the computation time. You can then move on to the **"Evaluation"** menu.

8.6.6 "Undo" menu item

If you have carried out any changes to a dialog box input field or moved objects to a different position on the screen after selecting the **"Graphics preferences/Move objects"** menu item or using the [F11] function key, this menu item will allow you to undo the movements. This function can also be reached by using the key combination [Alt] + [Back] or the appropriate tool in the toolbar (see Section 8.7.7).

8.6.7 "Restore" menu item

When this menu item is selected the last change made in a dialog box input field or the last change in the position of objects, which you undid using the menu item **"System/Undo"** will be restored. This function can also be reached by using the key combination [Ctrl] + [Back] or the appropriate tool in the toolbar (see Section 8.7.7).

8.6.8 "Preferences" menu item

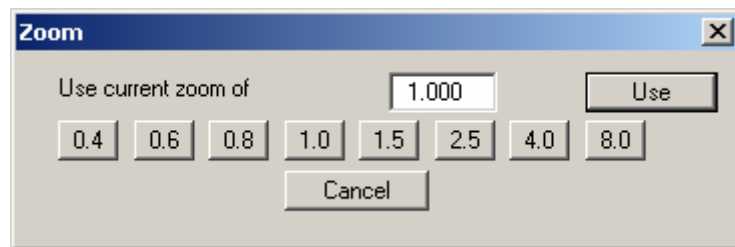
You can activate or deactivate the undo functions.

8.7 Graphics preferences menu

8.7.1 "Refresh and zoom" menu item

The program works on the principle of *What you see is what you get*. This means that the screen presentation represents, overall, what you will see on your printer. In the last consequence, this would mean that the screen presentation would have to be refreshed after every alteration you make. For reasons of efficiency and as this can take several seconds for complex screen contents, the screen is not refreshed after every alteration.

If, e.g., after using the zoom function (see below), only part of the image is visible, you can achieve a complete view using this menu item.



A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "Use" to exit the box the current factor is accepted. By clicking on the "0.4", "0.6", etc. buttons, the selected factor is used directly and the dialog box closed.

It is much simpler, however, to get a complete overview using [Esc]. Pressing [Esc] allows a complete screen presentation using the zoom factor specified in this menu item. The [F2] key allows screen refreshing without altering the coordinates and zoom factor.

8.7.2 "Zoom info" menu item

By clicking two diametrically opposed points you can enlarge a section of the screen in order to view details better. An information box provides information on activating the zoom function and on available options.

8.7.3 "Pen colour and width" menu item

In order to enhance the clarity of the graphics you can edit the pen settings for various graphic elements (e.g. potential, sources, etc.). You can edit the pen widths for the elements shown in the dialog box; by clicking on the button with the element designation you can also edit the pen or fill colours.


On *monochrome printers* (e.g. laser printers), colours are shown in a corresponding grey scale. Graphic elements employing very light colours may be difficult to see. In such cases it makes sense to edit the colour preferences.

8.7.4 "Legend font selection" menu item

With this menu item you can switch to a different true-type font. All available true-type fonts are displayed in the dialog box.

8.7.5 "Draw Mini-CAD objects first" menu item

Originally, the display sequence could be modified using this menu item. In the program's default settings **Mini-CAD** objects are above system graphics; they may therefore cover it.

In the integrated **Mini-CAD** module you can work on a total 20 different layers. To move **Mini-CAD** objects behind the main program system graphics click the "**Layers**" icon  in the **Mini-CAD** popup menu and deactivate the required layers in the "**Visualisation**" button dialog box. This draws the **Mini-CAD** objects in these layers first; they are therefore behind your system section, for example.

8.7.6 "Mini-CAD toolbar" and "Header toolbar" menu items

Using these two menu items you can add free text, lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.) to the main program graphics. PDF files can also be imported as images. The same pop-up menu opens for both menu items, the icons and functions used are described in more detail in the **Mini-CAD** manual provided. The differences between the Mini-CAD and Header CAD are as follows:

- Objects created with **Mini-CAD** are based on the coordinate system (generally in metres), in which the drawing is produced, and are shown accordingly. You should use the "**Mini-CAD toolbar**" when you wish to add information to the system (for example, labelling of slope inclinations or the location of any foundations).
- Objects created with the **Header CAD** are based on the page format (in mm). This makes you independent of the coordinate system and keeps you in the same position on the page. You should select the "**Header toolbar**" if you wish to place general information on the drawing (company logo, report numbers, plan numbers, stamp etc.). Once you have saved the header information to disk (see **Mini-CAD** user manual), you can load it into completely different systems (with different system coordinates). The saved header information will appear in exactly the same position on the page, which greatly simplifies the creation of general page information.

8.7.7 "Toolbar preferences" menu item

After starting the program a horizontal toolbar for menu items appears below the program menu bar. If you would rather work with a popup window with several columns, you can specify your preferences using this menu item. The smart icons can also be switched off.

At the bottom of the program window you find a status bar with further information. You can also activate or switch off the status bar here. The preferences will be saved in the "**GGU-3D-SSFLOW.alg**" file (see menu item "**Graphics preferences/Save graphics preferences**") and will be active at the next time the program is started.

Most of the program functions can be reached directly by clicking on the menu item tools (smart icons). If you hover with the mouse pointer over the tools the meaning of the smart icons appears in a text box. Some of the tool functions can be activated from the normal menu items.



"Next page"/"Previous page"

Using this icon, you can navigate between the individual pages in the *tabular representation*.



"Select page"

If you are in the *tabular representation*, you can use this icon to jump to a specific page or to return to the *normal representation*, that is, to the graphics.



"Zoom out"

If you have previously *zoomed in*, this tool returns to a full screen display.



"Zoom (-)"/"Zoom (+)"

With the zoom functions you can zoom in or out of parts of the image, by clicking the left mouse button.



"Colour on/off"

If you need to remove the colour from the system presentation, to create a black and white print-out, for example, use this on/off switch.



"Copy/print area"

Use this tool to copy only parts of the graphics in order to paste them, e.g. to a report. You will see information on this function and can then mark an area, which is copied to the clipboard or can be saved in a file. Alternatively you can send the marked area directly to your printer (see "**Tips and tricks**", Section 9.3).



"Undo"

By clicking this icon the last change made in a dialog box input field or the last performed movement of graphical objects made using the [F11] function key or the menu item "**Graphics preferences/Move objects**" can be undone.




"Restore"

By clicking this symbol, the last change made in a dialog box input field or in the position of objects carried out using "**Undo**" can be restored.


8.7.8 "3D toolbar" menu item

This menu item allows you to choose a pop-up window with tools for advanced editing of the three-dimensional graphics. If you hover with the mouse pointer over the tools, the meaning of the Smart icons appears in a text box.

The "**Rotation by ...**" tool allows you to rotate the graphics around one of the three axes. The plus and minus signs designate the direction of rotation. The angle of rotation can be adjusted as

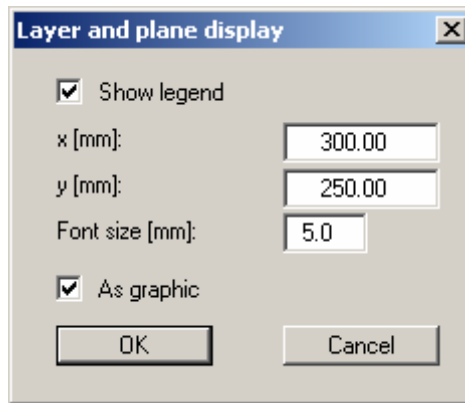
wished using the  tool.



The  tools allow you to move from one plane to the next. In addition, a number of menu items from the "**3D**" menu and the section definitions have been included in this toolbar.

8.7.9 "Layer/plane legend" menu item

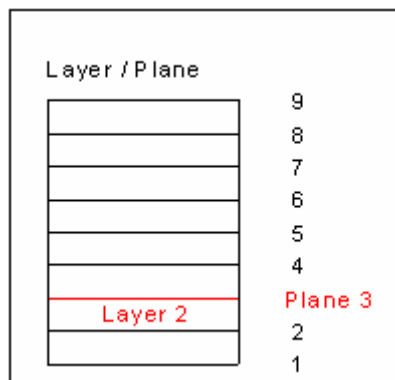
During input and graphical system evaluation you always find yourself in certain planes and layers. The numbers of the plane and layer are entered in a legend, if you have activated the "**Show legend**" check box in the dialog box for this menu item.



You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Font size**".

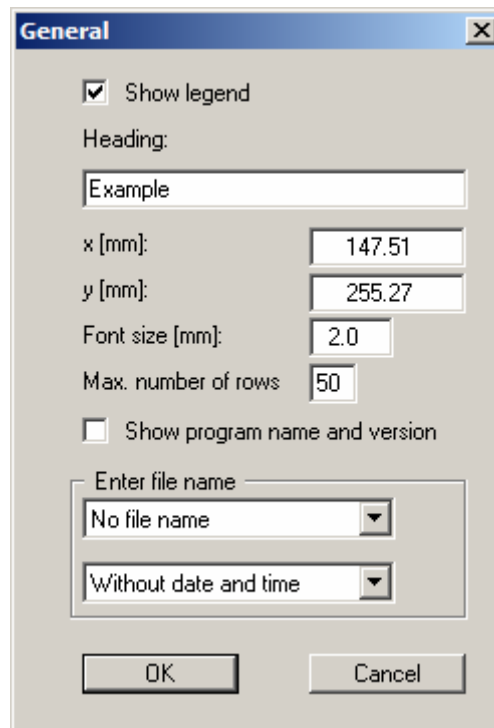
The fastest way to modify the position of the legend is to press the [F11] function key and then to pull the legend to the new position while holding the left mouse button.

If the "**As graphic**" check box is activated, the information will be displayed as a small graphical schematic as shown here.



8.7.10 "General legend" menu item

A legend with general properties will be displayed on your output sheet if you have activated the "**Show legend**" check box. Using this menu item you can alter the type of presentation.



In addition to the heading, this legend contains information on the current graphics and preferences. For example, if you display the FEM mesh you will see information on the number of triangles and nodes.

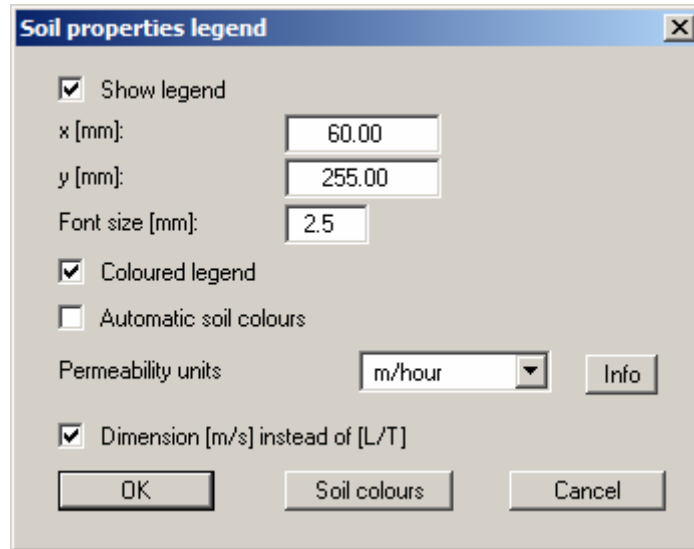
You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used.

The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position with the left mouse button pressed.

In the general legend you can, if wished, display information on the program (name and version) and on the current file. Any project identification entered (see Section 8.6.2) will also be shown in the general legend.

8.7.11 "Soil properties legend" menu item

A legend with the soil properties will be displayed on your output sheet if you have clicked the **"Show legend"** check box. Using this menu item you can alter the type of presentation or turn off the legend completely.



You can define and edit the position of the legend using the values **"x"** and **"y"**. The size of the legend is controlled by the values for **"Font size"**.

The fastest way to modify the position of the legend is to press the **[F11]** function key and then to pull the legend to the new position while holding the left mouse button.

- **"Coloured legend "**
The soil colours are displayed in the legend. Otherwise, they will be numbered.
- **"Automatic soil colours"**
The soils are assigned soils colours automatically by the program. If the check box is not selected, the soil colours individually defined using the **"Soil colours"** button will be adopted.
- **"Soil colours"**
You will see a dialog box, in which you can define your preferences. After clicking the button with the desired number you can assign each soil layer a new number or reorganise using the **"Soil colours/Reorganise"** command button. You can save your colour preferences to a file with **"Soil colours/Save"** and use them for different systems by means of the **"Soil colours/Load"** command button. In the lower group box you can also transfer the colour preferences to the Windows colour management dialog box, or vice versa, as user-defined colour preferences for example. You can read a further description by pressing the **"Info"** button.
- **"Permeability units"**
It often makes more sense in a transient analysis to enter the permeability in **"m/hour"** or **"m/day"**. To avoid unnecessary explanations when submitting your analyses, the k-value is shown in the soil properties legend in **"m/second"**. To achieve this, the program internally converts the entered coefficient of permeability from your selected unit back to m/s (see **"Info"** button).

- **"Dimension [m/s] instead of [L/T]"**

The program uses [m/s] as the default unit. As explained above, you can also convert to [m/h] and [m/d]. If different units are required (e.g. yards and minutes), the general format length [L] and time [T] can be adopted. Just deactivate this check box.

8.7.12 "Section legend" menu item

If you activate the **"Show legend"** check box in the dialog box for this menu item, a legend containing a sketch showing the location of the section is displayed on the screen. The position of the legend can be defined and edited using the values **"x"** and **"y"**, **"Width"** and **"Height"**. The font size can be specified for the heading and labelling of the points.

8.7.13 "kr = f(u) function legend" menu item

The value k_r is a dimensionless coefficient, representing the change in permeability in the unsaturated zone. It is specified in the **"System/Soil properties"** menu item for various soils on the basis of modelling points and used to calculate the phreatic line, among other things.

If you activate the **"Show legend"** check box in this menu item, the function is displayed as a graphic in a legend on the screen. The presentation can be modified to suit your needs in the dialog box.

The position of the legend can be defined or edited by means of the variables **"x"** and **"y"**, **"Width"** and **"Height"**. If you edit the font sizes you should also always increase the width and height correspondingly, otherwise the actual function graphic will appear smaller within the defined legend frame.

If you activate the **"Automatic axes"** check box, the program will use a presentation scale including all given values for u . If the check box is deactivated, visualisation of the x-axis can be restricted to a user-defined **"Max. u"** value.

If the **"Use soil colours"** check box is activated, the individual graphs are colour-differentiated. However, if the graph values are the same for all soils, the function graph is automatically drawn in black and a corresponding note added to the graphics.

8.7.14 **"Move objects" menu item**

Select this menu item in order to position legends or other graphical elements at the desired position on the output sheet. You can also move objects by pressing [**F11**] and then positioning the legend box with the left mouse button pressed. In that case an info-box appears no more.

8.7.15 **"Save graphics preferences" menu item**

Some of the preferences you made with the menu items of the **"Graphics preferences"** menu can be saved to a file. If you select **"GGU-3D-SSFLOW.alg"** as file name, and save the file on the same level as the program, the data will be automatically loaded the next time the program is started and need not be entered again.

If you do not go to **"File/New"** upon starting the program, but open a previously saved file instead, the preferences used at the time of saving are shown. If subsequent changes in the general preferences are to be used for existing files, these preferences must be imported using the menu item **"Graphics preferences/Load graphics preferences"**.

8.7.16 **"Load graphics preferences" menu item**

You can reload a graphics preferences file into the program, which was saved using the **"Graphics preferences/Save graphics preferences"** menu item. Only the corresponding data will be refreshed.

8.8 Page size + margins menu

8.8.1 "Auto-resize" menu item

This menu item provides a to-scale visualisation, in both x and y coordinates, of the system and result graphics. If you have previously altered the image coordinates graphically or via editor, you can quickly achieve a complete view using this menu item. This function can also be accessed using the [F9] function key.

8.8.2 "Manual resize (mouse)" menu item

You can use the coordinates of a section of the visualisation as the new image coordinates by marking the desired area with the mouse, pressing the left mouse button and holding the [Ctrl] and [Shift] keys. The scales of the x- and y-axes are adjusted accordingly. If the previous proportions (scale x-direction/scale y-direction) need to be retained, the "**Proportional section**" check box must be activated.

Alternatively, you can simply redefine *origin* of the visualisation. The previous scale preferences are not affected by this.

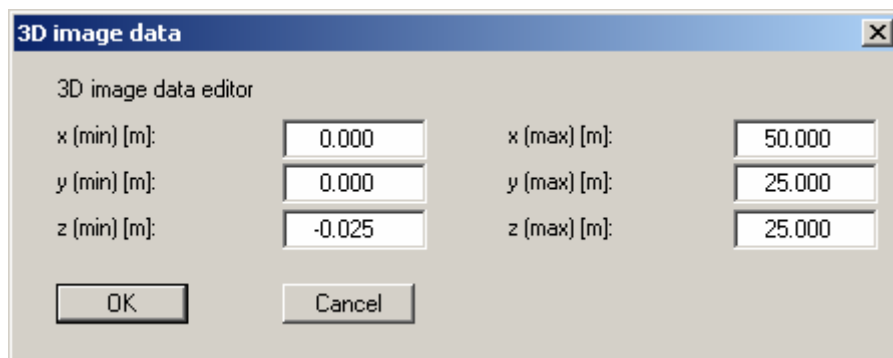
8.8.3 "Manual resize (editor)" menu item

You can alter the image coordinates by direct numerical input in a dialog box. This allows precise scale input. The coordinates refer to the *drawing area*. This can be defined in the "**Page size + margins/Page size and margins**" menu item by means of the plot margins (see Section 8.8.4).

The image coordinates entered here can be saved in a file with the extension ".bxy" and be re-loaded later for the same file or for different files.

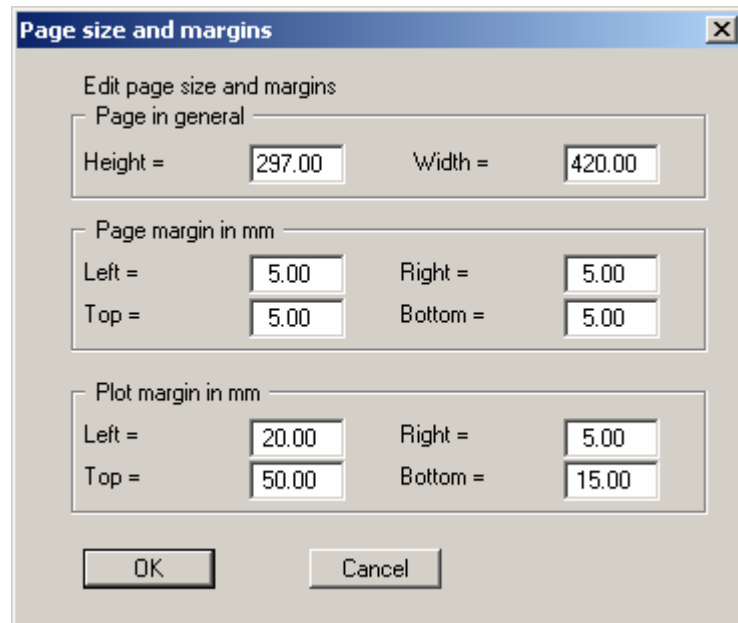
If you want to recover the previous values during input or use the menu item again after editing the coordinates, you can do this by pressing the "**Old values**" button.

In sections, you can edit the scale and boundary for the z axis; in 3-dimensional representations, you enter the lower and upper limit for the axes to be represented according to your wishes:



8.8.4 "Page size and margins" menu item

The default page set-up is A3 when the program is started. You can edit the page format in the following dialog box.



The dialog box titled "Page size and margins" contains the following settings:

Edit page size and margins	
Page in general	
Height =	297.00
Width =	420.00

Page margin in mm	
Left =	5.00
Right =	5.00
Top =	5.00
Bottom =	5.00

Plot margin in mm	
Left =	20.00
Right =	5.00
Top =	50.00
Bottom =	15.00

Buttons: OK, Cancel

- **"Page in general"** defines the size of the output sheet. The A3 format is set as default. The program automatically draws thin cutting borders around the page, which are required when using a plotter on paper rolls. The borders can be switched off using the **"With borders"** check box in the **"Page size + margins/Margins and borders"** menu item (see Section 8.8.6).
- **"Page margin"** defines the position of a frame as a distance to the margins. This frame encloses the subsequent diagram. You can switch off the frame deactivating the **"With margins"** check box in the **"Page size + margins/Margins and borders"** menu item (see Section 8.8.6).
- The **"Plot margin"** define a set distance between the page margin and the actual *drawing area* in which the graphical evaluation of your input is presented.

8.8.5 "Font size selection" menu item

You can edit font sizes for labelling the various graphical elements.

The font sizes of text within legends are edited in the respective legend editor. Just double-click in a legend to do this.

8.8.6 "Margins and borders" menu item

The program automatically draws thin cutting edges around the page, which are required when using a plotter on paper rolls. Page margins (see menu item "**Page size + margins/Page size and margins**") defines the position of a frame as a distance to the cutting border. This frame encloses the subsequent diagram. You can switch off the lines by deactivating the "**With margins**" and "**With borders**" check boxes.

8.8.7 "Designation of potential" menu item

The term "**Potential**" in the graphics can be replaced by "**Water level**" using this menu item.

8.9 Evaluation menu

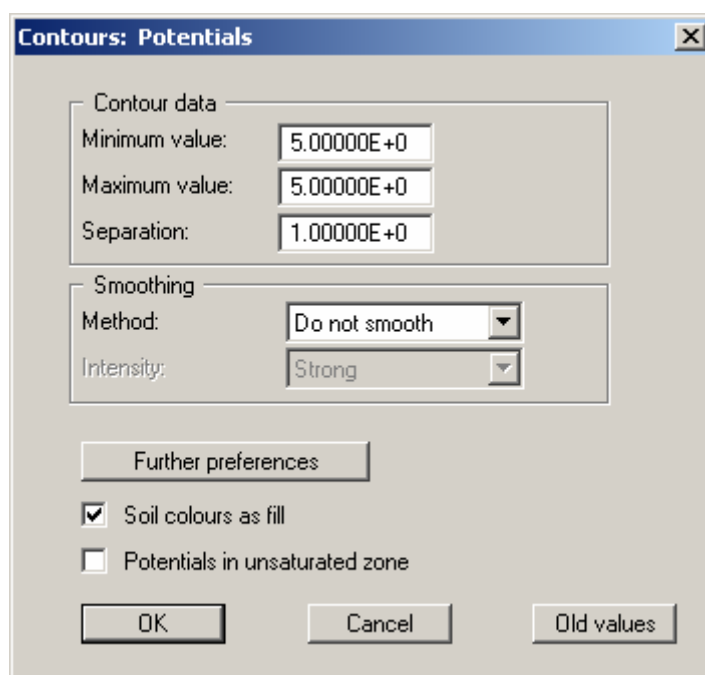
8.9.1 General notes on the "Evaluation" menu

The "Evaluation" menu, among others, is provided for graphical evaluation of the FEM model. The results are displayed for the currently active area (selected plane or activated section). If the area being considered here is a plane, you can move back and forwards between the planes by clicking the arrow buttons on the **3D toolbar** (see Section 8.7.8). The respective current layer and plane are shown in the **status bar** at the bottom of the window (see Section 8.7.7) or in the **Layer/plane legend**, if activated (see Section 8.7.9).

If you are currently within a system section view, the potential contour lines, discharges or velocities are shown in the section. If you are within the system section view and want to view in the node plane, first go to the "Mesh(x/y)/Mesh" menu item (see Section 8.2.2).

8.9.2 "Normal contours" menu item

Using this menu item it is possible to visualise the calculated potentials, velocities, or gradients as normal contour lines. A dialog box opens for selecting the required data. A further dialog box then opens, in which the type of visualisation can be modified (an example is shown below with "Potentials" selected).



The program shows you the existing smallest and largest values and the spacing of the contour lines. If you want the visualisation to begin at a different value, the initial value can be entered here. You can also vary the spacing, for example, in order to reduce the number of contour lines drawn.

When you select this menu item the settings displayed here are always those automatically selected by the program. Using the "Old values" button, the preferences used for the previous contour line diagram are adopted. This information is saved with the record.

You can also select from three smoothing out procedures:

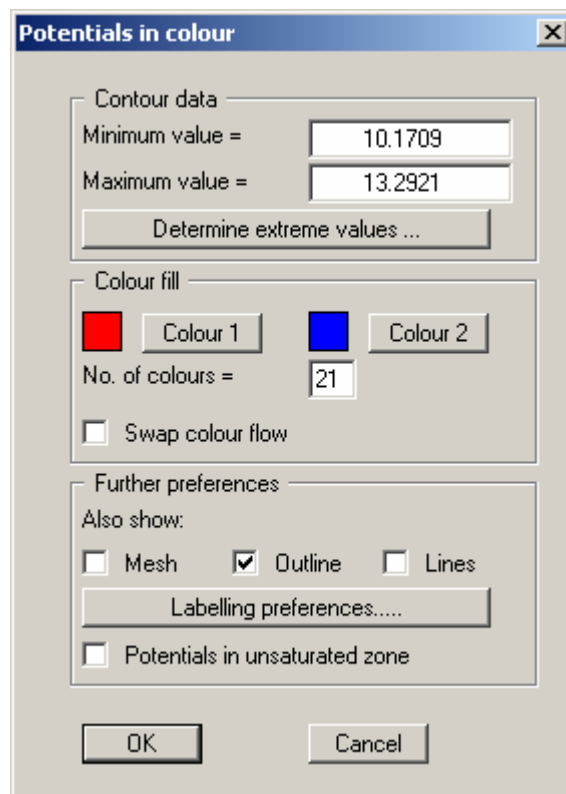
- **"Do not smooth"**
As a linear method is implemented within the triangle elements, this will provide analysis results without smoothing performed by the program.
- **"Method 1"** and **"Method 2"**
these smoothing out procedures employ two different Bezier splines. Smoothing out intensities can be given for both in order to acquire smooth contour lines. **Method 2** creates very **"round"** contours, with the danger that the true results may be falsified.

You can specify the type of visualisation for the contour lines by means of the **"Further preferences"** button. By activating the appropriate check box the soil colour can be used as fill.

As the program calculates with unsaturated permeabilities, potentials in the unsaturated zone are the result, which is physically absolutely correct. From experience, this confuses some clients. To save long discussions, switch off the potential lines in the unsaturated zone with the corresponding check box.

8.9.3 "Coloured" menu item

Using this menu item it is possible to visualise the calculated potentials, velocities or gradients as colour-filled contour lines. A dialog box opens for selecting the required data. A further dialog box then opens, in which the type of visualisation can be modified.



The following options are available in the dialog box above (the example shows **"Potentials"** selected):

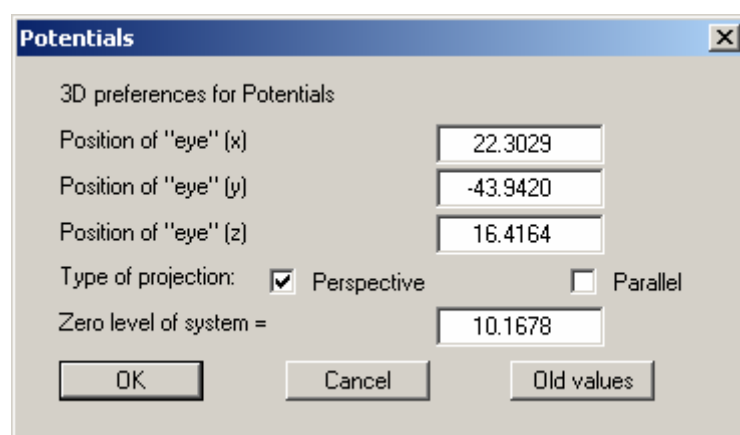
- **"Contour data"** group box
Press the **"Determine extreme values..."** button. The program then determines the minimum and maximum values for the corresponding layer base. You can then edit these values, for example in order achieve a defined start value.
- **"Colour fill"** group box
You can control the colour subdivisions of the contour diagram using **"No. of colours"**. In the example above, 16 colours will be displayed between **"Colour 1"** and **"Colour 2"**. The default setting is a colour course from red to blue. These colours can be edited as required after selecting the **"Colour 1"** and **"Colour 2"** buttons, or simply reverse the choice by selecting the **"Change colour series"** check box.
- **"Further preferences"** group box
In addition to the colour presentation you can also have the triangle mesh and/or the outline displayed. Additional contour lines can also be drawn. Line labelling preferences can be defined by means of the **"Labelling preferences"** button.
- **"OK"**
The colours will be drawn after confirmation.

A colour bar at the right edge of the sheet allows correlation between the colour and the corresponding value. If this colour bar is drawn in the right page margin, specify a larger value for the right plotting margin (e.g. 25 mm) in the **"Page size + margins/Page size and margins"** menu item (see Section 8.8.4).

As the program calculates with unsaturated permeabilities, potentials in the unsaturated zone are the result, which is physically absolutely correct. From experience, this confuses some clients. To save long discussions, switch off the potential lines in the unsaturated zone with the corresponding check box.

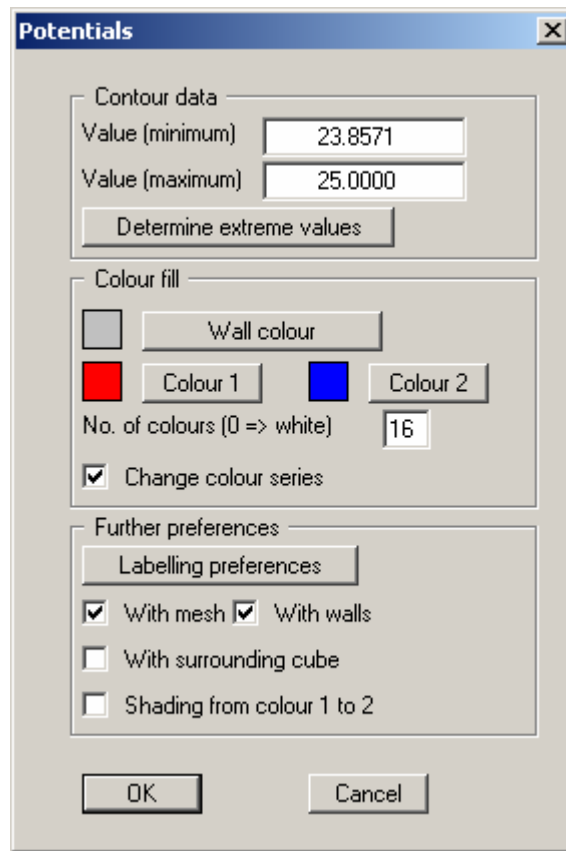
8.9.4 "3D" menu item

Using this menu item it is possible to visualise lines of equal water level (potentials) for the existing nodes in three dimensions. In the following dialog box you can choose between perspective and parallel projection, as well as selecting the position of the *eye*.



The program will generally make sensible suggestions for the input values in the dialog box, so you will not normally need to make alterations.

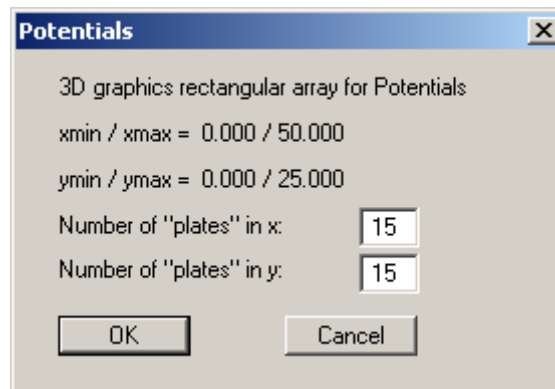
After leaving the dialog box by pressing "OK" you will see the following dialog box. It greatly resembles the box shown in the menu item "Evaluation/Coloured".



You can also specify whether an enclosing cube should be displayed; in some cases this improves the 3D effect. The "**Shade from colour 1 to colour 2**" check box represents a bit of a special effect. If this check box is activated a light source is simulated in the region of the "eye". The angle between the light beam and the respective 3D surface represents a measure of the reflection. The area is shaded in accordance with colours 1 and 2. A good choice of colours, for example, would be dark grey for colour 1 and pale grey for colour 2.

8.9.5 "3D array" menu item

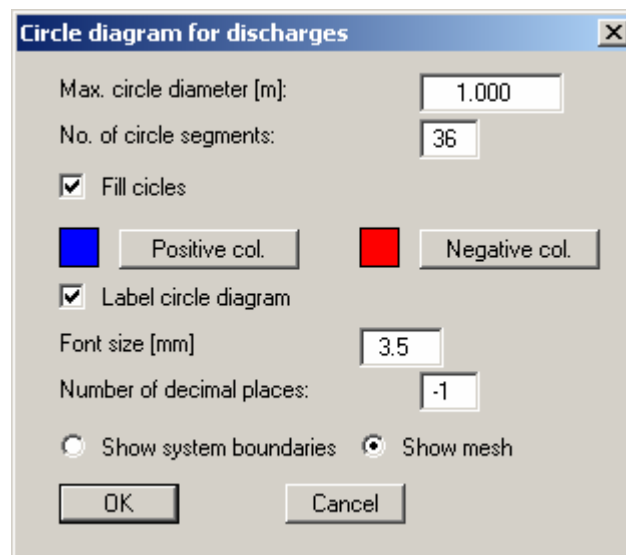
The 3D effect is usually lost in an irregular triangular mesh. The **"Evaluation/3D array"** menu item is useful here. The following dialog box opens where you can define the array.



After leaving the dialog box by pressing **"OK"** you will at first see the same box as that described in **"Evaluation/3D"**. The program calculates the state variables at the array points by linear interpolation of the results of the FEM analysis. You then see the dialog box for coloured contours preferences as described in **"Evaluation/3D"** (see Section 8.9.4).

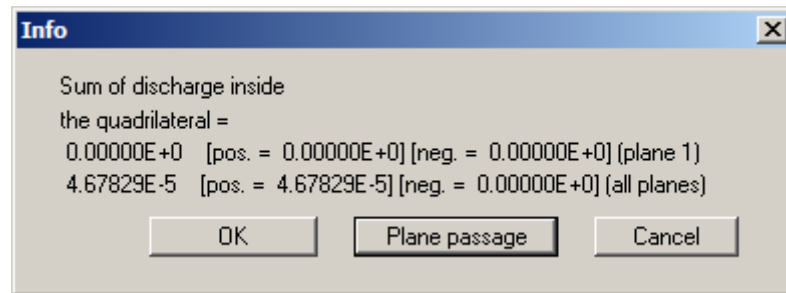
8.9.6 "Discharges" menu item

The FEM mesh will be displayed at the nodes, together with the calculated discharge. Discharges other than zero are present at the nodes at which you have defined a water level boundary condition or a source boundary condition. At all other nodes the discharge equals zero (continuity condition). Using the dialog box you can specify the type of presentation.



After leaving the box using **"OK"** the discharges are displayed as variable sized circles. Positive and negative discharges can be displayed in different colours in filled circles.

By specifying appropriate areas of the screen, the sum of the discharges in these areas can be calculated by the program. By clicking any four points anti-clockwise the discharge in the enclosing polygon is determined for the selected plane and for all other planes and displayed in a message box.



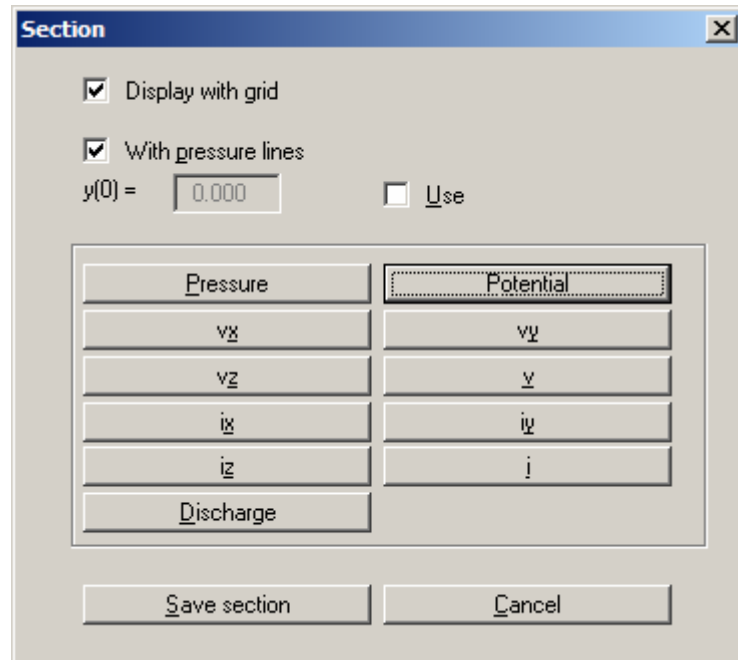
You can then calculate the plane discharge for the selected area. The plane discharge is the discharge flowing through the triangles contained within the quadrilateral of the current plane. It is calculated from the node discharges of the triangular prisms located below the plane. Under certain circumstances, the boundary nodes may include a component belonging to a neighbouring prism not included within the quadrilateral, leading to slight inaccuracy.

8.9.7 "Discharge in vertical section" menu item

After defining a section course, the total discharge in the defined section and the partial discharges from the active to the uppermost plane are computed. The result is presented in a message box. Here, you can opt to save the current section in a ".fes" file.


8.9.8 "Values along line" menu item

You can define a section using the left mouse button by clicking the appropriate nodes. Intermediate points are automatically supplemented by the program. A click with the right mouse button deletes the last point from the list. When the section is located where you want it press **[Return]**. You will see a dialog box where you can define preferences for the type of visualisation.





After clicking the parameter to be visualised the values along the defined section will be displayed graphically. You can save the previously defined section to a file. It can then be opened again if you return to this menu item. If you used this menu item immediately preceding this point, you can also open the last section using the "**Prev. section**" button.



8.9.9 "Values in section" menu item


This menu item is identical with the "**z/Define section**" menu item, but was included here again in order to clearly demonstrate the possibilities for section evaluations. You can also access this menu item using the  tool in the *3D toolbar*.

A section course can be defined in analogy to the "**Evaluation/Values along line**" menu item. After confirming by pressing [**Return**] the defined section is displayed on the screen. Further evaluations (e.g. displaying contours) are performed in this vertical section.

This menu item can be very impressively applied to represent a cut-away, 3-dimensional view of the system, including potential lines, etc. In order to make the system visible with the various soils, select the  tool from the *3D toolbar* ("**3D/External model**" menu item).

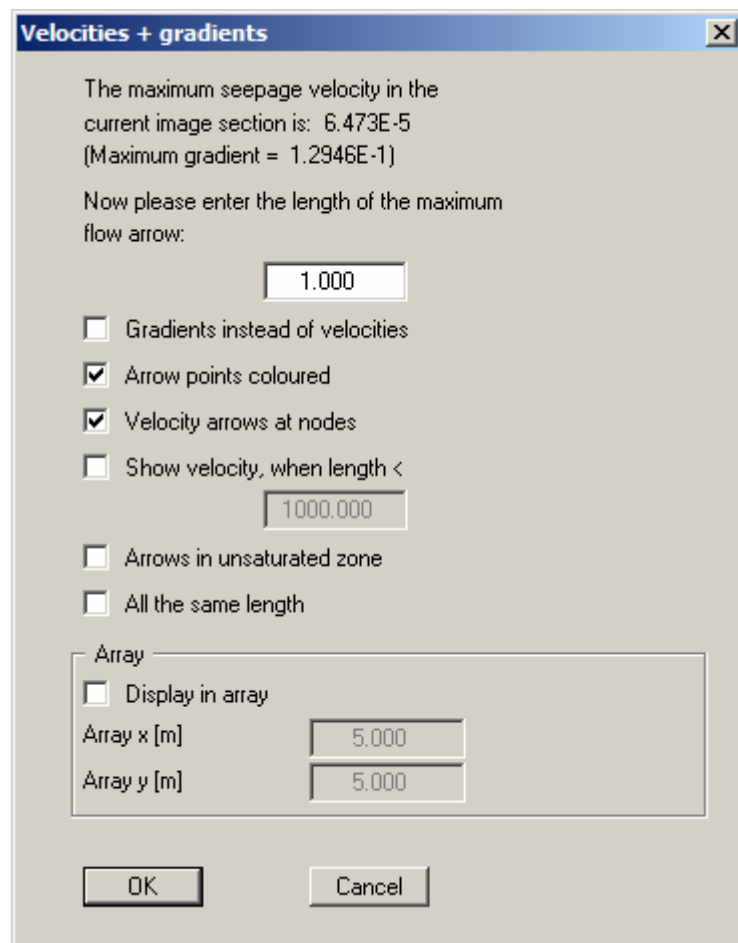
You can also define a *circular section*, ending back at the start point. If the circular section is defined anti-clockwise, it protrudes similar to a tower. If the circular section is defined clockwise, you will see a *depression*; you can "see inside" this "hole". If you change the active plane (e.g. using the  tool), the view also changes. Try it!

If you select the  tool ("**3D/Potential external model**" menu item) instead of , the potentials on the section faces are displayed.

If you are viewing a section and wish to return to the plan visualisation or evaluation, go to the "**Mesh(x/y)/Mesh**" menu item or use the  tool in the *menu toolbar*.

8.9.10 "Velocities + gradients" menu item

The velocity field can be visualised using this menu item. The size and direction of the seepage velocity are shown ($v_s = (k \cdot i)/n_{eff}$).



First, enter the length of the maximum seepage velocity to be displayed. You can then decide whether the velocities or the gradients are to be displayed.

By selecting the "**Velocity arrows at nodes**" check box, you can determine whether or not the velocities are to be displayed at the nodes or at the centroids of the triangular elements. The velocity at the centroid is determined by the program from the FEM calculation. The velocities at the element nodes are determined as a mean of the *centroid velocities* of the neighbouring triangular elements. The length of the maximum velocity vector must be given in the user-defined length unit (generally metres). In the following presentation you will see, at the top left, scale information for length/time units. The colour of the arrowheads can be edited using the "**Graphics preferences/Pen colour and width**" menu item (see Section 8.7.3).

For systems with, e.g., wells, very high velocities occur in the area of the well, which may possibly *ruin* the graphics. Select a length for which velocities should still be presented and activate the corresponding check box.

Additional visualisation preferences can be specified at the bottom of the dialog box. For example, all arrows may be drawn the same length or a small animation be displayed. The arrows can also be visualised in an array, if necessary corrected by the program to suit the geometry of the triangular elements.

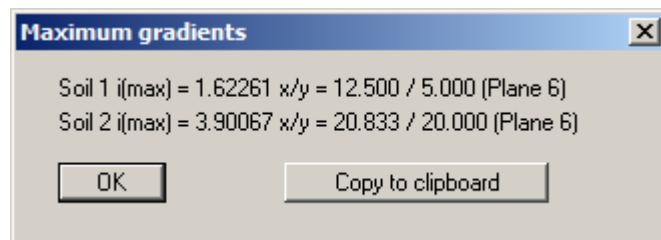
As the program calculates with unsaturated permeabilities, potentials in the unsaturated zone are the result, which is physically absolutely correct. From experience, this confuses some clients. To save long discussions, switch off the potential lines in the unsaturated zone with the corresponding check box.

8.9.11 "Potentials + velocities" menu item

Potentials and flow velocities can also be displayed simultaneously, thus illustrating their interdependency. The dialog boxes for contour line and velocity display preferences are shown in sequence. The dialog box descriptions can be found in Sections 8.9.2 und 8.9.10.

8.9.12 "Maximum gradients" menu item

The maximum gradients are displayed in a message box.



Using the "Copy to Clipboard" button the data displayed in the box can be very simply pasted into MS-WORD, for example, via the Windows Clipboard.

8.9.13 "Individual values" menu item

By left-clicking an FEM node all modelling results can be viewed for the respective node in the active plane. If you need to view the values for the same node in a different plane, first move to the plane required using the arrow buttons in the **3D toolbar** and click the node again. If you have defined a section you can directly click the nodes in the various planes.

8.9.14 "Groundwater-surface distance" menu item

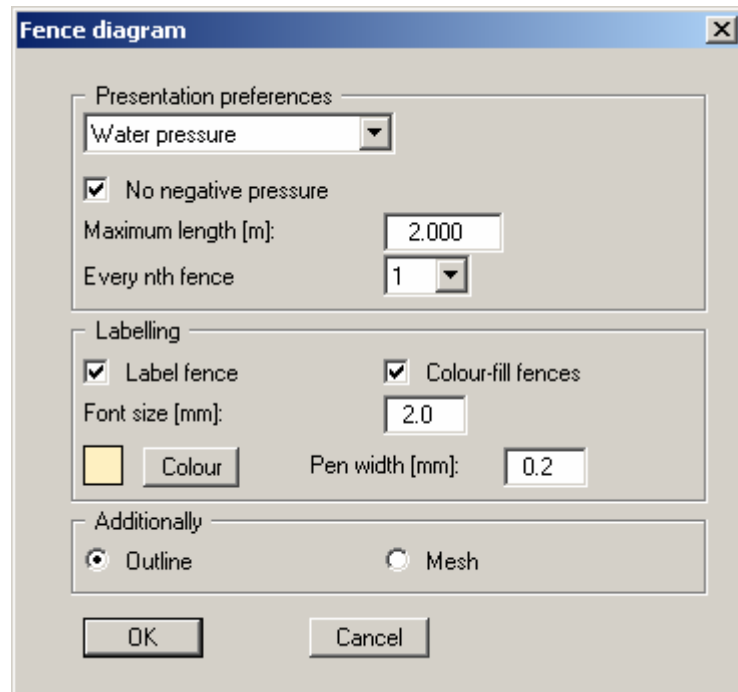
A normal or colour-filled contour diagram showing lines of equal groundwater-surface distance is generated (see menu items "Evaluation/Normal contours", Section 8.9.2 and "Evaluation/ Coloured", Section 8.9.3).

8.9.15 "Groundwater thickness" menu item

A normal or colour-filled contour diagram showing the groundwater thicknesses is generated (see menu items "Evaluation/Normal contours", Section 8.9.2 and "Evaluation/ Coloured", Section 8.9.3).

8.9.16 "Fence diagram" menu item

You can create a fence plot for computed quantities in a section using this menu item. The section is defined using the "**z/Define section**" menu item or the menu item "**Evaluation/Values in section**".



Choose visualisation of water pressure or potential in the dialog box. You can then specify preferences in accordance with the selected quantity.

8.9.17 "Difference" menu item

A contour diagram of equal water level differences can be generated using this menu item. To do this, append to the current data set an FEM data set previously computed and saved with the results. The program then adopts the potential differences between the two computations. More detailed descriptions (compatibility between data sets, positions of soils, boundary conditions) can be taken from the dialog box for this menu item.

The sign can be reversed for subsequent visualisation of the difference lines. You can also specify the type of contour line presentation by activating or deactivating the "**Coloured**" check box in the dialog box. If you exit the dialog box using "**OK**", you first import the second file for difference formation via a file selector. Once the program has completed the difference calculation, you will see dialog boxes for specifying contour line preferences in accordance with your selection (see Sections 8.9.2 and 8.9.3).

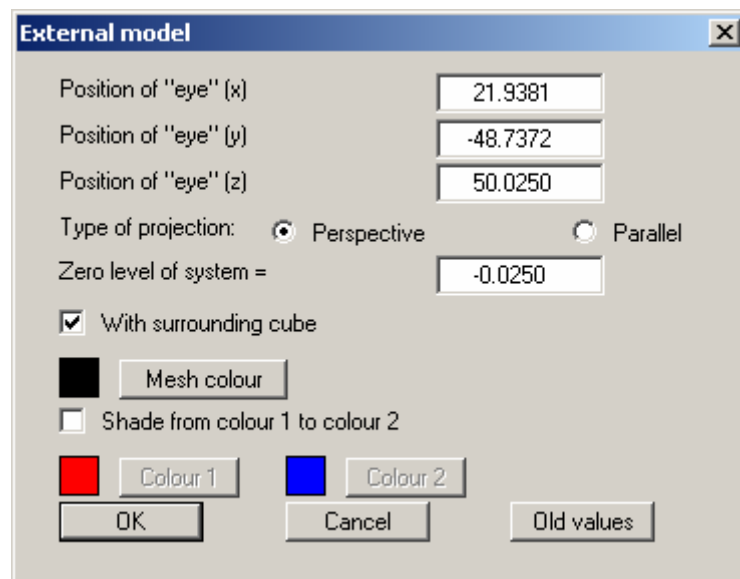
8.10 3D menu

8.10.1 "Wire model" menu item

After specifying the type of projection and visualisation, the system is displayed as a wire model, i.e. all nodes are connected by lines. To improve clarity, visualisation of the plane nodes can be restricted to the top and bottom plane by activating the "**Top and bottom only**" check box.

8.10.2 "External model" menu item

An external view of the model can be displayed. The surfaces are displayed using the soil type colours defined in "**Graphics preferences/Soil properties legend**".



If you activate the "**Shade from colour 1 to colour 2**" check box, visualisation does not use the soil colours but instead uses shading in the two specified colours. If you have previously defined a section, you will see a cut-away 3-dimensional system.

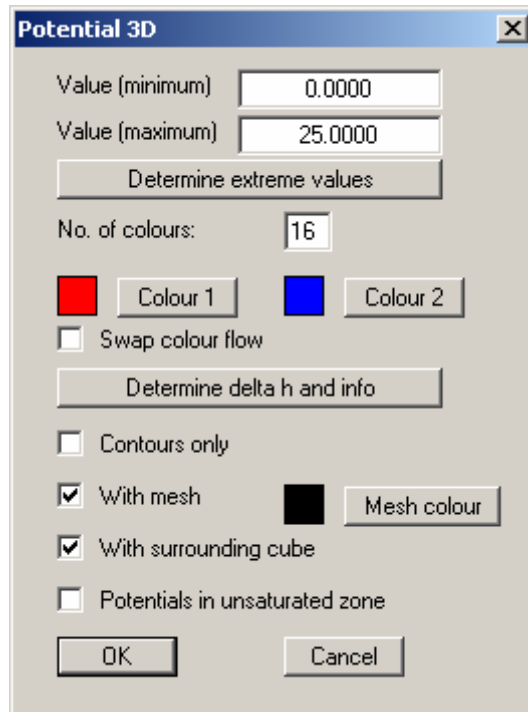
The outside of the system is covered by the mesh to increase the 3-dimensional impression. The colour of the mesh can be altered using the "**Mesh colour**" button.

After confirming your preferences using "**OK**", the 3D model is displayed on the screen. The angle of view of the model can be altered using the rotation tools in the **3D toolbar**. You can view the original visualisation once more by returning to this menu item and clicking the "**Old values**" button. If the model leaves the screen while it is being rotated, press the [F9] function key once and the screen coordinates will be recalculated.

You can move between the planes of your system using the arrow buttons on the **3D toolbar**. However, this only makes sense for cut-away system visualisation (external model in section).

8.10.3 "Potential external model" menu item

You can display an external view of the model that includes the potentials. After defining the angle of view and the type of projection you will see a dialog box for specifying potential visualisation preferences.



After determining the extreme values you define the boundary colours for the continuous colour flow using the "Colour 1" and "Colour 2" buttons. If you have previously defined a section, you will see a cut-away 3-dimensional system, including the potentials on the section surfaces.

If you activate the "Contours only" check box, no colour fill will occur and only contour lines will be drawn. The contour lines are then allocated "Colour 1". The spacing of the contour lines is determined by "Value (minimum)" and "Value (maximum)", as well as the number of colours specified. To save bothersome calculations you can display the spacing beforehand using the "Determine delta h and info" button. If necessary, change the number of colours.

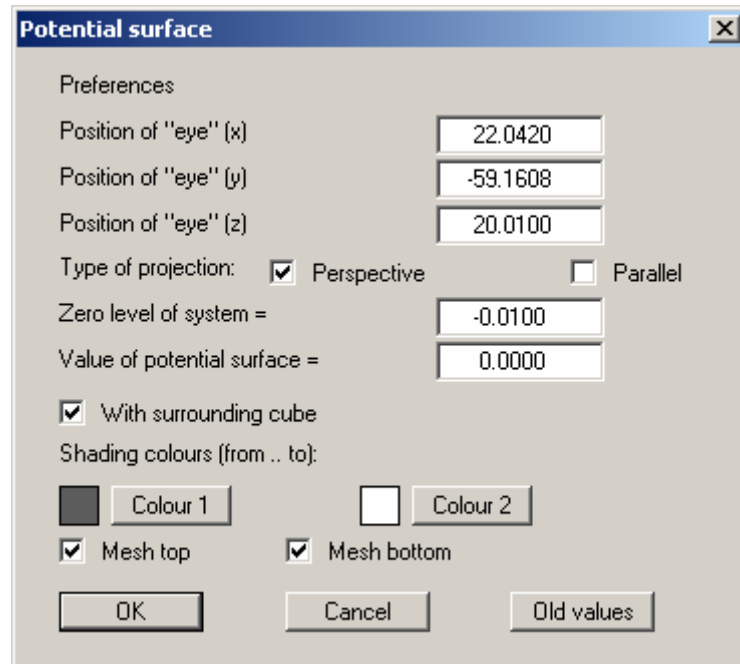
The outside of the system is covered by the mesh to increase the 3-dimensional impression. The colour of the mesh can be altered by means of the "Mesh colour" button.

After confirming your preferences using "OK", the 3D model is displayed on the screen. The angle of view of the model can be altered using the rotation tools in the *3D toolbar*. You can view the original visualisation once more by returning to this menu item and clicking the "Old values" button in the initial dialog box. If the model leaves the screen while it is being rotated, press the [F9] function key once and the screen coordinates will be recalculated.

You can move between the planes of your system using the arrow buttons on the *3D toolbar*. However, this only makes sense for cut-away system visualisation (external potential model in section).

8.10.4 "Potential surface" menu item

Surfaces with equal potentials, specified by user-input in "**Value of potential surface**", can be visualised in three dimensions in your system.



Shading corresponds to illumination of the system originating at the eye.

8.10.5 "Seepage surface" menu item

You can display your system's seepage surface in three dimensions. You will see a dialog box analogous to the box in the previous menu item (see Section 8.10.4).

8.10.6 "Flow lines preferences" menu item

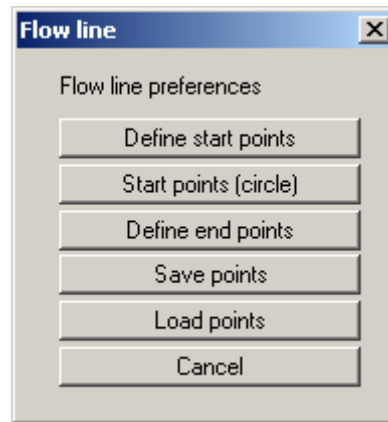
8.10.6.1 Principles

Flow lines can be drawn from the velocities calculated by the finite-element program. The principle used by the program is also known as the course line method or particle-tracking method. With this method a water particle, which wanders through the system from any start point, is observed. The program determines size and direction of seepage from this start point. With this velocity vector, the new position of the particle can be determined, whereby either a "**distance increment**" or a "**time increment**" can be selected for calculation of velocity vector length. The new position of the water particle is then the start point for the next calculation step. This procedure is carried on as long as a user-defined end-time is not exceeded, or the particle leaves the system. Small increments mean high precision and long calculation time.

If the increments are too large, nonsensical results can occur. Calculations are performed using the seepage velocity ($v_s = (k \cdot i)/n_{eff}$). Using this program, you can also follow a particle back (opposite to flow direction).

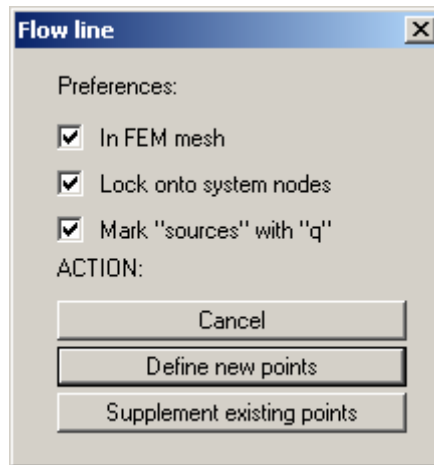
8.10.6.2 Preferences

After going to this menu item the flow modelling points are defined using the buttons in the following dialog box.



- **"Define start points"**
You can define the start points of water particles.
- **"Start point (circle)"**
You can define several start points, which can be grouped in a circle around the point you have determined. This can simplify the input of water particles, e.g., for wells within the system (key word: **"50 day line"**).
- **"Define end points"**
If depressions (extraction wells) are present within a system, the result may, in certain circumstances, be an oscillation of the particle track around the well, as all velocities are directed towards the well. In this case, end points can be defined in order to avoid unnecessarily long computation times. A square area must be defined by the user around the end points. If a water particle *enters* this square, computation of the corresponding particle track is ended.
- **"Save points"**
You can save start points to a file, in order to have them available for a different system.
- **"Load points"**
Start points saved using the previous button can be reloaded into the program using this button.

After selecting one of the buttons for defining points, the following box opens:



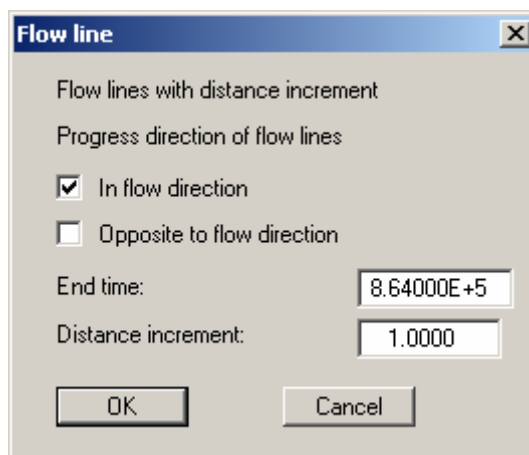
The "**In FEM mesh**" check box displays the FEM mesh when specifying points; otherwise, only the system outline is displayed. Defined points will be locked onto the system nodes closest to the mouse pointer when clicked, if the "**Lock onto system nodes**" check box is activated. Otherwise the points can be freely positioned within the system.

If some points are already defined, further points may be added by pressing the "**Supplement existing points**" button. Existing points are deleted if you exit the box via the "**Define new points**" button.

8.10.7 "Distance increment" menu item

You can start modelling the particle path(s) using a distance increment. You first define the angle of view and the type of projection in a dialog box. To improve clarity, visualisation of the 3D model can be restricted to the top and bottom plane by activating the "**Top and bottom only**" check box.

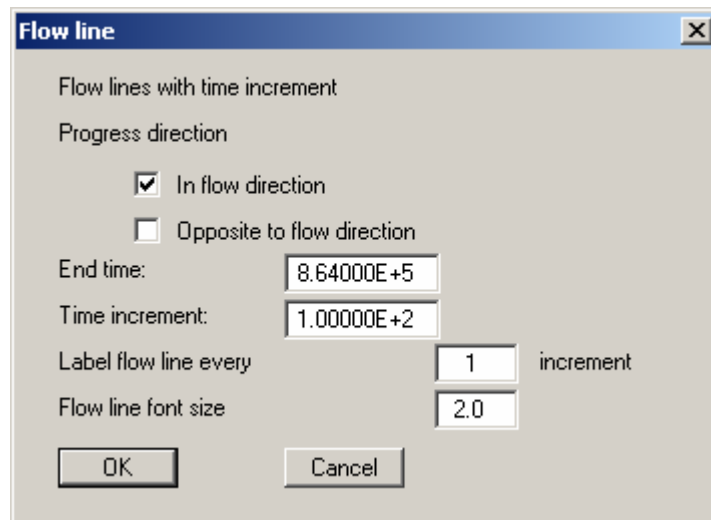
In the following dialog box the modelling parameters and the type of subsequent presentation can be confirmed or edited. Activate the "**Opposite to flow direction**" check box in order to retrace the path of a particle.



8.10.8 "Time increment" menu item

You can start modelling the particle path(s) using a time increment. You first define the angle of view and the type of projection in a dialog box. To improve clarity, visualisation of the 3D model can be restricted to the top and bottom plane by activating the **"Top and bottom only"** check box.

In the following dialog box the modelling parameters and the type of subsequent presentation can be confirmed or edited. Activate the **"Opposite to flow direction"** check box in order to retrace the path of a particle.



8.11 Info menu

8.11.1 "Copyright" menu item

You will see a copyright message and information on the program version number.

The **"System"** button shows information on your computer configuration and the folders used by GGU-3D-SSFLOW.

8.11.2 "Maxima" menu item

A box opens showing the maximum values allowed by the program for the nodes and elements of the FEM mesh (3D and 2D) .

8.11.3 "Help" menu item

The **GGU-3D-SSFLOW** manual is opened as a PDF document. The help function can also be accessed using the **[F1]** function key.

8.11.4 "GGU on the web" menu item

Using this menu item you can access the GGU Software website: www.ggu-software.com. Keep in touch with new program versions and the regular **download** offers.

If you would like to be automatically notified about program innovations, please register for the Newsletter in our Knowledge Base. Go to the following website: <http://kbase.civilserve.com>.

8.11.5 "GGU support" menu item

This menu item takes to the GGU-Software [Support area](http://www.ggu-software.com/support) at www.ggu-software.com.

8.11.6 "What's new?" menu item

You will see information on program improvements in comparison to older versions.

8.11.7 "Language preferences" menu item

This menu item allows you to switch the menus and the graphics from German to English and vice versa. To work in German, deactivate the two check boxes **"Dialoge + Menüs übersetzen (translate dialogues, menus)"** und **"Graphiktexte übersetzen (translate graphics)"**.

Alternatively, you can work bilingually, e.g. with German dialog boxes but with graphic output in English. The program always starts with the language setting applicable when it was last ended.

9 Tips and tricks

9.1 Keyboard and mouse

You can scroll the screen with the keyboard using the cursor keys and the **[Page up]** and **[Page down]** keys. By clicking and pulling with the mouse, with **[Ctrl]** pressed, you activate the zoom function, i.e. the selected section will fill the screen. Use the mouse wheel to zoom in or out of the screen view or to pan.

In addition, scale and coordinates of the system graphics (drawing area within the plotting margins) can be altered directly using the mouse wheel. The following mouse wheel functions are available

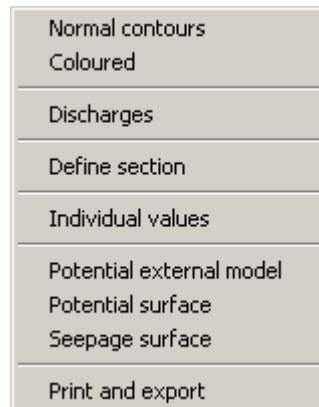
Change system graphics (new values can be checked in "Page size + margins/Manual resize (editor)"):

- **[Ctrl]** + mouse wheel up = enlarge system graphics (change of scale)
- **[Ctrl]** + mouse wheel down = shrink system graphics (change of scale)
- **[Shift]** + mouse wheel up = move system graphics up
(change in system coordinates)
- **[Shift]** + mouse wheel down = move system graphics down
(change in system coordinates)
- **[Shift]** + **[Ctrl]** + mouse wheel up = move system graphics right
(change in system coordinates)
- **[Shift]** + **[Ctrl]** + mouse wheel down = move system graphics left
(change in system coordinates)

Change screen coordinates:

- Mouse wheel up = move screen image up
- Mouse wheel down = move screen image down
- **[Alt]** + **[Ctrl]** + mouse wheel up = enlarge screen image (zoom in)
- **[Alt]** + **[Ctrl]** + mouse wheel down = shrink screen image (zoom out)
- **[Alt]** + **[Shift]** + mouse wheel up = move screen image right
- **[Alt]** + **[Shift]** + mouse wheel down = move screen image left

If you click the right mouse button anywhere on the screen a context menu containing the principal menu items opens.




By double-clicking the left mouse button on legends or **Mini-CAD** objects, the editor for the selected element immediately opens, allowing it to be edited.

9.2 Function keys

Some of the function keys are assigned program functions. The allocations are noted after the corresponding menu items. The individual function key allocations are:

- [Esc] refreshes the screen contents and sets the screen back to the given format. This is useful if, for example, you have used the zoom function to display parts of the screen and would like to quickly return to a complete overview.
- [F1] opens the manual file.
- [F2] refreshes the screen without altering the current magnification.
- [F3] opens the menu item "**Mesh(x/y)/Define nodes**".
- [F4] opens the menu item "**Mesh(x/y)/Manual mesh**".
- [F5] opens the menu item "**System/Modelling**".
- [F6] opens the menu item "**Mesh(x/y)/(Refine) Section**".
- [F7] opens the menu item "**Mesh(x/y)/(Refine) All**".
- [F9] activates the menu item "**Page size + margins/Auto-resize**".
- [F11] activates the menu item "**Graphics preferences/Move objects**".

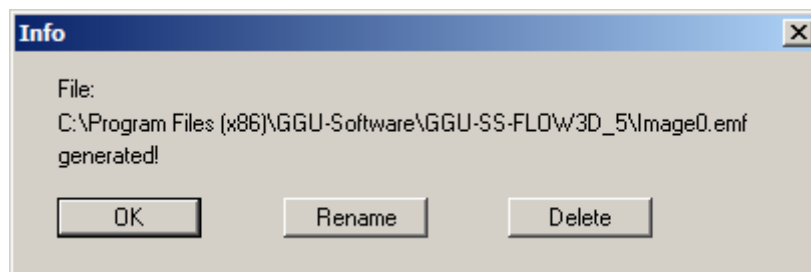
9.3 "Copy/print area" icon

A dialog box opens when the "Copy/print area" icon  in the menu toolbar is clicked, describing the options available for this function. For example, using this icon it is possible to either copy areas of the screen graphics and paste them into the report, or send them directly to a printer.

In the dialog box, first select where the copied area should be transferred to: "**Clipboard**", "**File**" or "**Printer**". The cursor is displayed as a cross after leaving the dialog box and, keeping the left mouse button pressed, the required area may be enclosed. If the marked area does not suit your requirements, abort the subsequent boxes and restart the function by clicking the icon again.

If "**Clipboard**" was selected, move to the MS Word document (for example) after marking the area and paste the copied graphics using "*Edit/Paste*".

If "**File**" was selected, the following dialog box opens once the area has been defined:



The default location of the file is the folder from which the program is started and, if several files are created, the file is given the file name "**Image0.emf**" with sequential numbering. If the "**Rename**" button in the dialog box is clicked, a file selector box opens and the copied area can be saved under a different name in a user-defined folder. Saving can be aborted by pressing the "**Delete**" button.

If the "**Printer**" button was pressed in the first dialog box, a dialog box for defining the printer settings opens after marking the area. Following this, a dialog box for defining the image output settings opens. After confirming the settings the defined area is output to the selected printer.

10 Worked example for a flood protection embankment

10.1 Step 1: Define the system

We will investigate a flood protection embankment with a 1 m thick sealing layer in the embankment foreland and on the slope towards the water. There is a weak area in the cover layer of the embankment foreland covering an area of $8.33 \text{ m} \cdot 10.0 \text{ m}$. The system is symmetrical perpendicular to the embankment axis, so only half the system need be investigated. The system looks like this in a 3-dimensional view:

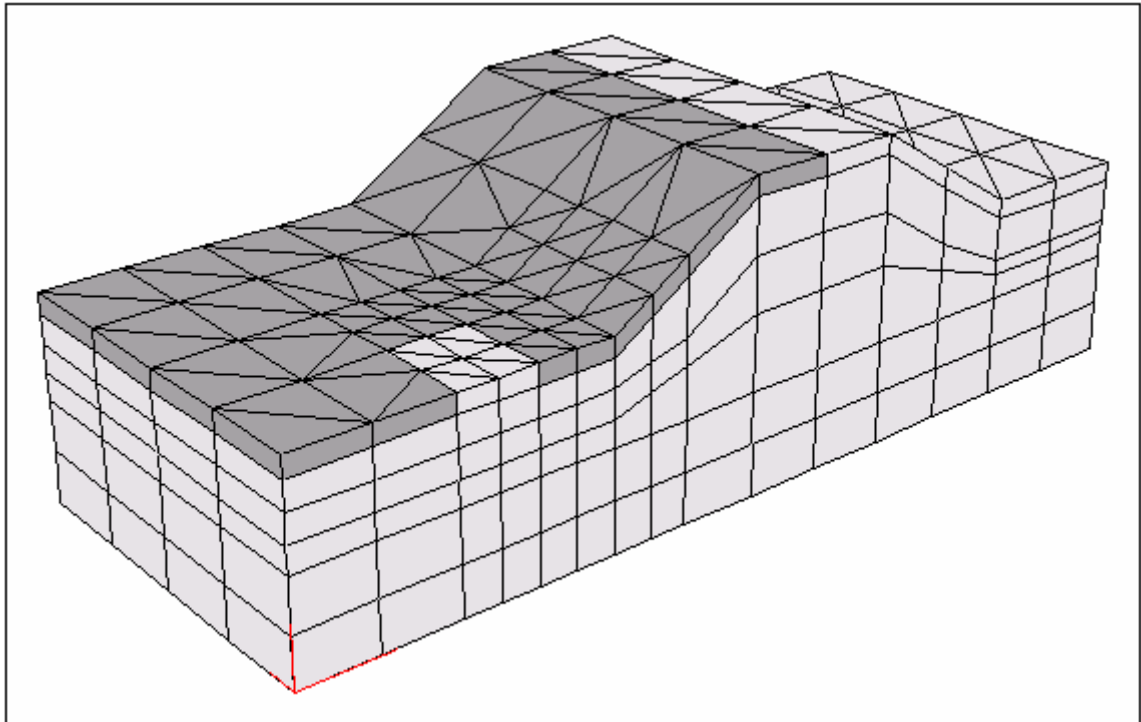


Figure 10 Worked example in 3D

The visible longitudinal side of the system is the symmetry plane. It looks like this in section:

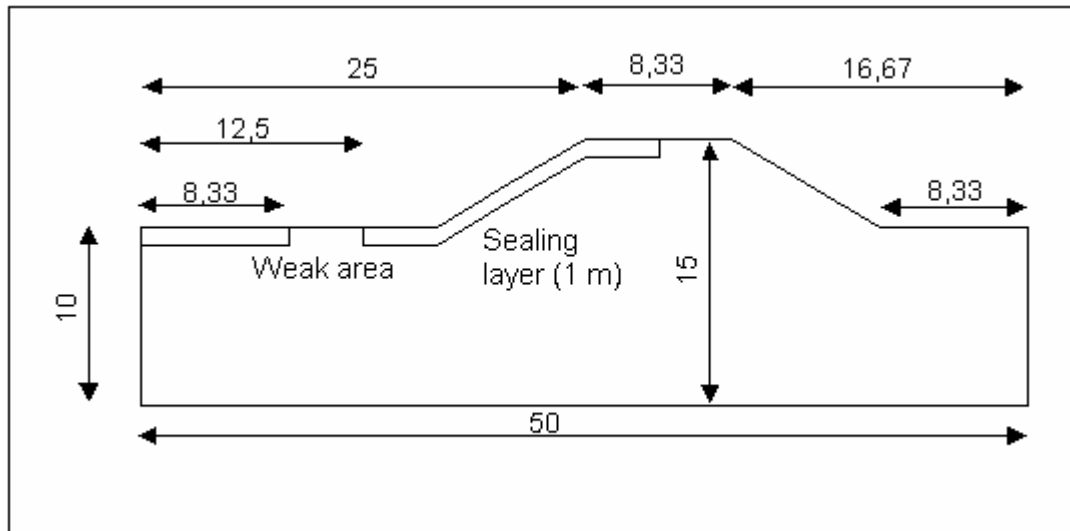


Figure 11 Worked example in section

Perpendicular to the section the weak area has a width of 10.0 m (5.0 m in the symmetrical system). The embankment will be modelled along a length of 40.0 m (20 m in the symmetrical system). The dimensions have been selected so that the subsequent mesh generation remains simple and does not have too much unnecessary ballast.

The following permeabilities are given:

- Embankment core and subsurface: $k = 10^{-4}$ m/s
- Sealing layer: $k = 10^{-7}$ m/s

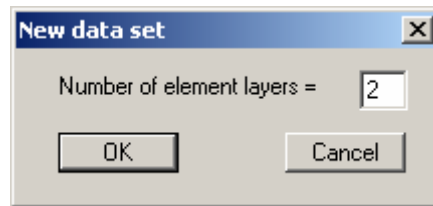
The upstream pondage of the embankment is at 15 m AD. As on many *proprietary* embankments, no drainage systems is installed on the *air side*. Drainage is by means of the hinterland, which is at 10 m AD.

The discharge through the embankment will be determined for the following cases:

- Case A with weak area
- Case B without weak area

10.2 Step 2: Create record

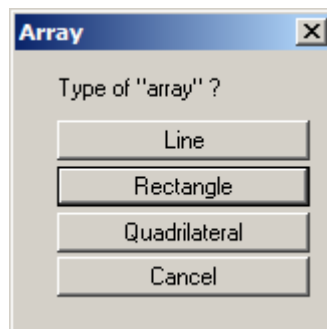
Start **GGU-3D-SSFLOW** and go to the **"File/New"** menu item. As you do not wish to proceed via **"Common systems"** answer no to the question and enter the number of element layers.



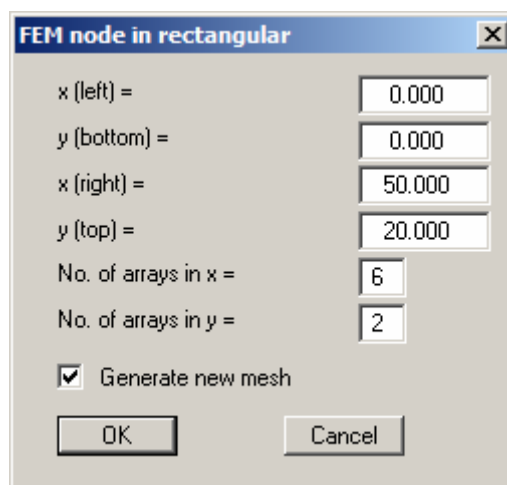
In vertical sections of the embankment a maximum of two soils need to be differentiated. It is therefore sufficient for mesh generation to enter two element layers.

10.3 Step 3: Generate FEM nodes and mesh

The rough structure of the system consists of the four boundary points. Mesh generation is therefore simplest using **"Mesh(x/y)/Array"** menu item. Select **"Regular"** as type of array. You will see the following dialog box:



Select the **"Rectangle"** button and get the following box:

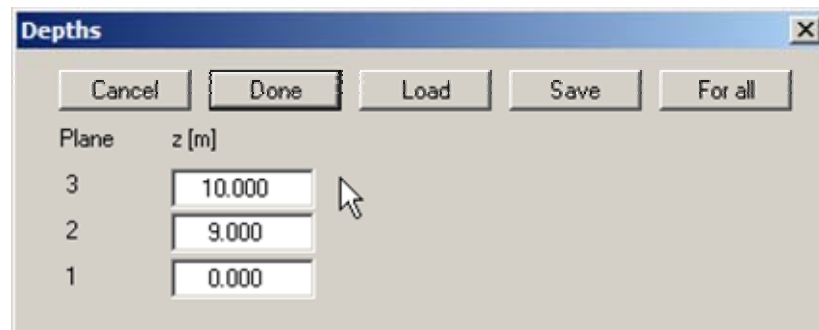


Enter the data in the dialog box and confirm by pressing "OK". If the "Generate new mesh" button is activated, the user-defined FEM nodes are automatically connected to form a triangular mesh. Otherwise, select the "Mesh(x/y)/Automatic" menu item.

To achieve a screen-filling representation of the system got to the "Page size + margins/Auto-resize" menu item. If necessary, move the legend(s) outside of the mesh using the [F11] function key.

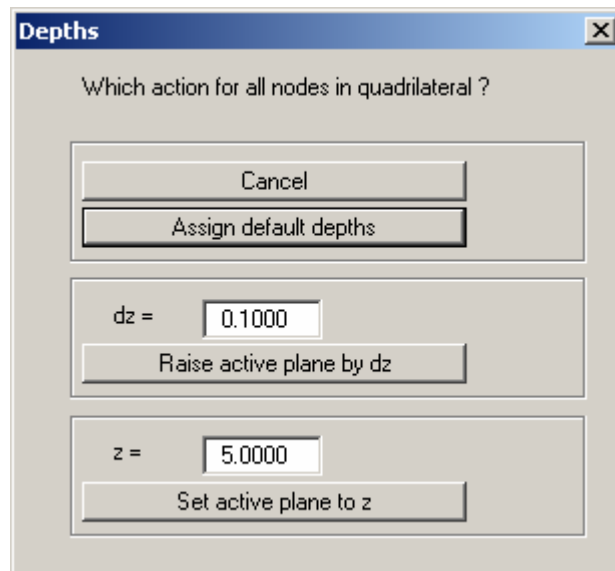
10.4 Step 4: Assign z ordinates for the embankment foreland and hinterland

Now that the rough system structure has been generated, we must assign the nodes a z ordinate. Go to "z/Default depths" to initially generate the values for the embankment foreland.



Plane	z [m]
3	10.000
2	9.000
1	0.000

Enter the values of the dialog box above and confirm with "Done". The value "10" stands for the surface of the embankment foreland and "9" is the top of the sealing layer. The value "0" designates the base of the system. Now go to "z/Depths in section" and travel around the left nine points of the FEM mesh (embankment foreland).



Which action for all nodes in quadrilateral ?

Cancel

Assign default depths

dz = 0.1000

Raise active plane by dz

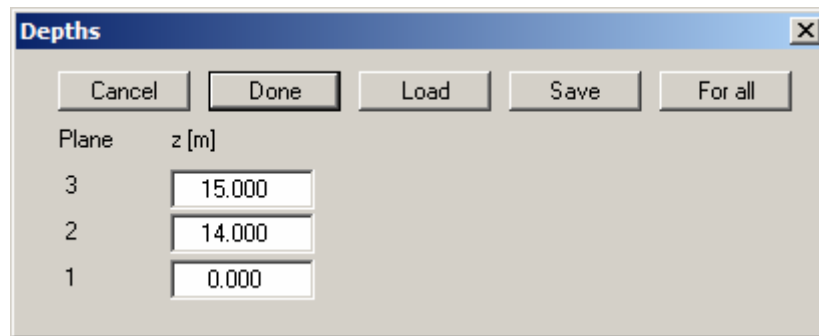
z = 5.0000

Set active plane to z

Press the "Assign default depths" button. Travel around the six right nodes (embankment hinterland) of the FEM mesh and assign these the default depth.

10.5 Step 5: Assign z ordinates for the top of the embankment

Go to "**z/Default depths**" to generate the values for the top of the embankment.



The screenshot shows a dialog box titled "Depths" with a close button (X) in the top right corner. Below the title bar are five buttons: "Cancel", "Done", "Load", "Save", and "For all". The main area of the dialog contains a table with two columns: "Plane" and "z [m]". The table has three rows of data.

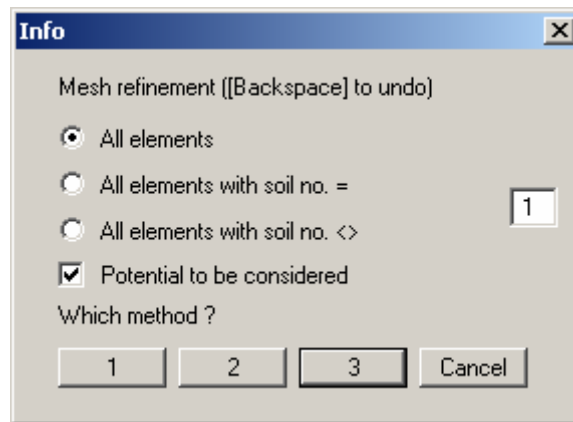
Plane	z [m]
3	15.000
2	14.000
1	0.000

Enter the values of the dialog box above and confirm with "**Done**". The value "**15**" stands for the surface of the embankment and "**14**" is the base of the sealing layer at the top of the embankment. The value "**0**" designates the base of the system. Now go to "**z/Depths in section**" and travel around the six points of the FEM mesh representing the top of the embankment.

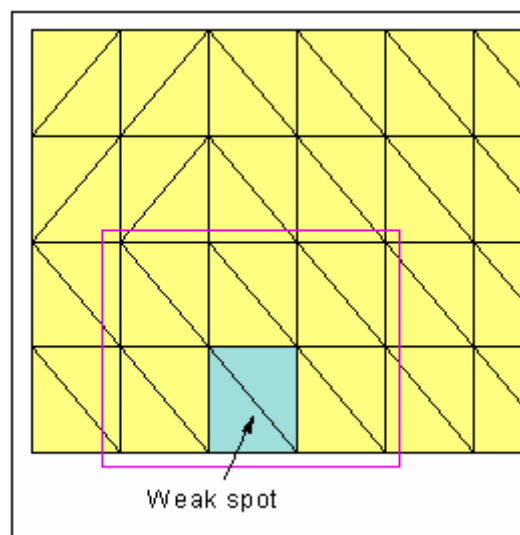
Check your input using "**z/Define section**" to define a section along the centre of the system. After pressing [**Return**], a sectional representation should be visible, approximating the Figure 11 above. Everything okay?

10.6 Step 6: Refine and optimise mesh in x/y-direction

Now, the mesh can be refined. To do this, go to "**Mesh(x/y)/(Refine)All**".




Press button "3" . Local refinement will be performed in the region of the weak spot. Go to "**Mesh(x/y)/(Refine)Section**". Press button "3" and travel with the mouse around the weak spot, so that a total of six rectangles with two triangles each are included (see the following figure).

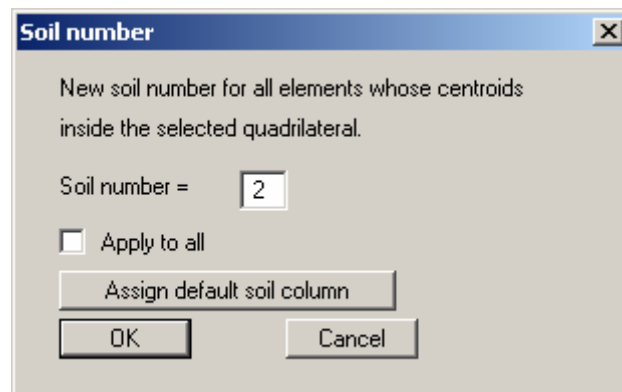


The six rectangles consist of the rectangle representing the weak spot and the five neighbouring rectangles.

Now go to "**Mesh(x/y)/Optimise**" and then press the "**Diagonals**" button. This is not a necessity but tends to generate a numerically more favourable mesh. The system should now look exactly like the system in the supplied "**MANUAL0.f3d**" file. You should now save all your data (menu item "**File/Save as**") and treat yourself to a coffee.

10.7 Step 7: Assign soil numbers

During the previous mesh generation, all new elements were assigned the soil number "1". In order to allow the sealing layer to later be assigned a different permeability, these elements must be assigned the soil number "2". Go to "**Boundary/(Soils) In section**". At the top right a schematic of the current plane and layer are shown. Use the  tool in the lower left *3D toolbar* until you have reached the uppermost layer, the sealing layer. Now travel around the entire left of the area as far as the centre of the embankment top. The sealing layer should end here.



Assign the soil number "2" and leave the dialog box with "**OK**". The "**Apply to all**" check box should not be activated under any circumstances, otherwise all layers in the defined region will be assigned the soil number "2".

The weak spot is given the same permeability as the subsurface of the embankment (soil number "1"). So, travel with the mouse around the weak spot and assign it the soil number "1".

Check your input using "**z/Define section**" to define a section along the lower system boundary (as seen in plan). Press **[Return]**. The FEM mesh is represented in section. If necessary, activate the check boxes "**Coloured**" and "**With soil no.**" in the "**Mesh(x/y)/Preferences**" menu item. Now go to "**z/Outline**". The sealing layer and the weak spot should now be visible. Do the same for the upper system boundary. Here, only the sealing layer should be visible.

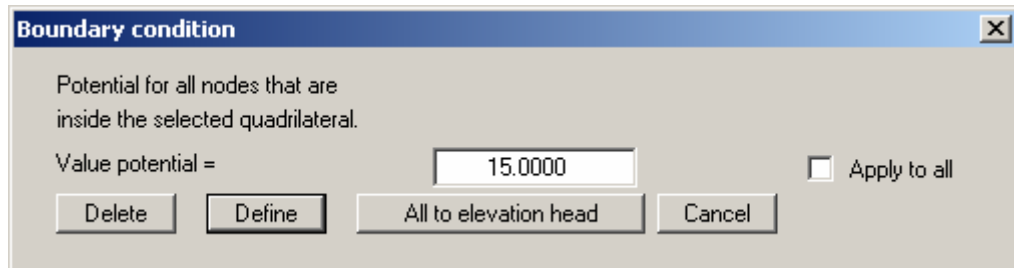
10.8 Step 8: Define boundary conditions

If you still find yourself in a sectional representation, go to "**Mesh(x/y)/Mesh**" before defining boundary conditions and return to the plan view.

To define boundary conditions go to "**Boundary/(Potentials) In section**" and travel around the FEM nodes on the left of the system, including the row of nodes making up the *left edge of the embankment top*. Check beforehand that you are still in the uppermost plane and if not use the



tool in the **3D toolbar** to get there.



Make sure you deactivate the "**Apply to all**" check box to ensure that only the nodes in the embankment foreland and on the upstream slope are assigned a potential of 15 m AD (= upstream water level).

Now travel round the nodes on the right of the system, including the row of nodes making up the *right edge of the embankment toe* and assign them a potential of 10 m AD (= surface of embankment hinterland).

Check your input using "**z/Define section**" to define a section along the lower system boundary (as seen in plan). Press [Return]. The FEM mesh is represented in section. Go to "**Boundary/Individual potentials**" or "**Boundary/(Potentials) In Section**" menu items. The potentials are represented in the selected section. In this section, too, you can edit the potential boundary conditions. However, this should not be necessary if everything was entered correctly.

Remark on boundary conditions:

The case of an impermeable boundary is considered automatically by the finite element method. All system boundaries, which do not possess source or potential boundary conditions, are automatically impermeable. In finite element theory, this type of boundary condition is also known as a natural boundary condition. The selected boundary conditions on the water-side assume that the embankment is not percolated from below by the river water, which may intersect the cover layer. If you want to model this, you will have to also create a continuous weak spot on the river-side.

If all input was correctly executed the system should now correspond to the system in the supplied "**MANUAL1.f3d**" file.

10.9 Step 9: Refine mesh in z-direction

First go to "**z/Define section**" to define a section through the system. The section is represented by pressing [**Return**]. It is apparent that the embankment subsurface is very coarsely modelled in z direction. Select the "**z/Refine**" menu item.

Click three times in total in the embankment subsurface, so that the original layer now consists of four layers. If you execute a wrong click, press [**Backspace**].

10.10 Step 10: Assign soil properties

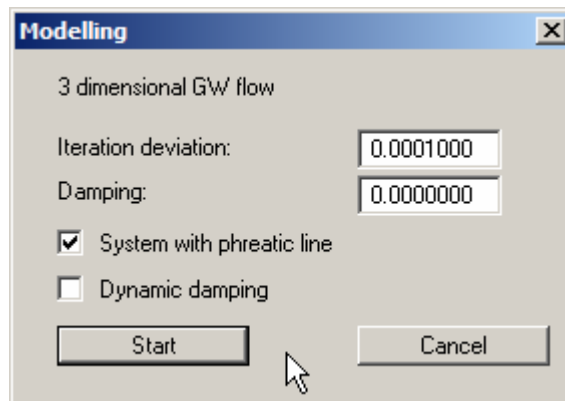
Go to "**System/Soil properties**". Enter the values of the following dialog box.

No.	kx	ky	kz	neff	kr = f(u)	Designation	Std
1	1.000E-4	1.000E-4	1.000E-4	0.20	kr = f(u)		Std
2	1.000E-7	1.000E-7	1.000E-7	0.05	kr = f(u)		Std

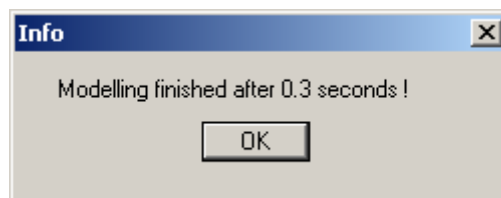
Because you have assigned two different soil numbers, the dialog box also now has two input rows for two soils. Enter the values of the dialog box above. If you do not place excessive demands on the location of the phreatic line you can do away with editing using the "**kr = f(u)**", otherwise, read the "**Theoretical principles**" Section 7.2. The effective pore space n_{eff} only plays a role in flow line representation and the representation of velocities. This value has no influence on potentials and discharges.

10.11 Step 11: Model system

Go to the "System/Model" menu item.



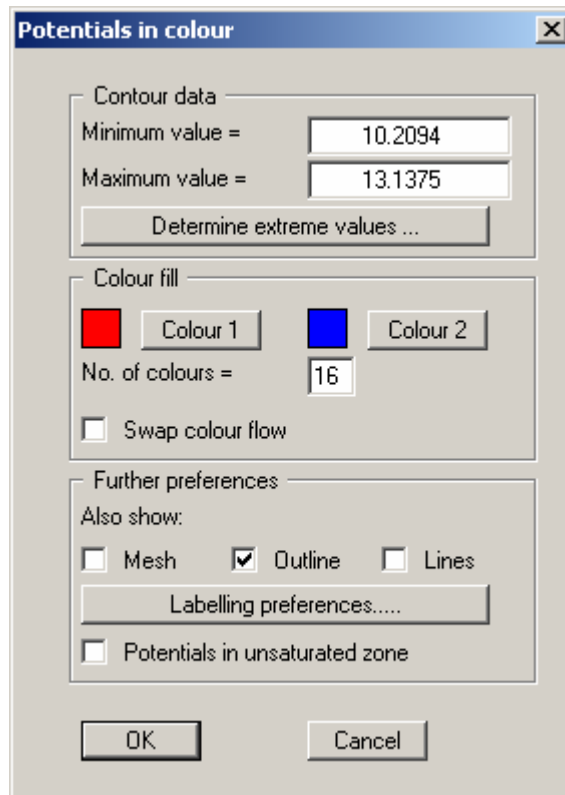
Accept the preferences and press the "**Start**" button. Depending on the power of your computer, you will be informed after a certain time that computation is complete.





The resulting system is saved as "**MANUAL2.f3d**".

10.12 Step 12: Evaluate potentials in "horizontal" planes

A contour plan of the potentials will be generated. If you currently find yourself in a sectional representation, go to "**Mesh(x/y)/Mesh**" menu item first to return to the plan view. Select the "**Evaluation/Coloured**" menu item.




Have the program determine the minimum and maximum contour line values by using the "**Determine extreme values ...**" button. In order to achieve clear grading in 25 cm steps in the colour bar, enter "**10**" as the minimum contour line value and as maximum "**15**". Set the number of colours to "**21**" and confirm the changes using "**OK**".

You will now see a contour plan of the current node plane. Move through the planes using the two tools  and  in the **3D toolbar** and observe the changes in the potential contours. You can send the resulting graphics to the printer.


If you prefer a simple line representation instead of coloured graphics, go to the "**Evaluation/Normal contours**" menu item (Section 8.9.2).

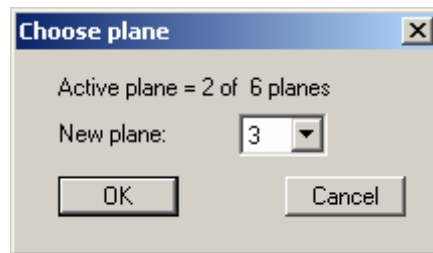
The term "**horizontal**" used in the heading for this section is incorrect inasmuch as the node planes need not necessarily be horizontal.

10.13 Step 13: Evaluate potentials in section

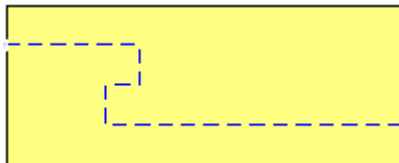
To define a section, go to "**Evaluation/Values in section**" or "**z/Define section**" or use the  tool in the **3D toolbar**. Press **[Return]**. Go to "**Evaluation/Coloured**". You will see the potentials in the defined section. If you lead the course of the section along the lower system boundary, the influence of the weak spot is easily visible.

10.14 Step 14: Represent a cut-away system

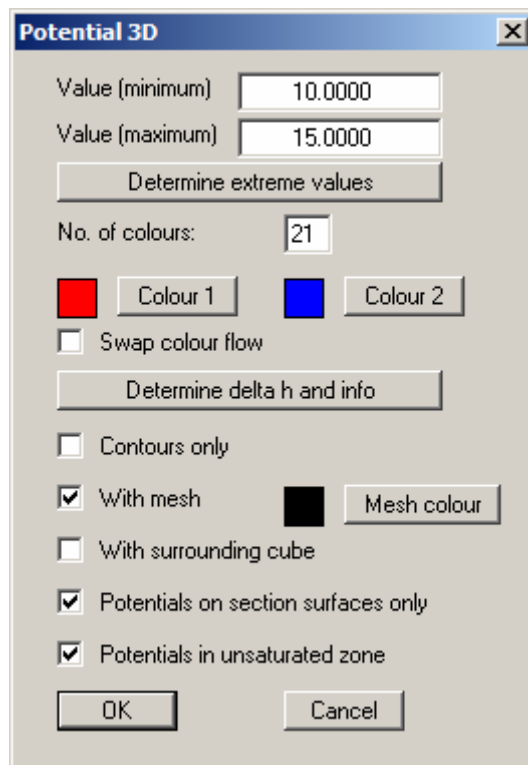
First, using the  tool in the **3D toolbar**, bring the current plane to "**3**" (not absolutely necessary).






Now define a section as shown in the figure below using the menu item "**Evaluation/Values in section**" or load the section provided in the "**MANUAL.fes**" file by pressing the "**Load section**" button.



Press the **[Return]** key. The section will be displayed. Now select the "**3D/Potential external model**" menu item and confirm the default projection preferences. The following dialog box appears:



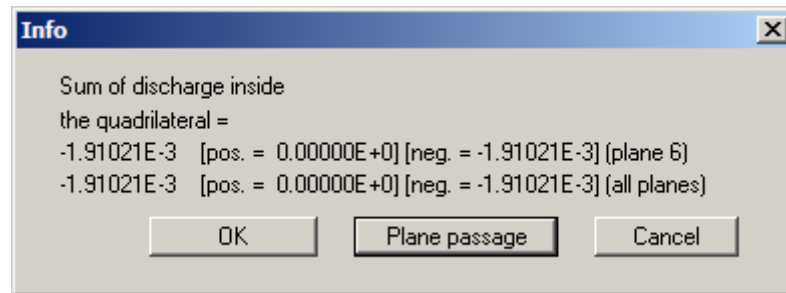
Confirm your entries with **"OK"**. You will see the cut-away system as a 3D representation. Move through the planes using the  and  tools in the **3D toolbar**. Select the  tool or the other rotation tools to modify your point of observation (position of the eye). Then press the **[F9]** function key or go to menu item **"Page size + margins/Auto-resize"** to adapt the graphics to the screen section.

If you select the **"External model"** menu item instead of the **"Potential external model"** you will see the soil colours or a shaded system of your 3D mesh instead of the potentials.

10.15 Step 15: Evaluate discharges

If you currently find yourself in a sectional representation, go to "**Mesh(x/y)/Mesh**" to return to the plan view.


Select the "**Evaluation/Discharges**" menu item and accept the preferences of the dialog box. Move to the uppermost node plane. All nodes with a potential boundary condition are labelled with their respective discharge. At all other nodes the discharge is "**0**". Travel around the nodes in the embankment hinterland.

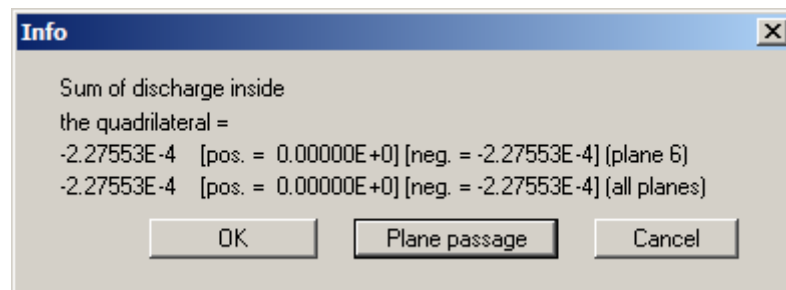


The summed discharge through the embankment is $1.9 \cdot 10^{-3} \text{ m}^3/\text{s}$ or $6.9 \text{ m}^3/\text{h}$. Because only half of the system was investigated, due to the symmetry, the total discharge is $13.8 \text{ m}^3/\text{h}$. This value is negative because the water leaves the system here. If you click the nodes in the embankment foreland and on the slope, you will get the same value but positive.

10.16 Step 16: Remove the weak spot

If you currently find yourself in a sectional representation, go to "**Mesh(x/y)/Mesh**" to return to the plan view.

Now go to the "**Boundary/(Soils) In section**" menu item. At the top right a schematic of the current plane and layer is shown as *Layer/plane legend*. Use the  tool in the *3D toolbar* until you have reached the uppermost layer, the sealing layer. The weak spot is now given the same permeability as the sealing layer (soil number "**2**"). So, travel with the mouse around the weak spot and assign it the soil number "**2**". Remodel the system ("**System/Model**") and then determine the discharge.



The discharge through the embankment is now $2.3 \cdot 10^{-4} \text{ m}^3/\text{s}$ or $0.8 \text{ m}^3/\text{h}$. Because only half of the system was investigated, due to the symmetry, the total discharge is $1.6 \text{ m}^3/\text{h}$. This shows that the defect removed in this step had led to an increase in the discharged water by a factor of 8.5.

11 Index

3

- 3D system,
 - display as cut-away external model 73
 - display as wire model..... 73

5

- 50 day line 76

A

- Analysis,
 - display results for individual FEM node 71
 - influence of mesh subdivisions 12
 - save results to file..... 16
- Angle of rotation, define for 3D graphics 53
- Approximation method..... 12
- Array for FEM nodes generation,
 - irregular array..... 29
 - regular array 28

B

- Bandwidth 49
- Boundary, impermeable..... 14, 45

C

- Clipboard..... 22
- CodeMeter stick..... 7
- Colour bar, define for contours..... 64
- Colours,
 - activate display in soil properties legend 56
 - define for interpolation/FEM mesh 41
 - define for soils..... 56
 - switch on/off 53
- Colours/pens, define for graphical elements..... 51
- Company letterhead, add via Mini-CAD..... 52
- Context menu, open..... 81
- Continuity condition..... 66
- Contours,
 - display for potentials as 3D array graphics .. 66
 - display for potentials as 3D graphics 64
 - edit preferences for colour-filled visualisation 63
 - edit preferences for normal visualisation 62
 - groundwater thickness..... 71
 - groundwater-surface distance..... 71
 - layer thickness..... 39
 - z values of interpolation mesh..... 42
- Coordinates of FEM mesh,
 - import as ASCII file 16
 - import via Windows clipboard..... 25
- Coordinates of graphics,
 - alter via editor 59
 - alter with mouse 59
 - optimise/reset 59
 - save/load 59

- Copy protection 7
- Copy/print area 22, 53, 82
- Course line method..... 75
- Cutting borders, switch on/off..... 61

D

- Damping 50
- Darcy's Law 12
- Data set description,
 - display in legend 55
 - enter..... 47
- Default depths, define..... 36
- Default soil column, define..... 37
- Delauney triangulation 32
- Delete,
 - all FEM nodes 25
 - diffuse sources..... 46
 - individual FEM elements 31
 - individual FEM nodes 25
 - linear sources..... 46
 - point sources 45
 - several FEM elements 32
- Depths,
 - edit for individual FEM nodes..... 37
 - edit for plane 38
 - edit for several FEM nodes 38
- Differential equation..... 13
- Diffuse sources, define/delete..... 46
- Diffuse sources, raise/reduce 46
- Dimension,
 - convert..... 48, 56
 - note for selection..... 49
- Discharges,
 - calculate for parts of the system..... 67
 - calculate in section 67
 - display at FEM nodes..... 66
- Drawing area, define..... 60
- DXF file,
 - export 22
 - import via Mini-CAD..... 7

E

- Editor window, output table 20
- Effective pore space 47
- Element layer,
 - delete 40
 - generate parallel 38
 - insert..... 40
 - numbering 15
- EMF format 22
- Equation solver, select for modelling 49
- Export,
 - data set interpolation mesh..... 43
 - FEM mesh coordinates..... 17
 - record for transient analysis 23
- External model, display 73
- Extraction well, take into account 76

F

FEM elements,	
assign soil column	37
assign soil properties	46
refinement methods	35
FEM mesh,	
define using mouse	31
delete FEM elements automatically	32
delete FEM elements using mouse	31
delete layer	40
generate automatically	32
insert layer	40
optimise FEM elements	33
refine certain FEM elements	34
refine individual FEM elements	34
refine several FEM elements	34
save coordinates as ASCII file	17
show max. number of nodes/elements	79
FEM nodes,	
adjust scale	26
assign depths	36
define depths in advance	36
define using array	27
define using editor	25
define z values in advance	37
define/delete using mouse	25
edit coordinates	27
edit depths	37
import coordinates via Windows clipboard ..	25
move using editor	26
Fence diagram, display in section	72
File,	
display name in legend	55
load/save	15
Finite-difference-methods	12
Finite-element-methods	12
Flow lines,	
define/delete points	77
display	75
using distance increment	77
using time increment	78
Font selection	51
Font size,	
define for general legend	55
define for graphical elements	60
define for $k_r = f(u)$ legend	57
define for layer/plane legend	54
define for section legend	57
define for soil properties legend	56
Footer, output table	18
Function keys	81
Function $k_r = f(u)$,	
curves for typical soils	13
define	47
display in legend	57

G

General page information,	
add via Mini-CAD	52
GGU-3D-TRANSIENT file, export	23
GGUCAD file, export	22
GGUMiniCAD file, export	23

Gradients,	
display as arrows	70
display as colour-filled contours	63
display as normal contours	62
display maximum values in a message box ..	71
Graphics, add via Mini-CAD	52
Groundwater thickness, as contours	71
Groundwater-surface distance, as contours	71

H

Header CAD, use	52
Header, output table	18
Hydraulic head, define	44

I

Impermeable boundary	45
Import,	
coordinates of FEM mesh	16
data set interpolation mesh	43
Installation	7
Interpolation mesh,	
contours of z values	42
edit mesh	42
edit nodes	41
for z values	41
modify z values using constant	42
Iteration,	
abort criteria	49
phreatic line calculation	49
unsaturated areas	14

K

Knowledge Base, access	79
k-value, edit unit	48, 56

L

Language preferences	8, 79
Layer,	
definition	9
delete	40
display in legend	54
generate parallel	38
insert	40
numbering	15
Layout,	
define for output sheet	60
define for output table	18
Legends, move with mouse	58
Licence protection	7
Linear sources, define/delete	46

M

Manual, open as PDF file	79
Maximum gradients, display/copy	71
Metafile, export	22
Mini-CAD,	
display objects behind system graphics	52
export file	23
use	52

Modelling results,	
display for individual FEM node	71
save to file	16
Mouse click functions.....	81
Mouse wheel functions.....	80

O

Objects, move with mouse.....	58
Output table,	
edit output as ASCII.....	20
edit output as graphics.....	18
navigation.....	19, 53
select the output format.....	17
switch to system graphics.....	19, 53

P

Page,	
copy/print section.....	53, 82
define format	60
define margins.....	60
switch on/off margins.....	61
Pagination, automatic	19, 20
Parallel layers, generate.....	38
Particle-tracking method.....	75
PDF file, import via Mini-CAD.....	7, 52
Pen preferences.....	51
Permeabilities,	
define to Darcy.....	47
edit unit	48, 56
Phreatic line,	
activate analysis	50
analysis for unsaturated areas.....	14
Piezometric heads,	
analysis.....	12
define	44
Plane,	
change	53, 62
definition.....	9
determine discharge	67
display in legend	54
numbering	15
Plot margins, define.....	60
Point sources, define/delete	45
Pore water pressure, analysis.....	13
Potential boundary condition,	
define linearly variable.....	45
define/delete for individual FEM nodes.....	44
define/delete for several FEM nodes.....	45
Potential surface, display	75
Potentials,	
display as colour-filled contours	63
display as cut-away external model	74
display as fence diagram	72
display as normal contours.....	62
replace designation in graphics	
by water level.....	61
Precipitation events, consider	46
Print,	
graphics	21
output table.....	22
section	22, 53, 82
several files	23

Printer preferences.....	20, 21
Program,	
display name in legend.....	55
save/load preferences	58
show improvements.....	79
show information	79
Project data, add via Mini-CAD	52
Project identification,	
display in legend	55
enter.....	47

Q

Quality of the solution	12
-------------------------------	----

R

Radius ratio, FEM elements	32
Refinement methods	35

S

Scale,	
alter with mouse	59
define via editor	59
determine automatically	59
Scroll the screen	80
Section course,	
define.....	39, 69
display in legend	57
Section,	
load.....	40
save	40
Seepage surface, display	75
Seepage velocity, display	70
Smart icons,	
edit for 3D graphics.....	53
menu items	52
Smoothing out procedures, select for contours	63
Soil colours,	
activate display in legend.....	56
define.....	56
use in $k_r = f(u)$ legend	58
Soil numbers,	
activate display in legend.....	56
assign for individual FEM elements.....	46
assign for several FEM elements.....	47
swap	48
Soil properties,	
define.....	47
display in legend	56
Status bar main program, activate.....	52
System coordinates,	
alter via editor	59
alter with mouse	59
optimise/reset	59
save/load	59
System,	
display properties in legend.....	55
generate automatically	30
input using DXF/BMP files.....	25
model.....	49
show information	79
test	47

T

Toolbar,	
activate for 3D graphics	53
edit for menu items.....	52
Transient 3D groundwater modelling,	
export file	23
Translation, activate	79
True-type font.....	51

U

Unit,	
convert.....	48, 56
note for selection	49
Unsaturated zone,	
define function $kr = f(u)$	47
switch off results visualisation	63, 64, 71

V

Velocities,	
display as arrows	70
display as colour-filled contours	63
display as normal contours.....	62
modelling	12
Velocity field, display	70
Version number,	
display in a message box	79
display in legend	55

W

Water level,	
activate designation in graphics	61
display differences	72
Water pressure, display as fence diagram.....	72
What you see is what you get	51
Wire model, display.....	73

Z

z values, assign using interpolation mesh.....	43
Zienkiewicz	7
Zoom factor, define for full-screen display	51
Zoom function, activate	51, 53, 80