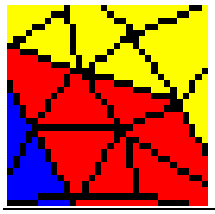
Modelling of groundwater flow using finite elements
in two-dimensional systems

GGU-SS-FLOW2D

VERSION 10



Last revision:	June 2015
Copyright:	Prof. Dr. Johann Buß
Technical implementation and sales:	Civilserve GmbH, Steinfeld

Contents:

List of Figures:

1 Preface

The **GGU-SS-FLOW2D** program allows modelling of steady-state groundwater flow in horizontal-plane, vertical-plane and axis-symmetrical groundwater systems using the finite-element method.

The program system includes a powerful mesh generator and easy-to-use routines for comfortable evaluation of the modelling results (contours, 3D graphics, etc.).

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 FE program can be taken from, e.g., J. Buß "*Unterströmung von Deichen*" (Announcement 92, Leichtweiß-Institut, TU Braunschweig).

Data input is in accordance with WINDOWS conventions and can therefore be learned almost without 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. DXF files can also be imported by means of the integrated Mini-CAD module (see the "**Mini-CAD**" manual).

The program system has been used in a large number of projects and has been thoroughly tested (using analytical solutions and in comparison with other FEM applications). 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-SS-FLOW2D** 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-SS-FLOW2D** 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-SS-FLOW2D is a multilingual 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 "**?**" menu, using the menu item "**Language preferences**" (in German: "**Spracheinstellung**", in Spanish: "**Configuración de idioma**").

4 Starting the program

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

- File
- ?

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/Common systems**" menu item after "**New**" is clicked (see Section 7.2.9). 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. Select your system and you will then return to the home screen. Depending on the type of system selected, you will now see either eight or ten menus along the menu bar:

- File
- Mesh
- z (for horizontal-plane systems only)
- Boundary
- System
- Graphics preferences
- Page size + margins
- Evaluation
- Special (for horizontal-plane systems only)
- ?

After clicking one of these menus, the menu items roll down, allowing you 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-SS-FLOW2D** 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 Short description

As reading manuals is, from personal experience, a bit of a chore, there will now follow a short description of the main program functions. After reading this section you will be in a position to analyse steady-state groundwater flow. Details can then be taken from further chapters. Worked examples can be found in Section 9.

- Design the system you wish to analyse.
- Start the **GGU-SS-FLOW2D** program and go to the menu item "**File/New**". Answer the question of the "**Common systems**" dialog box with "**No**". Select the type of system which you would like to work with.
- If necessary, fit the page coordinates to those of your system. For this, use the menu item "**Page size + margins/Manual resize (editor)**".
- Then go to the menu item "**Mesh/Define nodes**".
- Click on the principal nodes (points) of your system with the mouse. The points will be numbered. Alternatively, you can also enter the system nodes in tabular form, using the menu item "**Mesh/Change**". When defining nodes in horizontal-plane systems, new nodes will be assigned ***default depths*** (top and base of aquifer), which may be edited with the menu item "**z/Default depths**".
- If the nodes are outside of the page coordinates, go to the menu item "**Page size + margins/Auto-resize**" or use the [**F9**] key.
- Now go to the menu item "**Mesh/Manual mesh**" and combine the nodes, in groups of three, to triangular elements. In this way, you create a coarse structure for your system. Alternatively, you may have the program do this work for you, using the menu item "**Mesh/Automatic**".
- If you would like to edit the positions of mesh nodes, go to the menu items "**Mesh/Change**", "**Mesh/Move**" or "**Mesh/Edit**".
- If, for horizontal-plane systems, you would like to edit the positions of top and base of aquifer, use one of the menu items "**z/Individual depths**", "**z/Modify**" or "**z/In section**". Using "**z/Layer boundary contours**", you can get an overview of the positions of the top and base of the aquifer.
- If you would like to delete a triangular element, go to the menu item "**Mesh/Manual mesh**" again and click on the corner nodes of the appropriate element. Using this menu item, try double-clicking in a triangular element.
- The screen contents can be refreshed at any time using [**ESC**] or [**F2**].

- Create a fine structure for your system using the menu items "**Mesh/Refine individually**", "**Mesh/Section**" or "**Mesh/All**".
- Even after a mesh refinement, you can alter the system as you wish using "**Mesh/Define nodes**", "**Mesh/Manual mesh**", etc.
- For demonstration purposes, create one or more acute, and therefore numerically unfavourable, triangular elements using the menu item "**Mesh/Move**". Then go to the menu item "**Mesh/Optimise**" and follow the effects on the screen.
- Define the governing boundary conditions for the system using, e.g., the menu item "**Boundary/Individual potentials**".
- Edit, if wished, the soil numbers using the menu item "**Boundary/Individual soils**".
- Edit, if necessary, the soil properties using the menu item "**System/Soil properties**".
- When you have completed mesh generation, go to the menu item "**System/Analyse**" and begin the analysis. Before analysis starts the program will, if necessary, automatically carry out a bandwidth optimisation, in order to achieve a numerically favourably configured equation system.
- After analysis is complete you can, if wished, have the results printed as an output table, saved to a file or presented in a window (menu item "**File/Print output table**"). In general, however, this kind of result presentation will not very be satisfactory for the client.
- For this reason, you should immediately go to the "**Evaluation**" menu. The menu item "**Evaluation/Coloured**" is especially impressive or, for horizontal-plane systems, the menu item "**Evaluation/3D array**". The dialog boxes which follow can almost all be exited via the "**OK**" button, without further changes having to be made. The program usually makes sensible suggestions. Only the "**Determine extreme values …**" button should be clicked once, otherwise an error message will appear, with a correction note.
- If you have a colour printer installed you can point to the menu item "**File/Print and export**" and then the "**Printer**" button in the following dialog box, in order to get colour output to the printer. Grey scale will be used for ***black and white*** printers.
- Experiment with the example datasets.
- If you don't like manuals, you can get help using the menu item "**?/Help**" or the [**F1**] function key.

This short description demonstrates that only a few menu items must be selected in order to analyse a groundwater system. All further menu items are mainly for data saving, layout and any further evaluation of the analysis. A description is given in the following chapters.

6 Theoretical principles

6.1 General

An analytical solution is only possible for simple groundwater systems. When analysing 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-SS-FLOW2D** 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 on the **GGU-SS-FLOW2D** program 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 FEM mesh in these areas.

6.2 Horizontal-plane system

The program solves the differential equation:

$$T_x \cdot \partial^2 h/\partial x^2 + T_y \cdot \partial^2 h/\partial y^2 + Q = 0$$

Or, for a ***leaky aquifer***

$$T_x \cdot \partial^2 h/\partial x^2 + T_y \cdot \partial^2 h/\partial y^2 + k_v / d_v \cdot (H - h) + Q = 0$$

Where,

T_x, T_y	= Transmissibility in e.g. m²/s for x and y direction
h	= Piezometric head in e.g. m AD
k_v	= Permeability of blanket layer in e.g. m/s
d	= Thickness of blanket layer in e.g. m
Q	= Discharge in e.g. m³/s
x, y	= Coordinates e.g. x and y in m AD
H	= Water level above blanket layer in e.g. m AD

The transmissibilities result from the product of the coefficient of permeability, k_x or k_y, and the groundwater thickness in the aquifer d, in e.g. m. In the case of a confined aquifer, the value d is known in advance. In the case of a non-confined aquifer, the transmissibilities are not immediately known as, depending upon the position of the groundwater table, different levels will occur at different positions (see Figure 1). In this case an iteration is necessary to analyse each system, in order to determine the position of the phreatic line. This iteration process is carried out by the program automatically.

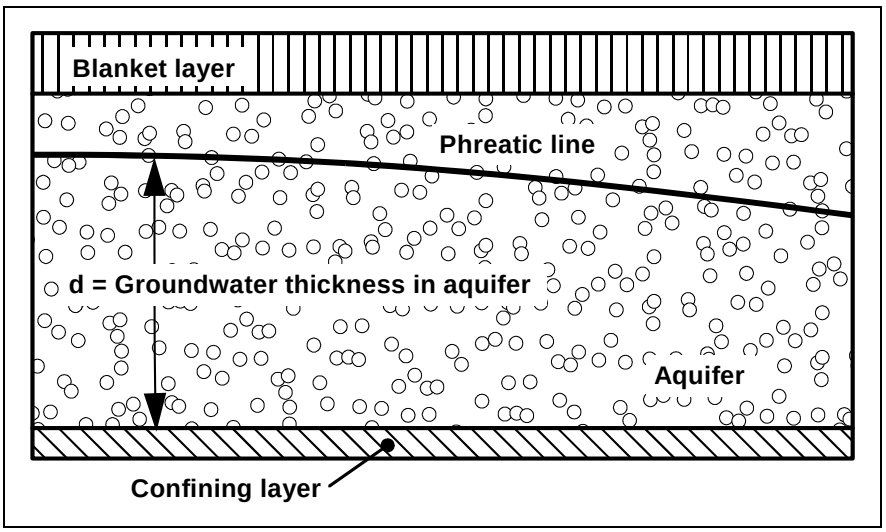


Figure 1 Groundwater thickness in aquifer

For the iterative determination of the groundwater thickness in the aquifer, the program first uses an estimation of the phreatic line. In the current program version this is taken as the top of the aquifer. With the resulting transmissibilities the program calculates, in the first iteration step, the piezometric head at all nodes. From these piezometric heads, the transmissibilities for the second iteration step are calculated. The process is carried on until the transmissibilities between iteration step (i) and (i - 1) fall below a user-defined deviation.

Note on "leaky aquifer":

The term ***leaky aquifer*** can be taken to mean ***semi-permeable*** aquifer. This means that water ingress into a lower groundwater system occurs through the blanket layer. The amount of water entering from ***above*** depends upon

- the water level difference between the water level above the blanket layer H and the water level in the aquifer h,
- the permeability of the blanket layer k_v,
- the thickness of the blanket layer d_v.

In the differential equation used for analysis (see above), this influence is taken into account in the term

$$\mathbf{k_v / d_v \cdot (H - h).}$$

This expression only creates an additional amount of water, any three-dimensional flow process cannot be modelled with this simple term. Investigations have shown that for permeability differences of

$$\mathbf{k / k_v > 10.0,}$$

between the aquifer k and the blanket layer k_v reliable results can be achieved. The **GGU-SS-FLOW2D** program reliably calculates even small values numerically, without questioning the physical sense of such input.

Note on boundary conditions:

The case of an impermeable boundary is automatically considered in 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.

6.3 Vertical-plane system and axis-symmetrical system

The program solves the differential equation:

$$\mathbf{k_r \cdot k_x \cdot \partial^2 h/\partial x^2 + k_r \cdot k_y \cdot \partial^2 h/\partial y^2 + Q = 0}$$

Where

k_x, k_y = Permeability in, e.g., m/s for x and y direction

h = Piezometric head in e.g. m

k_r = Coefficient for determination of permeability in the unsaturated zone (dimensionless)

Q = Discharge in e.g. m^3/s

x, y = Coordinates in e.g. m

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 y and the unit weight of water γ_w.

$$\mathbf{u = (h - y) \cdot \gamma_w}$$

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.

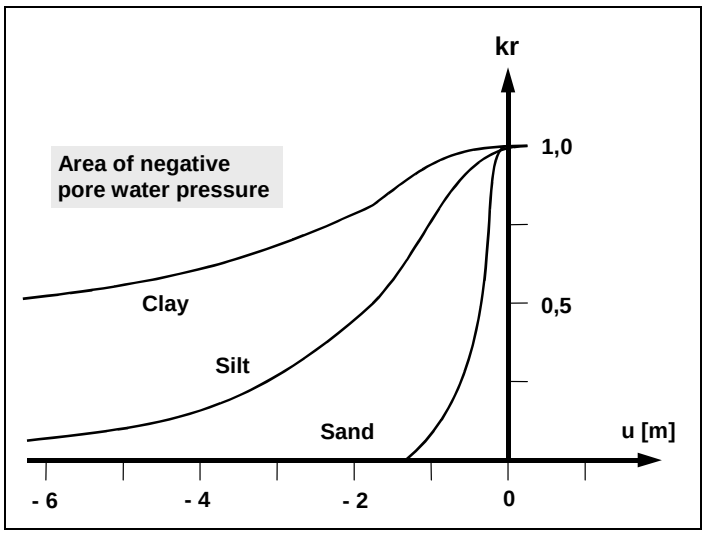


Figure 2 Function kr = f(u)

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

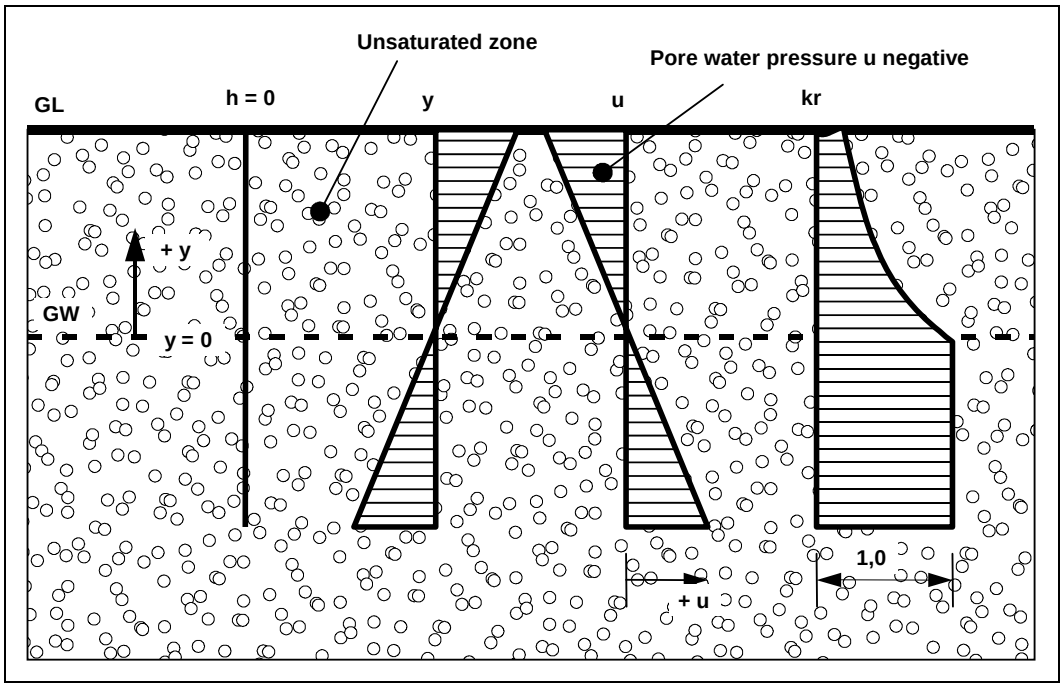


Figure 3 Unsaturated zone

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:

$$\mathbf{k_r = 1.0.}$$

After the initial calculation of the piezometric heads at the system nodes, the pore water pressures u are calculated and k_r determined. With the altered permeabilities, a new calculation of the system follows, with new piezometric heads and correspondingly new pore water pressures and k_r 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).

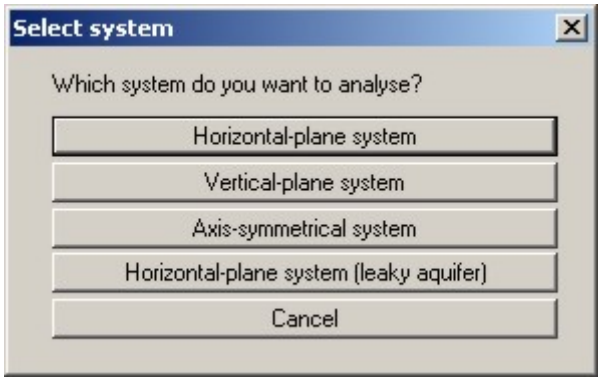
Note on boundary conditions:
The case of an impermeable boundary is automatically taken in to account by the finite-element-method. Valid is, that all system boundaries or partial system boundaries which have no water level or source boundary conditions, are automatically impermeable.

7 Description of menu items

7.1 File menu

7.1.1 "New" menu item

After pointing at the "**New**" menu item the subsequent dialog box allows you to define the type of groundwater system to be created.



If you have worked on a previous system you can, if desired, utilise the existing mesh, after confirming a query.

7.1.2 "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.

7.1.3 "Save" menu item

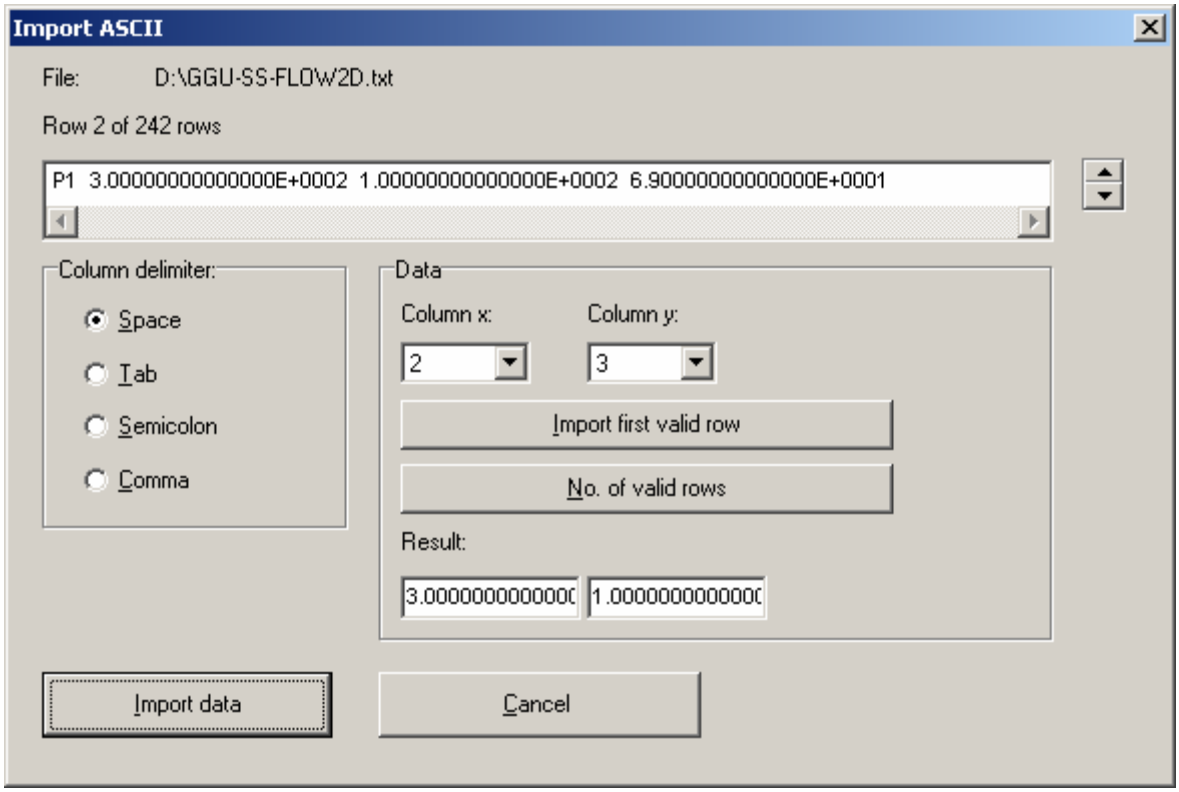
You can save data entered or edited during program use to a file, in order to have them available at a later date, or to archive them. The data is saved without prompting with the name of the current file. Loading again later creates exactly the same presentation as was present at the time of saving.

7.1.4 "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 "**.fen**" 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, "**.fen**" will be used automatically.

7.1.5 "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.



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.

7.1.6 "Save ASCII file" menu item

If an FEM mesh has been generated, the node coordinates can be saved to an ASCII file, allowing them to be imported into other programs where required.

If you are working with a vertical-plane system and the system has already been analysed, an ASCII record can be exported especially for the GGU's **GGU-STABILITY** slope stability application. This contains the calculated potentials, beside the node coordinates. These data can be directly utilised in **GGU-STABILITY** as a pore water pressure mesh.

7.1.7 "Export (GGU-3D-SSFLOW)" menu item

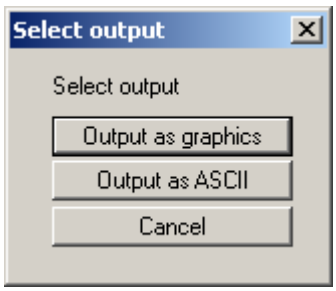
Using this menu item it is possible to export a previously analysed horizontal-plane system to the GGU's groundwater modelling program **GGU-3D-SSFLOW** for further processing. The **GGU-3D-SSFLOW** application allows analysis of groundwater flow using finite elements in three-dimensional systems.

7.1.8 "Print output table" menu item

7.1.8.1 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.

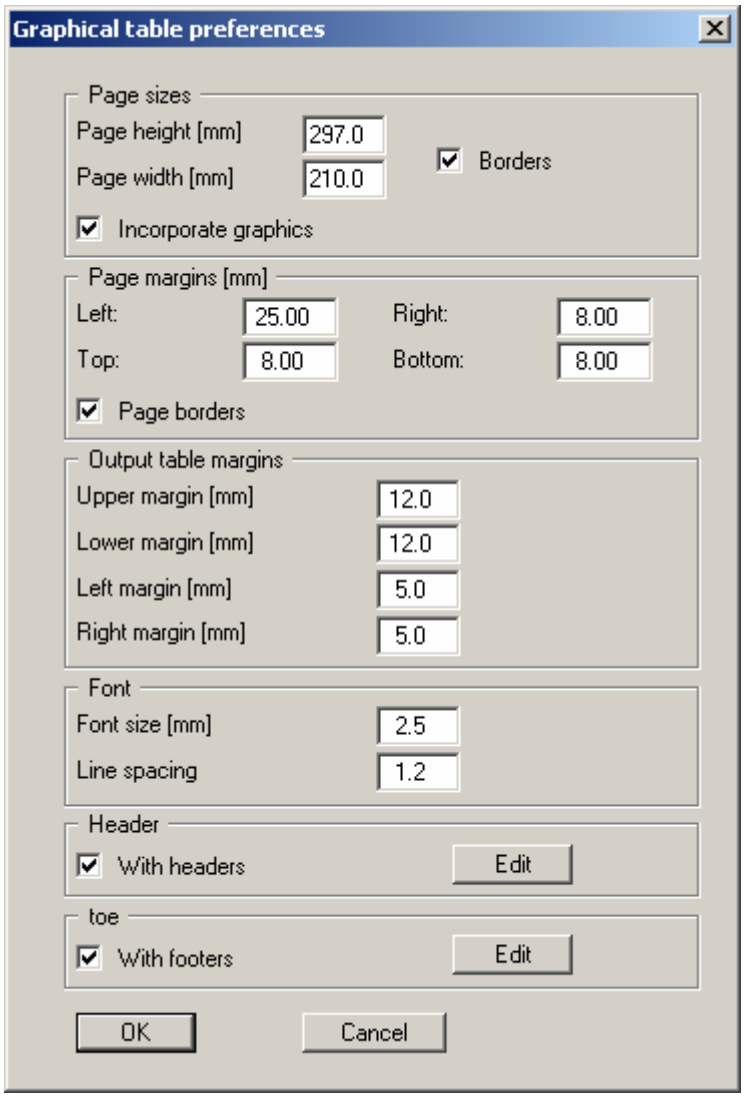
You have the option of designing and printing the output table as an annex to your report within the **GGU-SS-FLOW2D** program. To do this, select "**Output as graphics**" from the following options.



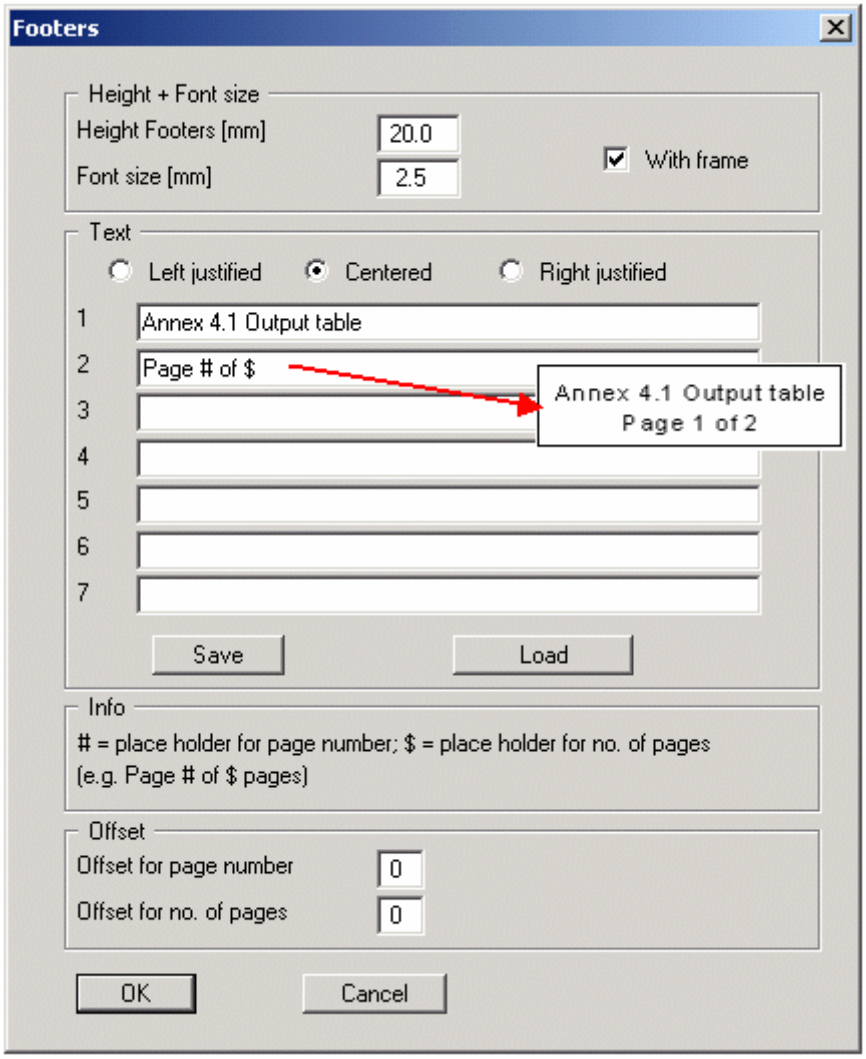
If you prefer to easily print or process the data in a different application, you can send them directly to the printer or save them to a file using the "**Output as ASCII**" button.

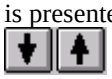
7.1.8.2 Button "Output as graphics"

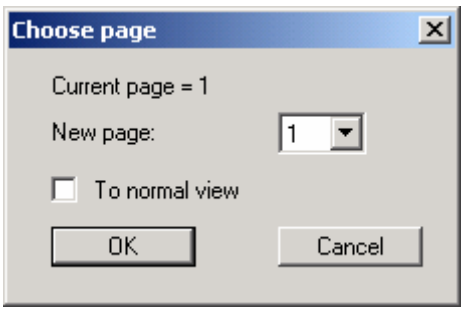
If you selected the "**Output as graphics**" button in the previous dialog box a further dialog box opens, in which you can define further preferences for result visualisation.



You can define the desired layout for the output table in various areas of the dialog box. By activating the "**Incorporate graphics**" button, a sketch of the system is integrated in the output table. 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.

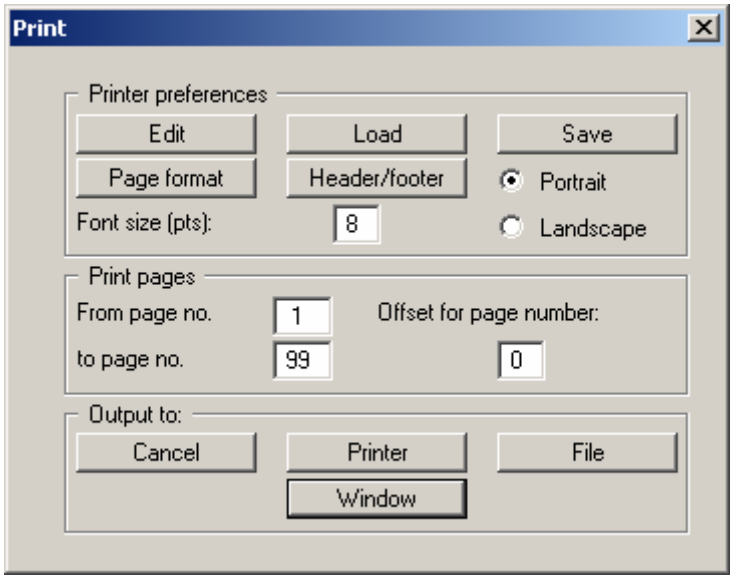


Automatic pagination can also be employed here if you work with the placeholders as described. After exiting the dialog boxes using "**OK**" you will see a further dialog box in which you can select the parameters to be used in the output table. If you click the "**Start**" button the output table is 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:



7.1.8.3 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. 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-SS-FLOW2D.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.

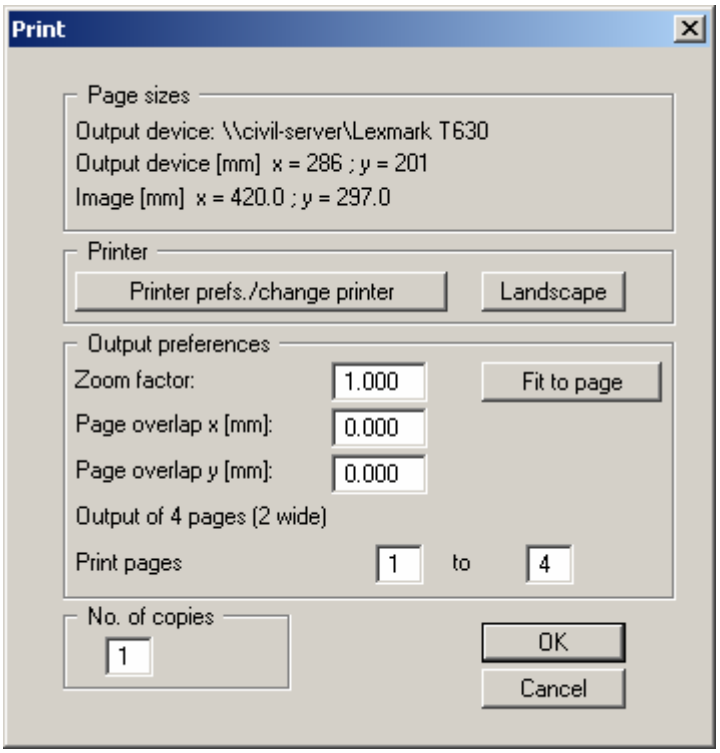
7.1.9 "Printer preferences" menu item

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

7.1.10 "Print and export" menu item

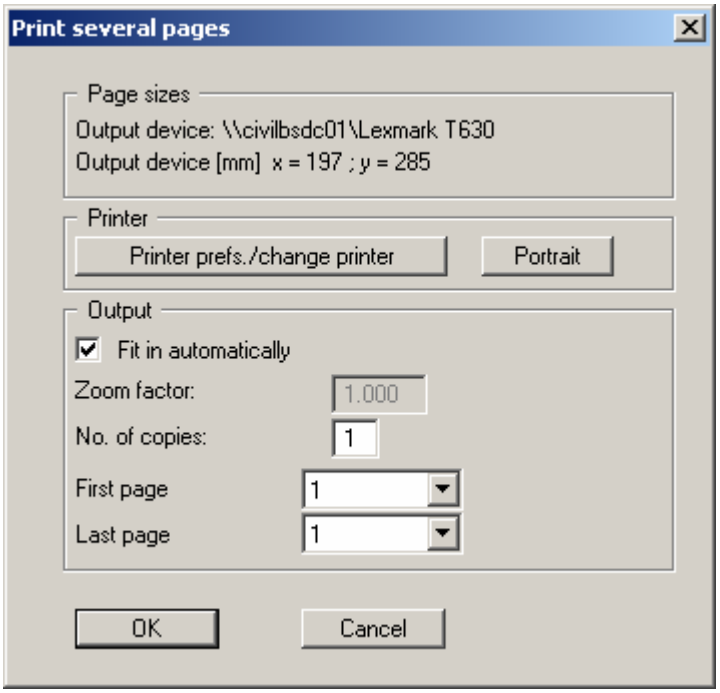
You can select your output format in a dialog box. You have the following possibilities:

- "**Printer**"
  allows graphic output of the current screen contents (***graphical representation***) to the WINDOWS default 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 ***tabular 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.

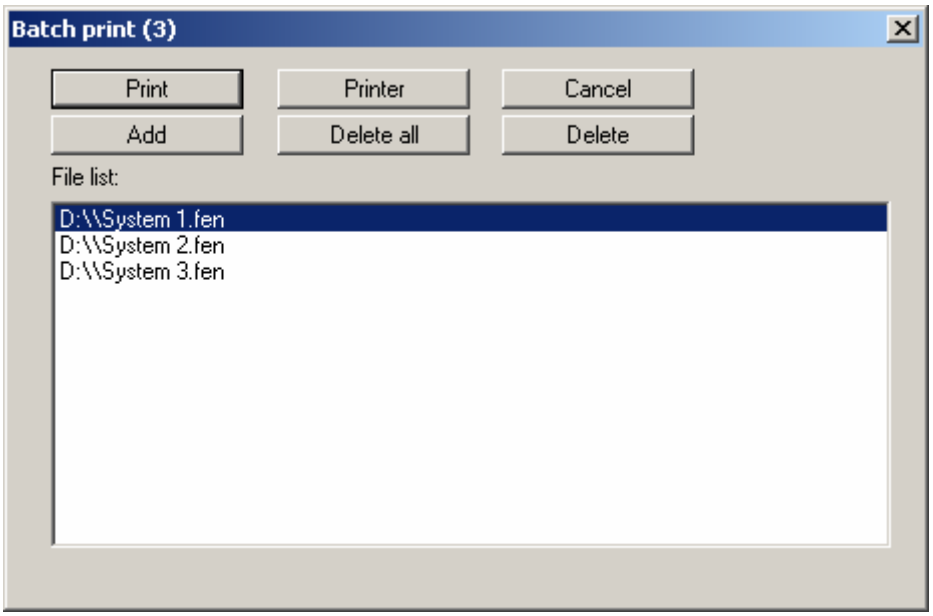
If you select the "**Copy/print area**" tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see "**Tips and tricks**", Section 8.3).

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.

7.1.11 "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.

7.1.12 "Exit" menu item

After a confirmation prompt, you can quit the program.

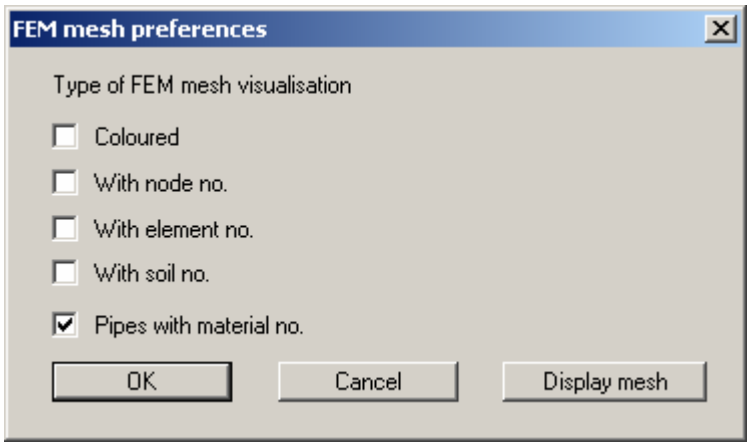
7.1.13 "1, 2, 3, 4" menu items

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.

7.2 Mesh menu

7.2.1 "Preferences" menu item

Using this menu item you can define the appearance of the FEM mesh on the screen. The element no. and the soil no. cannot be displayed simultaneously.



?

After leaving the dialog box via the "OK button the preferences will be adopted. The "**Display mesh**" button produces a direct representation of the FEM mesh using the selected preferences.

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

7.2.2 "Mesh" menu item

After going to this menu item the FEM mesh is displayed as defined in "**Mesh/Preferences**".

7.2.3 "Outline" menu item

After going to this menu item the outlines of the various soils used in the FEM mesh are displayed as defined in "**Mesh/Preferences**".

7.2.4 "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**]. If you are editing a horizontal-plane system, each new point created using this menu item will be assigned the default depths (see "**z/Default depths**" menu item, Section 7.3.2).

When setting new nodes, the current x- and y-coordinates of the mouse are shown in the status bar at the bottom of the window. If you have access to a scanner you can scan in the system to be processed and save it e.g. 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.

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 a check box with the option of locking on to **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 on to this point precisely; if a number of points are located within the rectangle, it will lock on to the one nearest the centre of the cross-hairs.

7.2.5 "Change" menu item

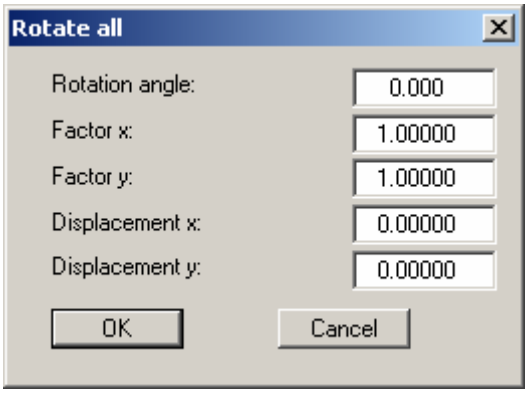
The coordinates of existing nodes can be edited. Three options are available for this:

- "**Via a table**"
  You can edit the coordinates of existing nodes in a dialog box 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**". If you are editing a horizontal-plane system, each new point created using this menu item will be assigned the default depths (see "**z/Default depths**" menu item, Section 7.3.2). If you set the number of nodes to **0** the FEM incidence table is deleted.

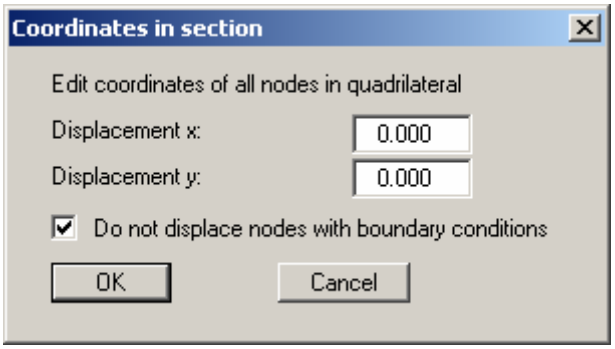
  It is even easier to import node coordinates via the Windows clipboard. For example, if the x-/y-coordinates of the FEM mesh nodes are available in an Excel table, it is possible to copy the two columns containing the data into the clipboard ("*Edit/Copy*") and then to paste them into the dialog box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



  The coordinates can be edited differing for x- and y-coordinates via the equation "**value(new) = factor · value(old) + displacement**". A rotation of the complete system can also be achieved using the entered rotation angle. If an interpolation mesh already exists an additional box containing the two "**FEM mesh**" and "**Interpolation mesh**" check boxes appears in the dialog box, allowing the required mesh to be activated.

- "**In section**"
  The coordinates for a selected area can be displaced by a fixed amount in the x- and the y-direction. Click 4 points counter-clockwise using the mouse. The required displacement can then be entered in the opened dialog box.



  If the "**Do not displace nodes with boundary conditions**" check box is activated these points will be excluded from any displacement.

7.2.6 "Move" menu item

The defined FEM system is displayed with the FEM elements after selecting this function. The nodes can be moved with the left mouse button pressed. The coordinates of the current node are displayed in the status bar. The last node movement can be undone using the [**Backspace**] key.

7.2.7 "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.

7.2.8 "Array" menu item

7.2.8.1 Select type of array

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

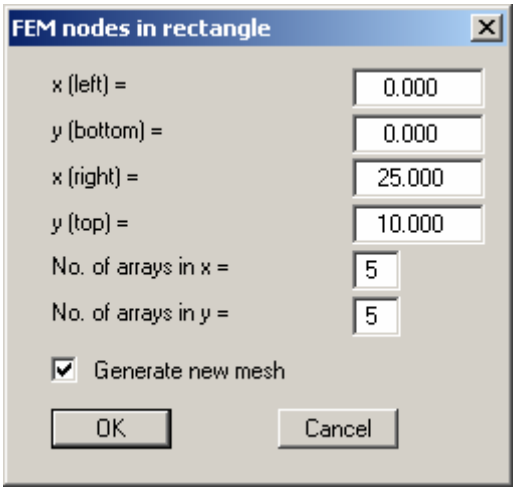
If you are editing a horizontal-plane system, each new point created with this menu item will be assigned the default depths (see "**z/Default depths**" menu item, Section 7.3.2).

7.2.8.2 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.

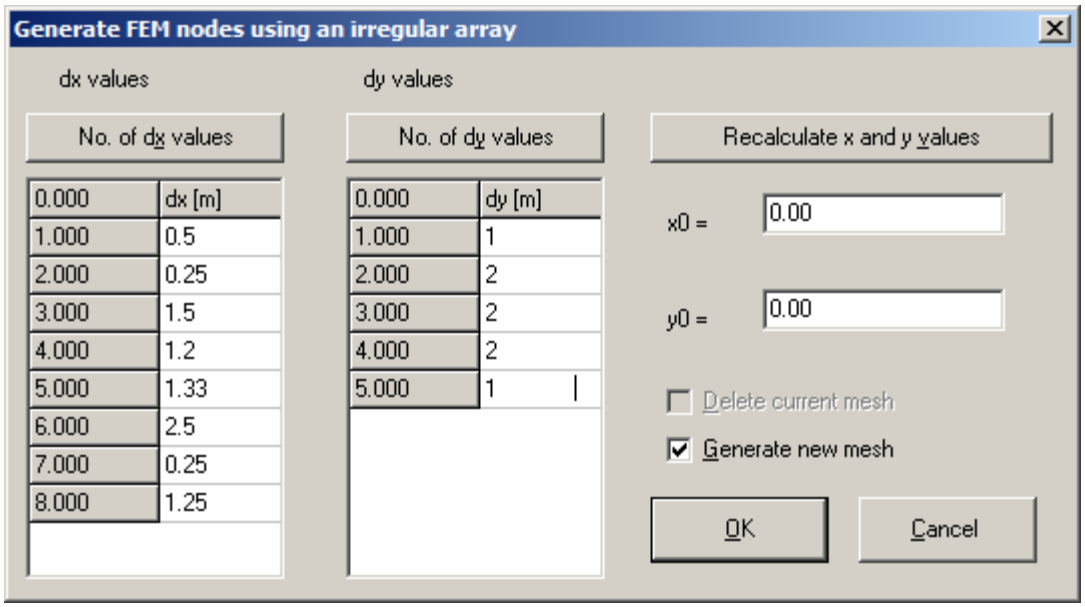
The procedure is similar for all three cases. Therefore, only the rectangles will be described.



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.

7.2.8.3 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.

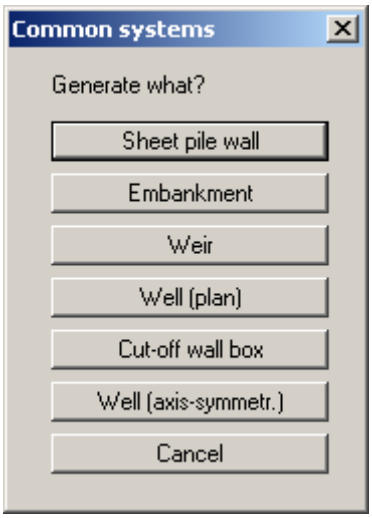


The array numbers can be defined using the "**No. of dx values**" and "**No. of dy values**" buttons. Enter the array spacing in "**dx**" and "**dy**". If you press the "**Recalculate x and y values**" button the first column to the left of "**dx**" and "**dy**" are recalculated. These are absolute values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

The program can generate an FEM mesh from the newly generated FEM nodes if the "**Generate new mesh**" check box is activated. If a mesh already exists, it can be deleted prior to generating the new one by activating the "**Delete current mesh**" check box.

7.2.9 "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.

7.2.10 "Manual mesh" menu item

After input of the mesh nodes this menu item is used to define the FEM mesh. 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. This menu item can also be reached using [**F4**].

An FEM element can be deleted by selecting the three corresponding nodes once again using the left mouse button.

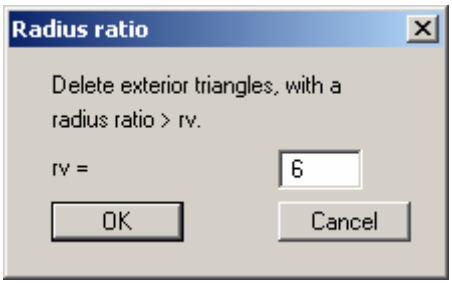
7.2.11 "Automatic" menu item

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 are assigned the soil number **1**.

7.2.12 "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 program first draws the maximum radius ratio of the most unfavourable triangle and displays the data in a message box. You then see a dialogue box, allowing you to define a maximum radius ratio above which all triangles with greater values are deleted.



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). In the example above, all external triangles with a radius ratio greater than 6 will be removed.

> In order to avoid ***interpolation holes*** in the triangle system only triangles at the boundaries are deleted.

7.2.13 "Delete" menu item

This menu item allows you to 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.

7.2.14 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:

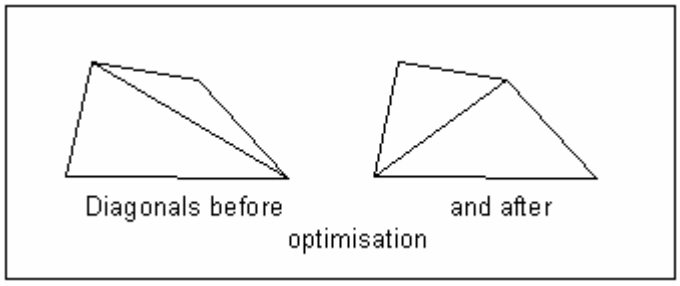


Figure 4 Optimisation of diagonals

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.

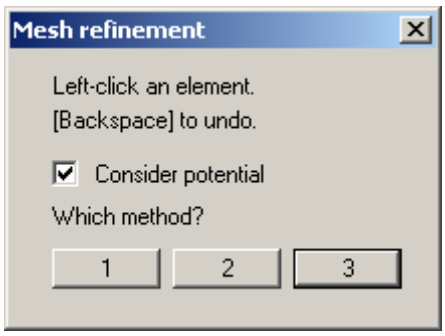
In horizontal-plane systems the problem arises that the values for aquifer base and aquifer top, as well as for top of blanket layer, are not adapted to the new location due to the displacement of individual nodes. If different values for aquifer base and aquifer top, as well as for top of blanket layer, exist in the system the program will issue a warning. In this case, activate the "**Adapt aq. base and aq. top**" or "**Adapt aq. base and aq. top and blanket top**"; the values are then adapted appropriately by the program using interpolation between the neighbouring nodes. It is possible to assign the precise values via an interpolation mesh following the optimisation (see Step 7 in Example 1, Section 9.1.8).

7.2.15 "Align" menu item

By clicking the left mouse button it is possible to select a node to be aligned depending on certain criteria (e.g. on a circle).

7.2.16 "Refine individually" menu item

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



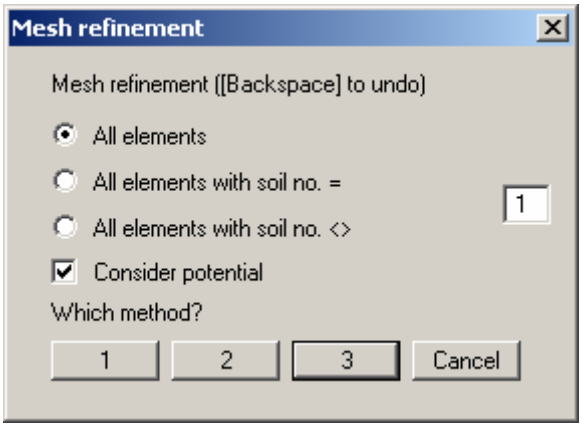
Upon activating the "**Consider potential**" check box, new nodes located immediately between two nodes with potential boundary conditions will be assigned the average of the two values. This procedure is not unequivocal when applied to ***source boundary conditions*** and can lead to misunderstandings. Source boundary conditions are therefore not refined in the course of mesh refinement. A description of the 3 refinement methods can be found in Section 7.2.19.

7.2.17 "Section" menu item

A number of elements previously enveloped in a polygon can be refined using this menu item or, alternatively, pressing [**F6**]. Potential boundary conditions can be taken into consideration (see "**Mesh/Refine individually**" menu item, Section 7.2.16). A description of the 3 refinement methods can be found in Section 7.2.19.

7.2.18 "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/Refine individually**" menu item, Section 7.2.16). A description of the 3 refinement methods can be found in Section 7.2.19.

7.2.19 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.

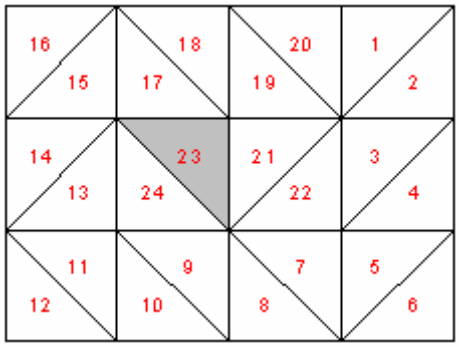


Figure 5 FEM refinement demonstration mesh

- "**Method 1**"
  An additional node is generated in the centroid of the selected triangle.

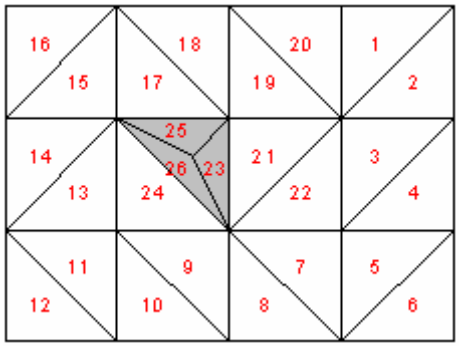


Figure 6 FEM mesh refinement using Method 1

- "**Method 2**"
  The selected triangle element and the neighbouring triangle element are halved.

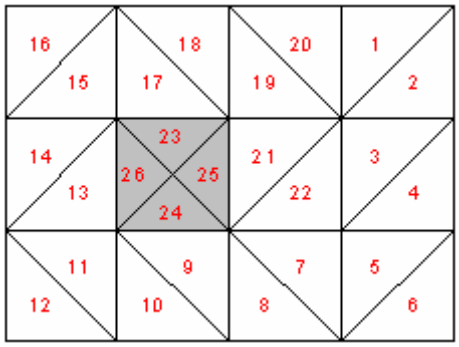


Figure 7 FEM mesh refinement using Method 2

- "**Method 3**"
  A new triangle element is inserted at the median of the clicked triangle element. The neighbouring triangle elements are halved.

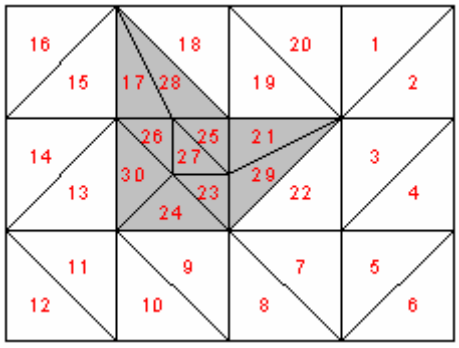


Figure 8 FEM mesh refinement using Method 3

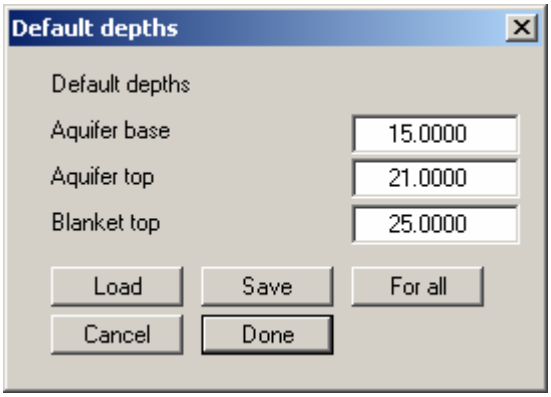
7.3 z menu (for horizontal-plane systems only)

7.3.1 General note

The "**z**" menu appears for horizontal plane systems only. It allows simple definition of the z-values of a system, i.e. the definition of the top of the aquifer, the base of the aquifer and, in addition, the top of the blanket for a "**leaky aquifer**".

7.3.2 "Default depths" menu item

The default depths defined in this dialog box are assigned to all new FEM nodes defined using the "**Mesh/Define nodes**" menu item. The additional entry "**Blanket top**" appears in the dialog box for a "**leaky aquifer**".



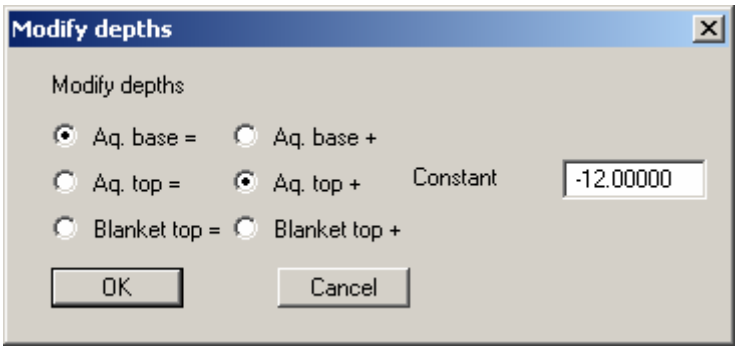
If you edit the given value and leave the dialog box using "**Done**" the newly defined depths will be adopted as of this moment for newly defined nodes. If nodes are already defined, they can also be retroactively assigned z values using the "**For all**" button.

7.3.3 "Individual depths" menu item

The z-values of nodes (top/bottom of aquifer, top of blanket layer) can be edited by double-clicking a node.

7.3.4 "Modify (depths)" menu item

You can modify all z-values as a function of the existing z-values. The following dialog box shows an example where the base of the aquifer lies at 12 m below the existing aquifer top at all nodes.

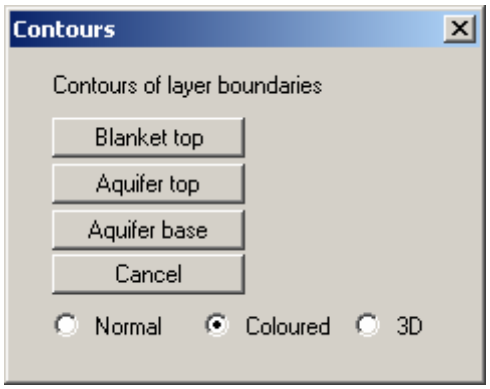


7.3.5 "In section" menu item

After defining a quadrilateral, all nodes within it can be assigned new z-values (top of aquifer, base of aquifer and, where required, top of blanket). If the "**Show depths**" check box is activated in the message box the nodes are labelled with the current depth. This provides a good overview of the possible depth ranges present.

7.3.6 "Layer boundary contours" menu item

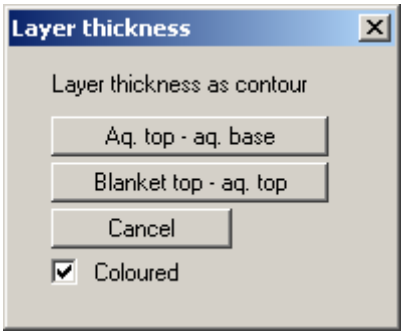
In order to clearly present the layer situation it is possible to generate a contour diagram of the top of the aquifer (aq. top) or the base of the aquifer (aq. base) or the top of the blanket (blanket top) for a ***leaky aquifer***.



Define the type of contour diagram using the "**Normal**", "**Coloured**" and "**3D**" option buttons. Details are described in Sections 7.8.1 to 7.8.3. Exit the dialog box by clicking the button for the layer for which the contour diagram is to be displayed.

7.3.7 "Layer thickness contours" menu item

You can create a contour diagram of the layer thicknesses.



If the "**Coloured**" check box is activated, colour filled contours are created.

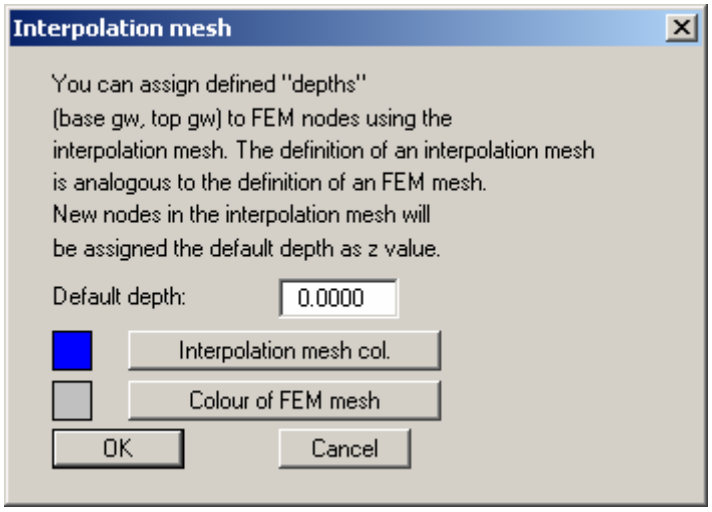
7.3.8 "Section" menu item

The layer boundaries for a given section can be displayed using this menu item. Left-click on system nodes to define a section. The layer boundaries are displayed after pressing [**Return**].

If you have evaluated a section for an analysed system using the "**Evaluation/Values in node section**" menu item and then saved the section, this section can be reloaded and utilised for layer boundary representation.

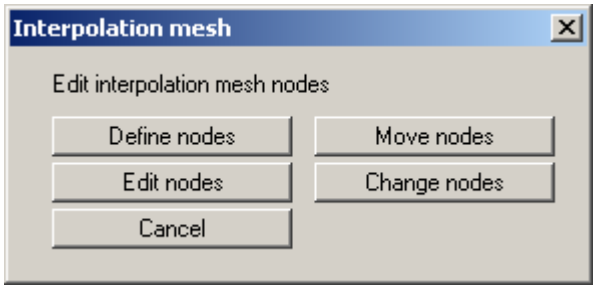
7.3.9 "Interpolation mesh" menu item

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 these z values to the FEM mesh as the base or top of the aquifer (or, for a "**leaky aquifer**", as blanket layer top). 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. In the dialog box yoou can define the colour of visualisation for interpolationa and FEM mesh using the corresponding buttons.



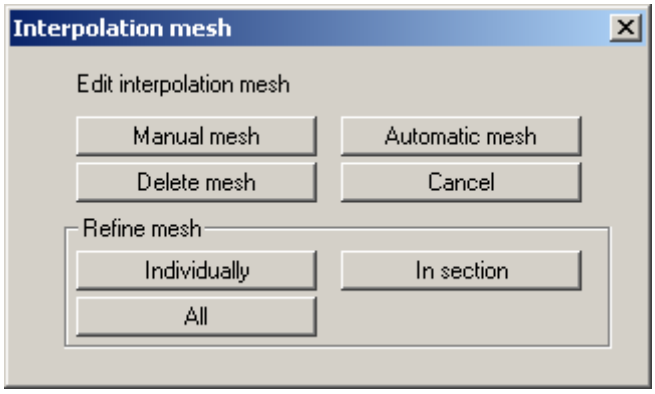
7.3.10 "Nodes" menu item

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



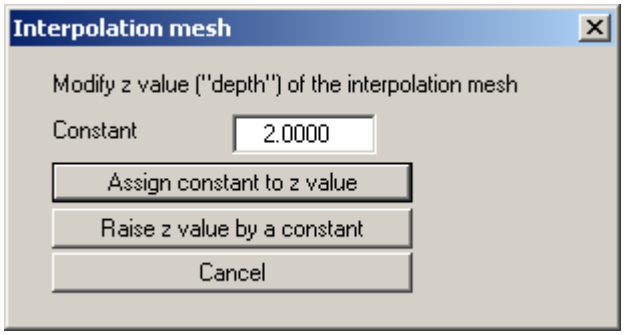
7.3.11 "Mesh" menu item

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



7.3.12 "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:



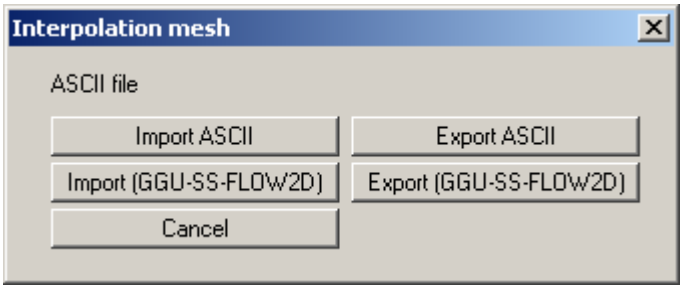
Once a constant has been entered the two options shown in the dialog box are available for modifying the z-values defined by the quadrilateral.

7.3.13 "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 7.8.2 and 7.8.3.

7.3.14 "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 7.1.5 opens for the "**Import ASCII**" option.



If the coordinates are exported using the **GGU-SS-FLOW2D** format, the information regarding triangles (incidence table) and z values will also be exported and can be imported again.

7.3.15 "Assign" menu item

After completely processing the interpolation mesh the z-values associated with the interpolation mesh can be assigned to the FEM mesh as the top of the aquifer, the base of the aquifer or the top of the blanket.

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

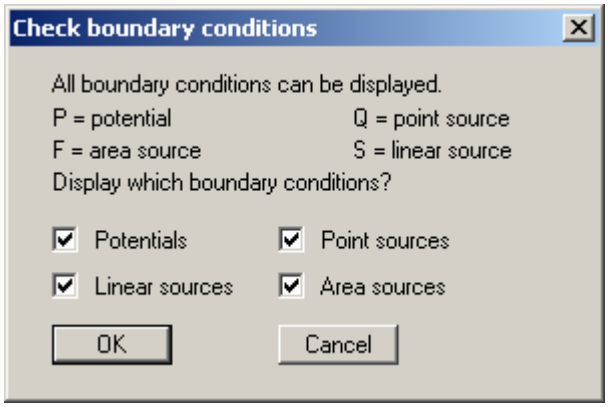
7.4 Boundary menu

7.4.1 "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.

7.4.2 "Check" menu item

This menu item provides a good overview of the locations of all boundary conditions by displaying the required boundary conditions on the screen.



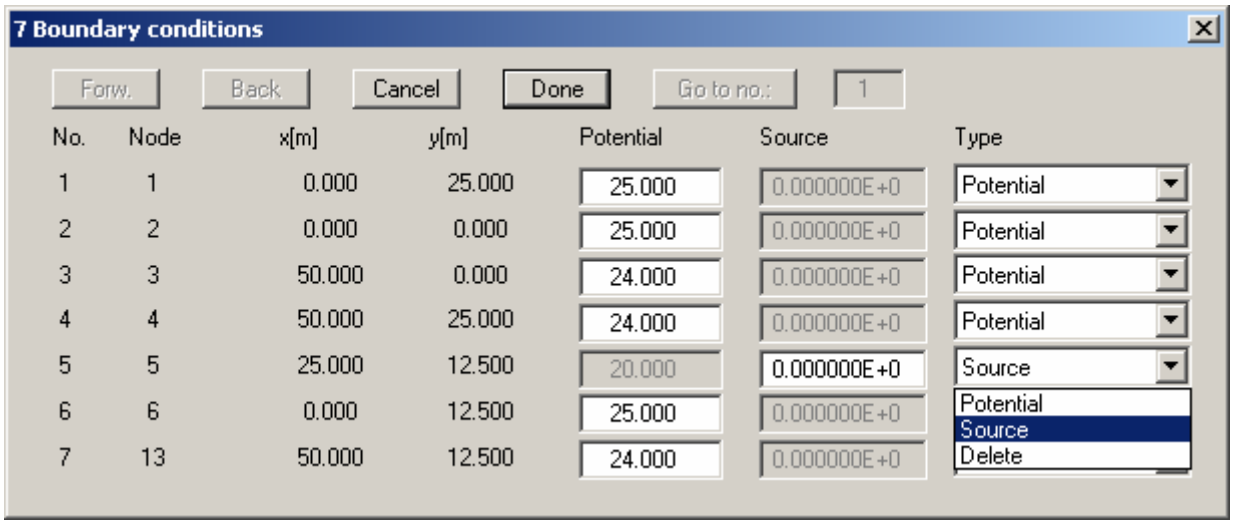
7.4.3 "Individual potentials" menu item

After going to this menu item you will first see the following dialog box:

After clicking the "**Graphically**" button the FEM mesh is displayed with any existing potentials. You can define a potential boundary condition at the FEM nodes using the left mouse button, or edit existing potentials. A right-click deletes existing potentials.

> **Note:**
> The piezometric head must be given when handling problems involving groundwater hydraulics. Die 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.

If potentials already exist at the individual FEM nodes, the "**Table**" button will also be available, leading to the following dialog box:



Here, the values of the potentials can be edited for several nodes simultaneously, potentials be converted to sources or potentials be deleted. The selected changes are carried out when you exit the dialog box using "**Done**".

7.4.4 "(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.

7.4.5 "(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 the [**Return**] key to then enter a start and an end potential.

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.

7.4.6 "Point sources" menu item

Using this menu items source boundary conditions are defined within the FEM system in analogy to the displacement boundary conditions. A source is positive if it brings water into the system.

ATTENTION:
When calculating ***axis-symmetrical*** groundwater flow these sources are multiplied internally with $2 \cdot \pi \cdot r$, so that no intervention by the user is needed in the running of the program.

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.

After clicking the "**Graphically**" button the FEM mesh is displayed with any existing sources. You can define a source boundary condition at the FEM nodes using the left mouse button, or edit existing sources. A right-click deletes existing sources.

After clicking the "**Table**" button the same dialog box opens as for the "**Boundary/Individual potentials**" menu item (see Section 7.4.3). You can edit the source values for various nodes, convert sources to potentials or delete sources.

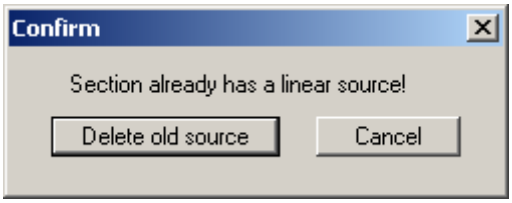
7.4.7 "Linear sources" menu item

General information on sources can be found in the menu item "**Boundary/Point sources**" (see Section 7.4.6).

The procedure for defining linear sources is explained in a dialog box. After clicking system nodes and confirming the selection with the [**Return**] key, a dialog box opens for defining the start and the end value. The intermediate boundaries will be assigned the appropriate value in a linear relationship to the point spacing.

If linear sources have already been defined an additional "**Delete all linear sources**" button appears in the info box after selecting this menu item again. If you exit the dialog box by using this button, all linear sources are deleted and a new linear source can be defined.

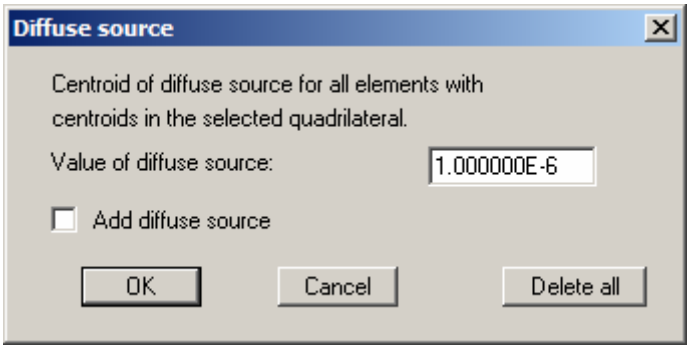
If you need to delete individual linear sources or sections of them, click the system nodes again, accept with the [**Return**] key and exit the start and end value dialog box by pressing "**OK**". The following dialog box now appears:



Click the "**Delete old source**" button until all sections of the marked linear source are deleted and then define the new linear source if required. If you exit the box by pressing "**Cancel**" before you are finished, the undeleted boundaries will remain in place.

7.4.8 "Diffuse sources" menu item

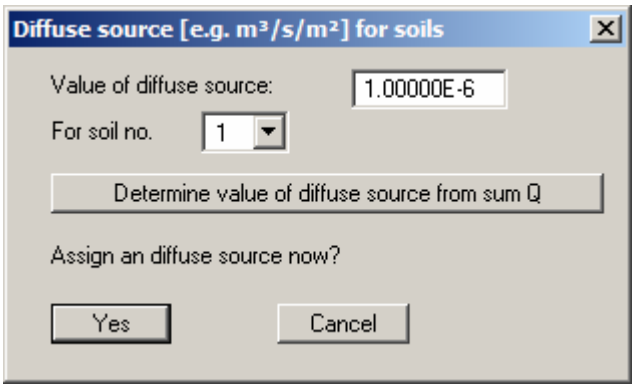
Certain elements can be assigned diffuse sourcesby defining the area required as a polygon using 4 counter-clockwise points. You will then see the following dialog box:



If you activate the "**Add diffuse source**" 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. If you click the "**Delete all**" button, all existing diffuse sources are deleted and you can define a new diffuse source after specifying a new polygon.

7.4.9 "Diffuse sources for soils" menu item

Individual soils can be defined as diffuse sources by assigning a discharge to the respective soil (e.g. $m^3/s/m^2$). Select the soil in the dialog box and enter the value for the diffuse source.



Alternatively, you can have the value of the diffuse source determined by the program. If you click the button "**Determine value of diffuse source from sum Q**" you must enter the discharge for the area of the selected soil in the new dialog box. The program then determines the value for the diffuse source from the data given. You can then accept this value

7.4.10 "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 7.5.3).

7.4.11 "(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 7.5.3).

7.4.12 "Define pipes" menu item

Left-click system nodes to define pipe elements which can then be assigned a material number after pressing the [**Return**] key. The defined pipes are shown with the appropriate material number in the graphics. The colour and width of the lines can be edited using the "**Graphics preferences/Pen colour and width**" menu item (see Section 7.6.3).

If you need to delete individual pipes or sections of them, click the system node again, accept with the [**Return**] key and exit the material number dialog box by pressing "**OK**". Click the "**Delete old pipe**" button until all sections of the marked pipe are deleted and then define the new pipe if required. If you exit the box by pressing "**Cancel**" before you are finished, the undeleted pipe sections will remain in place.

7.4.13 "Delete pipes" menu item

All defined pipes are deleted after a prompt.

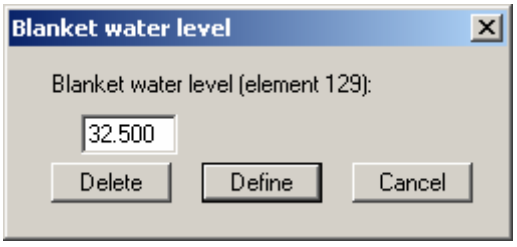
7.4.14 "Individual blankets" menu item (only for "leaky aquifer")

If you are analysing a horizontal-plane system with a "**leaky aquifer**", proceed as for a simple horizontal-plane system when generating the system. However, a ***water level above the blanket*** must be additionally assigned for elements below a blanket layer. If you forget to do this the system will be analysed as a horizontal-plane problem without a blanket. Regardless of this, the Aq. top entry (= top of aquifer) governs the analysis in terms of ***confined*** or ***unconfined***. If you do not assign a water above the blanket layer to individual elements, these elements are also dealt with as if the problem is a horizontal-plane without a blanket. This may well be the actual case in parts of the system and may therefore be very useful.

Note on water level above the blanket layer:
Although the designation ***Water level above the blanket layer*** has been used for this manual, this does not represent the value between the top of the blanket and the water level; instead, the ***absolute height*** is always meant.

For example, if the top of the blanket is at 31.08 m AD and the water level above the blanket is at 33.28 m AD, the water level above the blanket layer to be entered is 33.28 (m AD) and not the difference of 1.2 m between these two values.

The water level above the blanket is defined for each element. If you click a single element the following dialog box opens:

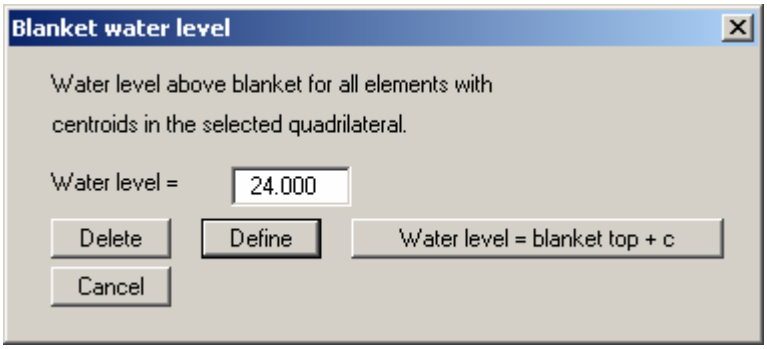


Enter the water level above the blanket and assign it to the selected element by clicking the "**Define**" button. Alternatively, you can also edit an existing water level or remove it completely using the "**Delete**" button.

7.4.15 "(Blankets) In section" menu item (only for "leaky aquifer")

General principles on the definition of blankets can be found in the "**Boundary/Individual blankets**" menu item (see Section 7.4.14).

After going to this menu item several FEM elements can be simultaneously assigned a blanket boundary condition in a defined section, or an existing boundary condition can be deleted. To do this, left-click four points counter-clockwise; a right mouse click resets the selected points. The definition refers to all nodes with their centroids within the quadrilateral. The following dialog box appears:



The "**Delete**" button deletes any existing water levels. The entered water level is assigned to the selected elements using the "**Define**" button. The "**Water level = blanket top + c**" button assigns the selected elements either a water level corresponding to the top of the blanket or the top of the blanket plus a constant c.

7.4.16 "(Blankets) Linear" menu item (only for "leaky aquifer")

General principles on the definition of blankets can be found in the "**Boundary/Individual blankets**" menu item (see Section 7.4.14).

Using this menu item it is possible to assign a linearly variable water level above the blanket to several elements. This can be useful for example if you need to model a ***semi-pervious*** water course where the bed has been subject to colmation, that is, it has a blanket layer and displays a variable water level within the system.

After going to this menu item you click the nodes of the FEM mesh to which a variable water level is to be assigned. Press the [**Return**] key to then enter a start and an end water level above the blanket.

The selected elements are assigned a water level proportional to the separation of the element centres, increasing from the start water level to the end water level.

7.5 System menu

7.5.1 "Info" menu item

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

7.5.2 "Project identification" menu item

You can enter a description of the current system; this will then be used in the general legend (see Section 7.6.9).

7.5.3 "Soil properties" menu item

You can edit the soil properties. A variety of dialog boxes can now appear depending on the selected basic system. The number of input rows corresponds to the soil numbers awarded using the menu items "**Boundary/Individual soils**" or "**Boundary/(Soils) In section**".

Horizontal-plane system

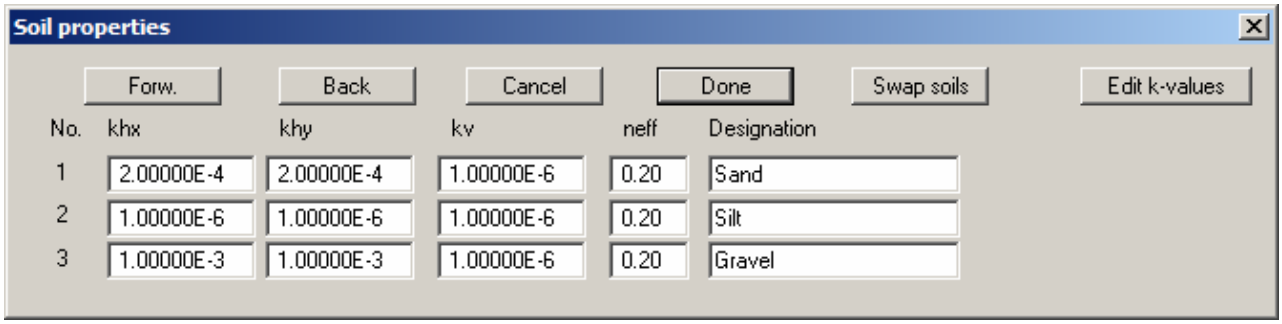
The Darcy permeabilities in *x*- and *y*-direction ("**khx**" and "**khy**") and the effective pore space ("**neff**") must be entered. The effective pore space is not incorporated in the basic differential equation. These values are only required for the subsequent evaluation of the data computed for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

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).

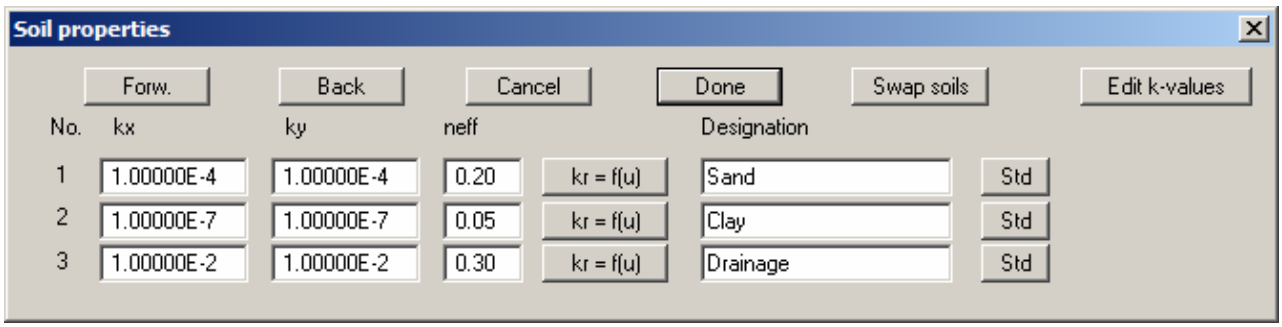
To avoid unnecessary explanations when submitting your analyses, the k-value is shown in the soil properties legend in "**m/second**". To achieve this, the program internally converts the entered coefficient of permeability from your selected unit back to m/s (see "**Info**" button).

System with "leaky aquifer"



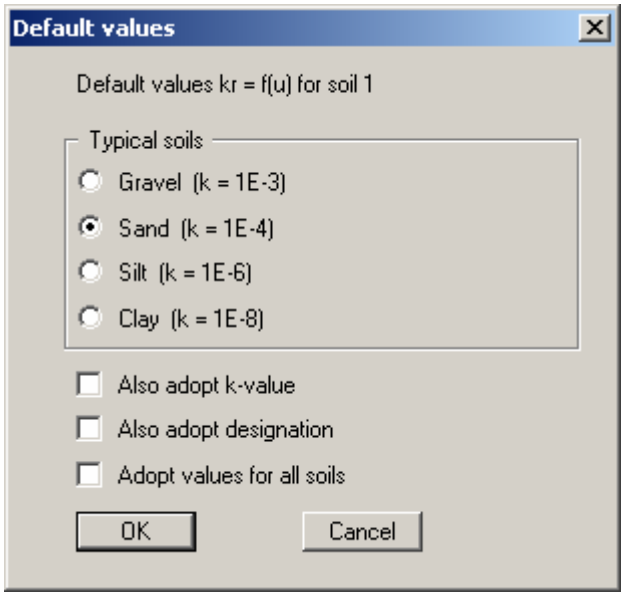
In addition, the vertical permeability of the blanket ("**kv**") is required.

Vertical-plane system and axis-symmetrical system



In addition to the Darcy permeability values "**khx**" and "**khy**" and the effective pore space "**neff**", the function "**kr = f(u)**" must be given. 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.

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.

7.5.4 "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.

7.5.5 "Analyse" menu item

7.5.5.1 Notes on bandwidth, units and equation solvers

After pointing to the "**System/Analyse**" 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.

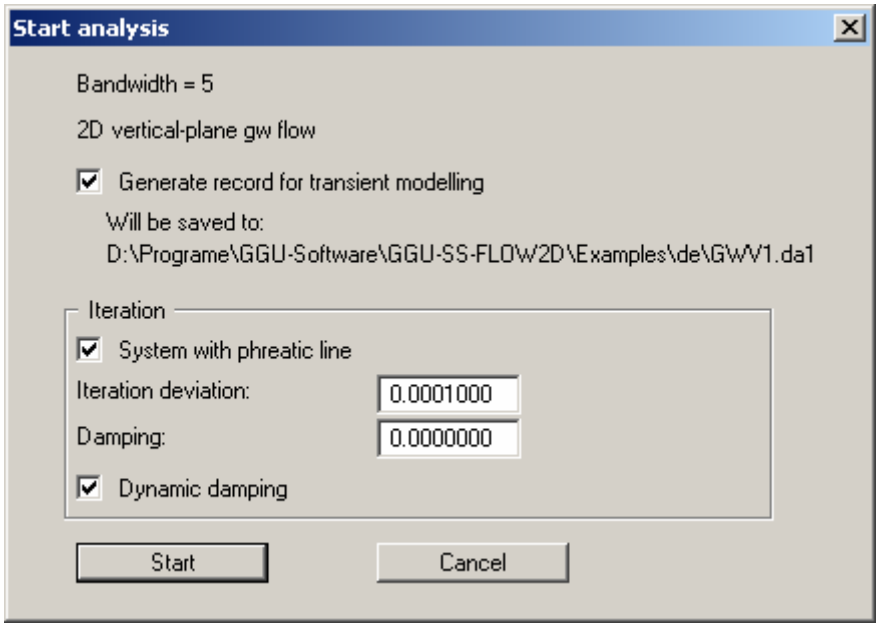
NOTE ON UNITS:
The program works true to your units. If you want to work with common length units (metres), it follows that all length input must then be made in metres. Beside the permeability and discharge, time is employed as an additional unit. Here too, only one unit of time *T*, e.g. seconds [*s*],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

Following the bandwidth optimisation a dialog box opens where "**Cholesky**" or "**Iterative**" can be selected as equation solver. Information on the differences between the two equation solvers can be read by pressing the "**Equation solver info**" button.

7.5.5.2 Start analysis

Once you have selected the equation solver start the analysis using "**OK**". Depending on the type of system employed slightly variable (shown below for a ***vertical-plane*** system) dialog boxes appear (horizontal-plane, vertical-plane, axis-symmetrical):



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

- **Generate record for transient modelling**
  All data and results are saved in a file for possible subsequent transient groundwater analysis (additional module **GGU-TRANSIENT**), contaminant transport analysis (additional module **GGU-CONTAM-FE/-RW**) or for subsequent graphical evaluation of the finite-element analysis. If this check box is activated the file name of the current file with the "**.da1**" extension is used by the program for the transient record. The file is always stored at the same level as the original steady-state file.
- **Iteration deviation**
  If unsaturated zones are present within the system and computation 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).

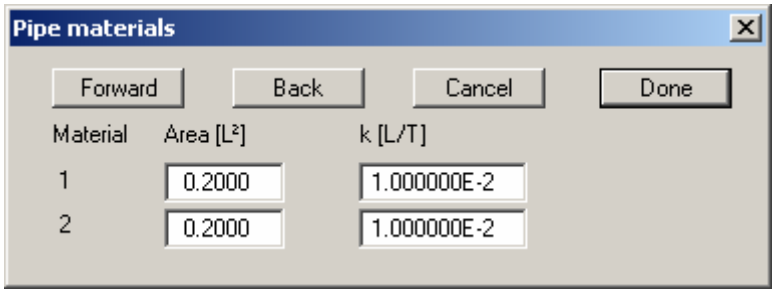
An additional button appears in the dialog box for ***vertical-plane*** or ***axis-symmetrical*** systems:

- **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.

7.5.6 "Pipe materials" menu item

You can edit the properties of any pipes defined. A dialog box opens containing a number of input rows dependent on the number of pipe materials specified in "**Boundary/Define pipes**" (Section 7.4.12).



7.5.7 "Undo" menu item

If you have carried out any changes to dialog boxes 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 7.6.7).

7.5.8 "Restore" menu item

When this menu item is selected the last change made in a dialog box 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 7.6.7).

7.5.9 "Preferences" menu item

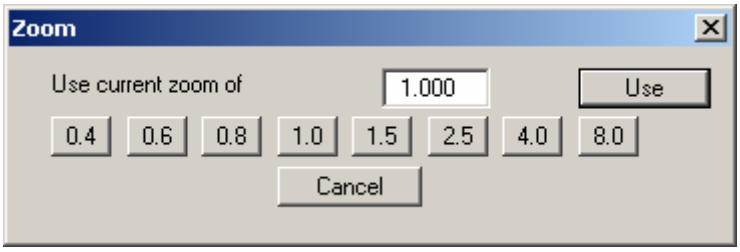
You can activate or deactivate the undo functions.

7.6 Graphics preferences menu

7.6.1 "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.

7.6.2 "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.

7.6.3 "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.

7.6.4 "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.

7.6.5 "Draw Mini-CAD elements 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 Version 7 of the integrated **Mini-CAD** module you can now 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.

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

Using these two menu items you can add free text to the graphics and add lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.). 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.

7.6.7 "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-SS-FLOW2D.alg**" file (see menu item "**Graphics preferences/Save graphics preferences**") and will be active at the next time the program is started.

By clicking on the tools (smart icons) for the menu items you can directly reach most of the program functions. The meaning of the smart icons appears as a text box if you hover with the mouse pointer over the tools. Some of the tool functions cannot 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 printout, 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 8.3).

 "**Undo move object**"
By clicking this icon the last performed movement of graphical elements made using the [**F11**] function key or the menu item "**Graphics preferences/Move objects**" can be undone.

 "**Restore move object**"
By clicking this symbol, the last object movement undo carried out using "**Undo move object**" can be restored.

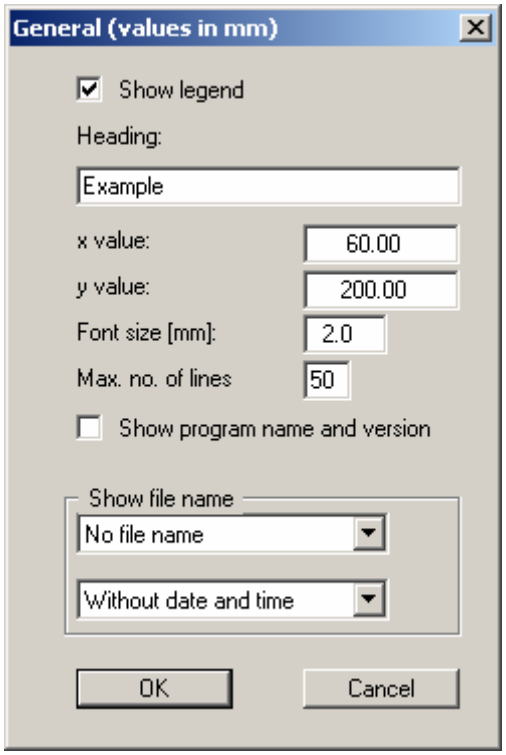
7.6.8 "3D toolbar" menu item

This menu item allows you to choose a pop-up window with tools for rotating and zooming three dimensional images.

The tools in this pop-up 3D window allow 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 (default: 45°) can be adjusted as wished using the  tool.

7.6.9 "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 velocity field you will see information on the velocities scale.

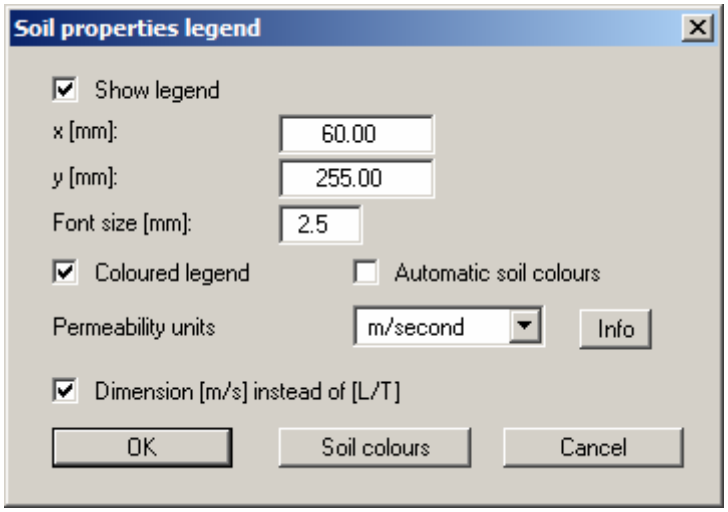
You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used.

> 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 7.5.2) will also be shown in the general legend.

7.6.10 "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.

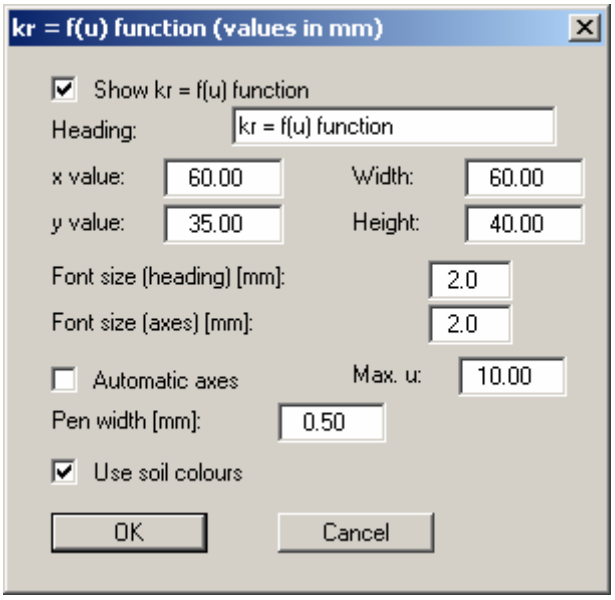
- "**Permeability units**"
  It often makes more sense in a transient analysis to enter the permeability in "**m/hour**" or "**m/day**". To avoid unnecessary explanations when submitting your analyses, the k-value is shown in the soil properties legend in "**m/second**". To achieve this, the program 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.

7.6.11 "Section legend" menu item

If you activate the "**Show section course**" 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 legend heading and font size can be modified.

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

If you activate the "**Show kr = f(u) function**" 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.

7.6.13 "Pipe legend" menu item

If you activate the "**Show legend**" check box in the dialog box for this menu item, a legend containing general information on any pipes is displayed on the screen. The position of the legend is defined by 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 with the left mouse button pressed.

7.6.14 "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.

7.6.15 "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-SS-FLOW2D.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**".

7.6.16 "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.

7.7 Page size + margins menu

7.7.1 "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.

7.7.2 "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.

7.7.3 "Manual resize (editor)" menu item

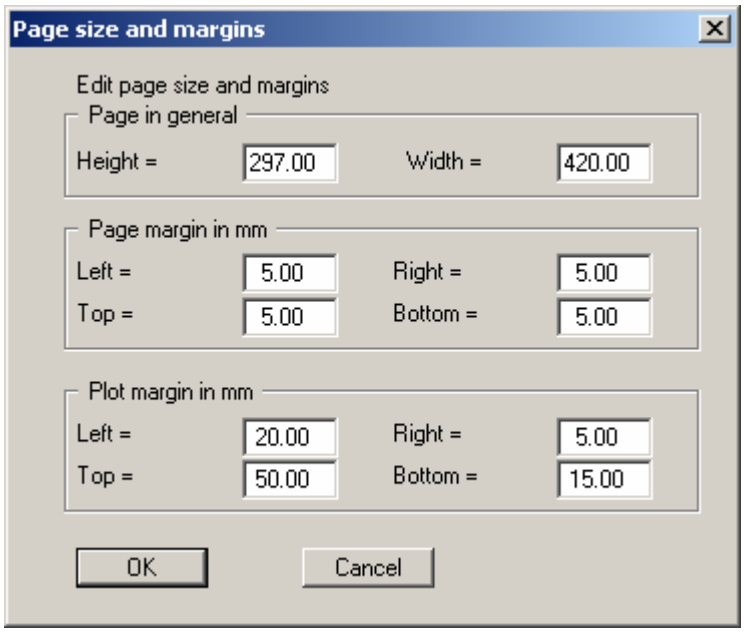
You can alter the image coordinates by direct numerical input in a dialog box. This allows precise scale input. The coordinates refer to the ***drawing area***. This can be defined in the "**Page size + margins/Page size and margins**" menu item by means of the plot margins (see Section 7.7.4).

The image coordinates entered here can be saved in a file with the extension "**.bxy**" and be reloaded 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.

7.7.4 "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.



- "**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 7.7.6).
- "**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 7.7.6).
- 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.

7.7.5 "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.

7.7.6 "Margins and borders" menu item

The program automatically draws thin cutting borders 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.

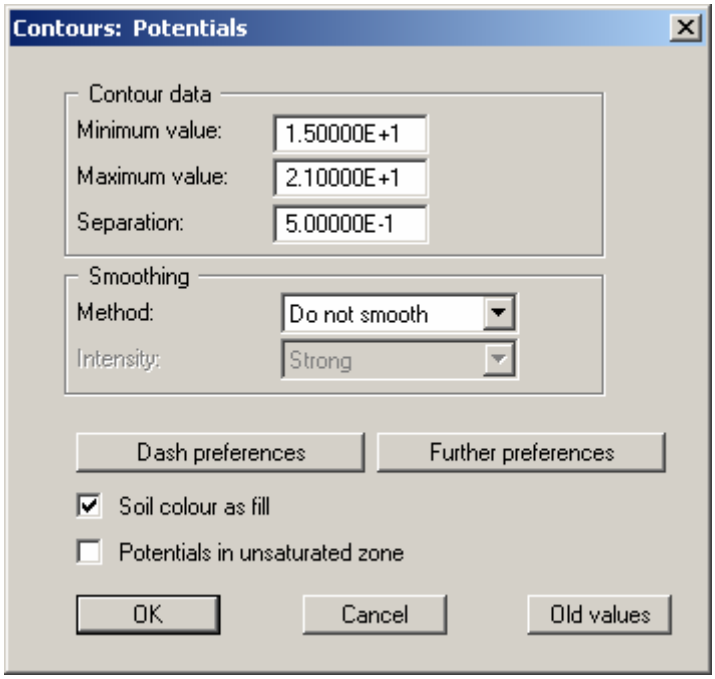
7.7.7 "Designation of potential" menu item

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

7.8 Evaluation menu

7.8.1 "Normal contours" menu item

Using this menu item it is possible to visualise the calculated potentials, velocities, gradients or the function kr for vertical-plane systems 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:

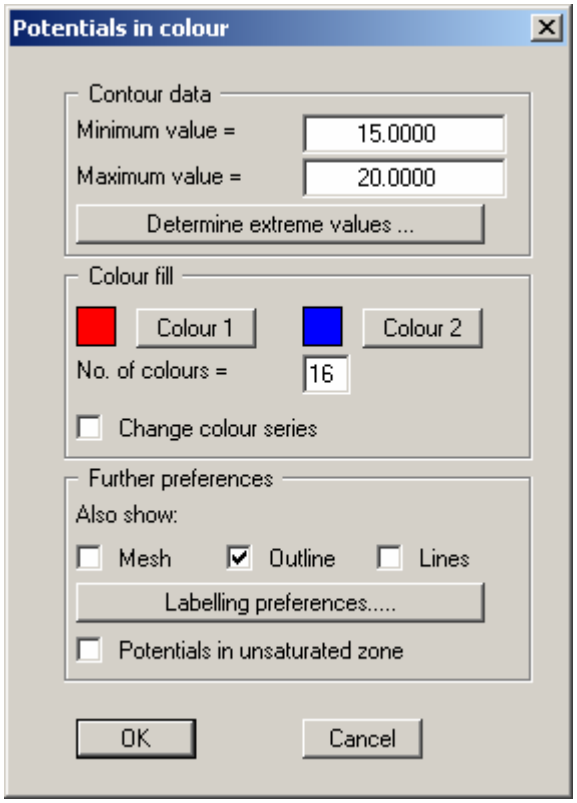
- "**Do not smooth**"
  As a linear method is implemented within the triangle elements, this will provide analysis results without smoothing performed by the program.
- "**Method 1**" and "**Method 2**"
  these smoothing out procedures employ two different Bezier splines. Smoothing out intensities can 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 "**Dash preferences**" and "**Further preferences**" buttons. By activating the appropriate check box the soil colour can be used as fill.

Because the program uses unsaturated permeabilities for its analysis of vertical-plane systems, potentials for the unsaturated zone are also acquired; physically, this is absolutely correct. Experience shows that this confuses some clients. In order to avoid lengthy discussions, simply switch off the potentials in the unsaturated zone using the appropriate check box.

7.8.2 "Coloured" menu item

Using this menu item it is possible to visualise the calculated potentials, velocities, gradients or the function kr for vertical-plane systems 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.

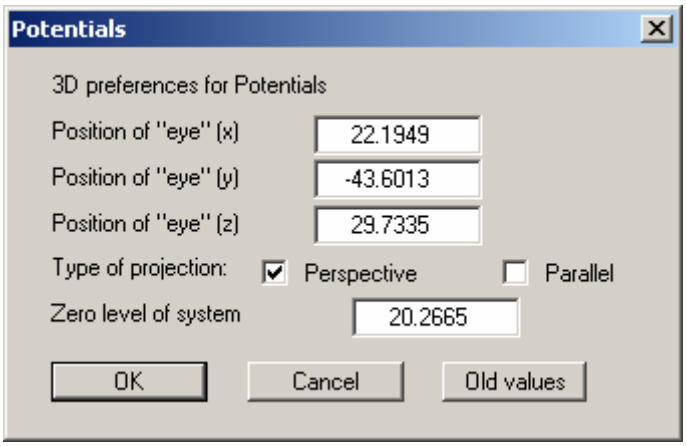
- "**OK**"
  The colours will be drawn after confirmation.

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 7.7.4).

> As the program calculates with unsaturated permeabilities for vertical-plane systems, 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.

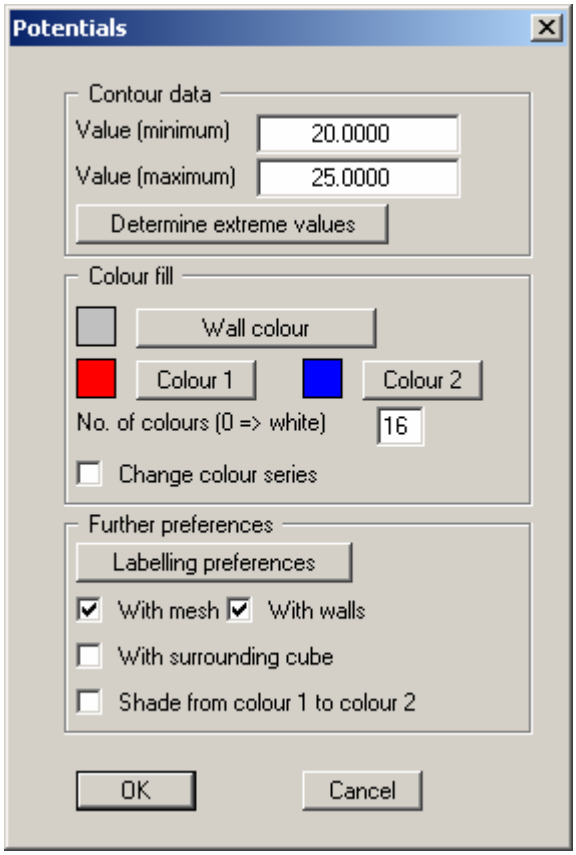
7.8.3 "3D" menu item

After selecting the required parameter the lines of equal state variables or the reinforcement parameters can be displayed in 3D for the existing nodes. 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.

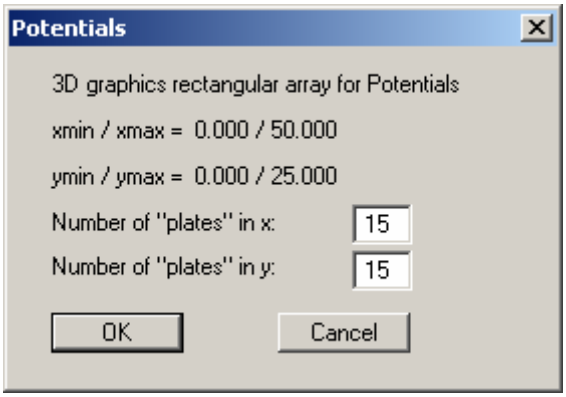
After leaving the dialog box by pressing "**OK**" you will see the following dialog box. It greatly resembles the box shown in the menu item "**Evaluation/Coloured**".



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.

7.8.4 "3D array" menu item

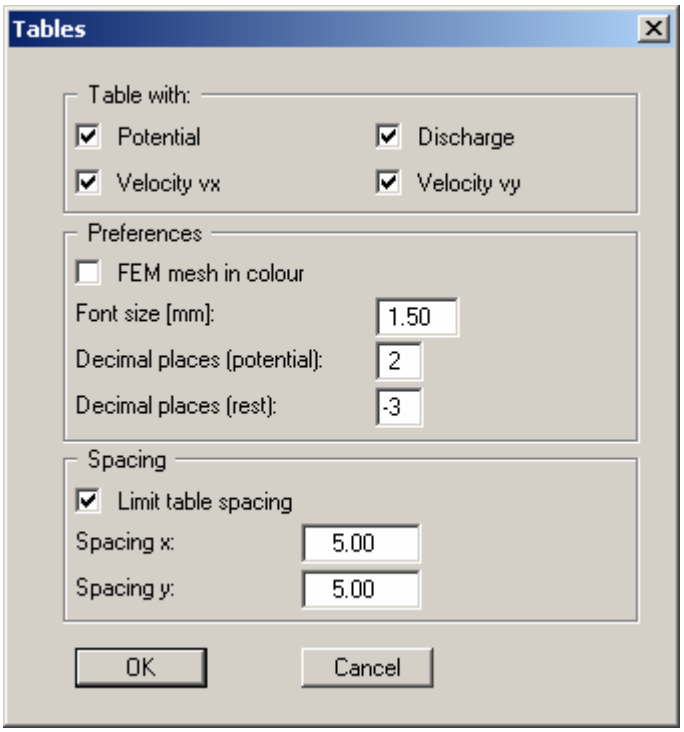
The 3D effect is usually lost in an irregular triangular mesh. The "**Evaluation/3D array**" menu item is useful here. After selecting the required parameter the following dialog box opens where you can define the array.



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 7.8.3).

7.8.5 "Tables" menu item

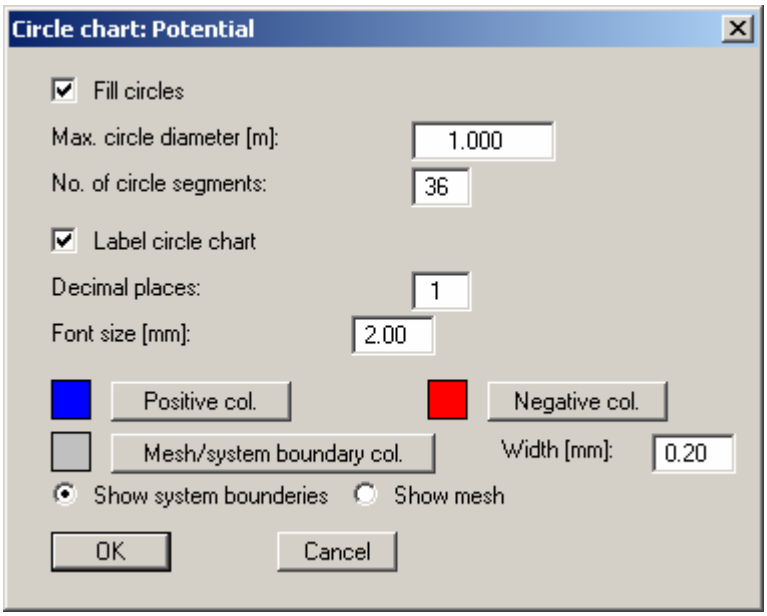
All important analysis results can be displayed graphically in a table. The parameters for visualisation are selected in the dialog box and their format can be modified.



The result tables are displayed in the graphics directly after using "**OK**" to exit the dialog box. A spacing can be defined to ensure that the tables do not overlap. The tables are then only displayed for the points of the defined grid.

7.8.6 "Circles" menu item

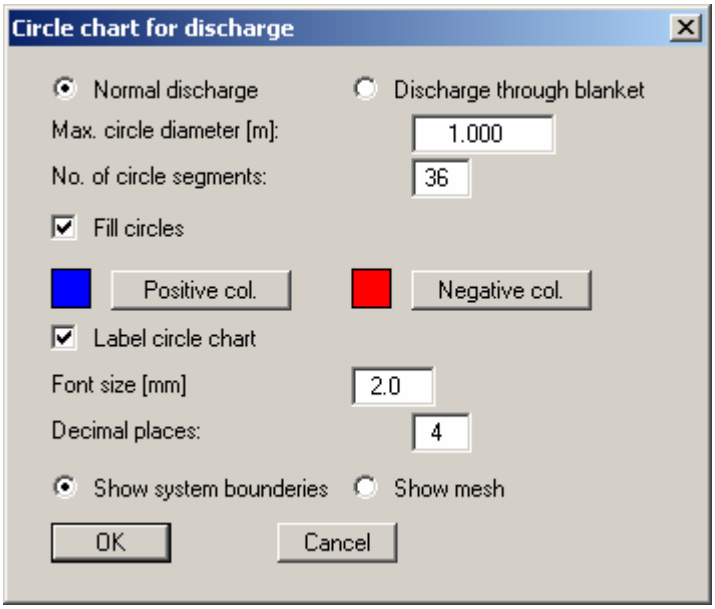
After selecting the required parameter, circle charts can be generated. The selected values (potentials, velocities or gradients) are displayed as circles of varying (value-dependent) circles.



If you activate the "**Fill circles**" check box the values are represented with a colour fill defined using the "**Positive colour**" or "**Negative colour**" buttons, corresponding to their sign. The "**Max. circle diameter**" defines the circles diameter for the largest value. The "**No. of circle segments**" specifies the resolution of the circle. For example, triangles are represented for a value of **3**. Each circle can be annotated with the corresponding value. The system boundaries or the FEM mesh can be drawn in addition to the circles. The colour and width of the pens can be specified for the graphical visualisation.

7.8.7 "Discharge" 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). By defining an appropriate image section you can have the sum of discharge present in that part of the system calculated by the program. The presentation can be modified in the dialog box.



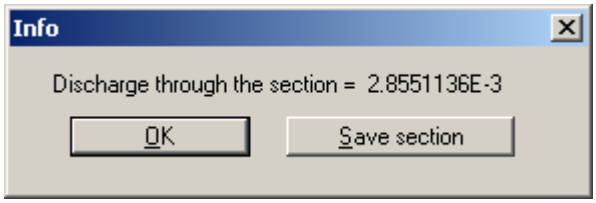
After using "**OK**" to exit the dialog box you define the area for which the discharge is to be calculated by clicking four point in an anti-clockwise direction. The calculated values are first displayed in a message box and then as a circle chart in your graphics.

If you are investigating a horizontal-plane system with a ***leaky aquifer***, you may also calculate the discharge for the blanket only by activating the additional "**Discharge through blanket**" check box.

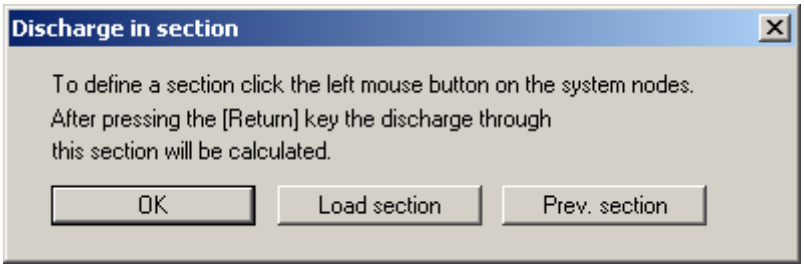
7.8.8 "Calculate in section" menu item

You can calculate the discharge along a section through the system.

To do this, define a section by clicking the appropriate nodes using the left mouse button. Intermediate points are inserted automatically by the program. The last point in the list is deleted by clicking the right mouse button. When the section is located where you want it press [**Return**]. The calculated discharge through this section will be displayed in an info box.



Here, you also have the option of saving the section to a file. You can repeat the procedure immediately for a different section after clicking on "**OK**". If you click this menu item a second time the following dialog box opens:

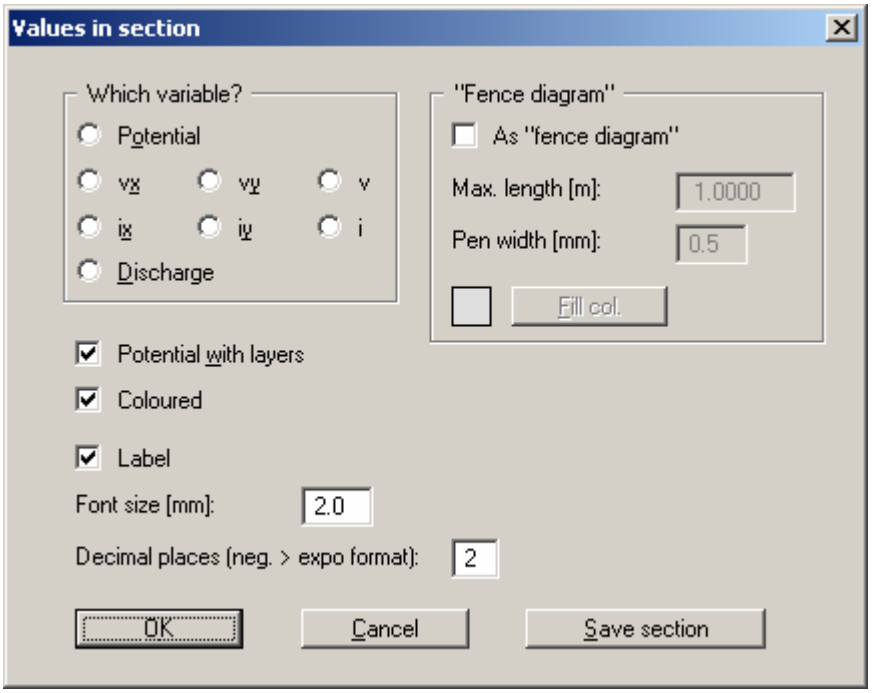


By pressing "**OK**" you can now create a new section or, using the other buttons, either return to the previous section or load one saved earlier.

Sections previously saved to a file can be loaded either via the menu item "**Evaluation/Values in node section**" or, if saved using that menu item, be loaded in this one (see Section 7.8.9).

7.8.9 "Values in node section" menu item

You can display calculated values, for example for potentials, in a section. Defining the section is done almost in complete analogy to the previous menu item. After pressing the [**Return**] key a dialog box opens allowing you to select the variable for display.



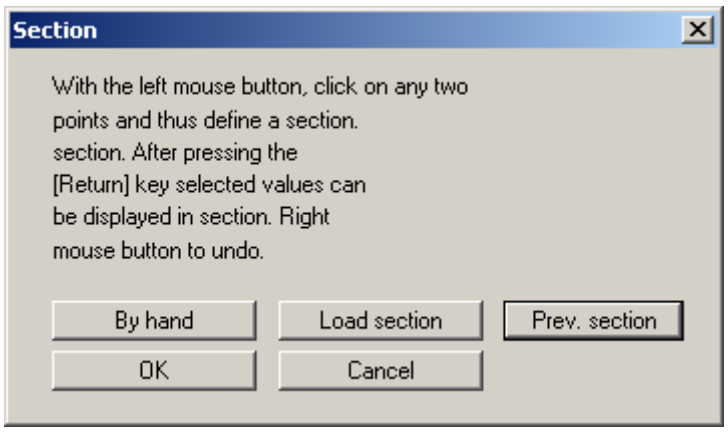
You can define preferences for the section as a function of the selected variable, for example labelling or a display of a ***fence diagram***. If you have selected "**Potential**", you can also choose to have the layers displayed, as well as the potentials.

Here you are given the opportunity to save your section to a file using the "**Save section**" button. This file can later be opened, for example using the menu item "**Evaluation/Calculate(discharge) in section**" (see Section 7.8.8).

If you have activated "**Show section course**" using the menu item "**Graphics preferences/Section legend**" (see Section 7.6.11), a location sketch is automatically displayed on the screen showing the system outline and the defined section.

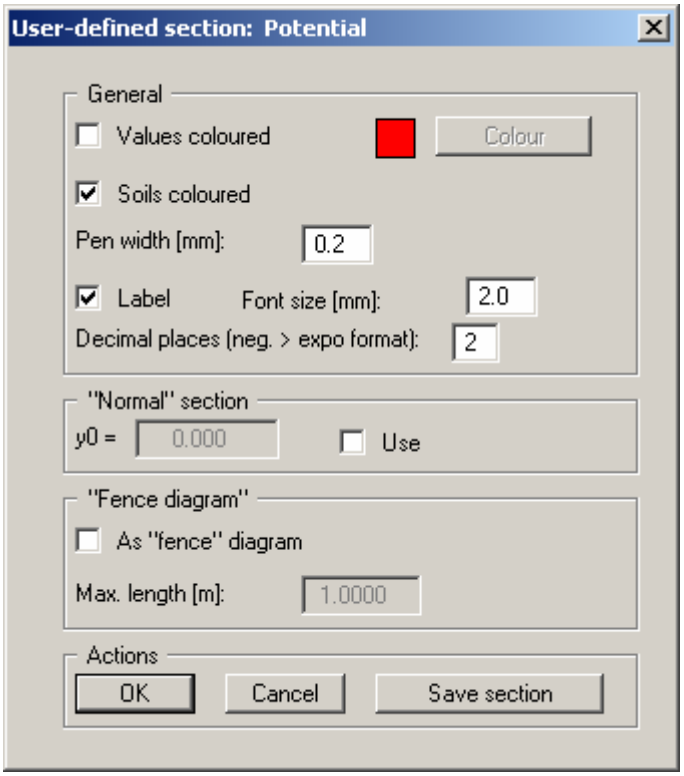
7.8.10 "User-defined section" menu item

After selecting this menu item you will see the following dialog box for defining the section:



For this type of evaluation the section course is no longer bound to the FEM nodes, but consists of a start and an end point. Using the "**By hand**" button you can specify the start and end points using numerical values instead of the mouse. 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.

Once the section definition is completed by pressing [**Return**] the number of subdivisions can be specified in a further dialog box and whether potential, velocity or gradient is to be visualised. The number of subdivisions defines the number of points required for calculation of the section values. After confirming with "**OK**" the following dialog box opens:

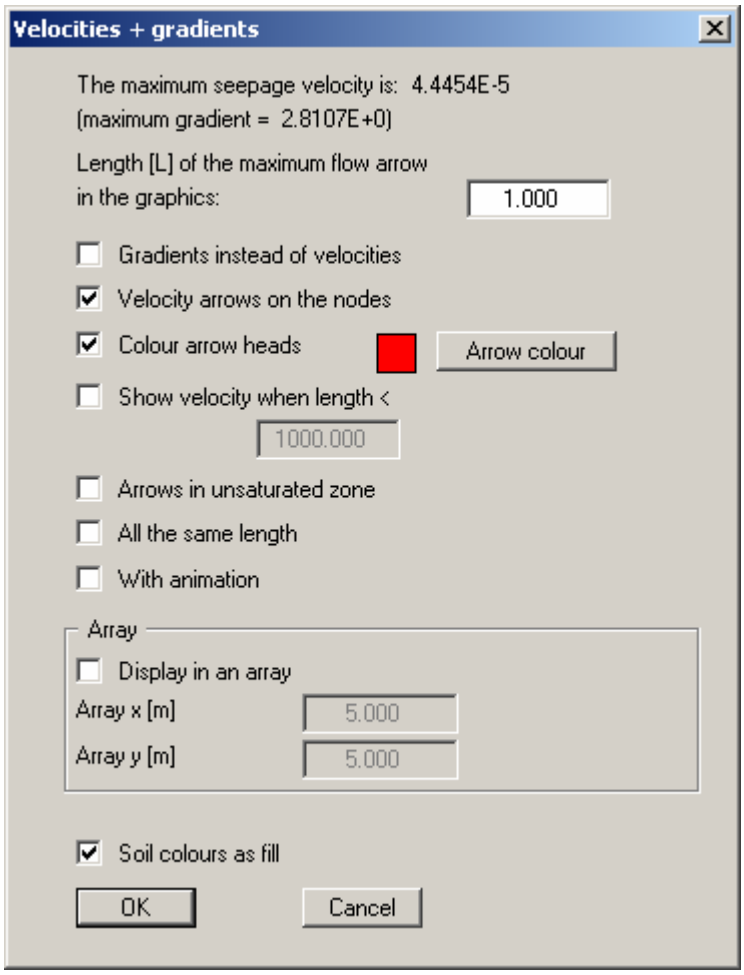


You can now specify whether the section should be displayed in colour. You can also define the intercept of the x-axis with the y-axis by entering a value for "**y0**". In this case the "**Use**" check box must be activated. If you do not use this option, the x axis will intercept the y axis at the smallest y value. The section can also be displayed as a ***fence diagram***. You can save the section in order to load it again at a later date (see above).

If you have activated "**Show section course**" using the menu item "**Graphics preferences/Section legend**" (see Section 7.6.11), a location sketch is automatically displayed on the screen showing the system outline and the defined section.

7.8.11 "Velocities + gradients" menu item

The velocity field can be visualised using this menu item. The size and direction of the seepage velocity are shown (vs = (k · i)/neff !!).



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 on the 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.

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 general 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, corrected by the program to suit the geometry of the triangular elements, if necessary. In addition, the soil colours can be used as a background.

> As the program analyses using unsaturated permeabilities for vertical-plane systems, 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.

7.8.12 "Contours + velocities" menu item

Using this menu item you can display on the screen a combination of normal contours and velocities. The dialog boxes for contour line and velocities preferences are shown in sequence. The dialog box descriptions can be found in Sections 7.8.1 und 7.8.11.

7.8.13 "Individual values" menu item

By clicking a FEM node using the left mouse button all analysis results for this node can be viewed.

7.8.14 "Flow lines preferences" menu item

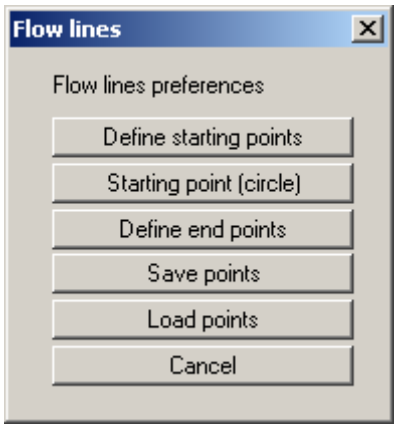
7.8.14.1 Principles

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 starting point, is observed. The program determines size and direction of seepage from this starting 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 starting 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 (vs = (k · i)/neff!). Using this program, you can also follow a particle back (against flow direction).

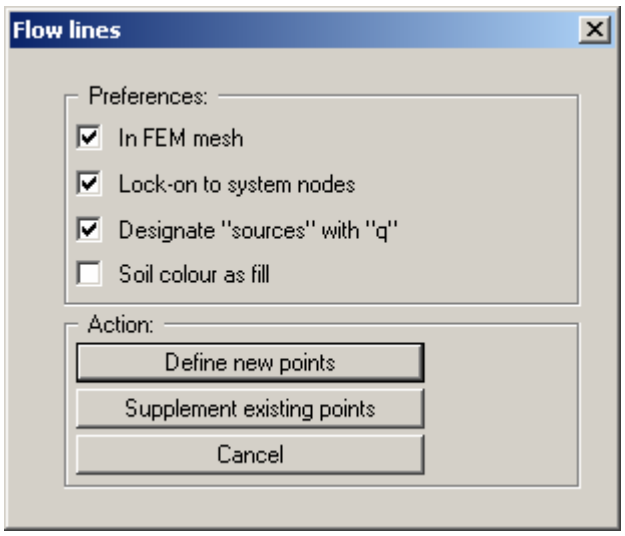
7.8.14.2 Preferences

After selecting this menu item you can enter various criteria for calculations in the following dialog box:



- "**Define starting points**"
  You can define the starting points of water particles.
- "**Starting point (circle)**"
  You can define several starting 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 starting points to a file, in order to have them available for a different system.
- "**Load points**"
  Starting 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:

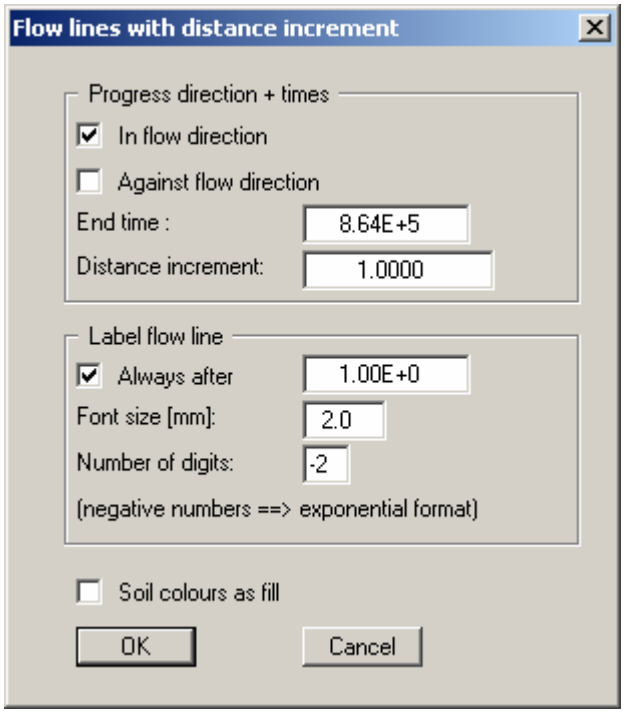


The "**In FEM mesh**" check box displays the FEM mesh when specifying points; otherwise, only the system outline is displayed. Defined points will be locked on to the system nodes closest to the mouse pointer when clicked, if the "**Lock-on to system nodes**" check box is activated. Otherwise the points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

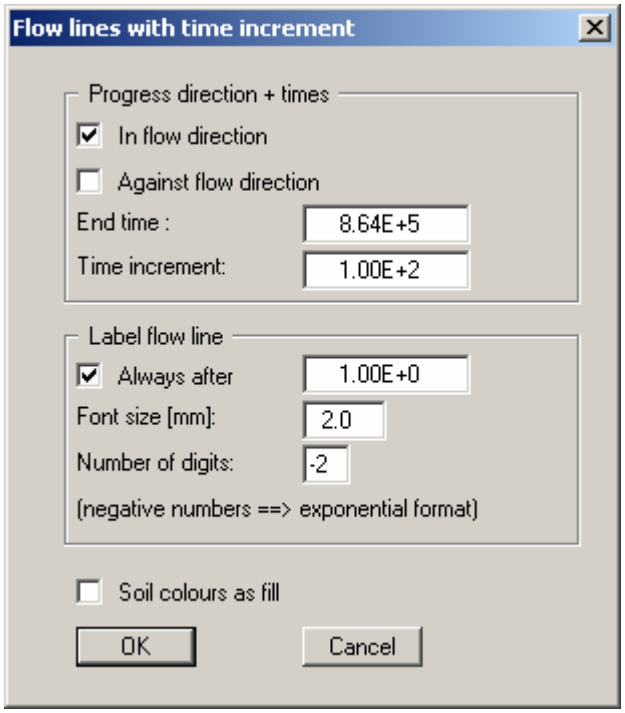
7.8.15 "Distance increment" menu item

You can commence the analysis of a particle track (or tracks). The analysis parameters and the type of subsequent presentation can be confirmed or edited in the dialog box. Activate the "**Against flow direction**" check box in order to retrace the path of a particle.



7.8.16 "Time increment" menu item

You can commence the analysis of a particle track (or tracks). The analysis parameters and the type of subsequent presentation can be confirmed or edited in the dialog box. Activate the "**Against flow direction**" check box in order to retrace the path of a particle.



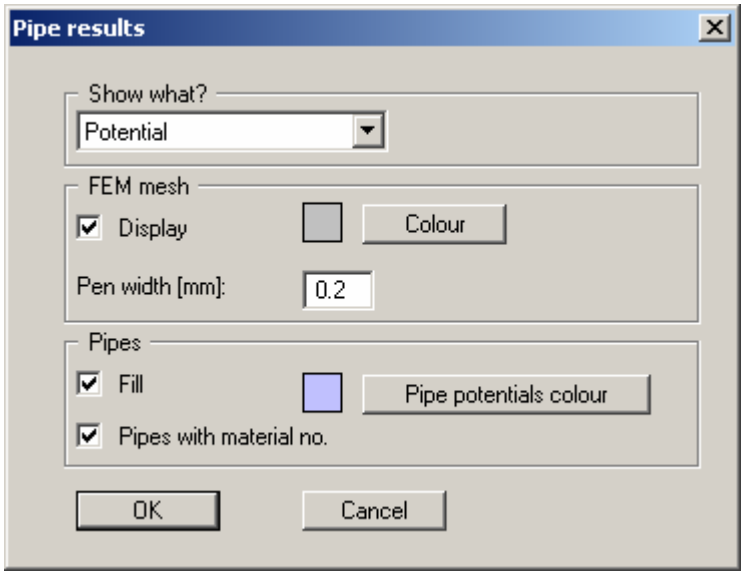
7.8.17 "Difference" menu item

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

Before leaving the dialog box, you can use the "**Normal**", "**Coloured**" and "**3D**" option buttons to specify the type of contour diagram. Once the program has finished the difference calculation you will see dialog boxes for specifying contour line preferences in accordance with your selection (see Sections 7.8.1 bis 7.8.3).

7.8.18 "Pipe results" menu item

If you have defined pipes with material numbers using "**Boundary/Define pipes**" you can display the results using this menu item. First, a dialog box opens for selecting between display of potentials, gradients, velocities or discharges.



You can the specify further general display preferences in the same dialog box. If you confirm the input with "**OK**", a special dialog box for the selected variable follows. After confirming the input with "**OK**", the results are displayed on the screen.

7.9 Special menu (for horizontal-plane systems only)

7.9.1 General note

The menu "**Special**" is only available for horizontal-plane systems. It allows further evaluation of such a system.

7.9.2 "Groundwater thickness" menu item

A contour diagram showing the groundwater thicknesses is generated (see menu item "**Evaluation/Normal contours**", Section 7.8.1).

7.9.3 "Coloured" menu item

A colour-filled contour diagram of the groundwater thicknesses is generated (see menu item "**Evaluation/Coloured**", Section 7.8.2).

7.9.4 "Volume" menu item

You see a message box containing information on the groundwater-filled volume of your system.

7.9.5 "Groundwater-surface distance" menu item

A contour diagram showing lines of equal groundwater-surface distance is generated (see menu item "**Evaluation/Normal contours**", Section 7.8.1).

7.9.6 "Coloured" menu item

A colour-filled contour diagram showing lines of equal groundwater-surface distance is generated (see menu item "**Evaluation/Coloured**", Section 7.8.2).

7.9.7 "Confined areas" menu item

Nodes in areas of the system where confined groundwater conditions are prevalent are marked with the letter "**G**". Alternatively, these areas can be colour-coded.

7.9.8 "Dry areas" menu item

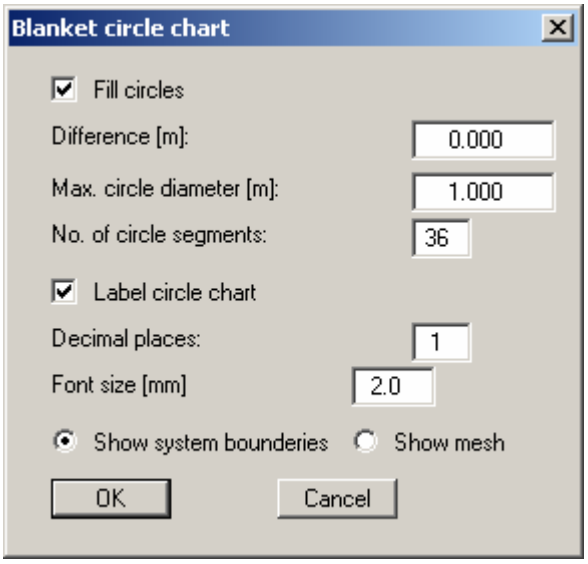
If the potential falls below the base of the aquifer in some areas of the system, then these areas are ***dry***. The nodes in these areas are marked with the letter "**T**". Alternatively, these areas can be colour-coded.

7.9.9 "Blanket layer areas" menu item

The water level in the user-defined blanket layer areas is entered in the corresponding FEM elements. The blanket areas can also be colour-coded.

7.9.10 "Blanket layer circle chart" menu item

The water level in the blanket layer is displayed as a circle chart (also see menu item "**Evaluation/Circles**", Section 7.8.6).



If your system contains several blanket layer zones with varying water levels, you can display circles with substantially different sizes, depending on the water level, by entering a value that is marginally larger than the smallest prevalent water level in the "**Difference**" input box.

7.9.11 "Discharge in blanket layer" menu item

The discharges in the blanket layer are displayed as coloured contour lines (see menu item "**Evaluation/Coloured**", Section 7.8.2).

7.10 ? menu

7.10.1 "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-SS-FLOW2D**.

7.10.2 "Maxima" menu item

A box opens showing the maximum values allowed by the program for the nodes and elements of the FEM mesh.

7.10.3 "Help" menu item

The **GGU-SS-FLOW2D** manual is opened as a PDF document. The help function can also be accessed using the [**F1**] function key.

7.10.4 "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.

7.10.5 "GGU support" menu item

This menu item takes to the GGU-Software Support area at www.ggu-software.com.

7.10.6 "What's new?" menu item

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

7.10.7 "Language preferences" menu item

This menu item allows you to switch the menus and graphics from German to English or Spanish and vice versa. The program always starts with the language setting applicable when it was last ended.

8 Tips and tricks

8.1 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

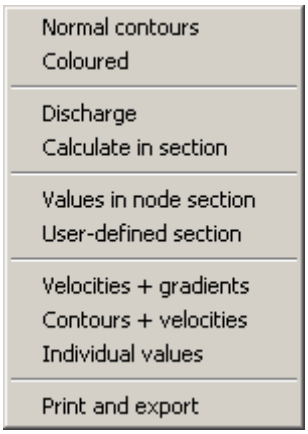
Change system graphics (new values can be checked in "Page size + margins/Manual resize (editor)"):

- [**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.

You can simplify system input by integrating graphics or DXF files into the system via the **Mini-CAD** module (see Mini-CAD manual). The size of the graphics or the DXF data can be imported to the correct scale. However, this is not absolutely necessary for mesh generation. If you do not model the system to scale using a graphics file, you can perform a scale correction after input is complete using the menu item "**Mesh/Change nodes**", "**Via equation**" button (see Section 7.2.5).

8.2 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/Define nodes**".
- [**F4**] opens the menu item "**Mesh/Manual mesh**".
- [**F5**] opens the menu item "**System/Analyse**".
- [**F6**] opens the menu item "**Mesh/(Refine) Section**".
- [**F7**] opens the menu item "**Mesh/(Refine) All**".
- [**F9**] activates the menu item "**Page size + margins/Auto-resize**".
- [**F11**] activates the menu item "**Graphics preferences/Move objects**".

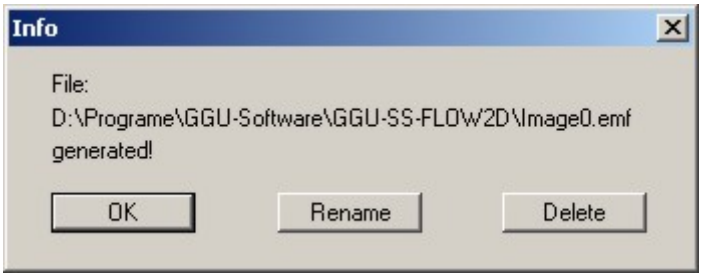
8.3 "Copy/print area" icon

A dialogue 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.

9 Worked examples

9.1 Example 1: Analysis of a horizontal-plane system

9.1.1 System description (Example 1)

Following, a concrete example of calculation of a horizontal-plane system will be shown. A system with the following dimensions is to be investigated:

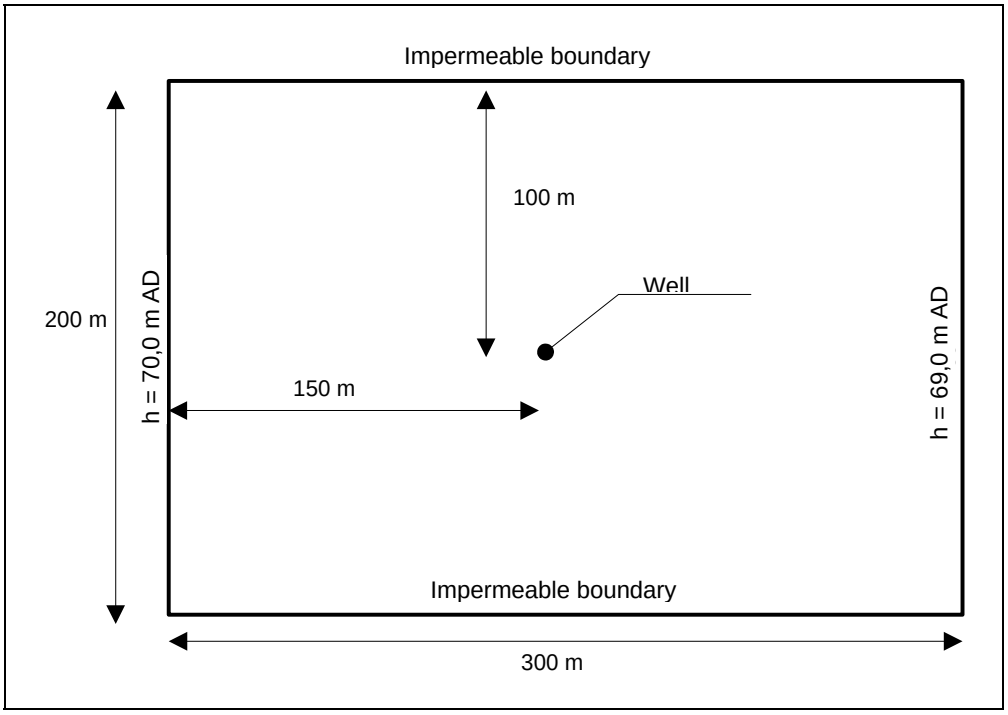


Figure 9 Example of a horizontal-plane system

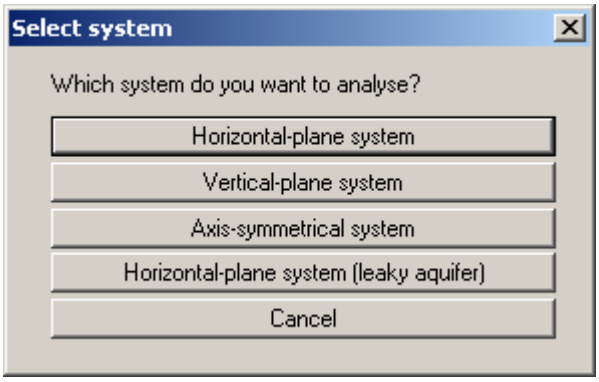
The system is depicted in the figure in plan. A section of the groundwater system of 200 · 300 m is to be considered. It is known from groundwater level measurements that without the influence of the well a parallel groundwater flow, from left to right, is present. The water level at the left system edge is 70 m AD and at the right system edge 69 m AD.

The subsurface consists of sand ($k = 2 \cdot 10^{-4}$ m/s), which is covered by a cohesive, slightly permeable valley loam. The top of the sand horizon is at approx. 68 m AD, so slightly confined conditions are met. The base of the sand horizon falls roughly linearly from 59 m AD at the left edge, to 51 m AD at the right edge.

The influence of the well, for a drawdown to 65 m AD, is to be investigated. Apart from the discharge, the areas which are no longer confined during well operations are to be investigated.

9.1.2 Step 1: Select the groundwater system (Example 1)

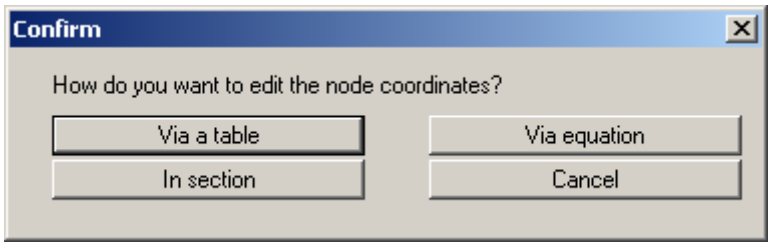
Start the program and go to the "**File/New**" menu item. Exit the "**Common systems**" dialog box using "**No**". You will then see the following dialog box:



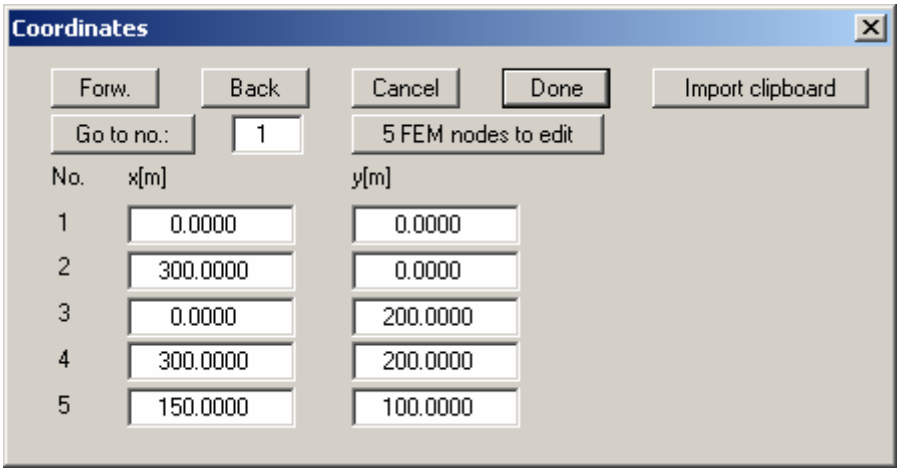
Select the "**Horizontal-plane system**" button from the dialog box.

9.1.3 Step 2: Define FEM nodes and mesh (Example 1)

The rough system structure consists of four boundary points and one point in the centre (well), so there are five points in all. The input of system points is therefore easiest using the menu item "**Mesh/Change**".



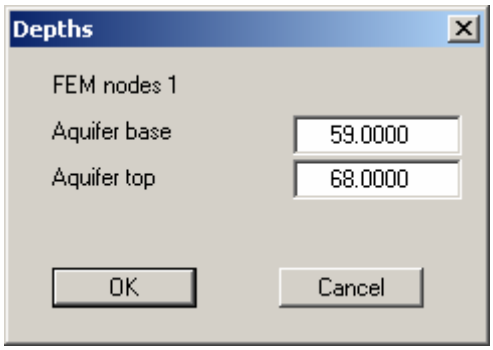
Select the "**Via a table**" button and the "**0 FEM nodes to edit**" button in the following dialog box. Enter **5** as the new number of FEM nodes and confirm the input. Then enter the five coordinate pairs in the table.



Now go to the "**Mesh/Automatic**" menu item and confirm all message boxes displayed. In order to achieve a screen-filling display click menu item "**Page size + margins/Auto-resize**" or simply press the [**F9**] function key.

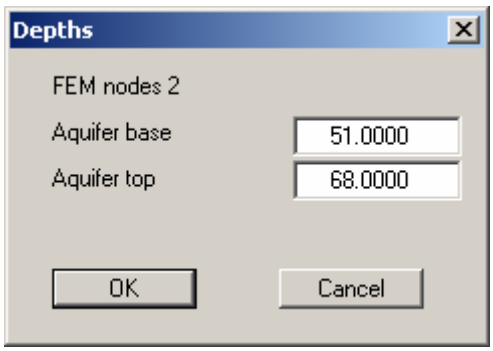
9.1.4 Step 3: Assign depth (Example 1)

Go to the "**z/Individual depths**" menu item in order to assign the five nodes the values for top of aquifer and base of aquifer. Double-click the bottom left node and enter the following data:

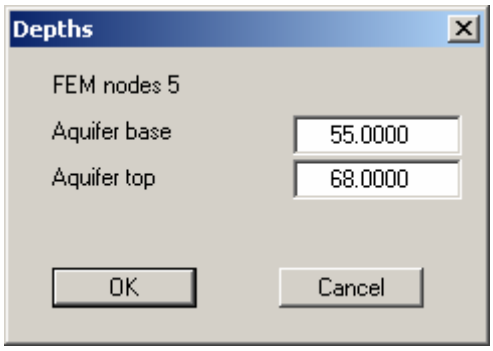


The node at the top left is given the same values.

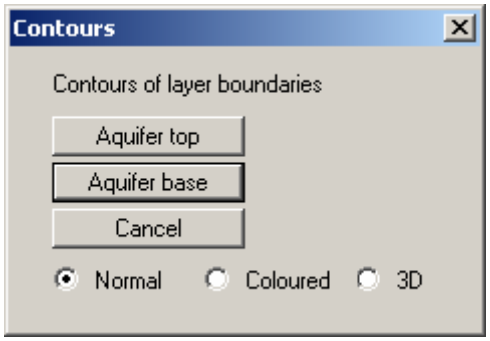
The two right-hand nodes are given the following values:



The central node is given the following values:



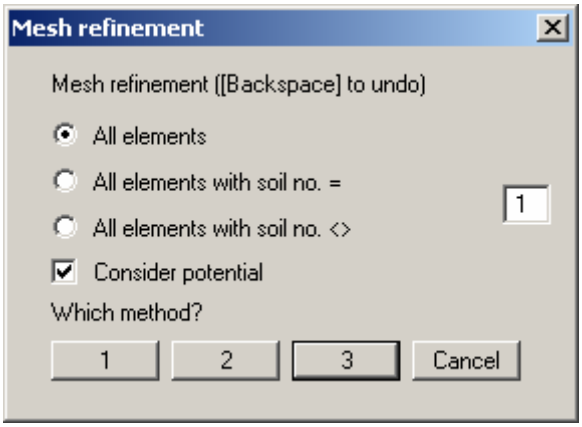
Check your input using the menu item "**z/Layer boundary contours**". Select the "**Aquifer base**" button in the dialog box.



If the aquifer base has been input correctly the contour lines should fall off towards the right.

9.1.5 Step 4: Refine mesh (Example 1)

The current system is still very coarse, so a mesh refinement will now follow. Go to the menu item "**Mesh/All**" and click on button "**3**" in the dialog box.



Repeat the procedure. You should now have a system with 64 triangles and 41 nodes (see menu item "**System/Info**"). A further refinement in the boundary areas need not be done yet. The area of the well should, however, be further refined, as this is where the subsurface hydraulics are most important. Select the menu item "**Mesh/Section**" and leave the dialogbue box clicking the button "**3**".

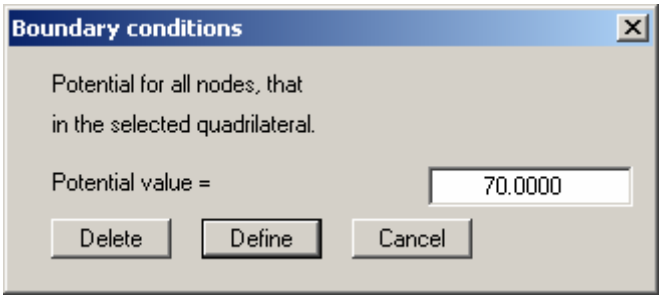
Go to the menu item "**Mesh/Section**" and exit the dialog box by pressing the "**3**" button. Now trace around the inner rectangle with its four triangles, which touch the (as yet non-existent) well in the centre of the system. Repeat the process twice for the respective inner rectangle with the 4 new respective triangles around the well.

Now perform an overall refinement of the FEM mesh using the menu item "**Mesh/All**". The system should now contain 448 triangles and 241 nodes; you can check this by pointing at the "**System/Info**" menu item (see Section 7.5.1).

During mesh refinement the program automatically computed the values of z for "**Aq. top**" and "**Aq. base**" for new nodes from a linear interpolation of the neighbouring nodes. The "**Aq. base**" z-value was defined in the initial coarse structure as decreasing linearly from 59 m AD (at the left edge) to 51 m AD (at the right edge). This linear course remains intact after mesh refinement. You can check this by again creating a contour diagram of the base of the aquifer (Aq. base) by pointing at the "**z/Layer boundary contours**" menu item. The contours are unaltered.

9.1.6 Step 5: Define boundary conditions (Example 1)

In order to set boundary conditions go to the menu item "**Boundary/(Potentials) In section**" and circumscribe the nine FEM nodes at the left side of the system.



Enter a potential of 70 m AD and accept it for the selected nodes by clicking the "**Define**" button. Trace around the nine FEM nodes on the right of the system and enter a potential of 69 m AD.

Point at the "**Boundary/Individual potentials**" menu item and click the "**Graphical**" button in the box. Now click the well, that is node 5 in the centre of the system, and enter a potential of 65 m AD. If you have problems actually hitting the node, enlarge the screen view by tracing the inner region with the left mouse button and the [**Ctrl**] key pressed. The [**Esc**] key resets the screen to an overall view.

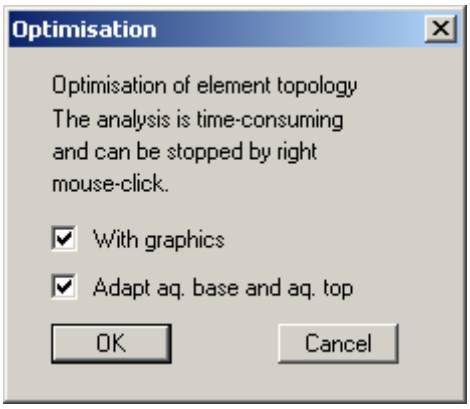
> **Note on boundary conditions:**
> The case of an impermeable boundary is automatically taken into consideration by the finite-element method. Valid is, that all system boundaries or partial boundaries, which do not posses source or potential boundary conditions, are automatically impermeable. In finite-element theory, this type of boundary condition is also known as a natural boundary condition.

9.1.7 Step 6: Optimise element topology (Example 1)

In principle, it is always a good idea to optimise the element topology before commencing modelling. However, in horizontal-plane systems this means that the values for aquifer base and aquifer top cannot be adapted to the new location due to the displacement of individual nodes. If the values for aquifer base and aquifer top vary within the system the program will issue a warning.

Generally, topology optimisation should always be performed after defining the potential boundary conditions, because the program does not displace nodes with potential boundary conditions during optimisation. If optimisation is carried out before defining the boundary conditions, the location of the well point in the centre may be altered.

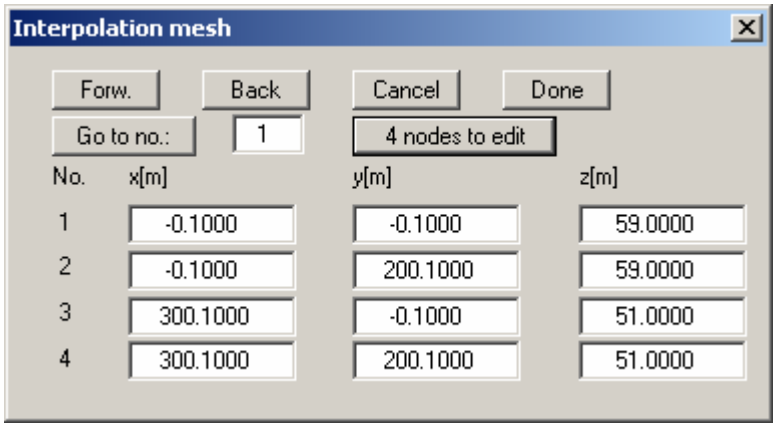
Go to the "**Mesh/Optimise**" menu item and click the "**Topology**" button in the dialog box. After acknowledging the warning message you will see the following dialog box:



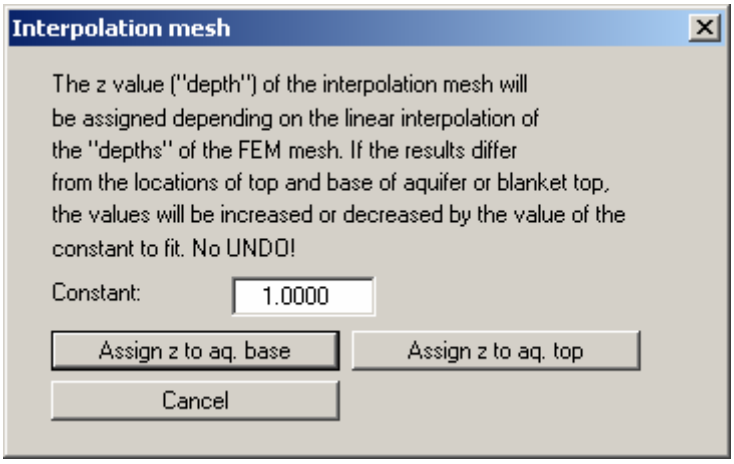
If you activate the "**Adapt aq. base and aq. top**" check box the values for aq. base and aq. top are adapted to the displaced nodes as well as possible during optimisation. The menu item "**z / Layer boundary contours**" allows you to convince yourself that the base of the aquifer no longer displays an exactly linear drop from left to right. In order to restore the exact course the values for aq. base and aq. top are defined in the next step via an interpolation mesh.

9.1.8 Step 7: Assign aquifer base using interpolation mesh (Example 1)

Go to the "**z/Nodes**" menu item and click "**x nodes to edit**". Change the number of nodes to **4** and enter the values from the dialog box below.



By doing this, you define the four points of the interpolation mesh. A small surcharge of 0.1 was used, with reference to the corner nodes of the FEM mesh, in order to ensure that the FEM mesh is completely enclosed. With the z values you define the gradient of the base of the aquifer. Now select the menu item "**z/Mesh**" and then the "**Automatic mesh**" button. The program connects the four nodes of the interpolation mesh to a triangular mesh with two triangles. Select the menu item "**z/Assign**".

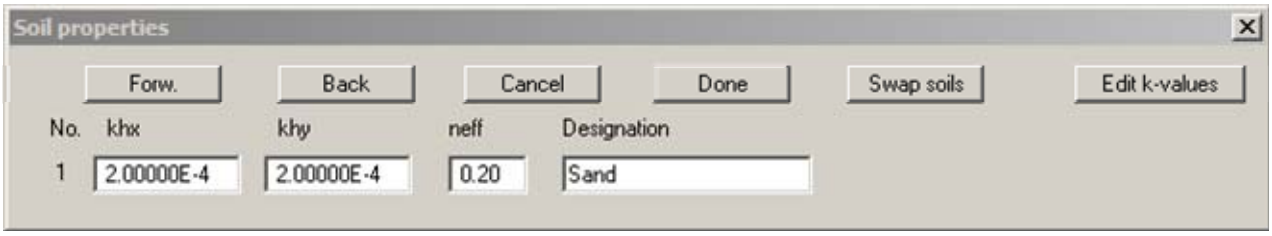


Press the "**Assign z to aq. base**" button.

Using the menu item "**z/Layer boundary contours**" you can convince yourself that the base of the aquifer has, once again, an exactly linear fall off from left to right.

9.1.9 Step 8: Assign soil properties (Example 1)

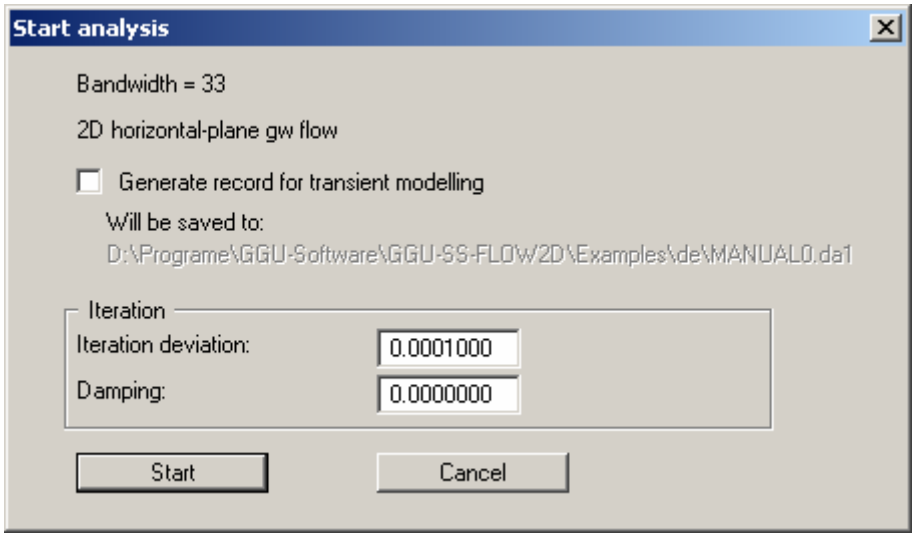
Go to the menu item "**System/Soil properties**" and enter the values of the dialog box.



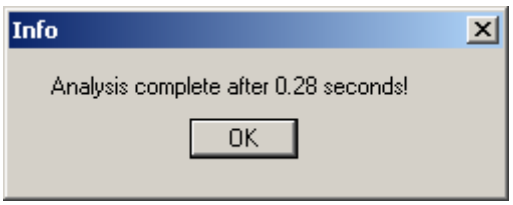
All triangular elements are assigned soil number **1** when the FEM mesh is automatically generated. The dialog box therefore only contains one row for one soil.

9.1.10 Step 9: Analyse system (Example 1)

Go to the menu item "**System/Analyse**" and confirm the iterative equation solver.

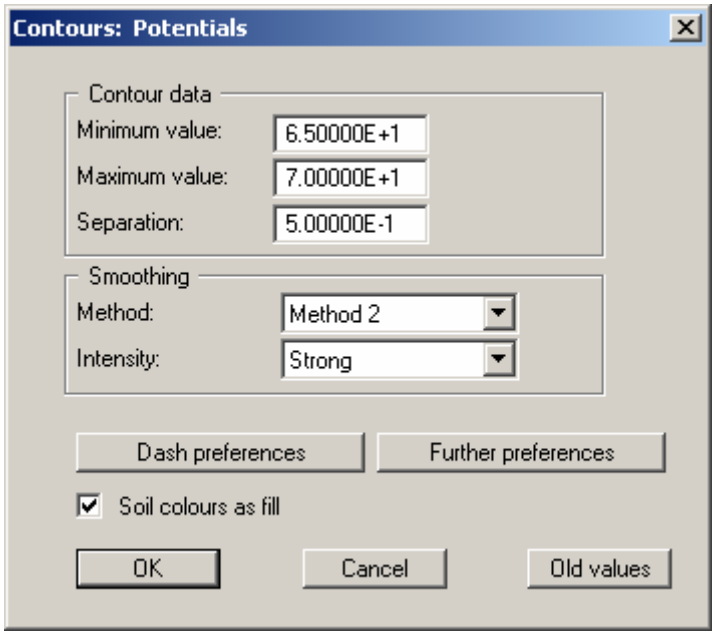


Press the "**Start**" button. After a certain time, depending upon the power of your computer, you will see information on analysis completion times.



9.1.11 Step 10: Evaluate potentials (Example 1)

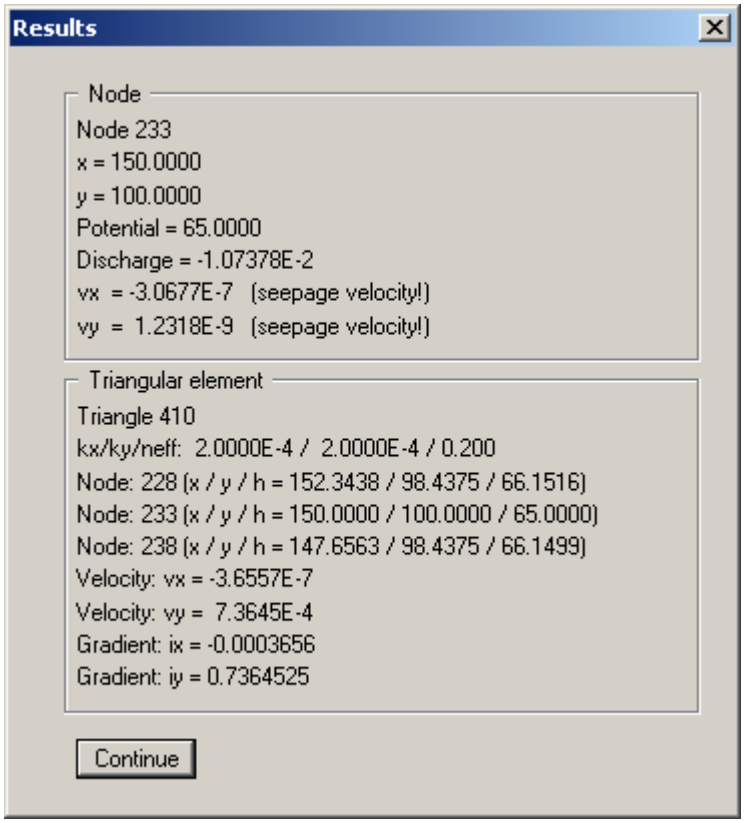
A graphic presentation of the contours is to be created. Go to the menu item "**Evaluation/Normal contours**".



Choose "**Method 2**" for smoothing and confirm your input with "**OK**". The lines of equal potentials are displayed on the screen. If you wish, you can print the resulting graphics.

9.1.12 Step 11: Evaluate well discharge (Example 1)

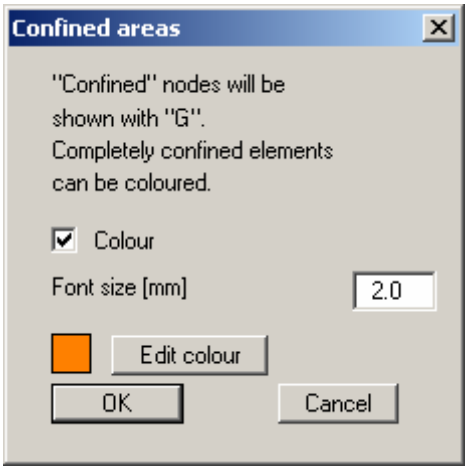
Go to the menu item "**Evaluation/Individual values**" and click the central FEM node (= well). If your screen does not have the resolution to allow you to see the point, activate the zoom by pressing the [**Ctrl**] key and the left mouse button and pulling open a rectangle around the point.



The discharge of the well is $1{,}1 \cdot 10^{-2}$ m³/s = 39,6 m³/h.

9.1.13 Step 12: Evaluate confined areas (Example 1)

Go to the menu item "**Special/Confined areas**" and confirm the with "**OK**".



It will become clear that an area of about 40 m around the well is no longer confined.

To check this, you can load the supplied file "**MANUAL0.FEN**", which contains the final system.

You can reanalyse the system with finer subdivisions by selecting the menu item "**Mesh/All**" once again, or several times, and selecting button "**3**". Compare the result quality for different mesh subdivisions.

9.2 Example 2: Analysis of a vertical-plane system

9.2.1 System description (Example 2)

An analysis of a vertical-plane system is demonstrated below using a practical example. An embankment with the given dimensions will be investigated:

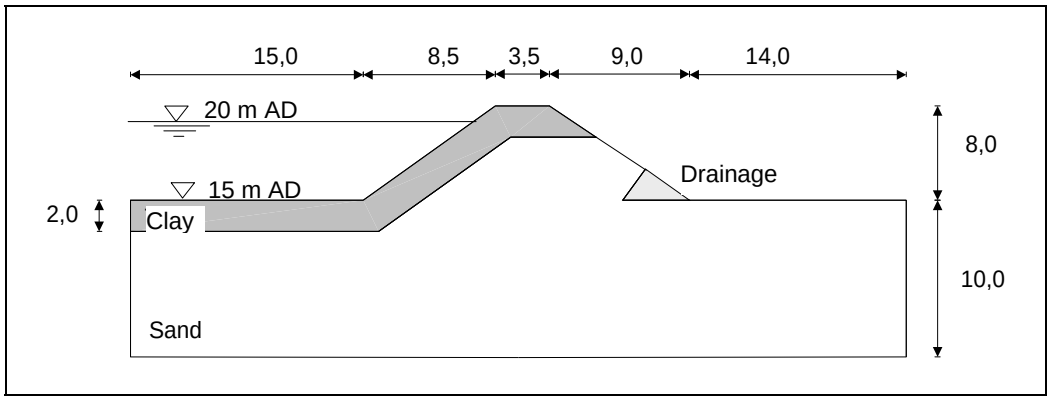


Figure 10 Example of a vertical-plane system

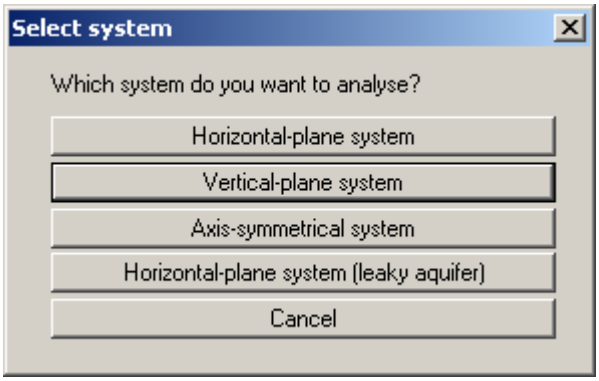
The following permeability data are given:

- Sand: k = 10^{-4} m/s
- Clay: k = 10^{-7} m/s
- Drainage: k = 10^{-2} m/s

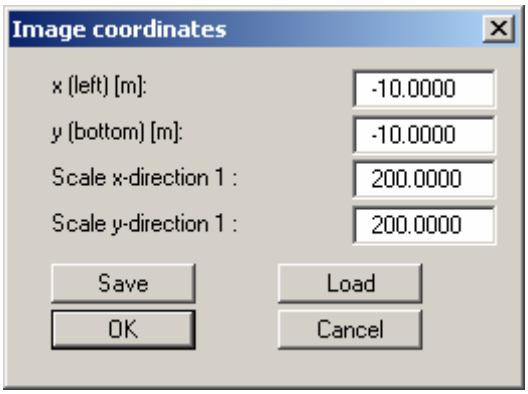
The pondage on the water side of the embankment is at 20 m AD. A drainage system is installed on the air side, draining to 15 m AD. The phreatic line and the volume of water percolating through the embankment will be determined for this system.

9.2.2 Step 1: Select the groundwater system (Example 2)

Start the program and go to the "**File/New**" menu item. Exit the "**Common systems**" dialog box using "**No**". You will then see the following dialog box; exit by clicking the "**Vertical-plane system**" button.



Go to the menu item "**Page size + margins/Manual resize (editor)**" and enter the following values:

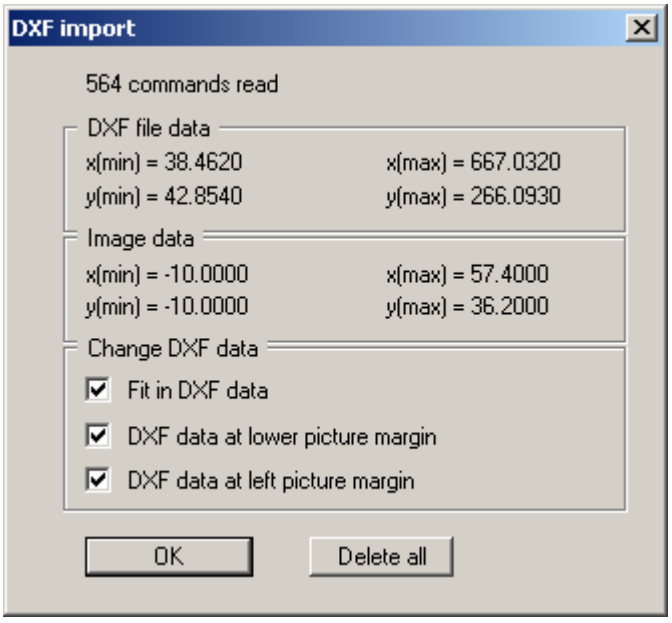


This input is not absolutely necessary for the program to operate. It merely ensures that you have the same data for the subsequent dialog boxes.

9.2.3 Step 2: Import and process a DXF file for system input (Example 2)

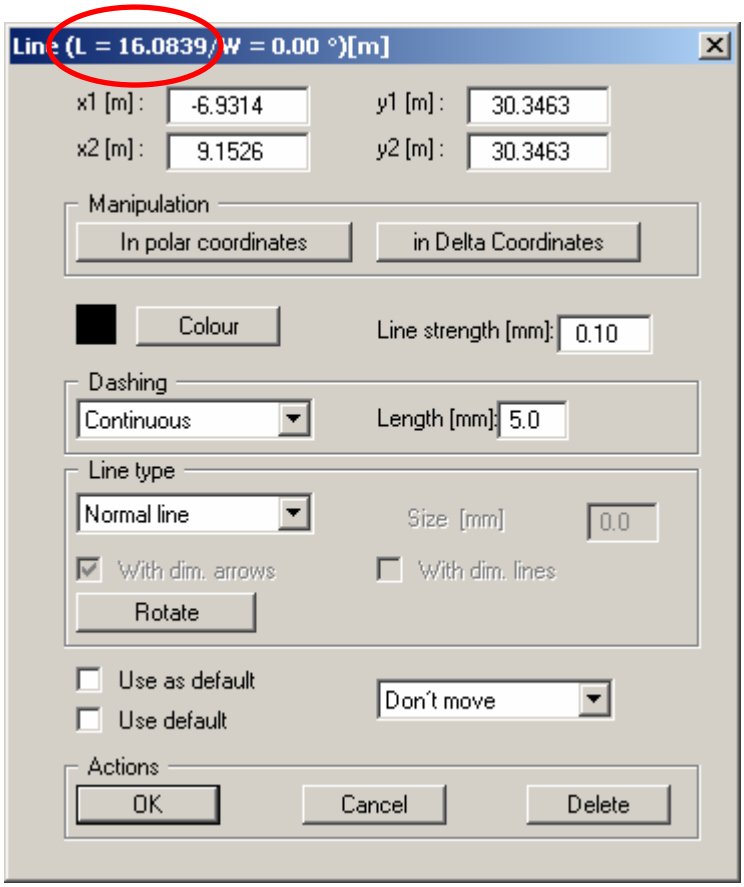
The system is available as the DXF file "**Manual.dxf**". Input of the coarse system structure can be considerably simplified by importing the DXF file into the graphics using the "**Mini-CAD**" module and then clicking the principal points with the mouse.

To do this, point at "**Graphics preferences/Mini-CAD toolbar**". A popup window with icons appears at the bottom right. Click the "**DXF import**" tool  and load the "**Manual.dxf**" file, without altering the preferences in the import dialog box. The DXF data are imported and the following editor box opens:



Normally, a DXF file can be imported without altering the DXF data (all 3 check boxes deactivated) and your system updated by pressing [**F9**]. However, the coordinates in the "**Manual.dxf**" example file are not correct. You therefore see a description of procedure for activated check boxes (= altering the DXF data). After importing the DXF file, confirm the above dialog box without alterations (= activated check boxes) by pressing "**OK**".

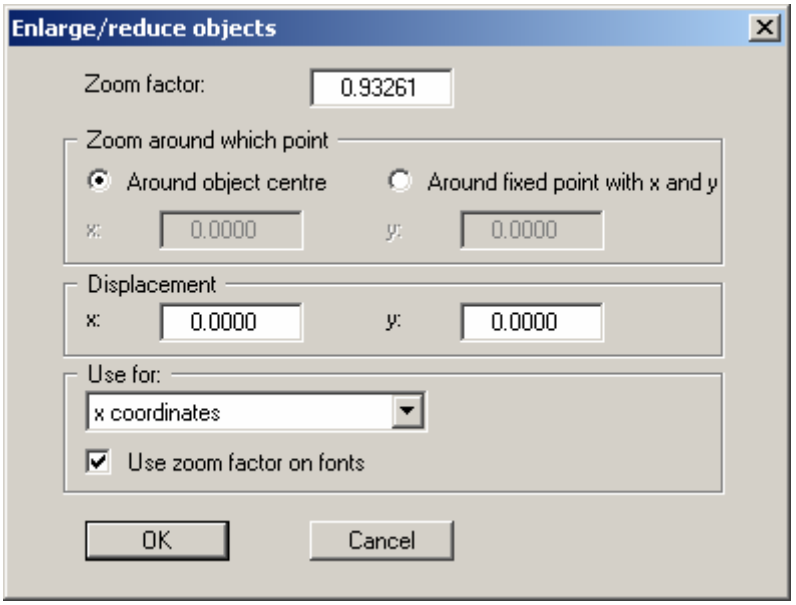
The system is displayed on the screen. However, it is not drawn to scale. To correct this, proceed as follows. A dimension line is shown at the top left; it should be 15 m long. Double-click on this line with the left mouse button to open the editor box.



The actual length of the line is given at the top edge of the dialog box (L = 16.0839). This allows a reduction factor *f* (target/actual) to be calculated:

$$f = 15.0/16.0839 = 0.93261.$$

In order to correspondingly reduce the drawing, select the "**Modify objects**" tool  from the Mini-CAD toolbar and enter the calculated value of **0.93261** in the "**Zoom factor**" dialog box.

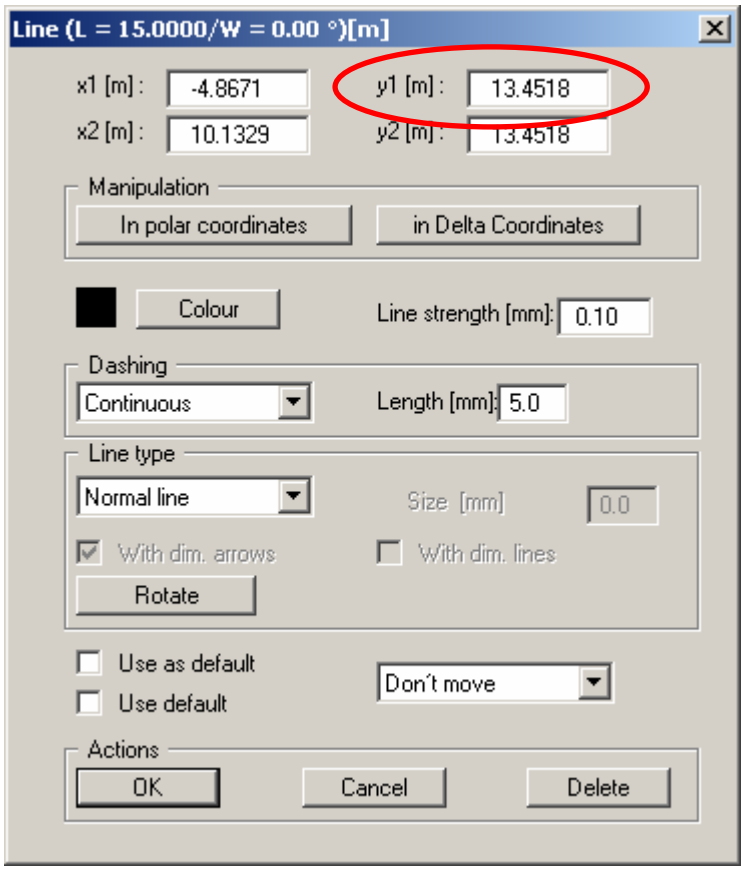


In the lower group box of the dialog box you will see "**Use for**"; select "**x coordinates**". The drawing is now to scale in x-direction and only needs to be adapted in y-direction.

A dimension line is shown at the right; it should be 10 m long. Double-click on this line with the left mouse button and read the actual length from the editor box once again (L = 20.6922). This allows a reduction factor *f* to be calculated:

$$f = 10.0/20.6922 = 0.48327.$$

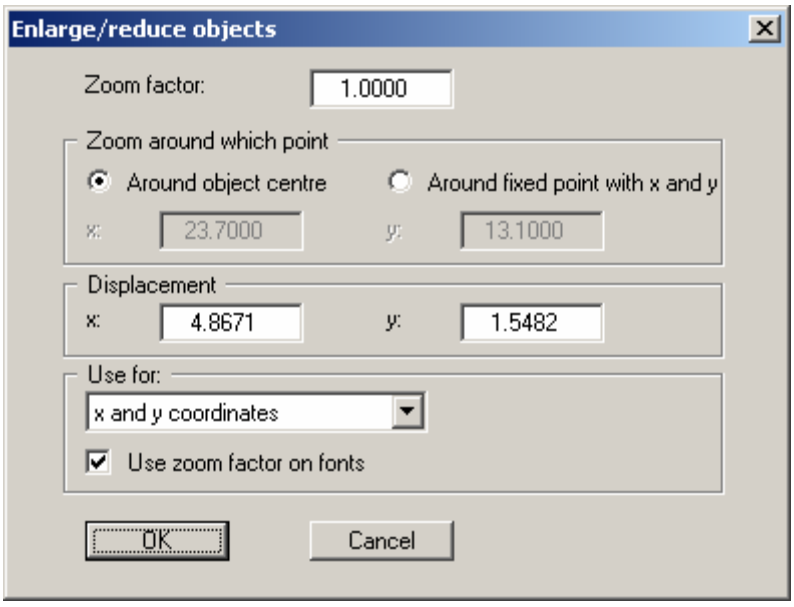
In order to correspondingly reduce the drawing, select the "**Modify objects**" tool  from the Mini-CAD toolbar and enter the calculated "**Zoom factor**" of **0.48327**. In the lower group box of the dialog box, this time select "**y coordinates**" for "**Use for**" and confirm your input with "**OK**". The system is now to scale; only the heights do not fit. So double-click the basin floor (line at the left of the system) with the left mouse button, because the height of this line is known.



The y-value of this line is shown after "**y1**" (= 13.4518). However, it should be at 15.0 m AD. This identifies the displacement in the y-direction; it is:

$$\Delta y = 15.0 - 13.4518 = 1.5482.$$

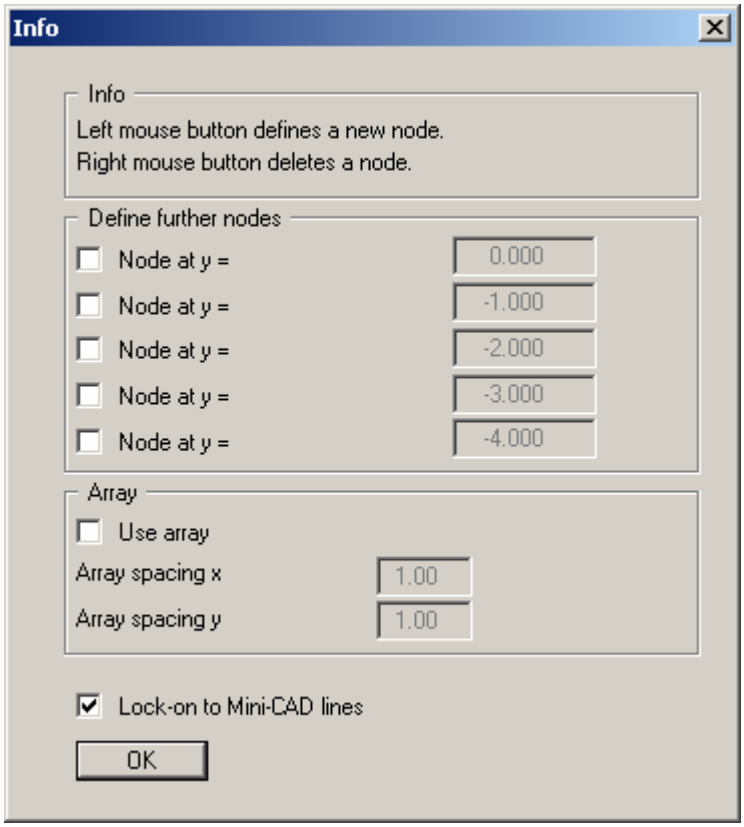
In order to correspondingly displace the drawing, select the "**Modify objects**" tool  once again from the Mini-CAD toolbar.



You must now enter a **1** for the zoom factor to avoid further reductions to the drawing. Enter the calculated **1.5482** as the value for "**y displacement**". In addition, you should also specify a displacement in x-direction of **4.8671** (= 0.0 - x1 value in the editor box for the left system boundary line) in order to ensure that the left system boundary lies at x = 0.0 (not absolutely necessary). This time, select "**x and y coordinates**" for "**Use for**". You have now adapted the DXF file data and can very simply click the principal points of the system with the mouse.

9.2.4 Step 3: Define FEM nodes, mesh and soils (Example 2)

Go to the "**Mesh/Define nodes**" menu item and activate the "**Lock-on to Mini-CAD lines**" check box in the dialog box.



After exiting the box by clicking "**OK**", the mouse cursor appears as a rectangle. If the end points of Mini-CAD lines are within this rectangle, the program will lock-on to these end points. You don't need to exactly hit the end points.

Now click all principal points in the system (including the intersection of water table and slope), in any order. Take care to ensure that the mouse cursor rectangle surrounds the necessary Mini-CAD line ends. In all, you must click on 15 points.

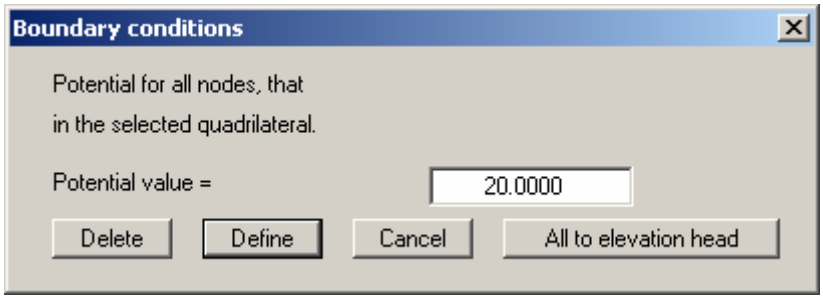
Finally, ***donate*** a further point at x = 25 and y = 5 (centre of bottom system edge). This auxiliary point only serves to give a ***nicer*** mesh for the following mesh creation. As none of the DXF lines end here, a node at this point will not exactly match the coordinates given above. In order to position this point exactly select the menu item "**Mesh/Edit**" and double-click on the nodes. Then enter the coordinates.

Now go to the menu item "**Mesh/Automatic**". The program connects the 16 nodes to a triangular mesh. The program creates a mesh which surrounds all nodes. For this reason, 2 to 3 ***superfluous*** elements are created on the water side and on the air side. To delete these, select the menu item "**Mesh/Delete**" and trace around the centroids of the superfluous elements with the rectangle.

The three different soils still need to be defined. All triangles were automatically assigned soil number "**1**" during mesh generation. This soil number will represent sand. Now go to the "**Boundary/(soils) In section**" menu item and trace with the rectangle around the areas to be defined as "**Clay**" (water-side seal) and assign the soil number "**2**". Do the same for the drainage and assign the soil number "**3**".

9.2.5 Step 4: Define boundary conditions (Example 2)

To define boundary conditions go to the menu item "**Boundary/(Potentials) In section**" and trace around the three points (two at the bottom of the basin and the intersection of the water surface with the slope) on the left side of the system.



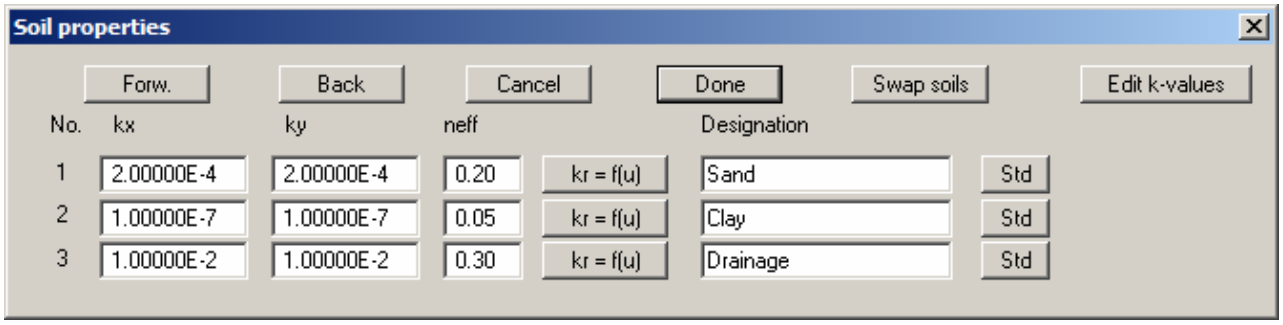
Enter a potential of 20 m AD and assign it to the points by clicking the "**Define**" button. Trace around the two nodes on the air side (base area) of the system and assign the points a potential of 15 m AD as already described.

> **Note on boundary conditions:**
> The case of an impermeable boundary is automatically taken into consideration by the finite-element method. Valid is, that all system boundaries or partial boundaries, which do not posses 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.

Save the file (menu item "**File/Save as**"). For control purposes, you can load the supplied "**MANUAL1.fen**" file, which contains this system.

9.2.6 Step 5: Assign soil properties (Example 2)

Go to the menu item "**System/Soil properties**".

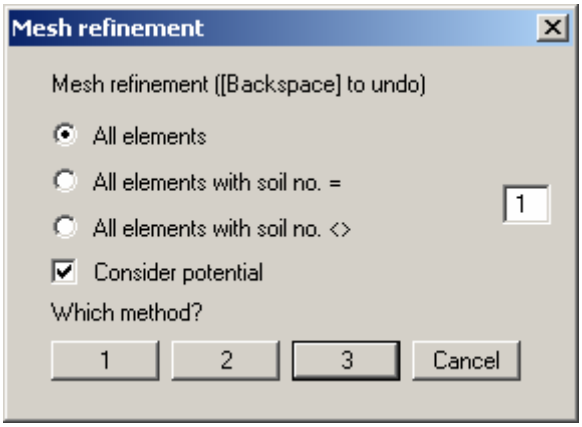


As you have assigned three different soil numbers, the dialog box now contains three input rows for soils. Enter the values in the dialog box. If you have no exaggerated demands on the position of the phreatic line, you can do without using the "**kr = f(u)**" buttons, otherwise read the section "**Theoretical principles/Vertical-plane system and axis-symmetrical system**" (see Section 6.3).

You can now delete the **Mini-CAD** data by clicking the "**Delete**" icon on the **Mini-CAD** toolbar.

9.2.7 Step 6: Refine and optimise mesh (Example 2)

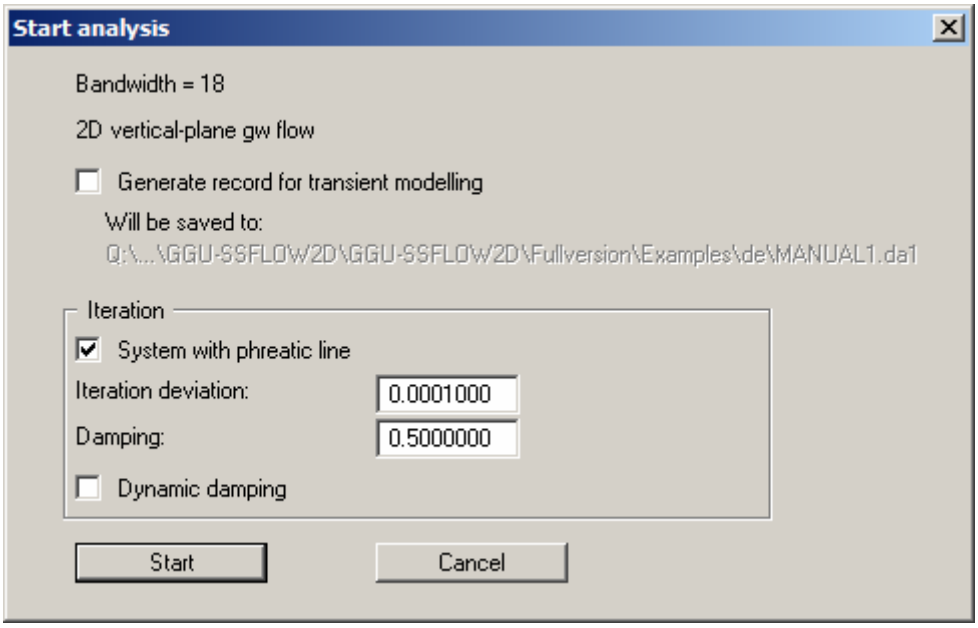
The system entered above has a rather coarse mesh. Although you could now analyse the existing system, carry out a mesh refinement first. Go to the menu item "**Mesh/(Refine) All**" and confirm by clicking button "**3**".



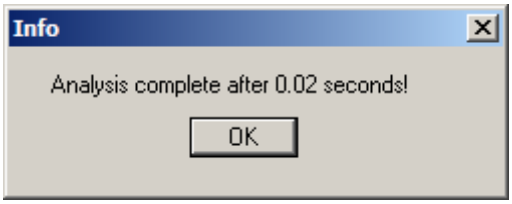
Go through this procedure a second time. You should now have a system with 272 triangles and 163 nodes. You can check this by going to the "**System/Info**" menu item. Go to the "**Mesh/Optimise**" menu item and then click the "**Topology**" button.

9.2.8 Step 7: Analyse system (Example 2)

Go to the menu item "**System/Analyse**" and confirm the iterative equation solver.

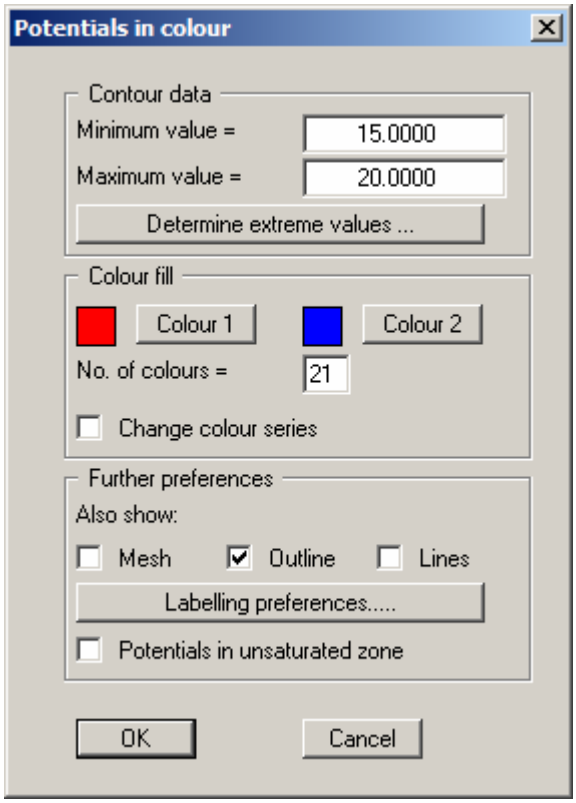


Enter a value of **0.5** for the damping and click the "**Start**" button. Depending on the power of your computer, you will informed after a certain time that analysis is complete.



9.2.9 Step 8: Evaluate potentials (Example 2)

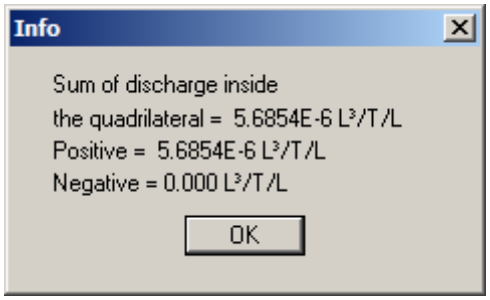
A colour-filled contour diagram of the potentials will be generated. Select the "**Evaluation/Coloured**" men u item and then "**Potentials**" from the subsequent option box. The following dialogue box then opens:



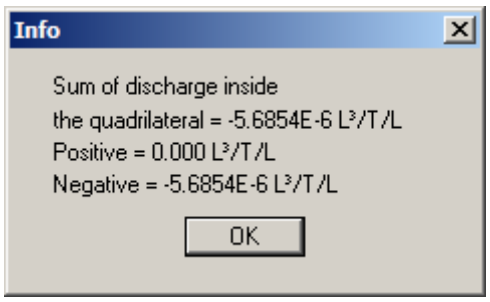
Press the "**Determine extreme values …**" button and then press "**OK**". If you wish, you can send the graphics to your printer.

9.2.10 Step 9: Evaluate discharges (Example 2)

Go to the "**Evaluation/Discharge**" menu item. Accept the display preferences and exit the dialog box using "**OK**". The system is now displayed with discharges. Using the rectangle, trace around the nodes on the water side that are labelled with a discharge. The sum of the discharges is displayed in a message box.



Trace around the nodes on the air side that are labelled with a discharge.



The same value is shown in the message box, but with a negative sign, because here the water exits the system.

The water percolating through the embankment is thus:

Q = 5.7 · 10^{-6} m³/s/running m of embankment = 0.5 m³/day/running m of embankment

9.2.11 Step 10: Analyse system without basin liner (Example 2)

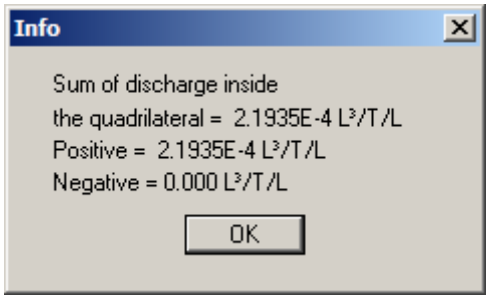
The system analysed in the previous section is to be investigated again, this time without a seal in the basin. The slope seal remains.

Go to the menu item "**Boundary/(Soils) In section**" and trace around the triangles at the base of the water side, which were defined as "**Clay**". Give these triangles the soil number "**1**" (= sand).

Go to the menu item "**System/Analyse**".

Display the potential contours once again (see Step 8, Section 9.2.9).

Determine the water percolating through the embankment (see Step 9, Section 9.2.10).



The discharge is, as expected, dramatically larger, by a factor of about 40:

Q = 2,2 · 10^{-4} m³/s/m embankment = 19 m³/d/m embankment

10 Index

E

F

G

H

I

K

L

M

N

O

P

Q

R

S

T

U

V

W

Z