Slope stability analysis and analysis of soil nailing and reinforced soil walls to DIN 4084 and EC 7

GGU-STABILITY

VERSION 14

Last revision: September 2022

Technical implementation, layout and sales: Civilserve GmbH, Steinfeld

Content:

List of Tables:

List of Figures:

1 Preface

The **GGU-STABILITY** program system allows slope failure investigations according to German Standard DIN 4084, DIN 4084:1996 and DIN 4084:2009, using circular slip surfaces (Bishop) and polygonal slip surfaces (Janbu, General Wedge method and Vertical slice method). Furthermore, it is possible to investigate soil nailing and reinforced soil walls. The formulas and relationships given in DIN 4084 are used (see "**Theoretical principles**" in Section 9).

The fundamentals of analysis using ***partial factors*** are given in EC 7/DIN 1054:2010. DIN 4084 contains the calculation procedures (Bishop, Janbu, General wedge method, etc.).

For bending design of the concrete shell according to EC 2, it is possible to export data to the **GGU-SLAB** program. Verification of punching according to EC 2 is implemented in the program module.

As a speciality, pore water pressure can be entered using not only the usual pore water pressure line, but also using a pore water pressure mesh. With a pore water pressure mesh, complicated flow conditions in a slope can be exactly modelled and taken into consideration for stability calculations. An interface to the groundwater program **GGU-2D-SSFLOW** is provided.

In 1996 Kölsch developed a constitutive equation for the analysis of slopes in waste materials (see 'The impact of fibrous constituents on the shear strength of domestic waste', Memoranda of the Leichtweiß Institute for Hydraulic Engineering at the TU Braunschweig, Bulletin 133/96 (*Der Einfluss der Faserbestandteile auf die Scherfestigkeit von Siedlungsabfall*). *Mitteilungen des Leichtweiß-Instituts für Wasserbau der TU Braunschweig, Heft 133/96*)). This constitutive equation is implemented in **GGU-STABILITY** as the "**Kölsch method**" and can be activated for the corresponding analysis. The theoretical principles are described in the section "**General notes on analysis with fibre cohesion**" (see Section 9.9), further literature can be found in Section 11.

The influence of vibrodisplacement compaction after Priebe can be investigated. It is also possible to include existing jointing. Moreover, you can have your system analysed using stabilisation columns after Neidhart/Gömmel.

The program system allows comfortable data input, which can be carried out almost completely with the mouse on the screen. Bothersome reading of the manual can be dispensed with, because

"**?**" buttons 

dealing with almost all geotechnical and program-specific problems are available in the dialog boxes. You are presented with the necessary information by clicking the "**?**" buttons (see also Section 6.1).

A variety of graphical presentation possibilities, to a high standard of quality, allow you to present the calculation results according to your wishes. Graphic 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, etc.) are supported. PDF and DXF files can also be imported by means of the integrated **Mini-CAD** module (see the "**Mini-CAD**" manual).

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

2 Capabilities

The **GGU-STABILITY** program has the following capabilities:

- Surface line with a maximum of 100 surface points
- 100 pore water pressure points
- 100 layers (no layer polygon input necessary !!!)
- 50 soil dowels
- 50 geosynthetics, can be directly selected as company products
- 150 tension members
- 150 steel strips
- 50 soils
- 50 point loads
- 40 permanent and/or live loads
- 50 horizontal distributed loads
- 1000 slices
- 5000 slip circle centre-points with any amount of radii
- 2500 polygonal slip bodies with a maximum of 200 polygon course points
- Polygonal slip surfaces after Janbu, General wedge method and Vertical slice method
- Safety statement after the principle of virtual movements for General wedge method (Goldscheider and Gudehus)
- Variation between two boundary slip bodies for polygonal slip bodies
- Exact consideration of pore water pressures via a pore water pressure mesh possible (optional)
- Interface to the groundwater modeller **GGU-2D-SSFLOW** for automatic import of pore water pressure mesh
- ASCII interface for automatic import of a pore water pressure mesh from third-party groundwater modellers
- Complete input and correction of system geometry possible using the mouse
- Consideration of passive earth pressure wedge (optional)
- Consideration of structural elements
- Consideration of vibrodisplacement compaction after Priebe
- Consideration of Neidhart/Gömmel stabilisation colums
- Use of any true-type fonts, guaranteeing an excellent layout
- Colour presentation of almost all system geometry. The colours can be freely specified by the user. In particular, the soil layers can be colour-filled. The colours can also be specified in accordance with DIN 4022 conventions.
- Zoom function
- "**Mini-CAD**" system (additional labelling, lines, rectangles, circles, graphics etc.)
- DXF import via "**Mini-CAD**"

- The program works on the principle of ***What you see is what you get***. This means that the screen presentation represents, on the whole, that which you will see on your printer. It also means that you can have the current screen contents sent to the printer at any stage during processing.

3 Licence protection

To protect the GGU software from unauthorised access, each GGU program is equipped with the ***CodeMeter software protection system*** from WIBU-Systems. Each GGU program is bound to a so-called ***CmContainer*** via a licence with the corresponding product code.

To be able to use the GGU licences in a CmContainer, a runtime environment, the ***CodeMeter Runtime Kit***, must be installed on your computer via driver software. To be simplified, we will refer to your computer with installed CodeMeter Runtime Kit and CmContainer as the ***CodeMeter licence server***.

We provide 3 alternative CmContainer types, that can be used on your CodeMeter licence server:

- CmStick
  The licence for your GGU program is stored in a USB dongle.
- CmActLicense (soft licence, not for virtual PC/servers)
  The licence for your GGU program is stored in a licence file that is bound to the hardware of a computer.
- CmCloudContainer
  The licence is located on a CmCloud server of WIBU-Systems and is mirrored to your CodeMeter licence server.

The GGU programs check at start-up and during runtime whether a corresponding license is available on a CmContainer.

4 Language selection

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

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

5 Starting the program

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

- File
- Info

After clicking the "**File**" menu, either an existing file can be loaded by means of the "**Load**" menu item, or a new system can be created using "**New**".

The program allows simple system input by moving directly to the "**Editor 1/Common systems**" menu item after "**New**" is clicked (see Section 10.2.10). The dialog box in this case is expanded somewhat to allow you to select the applicable standard and the required analysis method ("**Bishop**", etc.).

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. You will then return to the home screen. The program default standard is "**EC 7**" and the analysis method used employs circular slip surfaces after "**Bishop**". You then see ten menus in the menu bar:

- File
- Editor 1
- Editor 2
- Pwp mesh
- Centre-points/Slip body
- Safety factors/Utilisation factors
- Graphics preferences
- Page size + margins
- Nail wall
- Info

After clicking one of these menus, the so-called 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-STABILITY** 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.

6 Tips and tricks

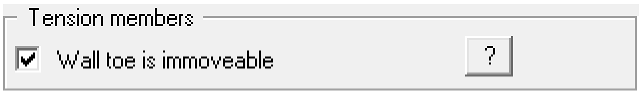
6.1 *"?" buttons*

Reading of the manual can mostly be dispensed with, because

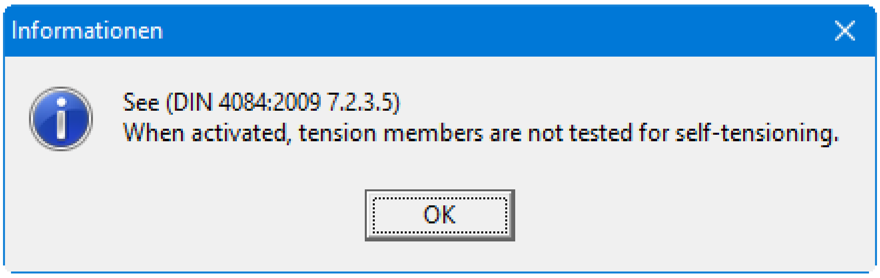
"**?**" buttons 

dealing with almost all geotechnical and program-specific problems are available in the dialog boxes. You are presented with the necessary information by clicking the "**?**" buttons.

For example, the following check box can be found in the "**Editor 1/Partial factors, …**" dialog box:



If you click on the question mark, you will see the following message box:



6.2 *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 "**Editor 2/Surface**".
- [**F5**] opens the menu items "**Safety/Utilisation factors (slip circles)/Analyse**" or "**Safety/Utilisation factors (polygonal)/Analyse slip bodies**".
- [**F6**] opens the menu item "**Editor 2/Layers**".
- [**F7**] opens the menu item "**Nail wall/Sliding, overturning, bearing failure**".
- [**F8**] opens the menu item "**Safety/Utilisation factors (slip circles)/Specific**".
- [**F9**] opens the menu item "**Page size + margins/Auto-resize**".
- [**F11**] activates the menu item "**Editor 1/Move objects**".

6.3 Keyboard and mouse

In most dialog boxes, the buttons for leaving the box or buttons for decisive functions are marked thickly and can be reached by clicking the [**Enter**]/[**Return**] key. In dialog boxes in which you have to make entries, e.g. change soil properties, the quickest way to jump to the next input box is to use the [**Tab**] key. The previous value is marked and can be overwritten directly with the new entry. You do not have to move to the box with the mouse and delete the old entry in advance.

You can scroll the screen with the keyboard using the cursor keys and the [**Page up**] and [**Page down**] keys. By clicking and dragging the mouse while holding down the [**Ctrl**] key, you activate the zoom function, i.e. the selected section is displayed full screen. By clicking and dragging the mouse while holding down the [**Shift**] and [**Ctrl**] keys, you zoom into your system graphic, i.e. you change the scale of your system representation.

You can also use the ***mouse wheel*** to zoom in or out or to move the screen display. When the program is started for the first time, the default setting for mouse wheel operation is activated according to Windows conventions

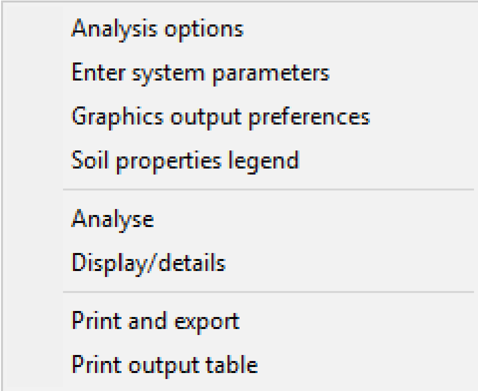
- Mouse wheel up = move screen image up
- Mouse wheel down = move screen image down
- [**Shift**] + mouse wheel up = move screen image right
- [**Shift**] + mouse wheel down = move screen image left
- [**Ctrl**] + mouse wheel up = enlarge screen image (zoom in)
- [**Ctrl**] + mouse wheel down = shrink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

If you want to change the system coordinates and the scale of your system using the mouse wheel, activate the "**Mouse wheel operation with global coordinates**" check box in the dialog box of the "**Graphics preferences/Zoom info**" menu item. ". If you close the program with this setting, the setting is still activated the next time you start it. You can now use the following mouse wheel functions to change the system:

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

From a zoomed representation you return to the full screen with [**Esc**]. You can undo a change in scale or system coordinates with [**F9**] (= menu item "**Page size + margins/Auto-resize**").

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 object immediately opens, allowing it to be edited.

After previously pressing the function key [**F11**], move legends and reference staffs to the desired position on the output sheet by holding down the left mouse button.

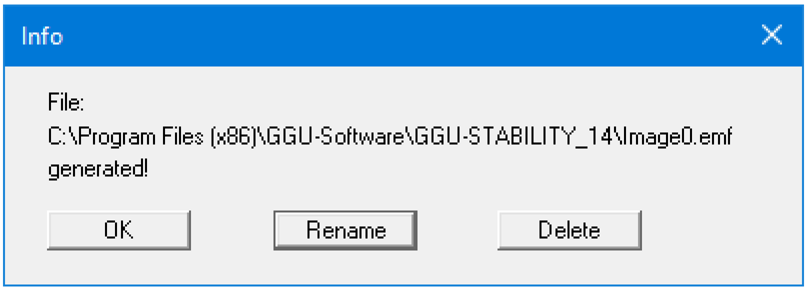
6.4 "Copy/print area" icon

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

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

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

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



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

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

7 Worked example 1: Data input with mouse

7.1 System description (Ex. 1)

The following simple slope shall be calculated using **Janbu**'s method:

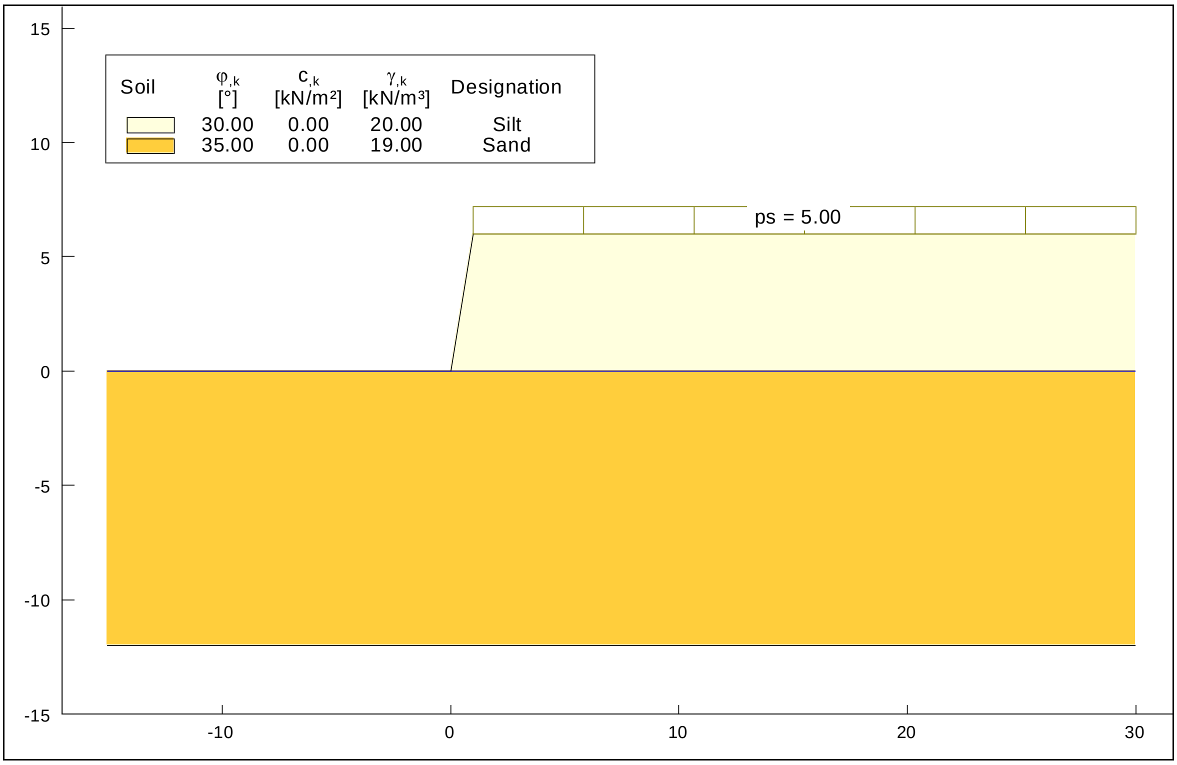


Figure 1 System of worked example 1

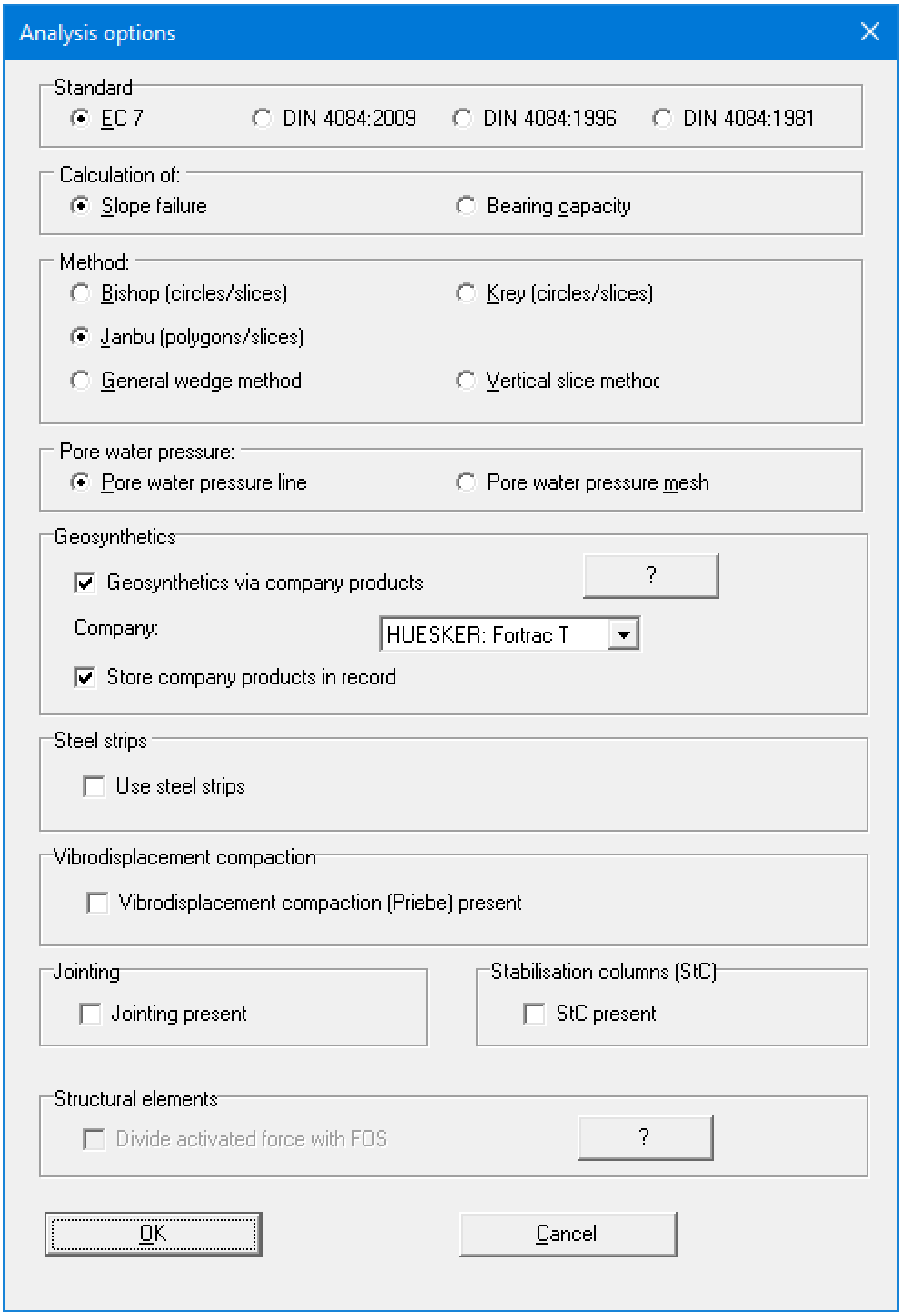
We have a 6.0 m high slope, to be reinforced with geosynthetics. The groundwater level is at surface level. The system has two differing soils. The soil properties are given in the legend. A permanent load of 5 kN/m² on the slope is to be considered.

In principle, system input can be carried out manually using direct numerical input, or with the mouse on the screen, or with a ***mixture*** of the two. Direct numerical system input is described in the "**Worked example 2: Data input via editor**" in Section 8. All further explanations concerning numerical system input can be found in Section 10.2.2 ("**Editor 1/Enter system parameters**" menu item).

Following, input with the mouse will be described.

7.2 Step 1: Select analysis options (Ex. 1)

After starting the program, the logo is at first displayed. Select the menu item "**File/New**" and leave the dialog box "**Common systems**" with "**No**". The "**Analysis options**" dialog box, which can also be accessed via the "**Editor 1**" menu, opens for input.



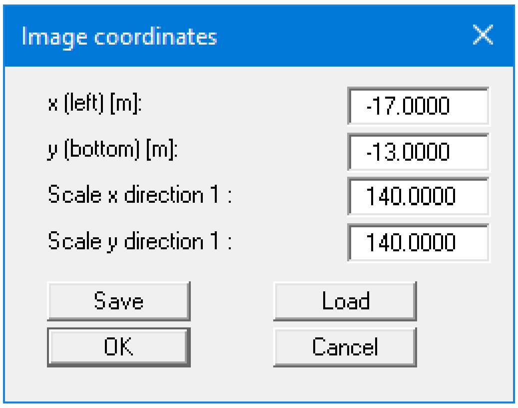
The default setting is the "**EC 7**" standard and the "**Bishop (circles/slices)**" calculation method.

Select the "**Janbu (polygons/slices)**" method for use with the example system. Because HUESKER geosynthetics are to be used later, in the "**Geosynthetics**" group box, "**Company:**" options menu, select "**HUESKER: Fortrac T**" from the displayed list of geosynthetics manufacturers and their product ranges.

After confirming with "**OK**" you will see a dialog box, in which you can enter the partial safety factors (see also Section 10.2.5). Using the "**To DIN 1054:2010**" button in the "**Default values**" group box, you can accept the partial factors for the various load cases. The partial safety factors for DS-P, which are selected by default, will be used in this example.

7.3 Step 2: Adjust page coordinates (Ex. 1)

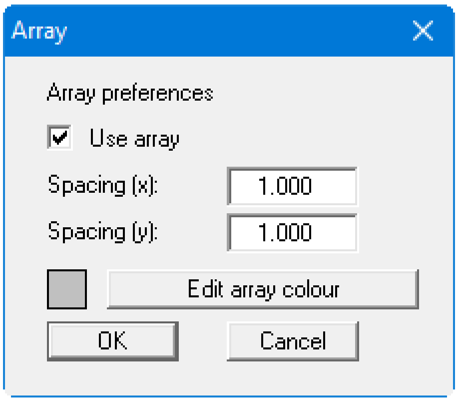
The program default coordinates do not fit the system and must be altered. Select the menu item "**Manual resize (editor)**" from the "**Page size + margins**" menu.



Enter the values from the above dialog box.

7.4 Step 3: Define array (Ex. 1)

Select the "**Array**" menu item from the "**Editor 2**" menu.



Activate the "**Use array**" check box. This ensures that the cross hairs snap onto these points during the following input with the mouse. This makes input of slope geometry easier. If you have a geometry which does not lend itself to being forced into such an array, do without activating the check box. Leave the dialog box with "**OK**". ***The array will be displayed after selecting one of the following menu items!***

If you own a scanner, you can produce a bitmap file of the system to be processed. This bitmap file can be imported using "**Graphics preferences/Mini-CAD toolbar**" menu item (see the "**Mini-CAD**" manual). You can thus further simplify system input.

7.5 Step 4: Enter surface and pore water pressure points (Ex. 1)

Select the "**Surface**" menu item from the "**Editor 2**" menu. You will first see an info box with possible ways of entering surface geometry. Then click on the surface line coordinates with the left mouse button. They will be continuously numbered from left to right. Erroneous input can be corrected by clicking with the right mouse button or be undone by using the [**Backspace**]-key.

For this example, click on the following four coordinate pairs:

x [m]	y [m]
-15.0	**0.0**
0.0	**0.0**
1.0	**6.0**
30.0	**6.0**

Table 1 Surface points of worked example 1

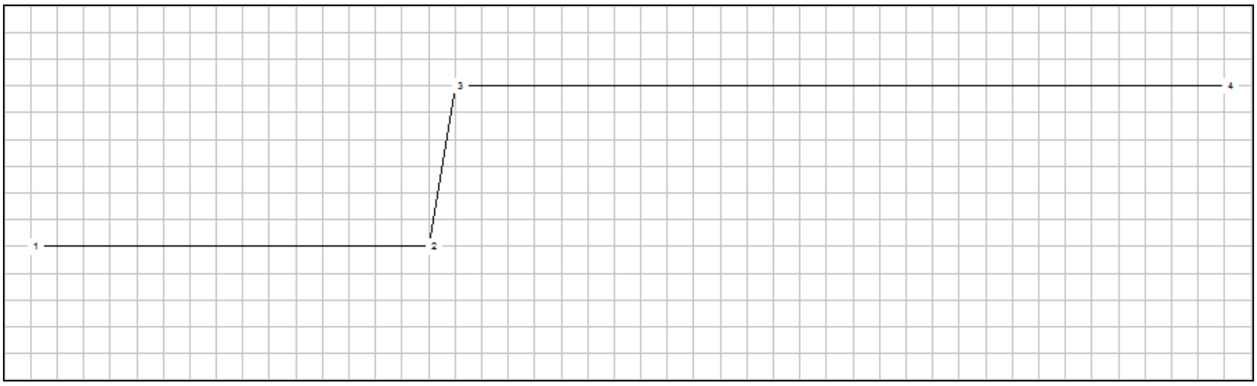


Figure 2 Surface points visualisation (worked example 1)

Select the "**Pore water pressure**" menu item from the "**Editor 2**" menu. Enter the pore water pressure line in analogy to the surface line. From the difference between a slice toe and the height position above this toe point of the pore water pressure line, the program later calculates the pore water pressure on the slice.

For this example, select a horizontal pore water pressure line at 0.0 m, which begins at x = -15 m and ends at x = +30 m and thus consists of two points only.

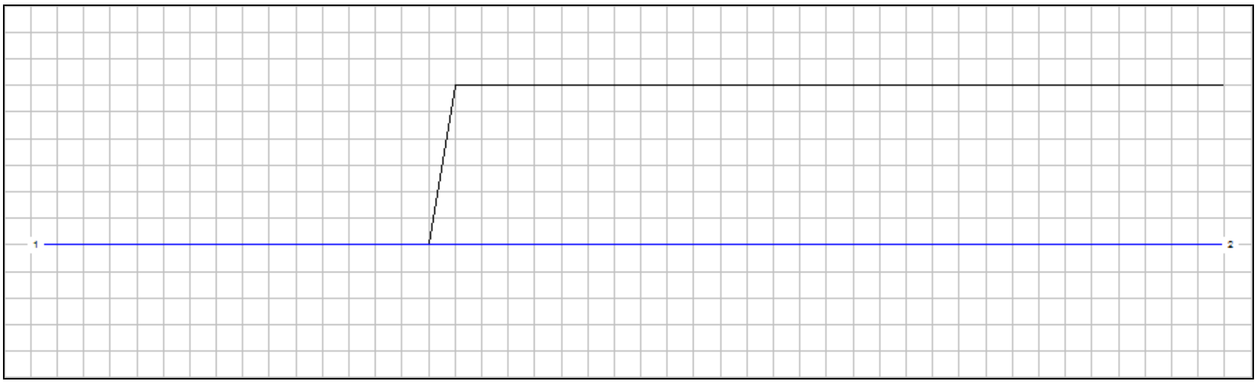


Figure 3 Pore water pressure visualisation (worked example 1)

7.6 Step 5: Enter soil layers (Ex. 1)

Select the "**Layers**" menu item from the "**Editor 2**" menu. A layer always consists of two coordinate pairs. Unlike many other programs, with **GGU-STABILITY** it is not necessary to enter a polygon course across the whole width of the system. Each layer has a soil number which is used to describe the soil properties (see second dialog box in Section 7.7). These soil properties are valid above the two coordinate pairs, as far as the surface line, or as far up as the base of a further layer. After input the layers are sorted automatically by depth and numbered. The soil number assigned to a layer is unaffected by the layer numbering (see Figure 4).

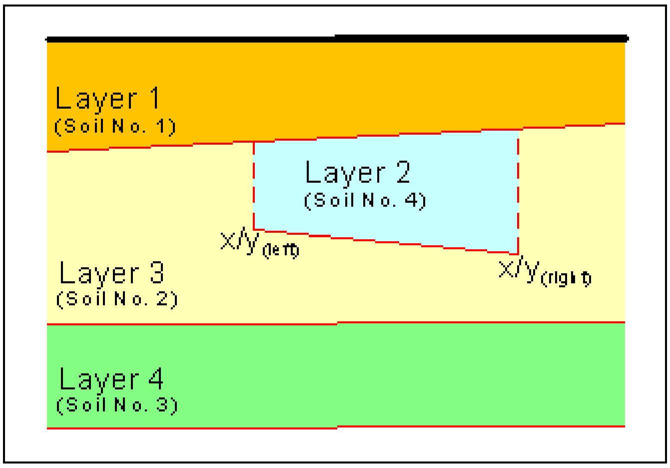


Figure 4 Layers and soil numbering

During input, observe that the layer boundaries may not cross. Nor are they allowed to intersect the surface. Both of these conditions are checked by the program immediately after layer input. Erroneous input is corrected or rejected, and an error message displayed.

For this example, select two layers with the coordinates:

- Layer 1 with soil number 1
  x/y (left) = 0.0/0.0
  x/y (right) = 30.0/0.0
- Layer 2 with soil number 2
  x/y (left) = -15.0/-12.0
  x/y (right) = 30.0/-12.0

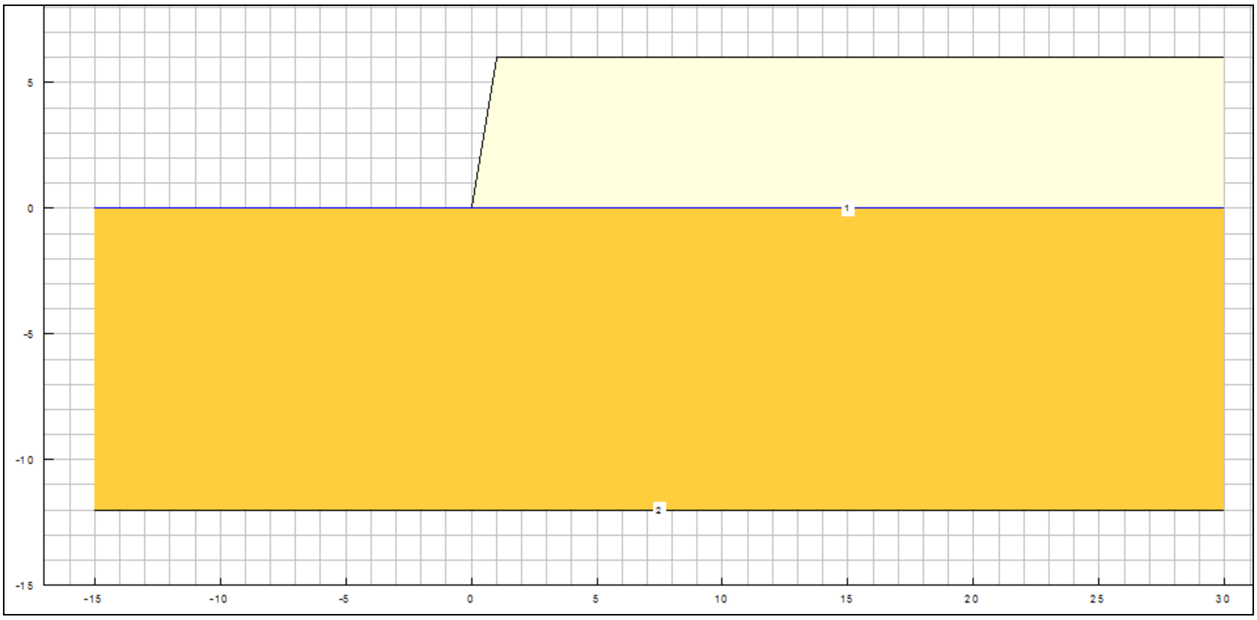


Figure 5 Soil layers visualisation (worked example 1)

The soil colours can be modified in the soil properties legend. Click in the legend using the left mouse button and then on the "**Colours**" button in the opened editor box (see menu item "**Editor 1/Soil properties legend**", Section 10.2.13).

Step 6: Enter water levels and loads (Ex. 1)

For calculation of the horizontal and the vertical water pressure above slope areas, the program also requires the water levels at the front ***left*** and front ***right*** of the slope. When the program starts the default values are 0.0 m (see first dialog box in Section 7.7). Water levels can be defined using the left or right mouse button after pointing to "**Editor 2/Water levels**" menu item. The example contains a slope without dammed water, so these values need not be altered.

The permanent load of 5 kN/m² for the example system is defined with the mouse in analogy to the previous input, using the menu item "**Editor 2/Loads/point loads**". First select the type of load from the dialog box. Click the "**Permanent loads**" button for the example used here. You will then see information on implementation; click the upper left and right slope points. Enter 5 kN/m² as the load for both sides in the following dialog box.

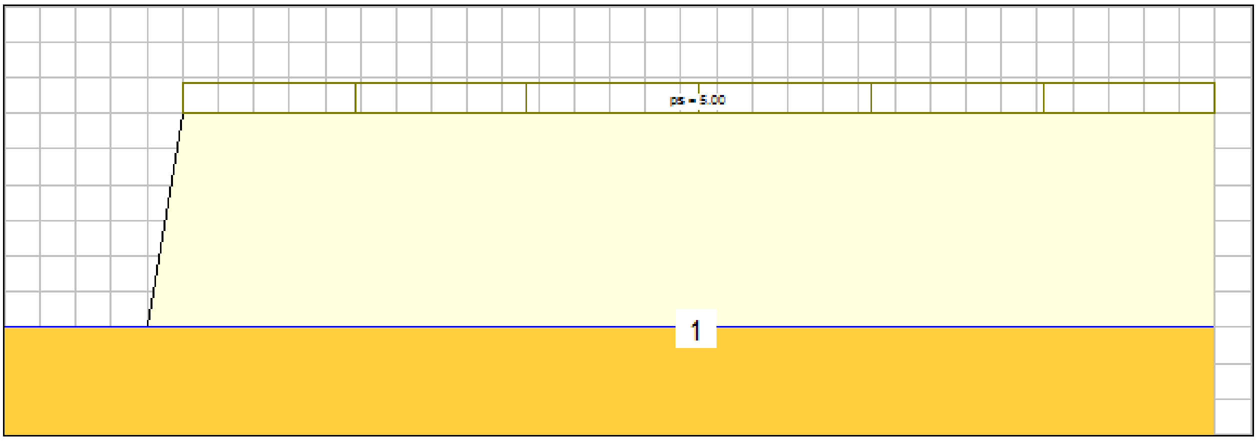
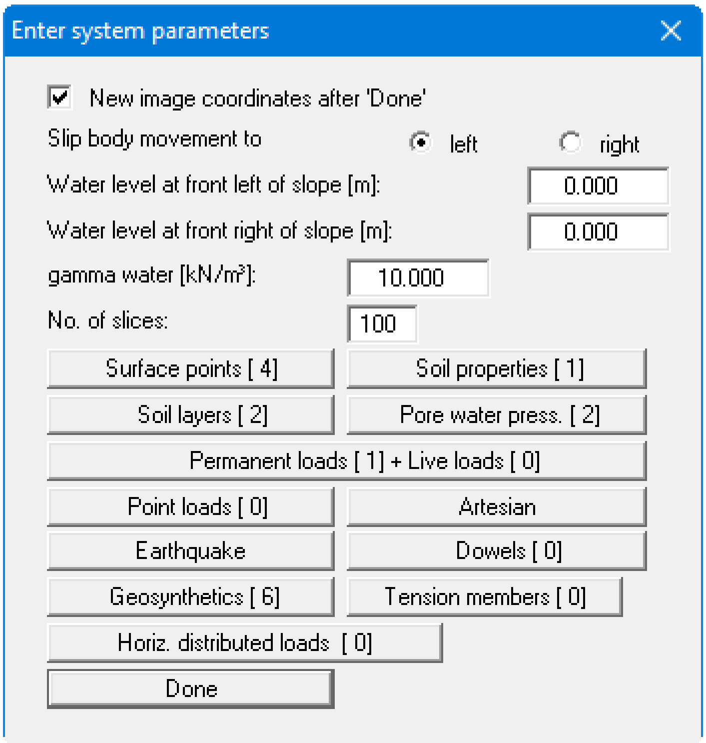


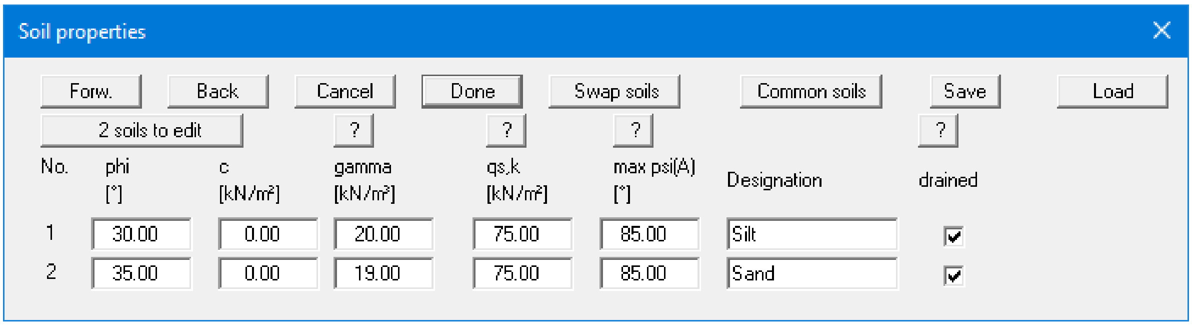
Figure 6 Permanent load visualisation (worked example 1)

7.7 Step 7: Enter soil properties (Ex. 1)

The slope geometry has now been described. Now, you only need to enter the soil properties. Select the "**Enter system parameters**" menu item from the "**Editor 1**" menu. You will see a dialog box for manual input of the system parameters. The values in brackets following the respective item show how many points, layers, etc. have been defined.



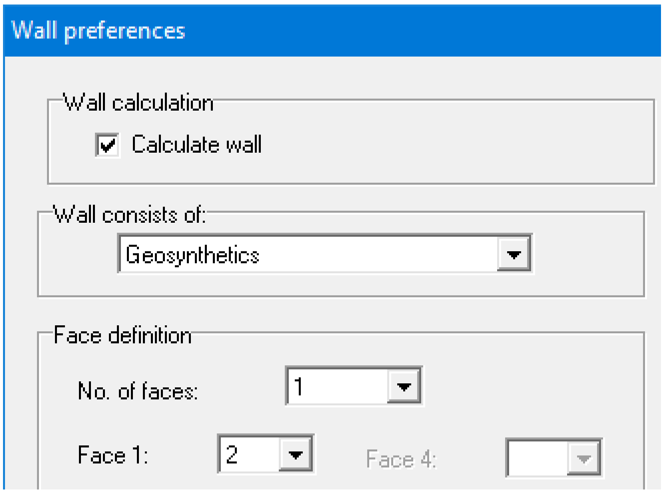
Select the "**Soil properties [x]**" button and click on the "**1 soil(s) to edit**" button in the dialog box. Enter the value "**2**" into the dialog box and confirm with "**OK**". Enter the soil properties given in the following dialog box.



When complete, leave this box and the box above using the "**Done**" button.

7.8 Step 8: Define reinforced soil wall preferences (Ex. 1)

Select the "**Nail wall/Preferences**" menu item and activate the "**Calculate wall**" check box. In the "**Wall consists of:**" group box select "**Geosynthetics**" as the nailing element.



The faces are sections of the defined terrain polygon course. They are continuously numbered from left to right. By assigning the (face) number you define which section of the terrain polygon course is the reinforced slope face (also see Figure 33 in Section 10.11.2). Assign the number ***2*** for "**Face 1**" because this is the reinforced slope face in the worked example. Confirm the remaining defaults with "**OK**".

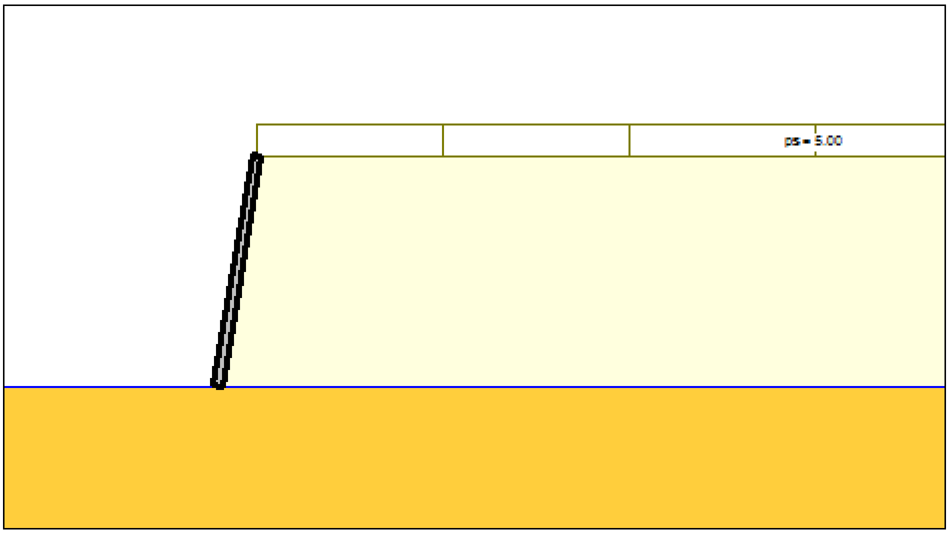
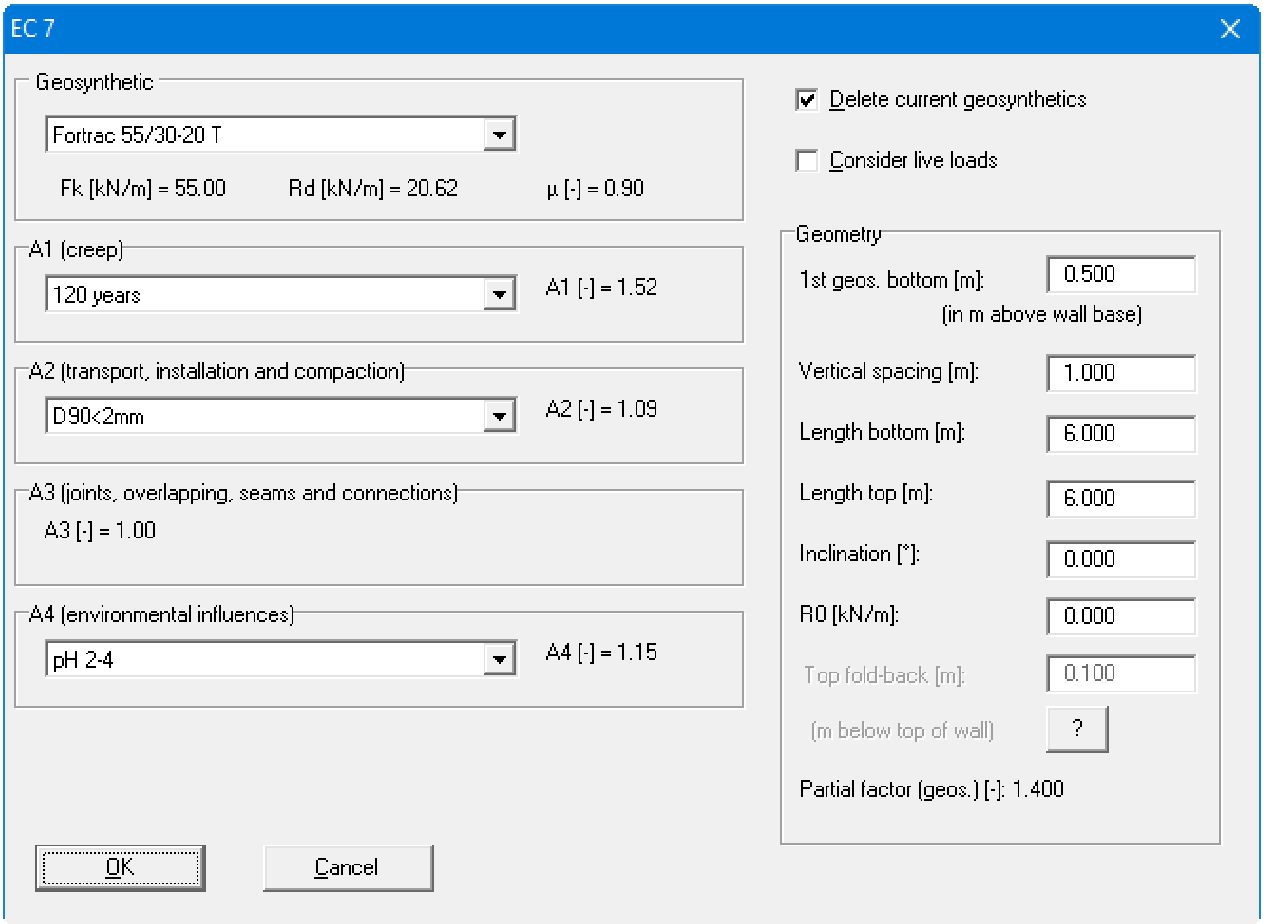


Figure 7 Reinforced slope face (worked example 1)

7.9 Step 9: Generate geosynthetics (Ex. 1)

Go to the "**Nail wall/Generate**" menu item. You will see the following dialog box. If geosynthetics have already been generated, activate the "**Delete current geosynthetics**" check box.

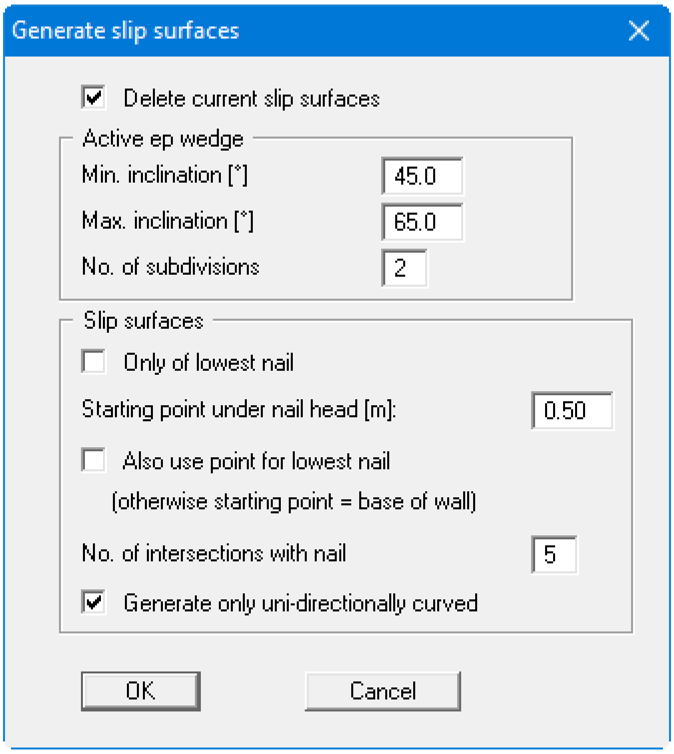


Enter the values shown in the dialog box and confirm with "**OK**". Six new geosynthetics are generated. After confirming the corresponding message box, a new prompt appears immediately. If you answer the "**Generate new slip surfaces?**" query with "**No**", the new geosynthetics are represented in the system and the defined values listed in a table in the "**Nail force legend**".

Normally and thus for the example, answer the query for the slip surface generation with "**Yes**". Generate an intersection if the geosynthetics starting points are not located at the wall.

7.10 Step 10: Generate slip surfaces (Ex. 1)

After answering the "**Generate new slip surfaces?**" query with "**Yes**" a dialog box opens, which can also be reached via the "**Nail wall/Generate slip surfaces**" menu item (see Section 10.11.11). Slip surfaces with 2 polygonal sections are generated (two-wedge failure mechanism).



In the upper group box of the dialog box, define the inclinations of the rear slip line, ***responsible*** for the active earth pressure acting on the system. The two inclinations given are used as limit values. The number of subdivisions used is given by "**No. of subdivisions**".

Define the location of the front slip line in the lower group box. The lowest slip lines always begin at the base of the wall and end at the tips of the nails. If the "**Only of lowest nail**" check box is deactivated, additional slip lines are also generated from the soil nails above. These lines will begin at the head of the respective nails or at a given distance below this; you can specify this distance with "**Starting point under nail head [m]**". The slip lines end at the nail foot if the slip line has a positive inclination; otherwise, the slip line is rejected (except for the lowest nail). If the "**No. of intersections with nail**" value is > 0, additional slip surfaces are generated, which intersect the respective nail.

The number of newly generated slip surfaces is shown in a message box. After closing the message box, the slip surfaces are immediately shown on the screen.

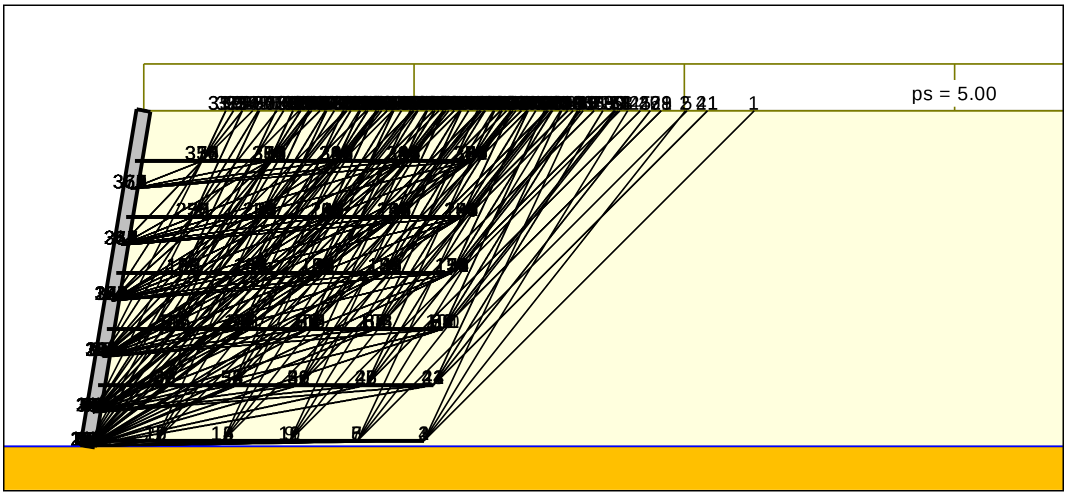
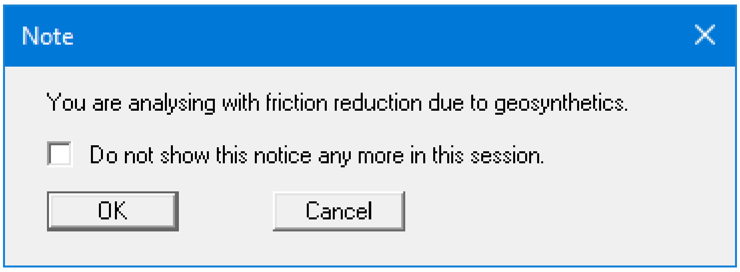


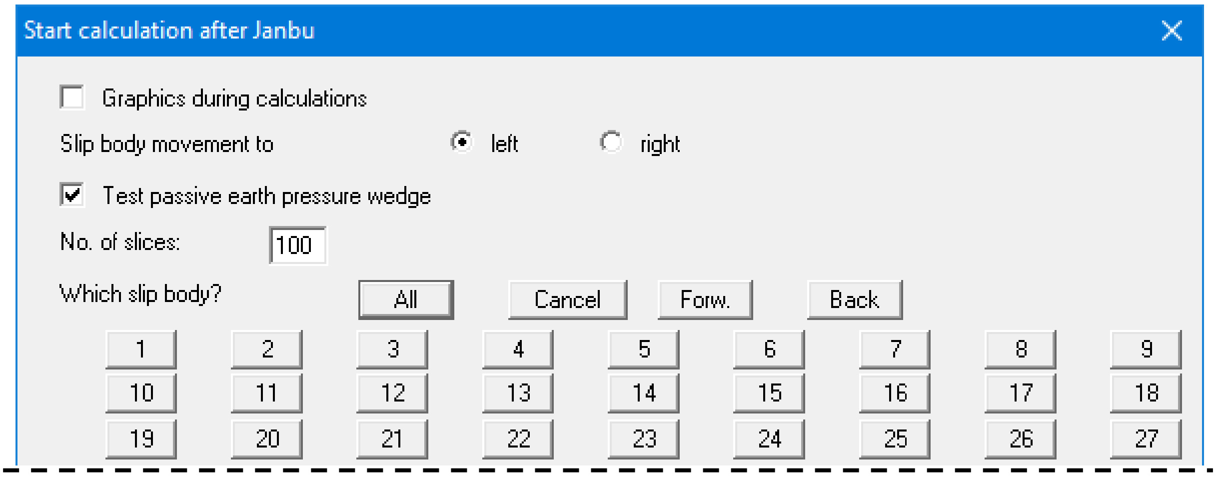
Figure 8 Slip surfaces in system (worked example 1)

7.11 Step 11: Analyse the slope (Ex. 1)

After defining the slip bodies the system can be analysed. Select the menu item "**Analyse slip bodies**" from the "**Utilisation factors**" menu. The following note is shown:



If the slip surface coincides with the geosynthetics, the tangent of the friction angle is multiplied by the geosynthetics reduction factor μ and correspondingly adopted for analysis. After confirming the note the following dialog box opens:



Adjust the number of slices to suit your wishes. If the "**Test passive earth pressure wedge**" check box is activated, the program will examine whether gradients greater than 45° - φ/2 occur in the passive earth pressure region of the respective slip body. These slip bodies will not be investigated. Slip body movement should to the "**left**". Begin calculation of all slip bodies using the "**All**" button. Once calculations are complete you will be shown some statistics.

Beside stability, the maximum geosynthetics forces resulting from slip body variation are determined. After confirming with "**OK**", the slip body with the lowest FOS is displayed. In the "**Utilisation factors**" menu, point at the "**Show individual slip bodies**" or "**All**" menu items in order to display other slip bodies or, indeed, all slip bodies.

7.12 Step 12: Evaluate and visualise the results (Ex. 1)

If you wish, you can now calculate safety against sliding, overturning and bearing capacity failure (menu item "**Nail wall/Sliding, overturning, bearing failure**", see Section 10.11.12). The active earth pressure on the reinforced soil wall, which is required for the three verifications, is then calculated with cohesion.

A further evaluation can be carried out using the menu items "**Nail wall/Maximum nail forces**" (Section 10.11.13) and "**Nail wall/Calculate earth pressure + weight**" (Section 10.11.15).

All principal data and results will be displayed in legends on the output sheet. Position the legends to suit your taste using the menu item "**Editor 1/Move objects**" (Section 10.2.15) or using the mouse after pressing [**F11**]. The legends can be most easily edited and modified by double-clicking with the left mouse button and then entering your preferences into the editor boxes.

Enter a company letterhead. To do this, activate the **Mini-CAD** system by pointing to "**Graphics preferences/Header toolbar**" and load the example file "**GGU-STABILITY-e.kpf**" using the "**Load**"  smart icon in the pop-up menu. If desired, save the file to your hard drive (menu item "**File/Save as**", Section 10.1.4).

8 Worked example 2: Data input via editor

8.1 Step 1: Select analysis options (Ex. 2)

Experience has shown that data input via editor is generally only used to correct geometry entered using the mouse. Nonetheless, this input method shall be demonstrated on a somewhat more complicated example. To improve understanding, you should follow the explanation in parallel on your computer.

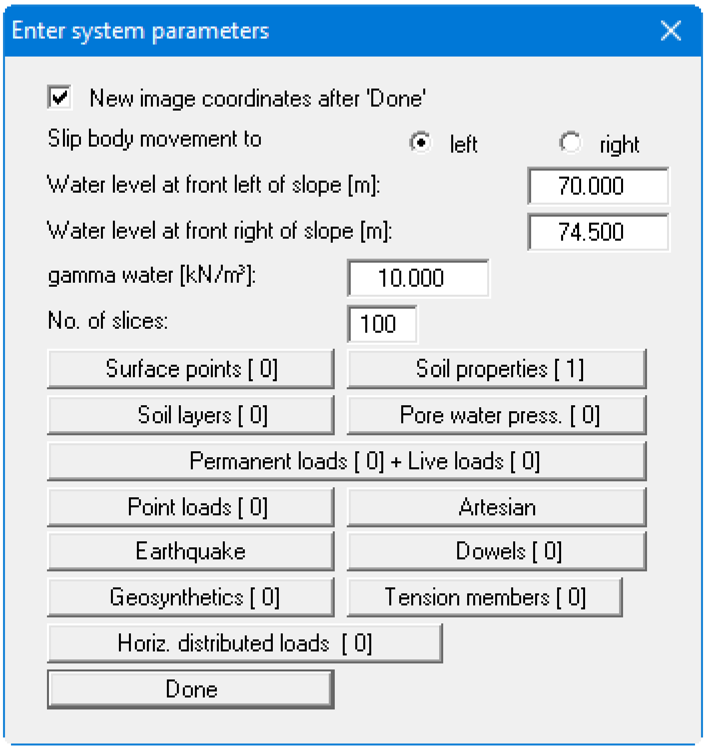
After starting the program, the logo is at first displayed. Select the menu item "**File**/**New**" and leave the dialog box "**Common systems**" with "**No**". The "**Analysis options**" dialog box, which can also be accessed via the "**Editor 1**" menu, opens for input. The default setting is the "**EC 7**" standard and the "**Bishop (circles/slices)**" calculation method.

After confirming with "**OK**" you will see a dialog box, in which you can enter the partial safety factors. Using the "**To DIN 1054:210**" button in the "**Default values**" group box, you can accept the partial factors given in the EC 7 for the various load cases. The partial factors for load case DS-P are used in this example.

If you have already worked with the program check and, where necessary, activate the mentioned switches in the "**Editor 1/Analysis options**" menu item.

8.2.1 Central dialog box

Go to the "**Enter system parameters**" item in the "**Editor 1**" menu. You will then see the central dialog box for the system to be processed. Now enter the water level values shown in the dialog box. Further system input is made by pressing the relevant buttons and entering the data in the subsequent dialog boxes. The following sections describe the individual buttons in more detail. After finishing data input, select the "**Done**" button.



For more inexperienced users it makes sense to have the input and settings displayed on the screen immediately after closing the above dialog box. The "**New image coordinates after 'Done'**" check box is therefore activated in the program's default settings. After an introductory phase this constant screen refresh may become onerous to experienced users, who can then deactivate this check box. The **GGU-STABILITY** program uses the latest settings next time it starts, regardless of whether an existing file is opened, or new input is made.

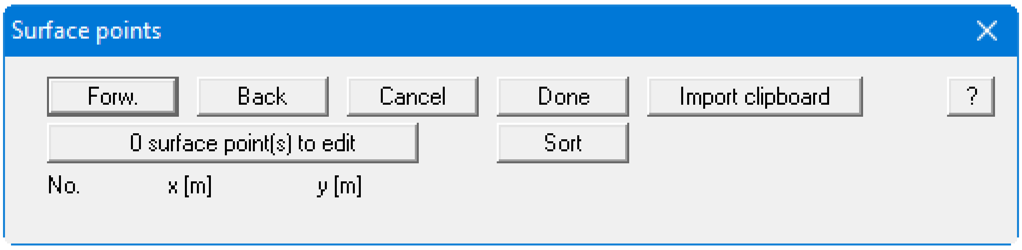
For wall analyses (see "**Nail wall**" menu"), only slopes failing to the left can currently be considered.

Note on water levels:
From the water levels the program calculates the water load in the area of a slice and the horizontal loading of the slope due to water pressure. If the water level is below surface level, it has no meaning for calculations.

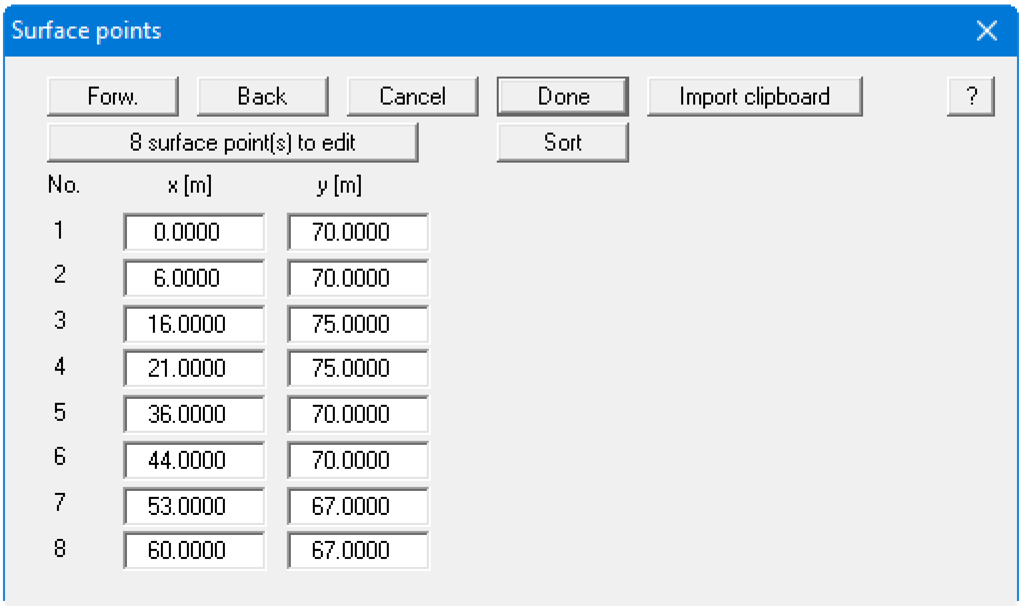
Note on number of slices:
Small numbers of slices mean low precision and shorter calculation times. Large slice numbers mean a correspondingly longer calculation time and higher precision. The minimum number of slices is also dependent upon the complexity of the slope. A slope which is heavily layered will require a larger number of slices than one which is homogenous. We recommend analysing using at least 50 slices.

8.2.2 Surface points (Ex. 2)

Next, the coordinates of the surface points must be specified. Select the "**Surface points**" button to do this.



The example system has 8 surface points. To edit the number of surface points, select "**0 surface point(s) to edit**". Then enter **8** as new number and confirm with "**OK**". You can now enter the 8 coordinate pairs (x/y). The example requires input of the following values:



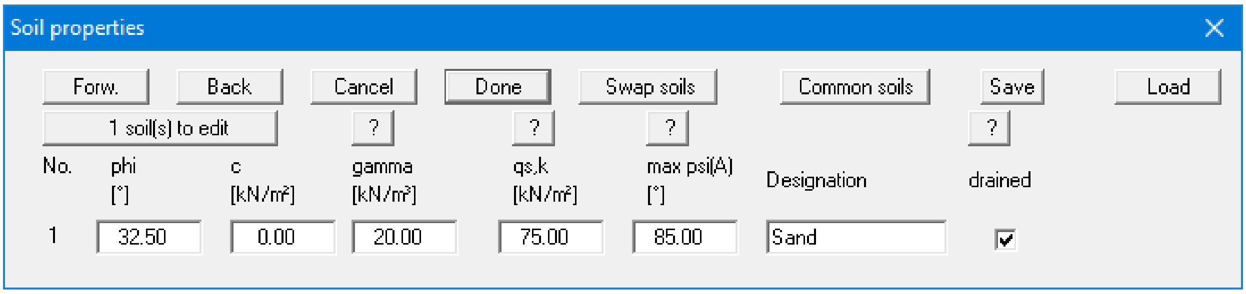
Note on surface points:
After leaving the dialog box (via the "**Done**" button), the coordinate pairs will be sorted according to increasing x-values. This allows simple deleting and inserting of surface points. You can delete a point by entering a "large" x-value and then reducing the number of points by 1. Inserting a point is achieved by increasing the number of points by 1 and then entering the coordinate pair at the end of the table. You can edit ***superfluous*** surface points even simpler using the mouse (see Section 7.5).

In the dialog box, 32 value pairs maximum are displayed at once. If the system has more than 32 value pairs, you can move through the table with the "**Forw.**" and "**Back**" buttons.

It is even easier to import surface points via the Windows clipboard. For example, if the x/y coordinates of the surface points 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 shown above by pressing "**Import clipboard**".

8.2.3 Soil properties (Ex. 2)

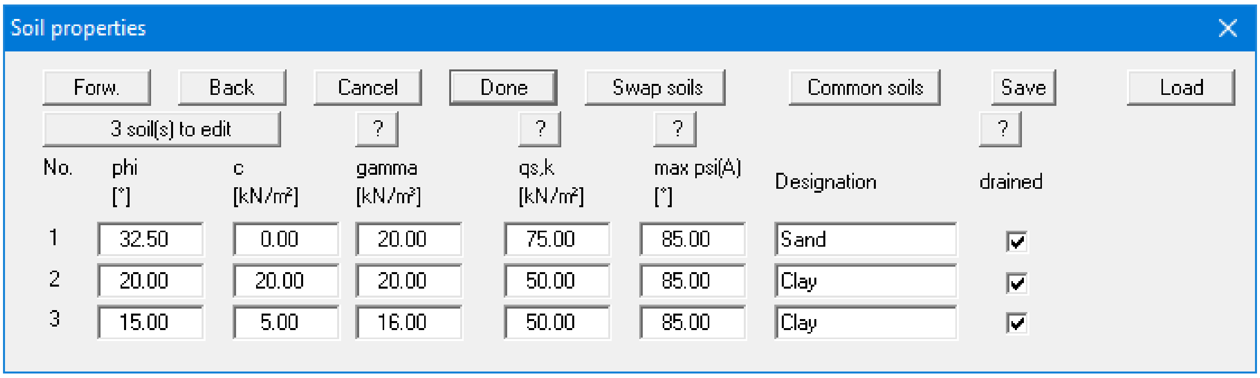
Next, the system soil properties must be specified. Press the "**Soil properties**" button in the central dialog box. The following dialog box opens:



For each soil information on friction angle, cohesion, unit weight and pore water pressure coefficient are needed.

> In saturated zones you enter unit weight of saturated soil. In unsaturated zones you enter unit weight of wet soil.
> Under no circumstances may, for the unit weight, the buoyant unit weight be entered, as the program considers vertical water pressures via pore water pressures, and thus achieves the most exact approach.

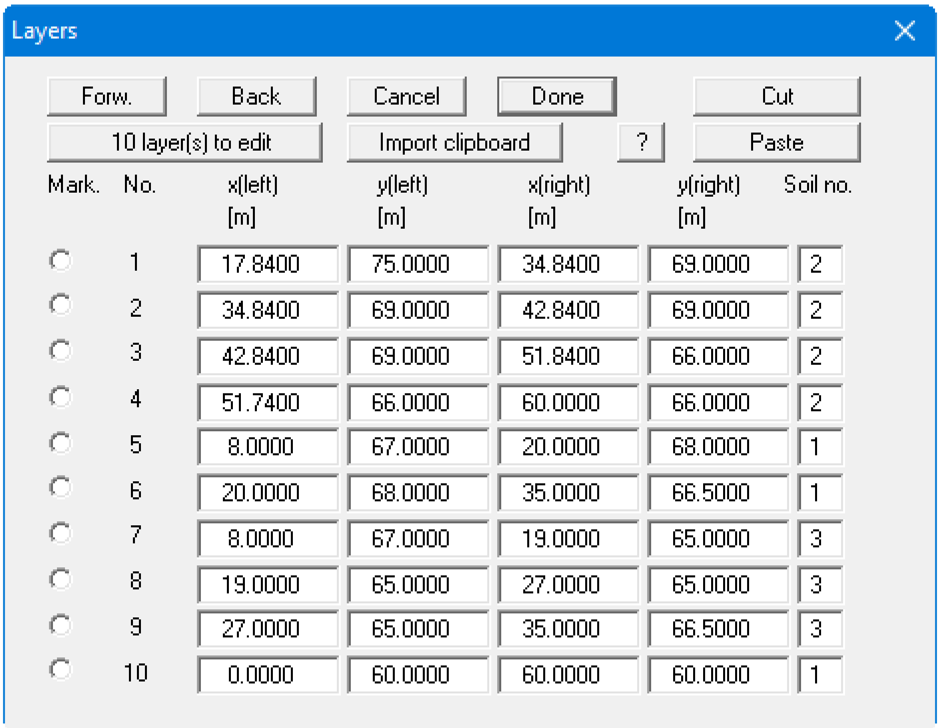
The example system has 3 differing soils. First, select the "**Soil properties**" button. To edit the number of soils, select "**x soil(s) to edit**". Then enter **3** as the new number and confirm with "**OK**". You can now enter the soil properties for example 2 as shown in the following dialog box:



8.2.4 Soil layers (Ex. 2)

The positions of the soil layers are now entered. The soil layers describe the stratification of the slope. A soil layer is defined by a polygon line with two x/y coordinate pairs as the lower edge of an overlying soil column. The area above these two coordinate pairs receives the soil properties of the soil also to be specified (as a number according to the numbering in the "**Soil properties**" dialog box). The soil layers (columns) extend upwards to the surface level or to the lower edge of any overlying soil layer. The soil layers must be entered in such a way that in all conceivable vertical sections the soil layer lying above another always has the lower layer number (layer number: do not confuse with soil number!!). This condition is checked by the program. If the condition is not met, an error message is displayed with the corresponding layer number and the possibility of correction. Incorrect entries are thus excluded.

The example system has 10 soil layers. First, select the "**Soil layers**" button. To edit the number of soil layers, select "**x layer(s) to edit**". Then enter **10** as the new number and confirm with "**OK**". The example requires input of the following values:



Mark.	No.	x(left) [m]	y(left) [m]	x(right) [m]	y(right) [m]	Soil no.
	1	17.8400	75.0000	34.8400	69.0000	2
	2	34.8400	69.0000	42.8400	69.0000	2
	3	42.8400	69.0000	51.8400	66.0000	2
	4	51.7400	66.0000	60.0000	66.0000	2
	5	8.0000	67.0000	20.0000	68.0000	1
	6	20.0000	68.0000	35.0000	66.5000	1
	7	8.0000	67.0000	19.0000	65.0000	3
	8	19.0000	65.0000	27.0000	65.0000	3
	9	27.0000	65.0000	35.0000	66.5000	3
	10	0.0000	60.0000	60.0000	60.0000	1

The dialog box for the soil layers has two additional buttons in comparison to the corresponding boxes for surface points and soil properties

- "**Cut**" and
- "**Paste**".

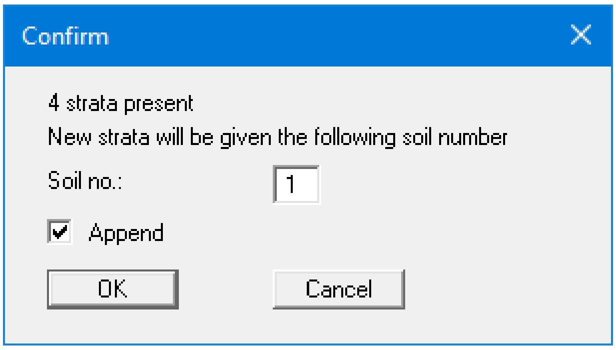
Further, small boxes are placed in front of each of the layer values. To delete a layer you must click on the box of the layer to be deleted and then select the "**Cut**" button. The layer will then be deleted. The layer values are copied to an internal buffer and used as the new values for a layer if a "**Paste**" is necessary. This makes it easy to reorganise layers. Pasting of layer is analogous to cutting. A new layer will be pasted in front of the marked layer. Correct sorting can also be left to the program by selecting the "**Test system**" menu item from the "**Editor 1**" menu. The program will find any irregularities and offer correction possibilities.

It is even easier to import soil layers via the Windows clipboard. If you have the information on the soil layers in an Excel spreadsheet, for example, you can copy the corresponding columns there to the clipboard ("*Edit/Copy*") and then enter these values using the "**Import clipboard**" button in the above dialog box.

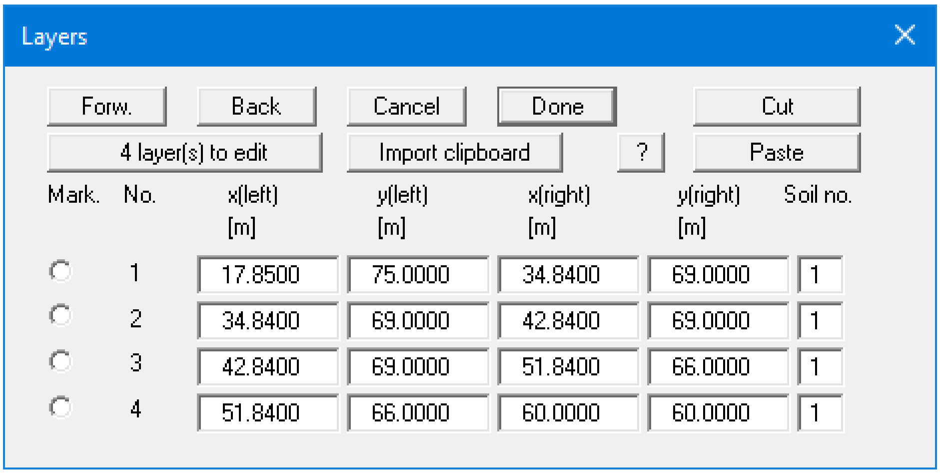
Soil no. 2		Soil no. 1		Soil no. 3		Soil no. 1	
x	y	x	y	x	y	x	y
17,85	75	8	67	8	67	0	60
34,84	69	20	68	19	65	60	60
42,84	69	35	66,5	27	65		
51,84	66			35	66,5		
60	60						

For contiguous soil layers the end point of the first line is automatically adopted as the start point of the new line. When reading in coordinate pairs a dialog box shows how many layers are defined. You also assign the soil number here, so that the soil properties are defined for this soil layer.

If you copy the coordinates for the layers that are to have soil no. 2 from the above Excel table and click on "**Import clipboard**", the following therefore appears. The layers are to be added to the already defined layers, hence the "**Append**" check box is activated.



Before reading in the coordinates, select the soil number for which you are inserting the layer data. For a completely new system the layers (soil columns) are taken over into the above dialog box and inserted as shown below:



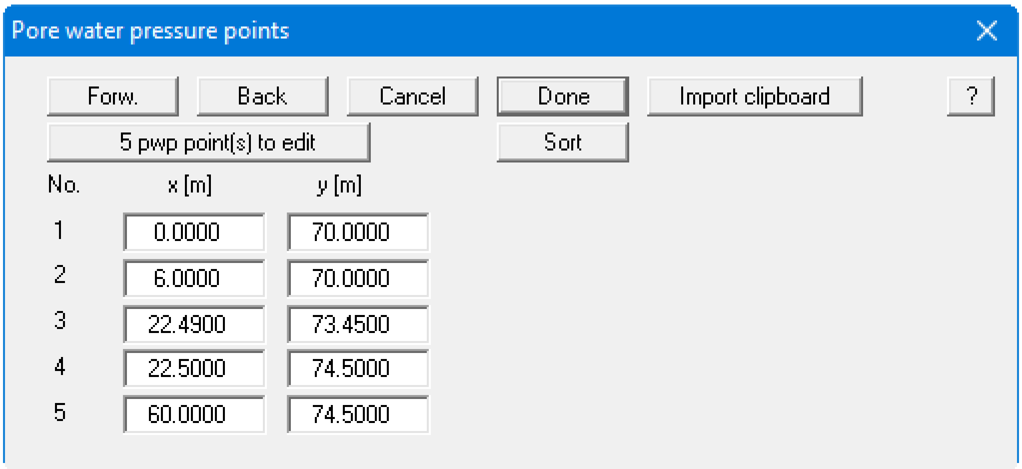
8.2.5 Pore water pressure (Ex. 2)

The coordinates of the pore water pressure line are entered as a polygon course. The x-values of the polygon course must increase from left to right. From the vertical distance between the slice foot and the point on the pore water pressure line above this, the program calculates the valid pore water pressure (u) for the slice. If the pore water pressure line lies below the slice foot, the pore water pressure will be set to zero.

The pore water pressure line must cover the whole of the area to be investigated in the calculations. If no pore water pressures are present you can simply define a pore water pressure line which consists of two points, and which runs below any possible y-values of the slice feet.

If a phreatic line is present within the slope, this line is generally a pore water pressure line.

The example system has 5 pore water pressure points. First, select the "**Pore water press.**" button in the central dialog box and then, alter the number of pore water pressure points to **5** using the "**x pwp point(s) to edit**" button. Enter the values shown in the following dialog box:

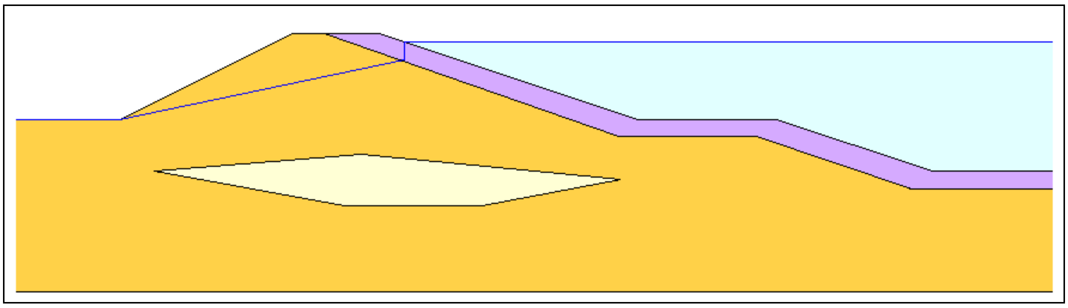


For deleting and pasting of pore water pressure points see the note in "**Surface points**" (Section 8.2.2).

The pore water pressure points can also be imported via the Windows clipboard. For example, if the x/y coordinates of the pore water pressure points are available in an Excel table, it is possible to copy the two columns containing the data into the Windows clipboard ("*Edit/Copy*") and then to paste them into the dialog box shown above by pressing "**Import clipboard**".

8.3 Step 3: Check and save system (Ex. 2)

After finishing input, you can select the "**Test system**" menu item from the "**Editor 1**" menu and have the ***correctness*** of your system input checked by the program. If errors are detected, an error message will be produced with a note on the faulty area of the data set. The program will then offer correction possibilities for all erroneous input. The following system will be shown on the screen:



You can now save the data set, in order to have it available again for a possible later sitting. Then select the menu item "**Save as**" from the "**File**" menu. A file requester box then appears: Enter the desired file name. If you do not enter a suffix when saving, "**.boe**" will be used automatically. You should always use the default suffix "**.boe**", as this is the filter used when loading files. This greatly increases the clarity of the file selector box.

8.4 Step 4: Define slip circles (Ex. 2)

8.4.1 Define centre-points (Ex. 2)

As the system input is complete, you must now describe the geometry of the slip circles. First, you have to define the centre-points. For this, the following menu items of the "**Centre-points**" menu are provided:

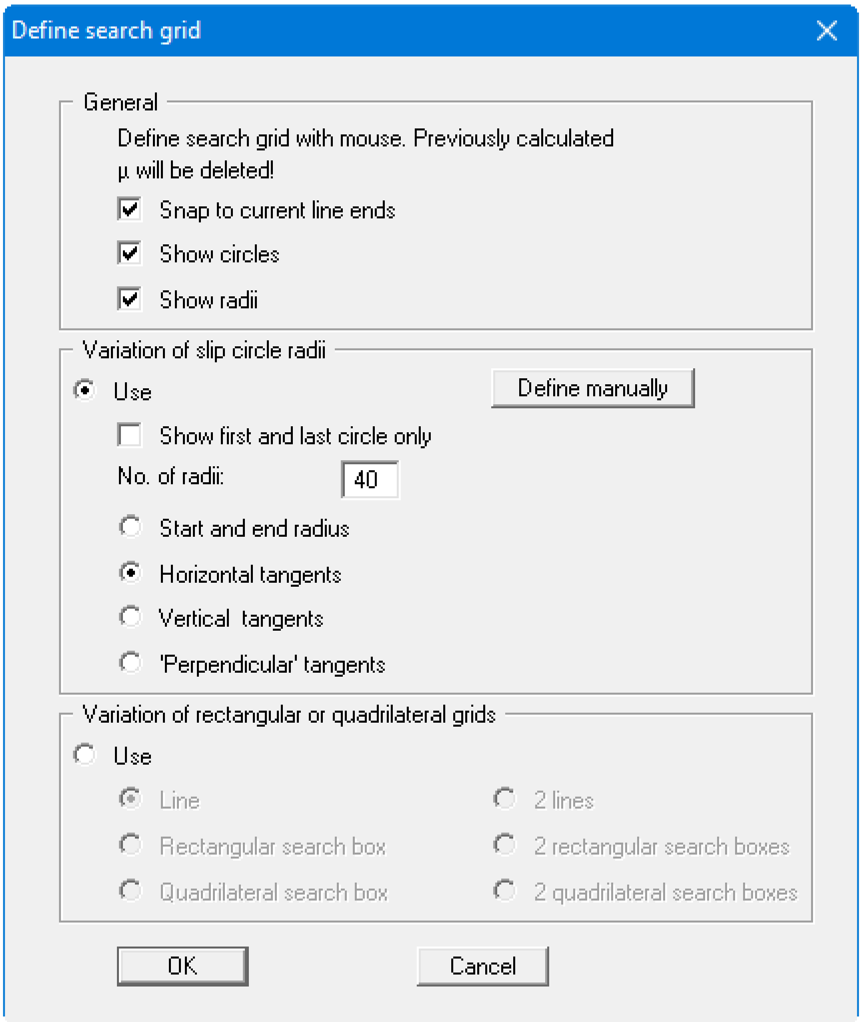
- "**Define in quadrilateral**"
  You click on the four corners of a quadrilateral with the mouse. You can then select an array subdivision. The centre-points will be set in accordance with the size of the quadrilateral (see also Section 10.5.1).
- "**In rectangle**"
  You click on two diametrically opposed points of a rectangle and enter an array subdivision. The centre-points will be set in accordance with the size of the rectangle (see also Section 10.5.2).
- "**Individually (graphically)**"
  You define the coordinates of individual centre-points by clicking with the left mouse button (see also Section 10.5.3).
- "**Individually (editor)**"
  You enter the centre-point coordinates directly using the keyboard (see also Section 10.5.4).

A current array can be supplemented and refined at any time. You can also swap between the described methods above. A total of 5000 slip circle centre-points can be entered.

The area above the slope which is ***clickable*** with the mouse is somewhat small. Select the "**Zoom**" menu item from the "**Page size + margins**" menu. Confirm the default zoom factor of 0.8 with "**OK**". Now select the "**Individually (graphically**)" menu item from the "**Centre-points**" menu and then click on some centre-points in front of the slope with the left mouse button. It is now up to you to try using one of the other methods for centre-point definition.

8.4.2 Define search grid (Ex. 2)

After completing the centre-point definition, a search grid must be specified. Select the "**Define search grid**" item from the "**Centre-points**" menu. The following dialog box appears:



In the "**General**" group box you can alter presentation preferences. In the other two input group boxes you can vary the search grid by means of slip circle radii or rectangular and quadrilateral search boxes. Activate the "**Use**" button in the required variation group box. In addition to the following discussion, further examples for defining the search grid can be found in the "**Centre-points/Define search grid**" menu item in Section 10.5.7.

For "**Variation of slip circle radii**" the search grid is characterised by two points A and B, which must be clicked by the mouse. There are four different types of definition:

- "**Start and end radius**"
  All radii pass through point A and point B. Subdivisions between these two are in accordance with the user-defined number of radii.

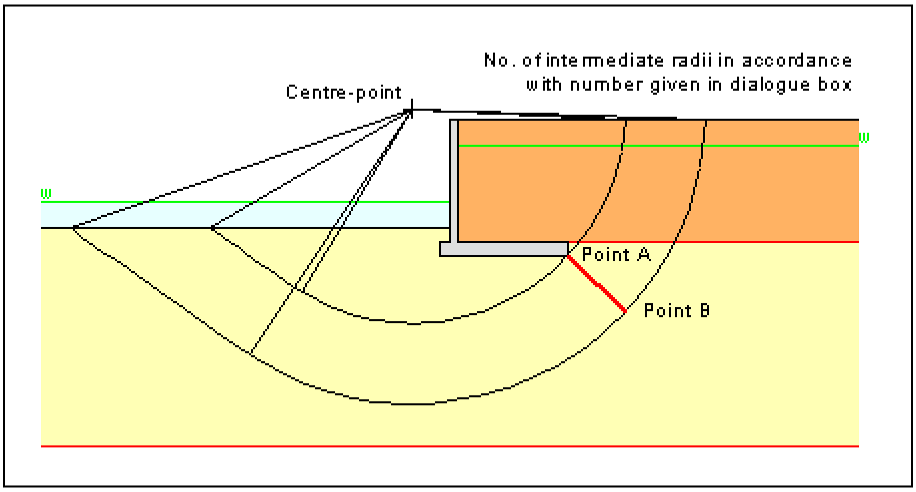


Figure 9 Search grid slip circle radii via start and end radius

- "**Horizontal tangents**"
  All radii touch a horizontal tangent, which is defined by the points A and B. Subdivisions between these two are in accordance with the user-specified number of radii.

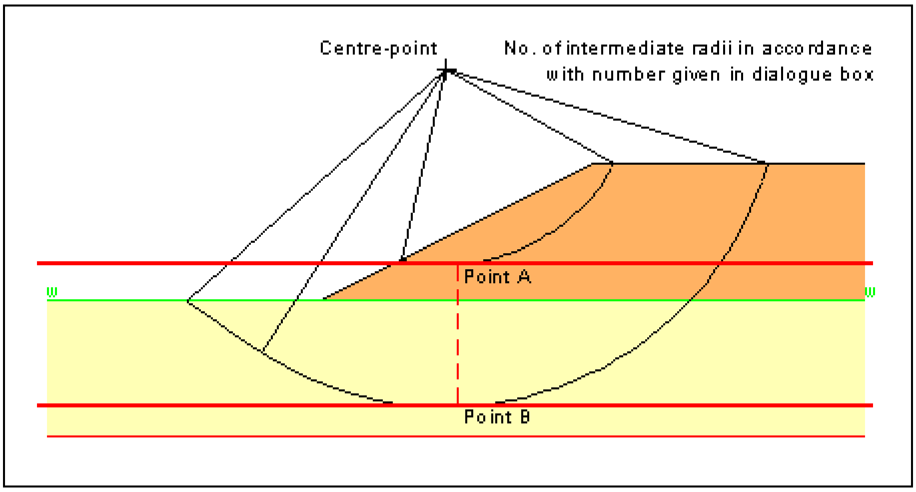


Figure 10 Search grid slip circle radii via horizontal tangents

If the location of the governing search grid cannot be unequivocally identified from the system, it makes sense to define ***Top of slope*** and ***Base of system*** as points A and B.

- "**Vertical tangents**"
  All radii touch a vertical tangent, which is defined by the points A and B. Subdivisions between these two are in accordance with the user-specified number of radii. This method is of only minor importance.

- "**Perpendicular tangents**"
  All radii touch a tangent, the rise of which is specified as being normal to a line connecting points A and B. Subdivisions between these two are in accordance with the user-specified number of radii. This method is of only minor importance.

After confirming with "**OK**", click on two points (A and B) within the slope. The boundary radii will then be displayed. If you are not satisfied with your choice, you can then immediately click on two new points.

> **Note:**
> For bearing capacity analysis with a footing within the slope, definition of a search grid is not necessary. The search grid will be set automatically. The left or right footing edge will be defined as search radius depending on slip body movement.

For "**Variation of rectangular or quadrilateral grids**" you also have the option of selecting between a number of methods. After selecting the method to be adopted and entering the subdivisions in a dialog box, the selected element (line, rectangular search box, etc.) is displayed on the screen. In this manner you have an example for application of the individual procedures and, in analogy, can subsequently try out your own definitions.

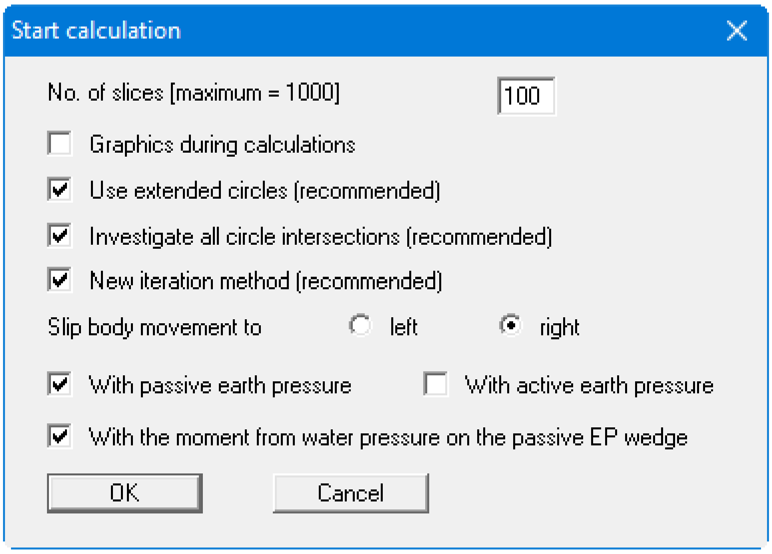
For example, select "**Rectangular search box**". The number and spacing of the slip circles is varied according to the defined array subdivision and the size of the search box subsequently represented on the screen. The search grid is restricted to the array used in the line or search box. To facilitate better understanding, try a gradual increase in subdivisions in x and y directions from ***1*** upwards.

If, in the above dialog box, you activate the "**Snap to current line ends**" check box, a small square is displayed around the mouse points. The mouse pointer then locks on to the end points of surface lines, etc., that lie within this square when clicked.

If, in the dialog box above, the "**Show circles**" and "**Show radii**" check boxes are deactivated, the limit radii will not be displayed when defining the search area. This generally only makes sense for very slow computers. This is similar to the "**Show first and last circle only**" check box, which allows only the limit radii of the first and last centre-points to be displayed. The configuration of these check boxes does not influence subsequent analysis evaluation.

8.5 Step 5: Analyse slope with circular slip surfaces (Ex. 2)

When the definition is according to your wishes, calculations can be started. Select the "**Analyse**" menu item from the "**Utilisation factors**" menu. You can also initiate the analysis using the [**F5**] function key and see the same start dialog box.



Adjust the number of slices to suit your wishes. Slip body movement should be to the "**right**". The "**With passive earth pressure**" check box should remain activated. This enables consideration of a passive earth pressure wedge in those areas in which the slip surface is steeper than 45° - φ/2. Confirm your input with "**OK**". Computation will then be performed using the data provided.

If you chose to view the graphical representation during computation you will now see the course of the calculations, together with safety factor data, on the screen. A large number of slip circles can greatly extend computation time. Deactivate the "**Graphics during calculations**" check box.

Once calculations are complete you will be shown some statistics. After confirming with "**OK**", the minimum safety factors corresponding to each centre-point will be displayed. The slip circle with the lowest safety will also be graphically displayed. If the most unfavourable slip circle centre-point is at the edge of all defined centre-points, and you would like to determine the position more precisely, go back to the "**Centre-points**" menu and supplement the centre-point array as desired.

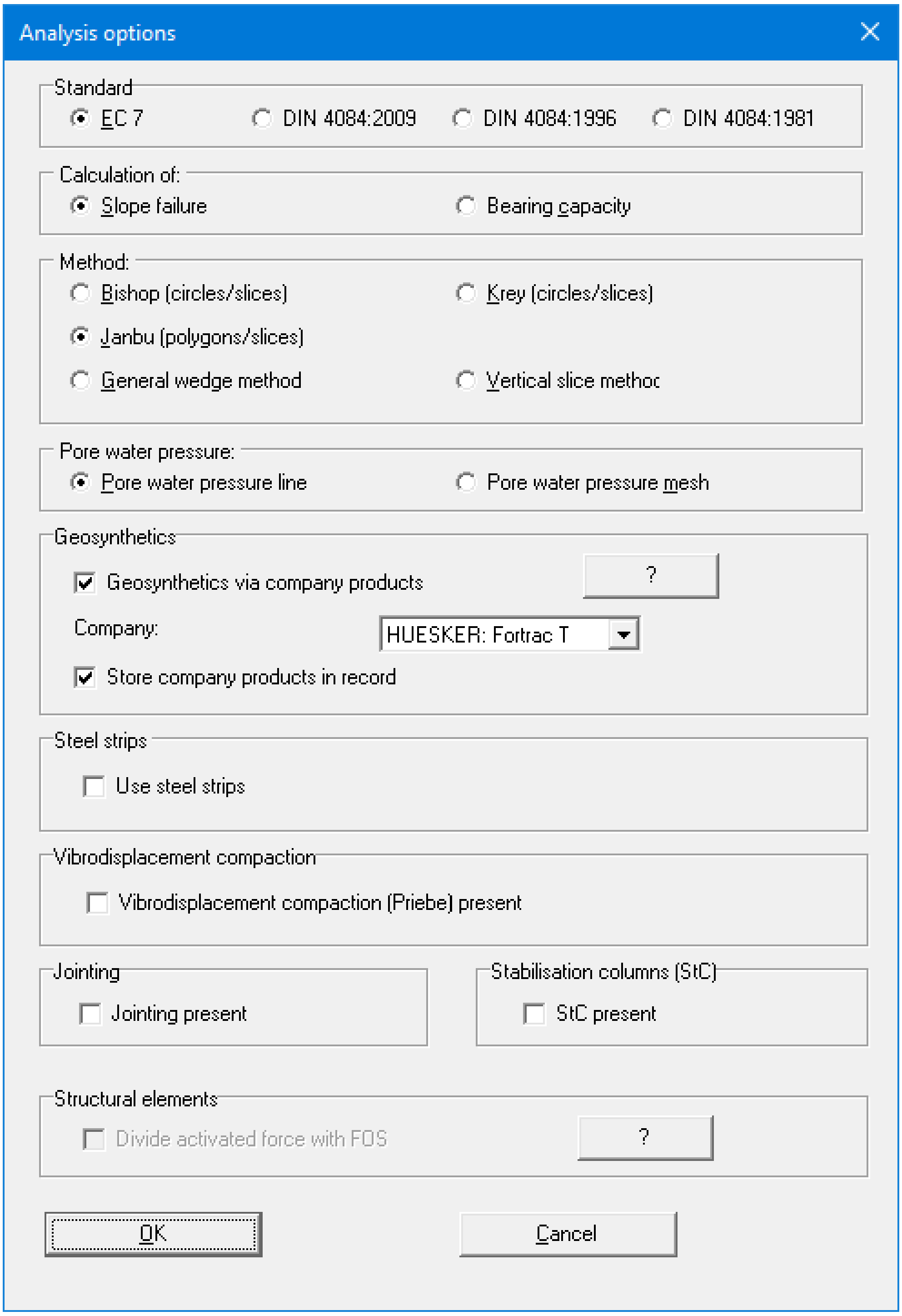
8.6 Step 6: Evaluate and visualise the results (Ex. 2)

When everything is according to your wishes, you have a variety of evaluation possibilities.

- If you have defined and calculated more than three slip circle centre-points, you can create contour graphics using the menu item "**Utilisation factors/Contours**" or "**Utilisation factors/Coloured**".
- Besides the most unfavourable slip circles, you can also have further slip circles displayed (menu item "**Utilisation factors/Specific**").
- You can have all slip circles displayed (menu item "**Utilisation factors/All**").
- You can have the pore water pressure course on the slip surface displayed (menu item "**Utilisation factors/Preferences**")
- You can have numerical values from calculations displayed for every slip circle using the menu item "**Utilisation factors/Display/Details**".
- You can have a complete calculation protocol sent to the printer (menu item "**File/Print output table**". The output can also be sent to a file, in order to be imported into, e.g., a word processor.
- The graphics can be supplemented with "**Mini-CAD**", and explanations added.
- You can save your work to a file. Select the menu item "**File/Save as**" to do this.
- The ***What you see is what you get*** principle provides the possibility of sending the current screen contents to the printer at any time during processing (including during input). Select the menu item "**File/Print and export**" and specify your output preferences for page format and presentation.

8.7.1 Select analysis method

If you would now like to calculate the slope with polygonal slip surfaces as well as circular slip surfaces, select the menu item "**Editor 1/Analysis options**":



Activate the "**Janbu (polygons/slices**)" switch in the upper part of the box. After confirmation with "**OK**" the menu bar will change slightly. Instead of the "**Centre-points**" menu, "**Slip body**" becomes visible. Further to this, in the "**Utilisation factors**" menu, some of the menu items have changed.

8.7.2 Define slip bodies using the mouse (Ex. 2)

To enter a slip body polygon, go to "**Define new**" in the "**Slip body**" menu (also see Section 10.6.2). Now click on the points of a slip body. If you have executed a wrong mouse click, which has "ruined" your slip body, simply press the [**Backspace**] key and you will return to the previous condition.

> After entering the final slip body point you must press the [**Return**] key to indicate to the program that the slip body entry is complete..

You can then immediately commence input of the next slip body. To do this you can also use the "**Slip body/Duplicate**" menu item to reproduce current slip bodies and then edit selected slip body points. The actions described can also be performed on previously analysed slip bodies.

To ensure an intersection with the surface is achieved, you can place the first and the last slip body point slightly above surface level. The program automatically generates the appropriate intersections once [**Return**] has been pressed. It is difficult to cleanly position slip body points on layer boundaries. To simplify this, you can also move the mouse pointer using the arrow keys. Pressing the left and right mouse buttons can also be carried out using the keyboard:

- [**Home**] key = press left mouse button
- [**End**] key = press right mouse button

If you cannot precisely determine which soil properties are valid at the mouse pointer, press [**Shift**] and the [**F4**] function key. The information will be presented in a dialog box. If you need to determine the inclination of slip body lines, press [**Shift**] and the [**F5**] function key. A further simplification of precise slip body point input can be had by activating the zoom function ("**Graphics preferences/Zoom info**" menu item).

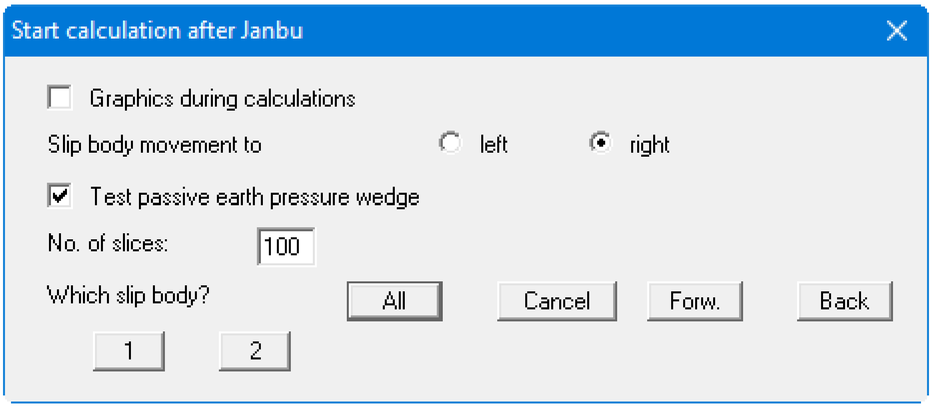
8.7.3 Define slip bodies via editor (Ex. 2)

Beside slip body input using the mouse you can also define a slip body be entering values into a table. Go to "**Edit**" in the "**Slip body**" menu.

First, click the "**New**" button in the combo box that opens and enter the required number of polygon points. You can then enter the slip body coordinates directly into the dialog box.

8.8 Step 8: Analyse slope with polygonal slip surfaces (Ex. 2)

After defining the slip bodies the system can be analysed (see also Section 10.8.2). Select the menu item "**Analyse slip bodies**" from the "**Utilisation factors**" menu.



Adjust the number of slices to suit your wishes. If the "**Test passive earth pressure wedge**" check box is activated, the program will examine whether gradients greater than 45° - φ/2 occur in the passive earth pressure region of the respective slip body. These slip bodies will not be investigated. Slip body movement should to the "**right**". Start calculation of all slip bodies using the "**All**" button.

Once calculations are complete you will be shown some statistics. After confirming with "**OK**", the slip body with the lowest FOS is displayed. In the "**Utilisation factors**" menu, point at the "**Show individual slip bodies**" or "**All**" menu items to display other slip bodies or all slip bodies. Point at the "**Display results**" menu item to display the principal analysis parameters and to print them.

9 Theoretical principles

9.1 DIN 4084:2009 and GGU-STABILITY

DIN 4084 (January 2009 edition) differs significantly from the previous DIN 4084 published in 1981.

DIN 4084-100 (referred to below as DIN 4084:1996!), which also included the new partial safety factor concept, was published as a draft in 1996. New analysis methods (general wedge mechanism and vertical slice methods) were described using examples in an unpublished supplement to this standard. These methods were implemented in the **GGU-STABILITY** program in 1996. These methods are not standard-dependent and therefore also apply to the global safety factor concept.

DIN 4084:1996 also included completely new designations for determination of stability using Bishop and Janbu methods (circular and non-circular slip surfaces). These equations were not initially implemented and were put on hold until the official introduction of the DIN 4084:1996. In **GGU-STABILITY**, therefore, the designation "**DIN 4084:1996**" means that stability is analysed using the equations published in DIN 4084 from 1981, but with the definitions of the partial safety concept. Analysis uses the design variables φ_d and c_d.

$$\tan \varphi_d = \tan \varphi_k / \gamma_\varphi$$

$$c_d = c_k / \gamma_c$$

φ_k and c_k = characteristic shear strength values
φ_d and c_d = shear strength design values

In addition, partial factors must be applied to live loads (e.g. 1.3 in Load Case 1). Stability is then analysed using the modified values. This gives the safety factor η in accordance with the global safety factor concept, but relative to the design values. The reciprocal of this "**global safety factor ηd**" gives the utilisation factor μ in accordance with the partial safety factor concept:

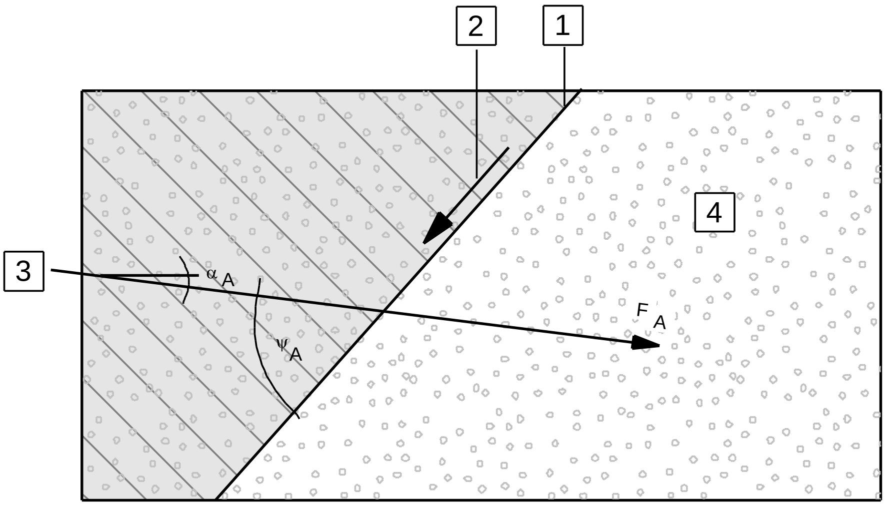
$$\mu = 1 / \eta_d$$

It was not until January 2009 that this concept was made mandatory. The new DIN 4084:2009 was primarily based on the relationships defined in DIN 4084:1996. However, it also includes considerable modifications. All new relationships are implemented in their entirety in the current version of the **GGU-STABILITY** program.

Comparative analyses using existing examples found in the literature and existing standards indicate very minor deviations between the old and the new standards.

Beside the new equations for Bishop and Janbu, one principal difference to the old standard is found in the definition of the passive tension member. It states that the force from a tension member may only be adopted if the boundary conditions shown in Figure 11 are adhered to.

Excerpt from DIN 4084:2009:



Legend:

1. Slip surface
2. Direction of slip body movement.
3. Tension member axis
4. Non-moving ground

Figure 11 Validity of a tension member (after Figure 2 in DIN 4084:2009)

*"A tension member is regarded as self-tensioning if the slip body in which the head of the tension member is located move approximately as a rigid body on a slip surface and the angle ψA between the tension member axis and the most unfavourable slip surface (*Figure 11*) achieves the following maximum values:*

- *for loose, cohesionless soils or soft, cohesive soils: ψA = 75°;*
- *for firm, cohesive soils: ψA = 80°;*
- *for medium-dense, cohesionless soils and stiff, cohesive soils: ψA = 85°;*
- *for dense, cohesionless soils: ψA = 90°.*

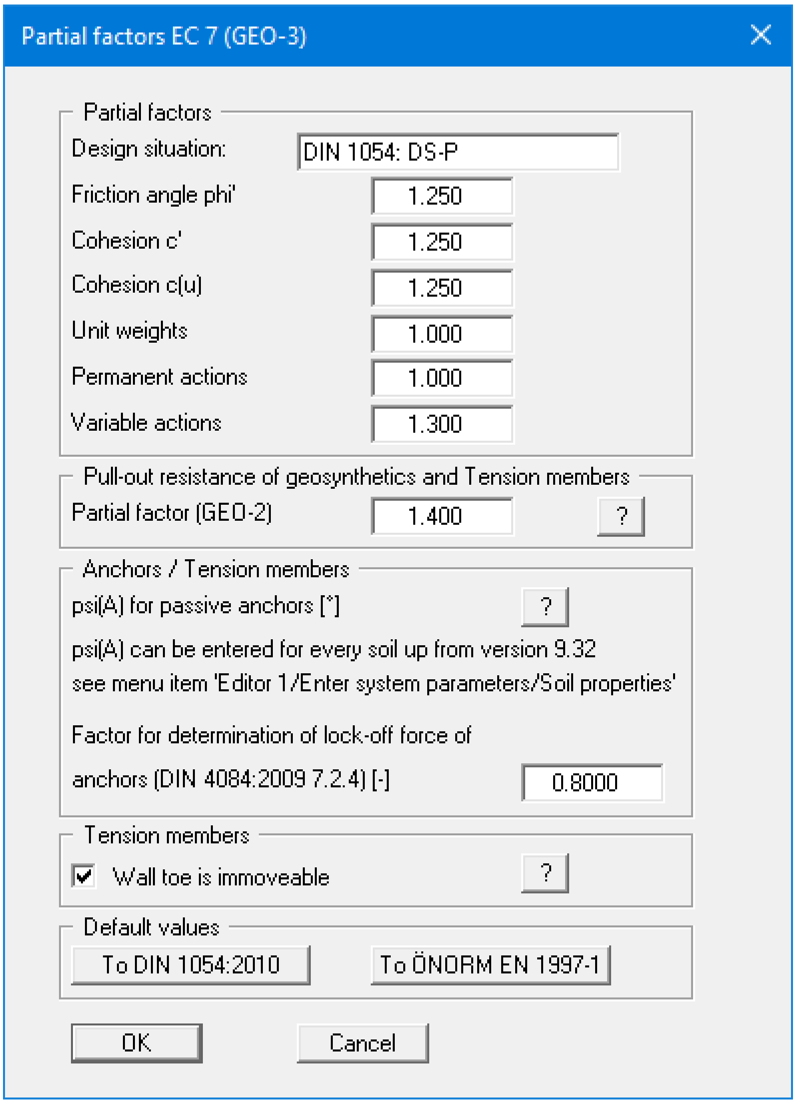
The tensile force of passive tension members must be determined to DIN 4084 7.2.1.

7.2.1 For analyses using this standard, the design values of the resisting forces exercised by tension members, dowels, piles and struts are acquired from the characteristic values for either the pull-out resistances, dowel resistances or penetration resistances, by dividing by the partial factors to DIN 1054 or the design values for allowable stresses according to the corresponding construction standards. The respectively smaller value is adopted. The acceptable forces on dead men or anchor slabs must be determined using the design values for ground resistances.

NOTE: *Non-passive tension members, which are not prestressed, have no effect.*

Where a wall is supported by tension members with angles ψA greater than the boundary values given in 7.2.3.4, a tension member may be taken into consideration at any angle if the wall can only rotate around a vertically and horizontally immoveable point at the base of the wall because of the support situation around the toe area. In this case the force necessary to accept the design values for earth and water pressures must be adopted as the tensile force for analysing sufficient global stability."

The settings described can be modified in the lower group box of the "**Editor 1/Partial factors, …**" dialog box and apply to all types of tension members definable in **GGU-STABILITY**.



If the "**Wall toe is immoveable**" check box is activated in **GGU-STABILITY**, the information to DIN 4084:2009 in Figure 11 is ignored.

Implementation of the new standard was carried out with great care over a period of approx. two months. Competent examination of the implementation was not possible due to the lack of eligible examples available. The examples relating to the new DIN 4084:2009 are to be published by the standardisation committee in 2013.

Seven examples are available for DIN 4084:1996; however, they were never published. These examples are provided as **GGU-STABILITY** files with the program.

"**DIN 4084-100-1996 Bsp 1.boe**"

to

"**DIN 4084-100-1996 Bsp 7.boe**"

The partial factors defined at that time envisaged different values for friction angle and cohesion:

- E.g. for Load Case 1
  γ_φ = 1.25 and
  γ_c = 1.60

Accordingly, the examples also include these different partial factors. The partial factors for friction angle and cohesion are the same in the new DIN 4084:2009.

- E.g. for Load Case 1
  γ_φ = 1.25 and
  γ_c = 1.25

The utilisation factors μ computed in the examples therefore do not correspond to those in the current version of the standard.

A comparison of the results in the examples (DIN 4084:1996) with those given by **GGU-STABILITY** displays an excellent correlation.

9.2 General information on Janbu and Bishop

The program is based on:

- Bishop (circular slip surfaces)

$$\eta = \frac{r \cdot \sum T_i + \sum M_S}{r \cdot \sum G_i \cdot \sin \vartheta_i + \sum M}$$

with

$$T_i = \frac{\left[G_i - \left(u_i + \Delta u_i\right) \cdot b_i\right] \cdot \tan \varphi + c_i \cdot b_i}{\cos \vartheta_i + \frac{1}{\eta} \tan \varphi_i \cdot \sin \vartheta_i}$$

- Janbu (polygonal slip surfaces)

$$\eta = \frac{\sum T_i + \sum H_S}{\cdot \sum G_i \cdot \tan \vartheta_i + \sum H}$$

with

$$T_i = \frac{\left[G_i - \left(u_i + \Delta u_i\right) \cdot b_i\right] \cdot \tan \varphi_i + c_i \cdot b_i}{\cos^2 \vartheta_i \cdot \left(1 + \frac{1}{\eta} \tan \varphi_i \cdot \tan \vartheta_i\right)}$$

from DIN 4084. Where:

- η terrain or slope failure safety factor
- G_i self weight of an individual slice in kN/m with consideration of the soil unit weight estimates from Table 1, including surcharges
- M moments of loads and forces not included in G_i around the centre-point of the slip circle in kNm/m, positive when acting excitingly (H for Janbu analogous)
- M_S moments around the centre-point of the slip circle in kNm/m from forces after Section 6e (DIN4084), which are not considered in T_i (HS for Janbu analogous)
- T_i the resisting tangential force of the soil at the slip surface for each slice in kN/m (for polygonal slip surfaces the horizontal component)
- ϑ_i tangential angle of the slice to the horizontal in degrees, which for circles is equal to the polar coordinates
- r radius of slip circle in m
- b_i width of slice in m
- φ_i the decisive friction angle, in degrees, for the individual slice after Section 8 (DIN 4084)
- c_i the decisive cohesion, in kN/m², for the slice after Section 8 (DIN 4084)
- u_i the decisive pore water pressure, in kN/m², for the individual slice

Δu_i the decisive pore water pressure for the slice, in kN/m², as a result of soil consolidation. In the **GGU-STABILITY** program, Δu_i is calculated by multiplying the pore water pressure coefficient and the effective vertical stresses. Alternatively, you have the possibility of defining so-called consolidation layers. Using the required input data, the program carries out a one-dimensional consolidation calculation.

The relationships are described in detail in DIN 4084 and DIN 4084:2009. We need therefore only deal with a few special cases here. Partial factors are used in DIN 4084:2009. The safety factors are thus already incorporated in the soil properties, loads, etc. The term "**safety factor**" is thus already allocated. Instead of "**η**", then, "**μ**" must be adopted in the above relationships, known as the "**utilisation factor**".

For pre-stressed tension members, the friction force activated by the normal component may be considered, in accordance with DIN 4084 and DIN 4084:2009 (Section 7.2.3.3). This is implemented for tension members in **GGU-STABILITY**.

In DIN 4084:2009, Section 7.2.3 the term ***self-tensioning*** is defined. The angle ψA between the tension member axis and the slip surface is decisive for this (see Figure 2 in DIN 4084:2009). This condition is calculated by the program (only when DIN 4084:2009 or EC 7 is specified). The value for ψA can be entered for each soil in the menu item "**Editor 1/Partial factors...**".

In DIN 4084:2009, Section 6.2.3 it is proposed that for pre-stressed tension members which are not self-tensioning, but which act favourably, the fixing force of the tension member and not the design value of the tension member is to be used into the calculation. If you design according to DIN 4084:2009, the program checks for ***non-self-tensioning*** and ***favourable***, and then multiplies the design value of the tension member with a factor, which can be entered globally for all tension members in the menu item "**Editor 1/Partial factors…**".

For analysis according to DIN 4084:2009 the design values are to be entered for soil dowels, geosynthetics and tension members. For the soil properties you enter the characteristic values, from which the program calculates the design values using the partial factors.

During input, partial factors must be differentiated for drained and undrained soils. Correspondingly, a "**drained**" check box is provided for input of soil properties - except for analysis using the global factors to DIN 4084 (old) - so that the program can calculate the correct design value for each soil.

9.3 General information on General wedge method and Vertical slice method

Calculations according to General wedge method or the Vertical slice method are explained in the DIN 4084:1996. With these procedures force polygons are constructed. The force polygon is only closed in special cases (η = 1.0 or 1/f = 1,0). Otherwise, an additional force dT is required to close the polygon. If this force dT acts excitingly, the safety is larger than 1.0. If the force dT acts resistingly, the safety is less than 1.0.

For calculations with partial factors, it is sufficient for verification that the safety factor is > 1.0 or the utilisation factor < 1.0. However, one is generally also interested in the ***safe distance*** to the failure condition. In order to calculate this ***safe distance*** and/or to carry out a quantitative comparison of various failure bodies, the force dT could, in principle, be brought in. The size of this force is, however, mainly dependent upon the size of the investigated slip body. A comparison of two very differently sized slip bodies using the force dT can easily lead to a false impression. The program therefore not only calculates the force dT, but reduces or increases the friction angle and cohesion as long as is necessary to close the force polygon. The safety factor η or the utilisation factor 1/f is then calculated from

$$\boldsymbol{\eta \text{ or } \mu = \tan \varphi_{pres.} / \tan \varphi_{eq.}}$$

For calculations after General wedge method or the Vertical slice method, as opposed to the procedure after Janbu, the shear forces in the intermediate slip surfaces are considered.

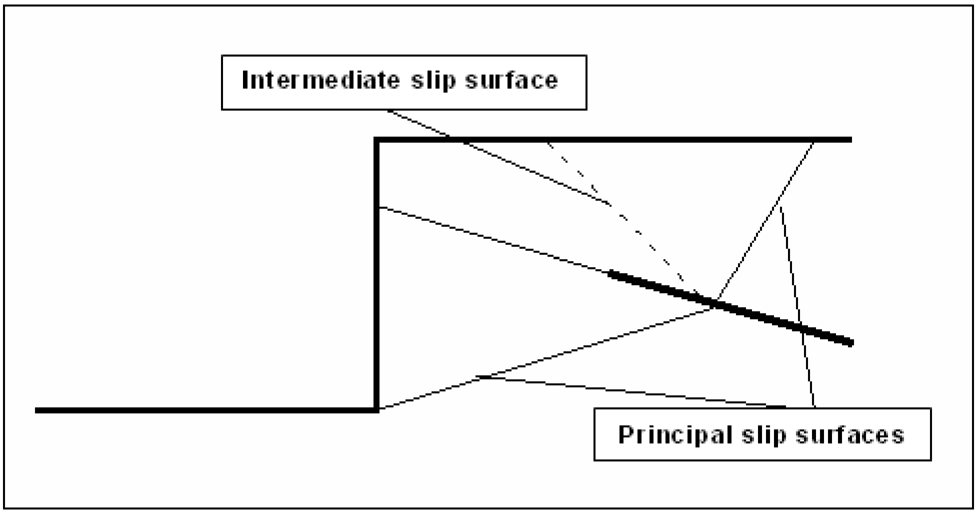


Figure 12 Intermediate slip surface and principal slip surfaces

This means that, in comparison to calculations after Janbu, higher safety factors are achieved. The program also offers the possibility, however, of setting the shear forces in the intermediate slip surfaces to "**0**".

When calculating according to General wedge method, if several soils are defined within a principle or intermediate slip surface, the program calculates with averaged values for the friction angle and cohesion. If an intermediate slip surface is vertical, the shear forces will not be considered in any case.

When calculating with the Vertical slice method, the intermediate slip surfaces are vertical and earth pressure forces in the slice flanks are considered, the inclination of which results from the pressure line. According to DIN 4084:1996, the pressure line may run between the centre and the lower third of the slice flank. The program therefore calculates the pressure line at a height h to:

$$\mathbf{h = 0{,}416 \cdot H = H \cdot (0.5 + 0{,}333)/2}$$

H = height of slice flank

If the inclination of the pressure line, and thus the inclination of the earth pressure force, is larger than the average friction angle in the corresponding slice flank, the average friction angle will be used for calculations. In the examples in the DIN 4084:1996 the friction angle is averaged using the earth pressure coefficients. The program does this simply via soil thickness.

9.4 Consolidation theory

If cohesive layers are loaded faster than they can release their pore water, excess pore water pressures result, which are only gradually dissipated. This process is known as consolidation. For a one-dimensional case a closed solution exists (see e.g. Das; Advanced Soil Mechanics; McGraw Hill). The following input values are required:

- Δu = excess pore water pressure at time t = 0 (constant for the whole layer depth)
- E_s = constrained modulus of layer
- k = permeability of layer
- d = thickness of layer
- t = time at which the excess pore water pressure is to be determined.

Further to this, the drainage conditions of the layer are to be considered:

- draining to the top and bottom
- draining to the top only
- draining to the bottom only

The program allows input of these values via so-called consolidation layers.

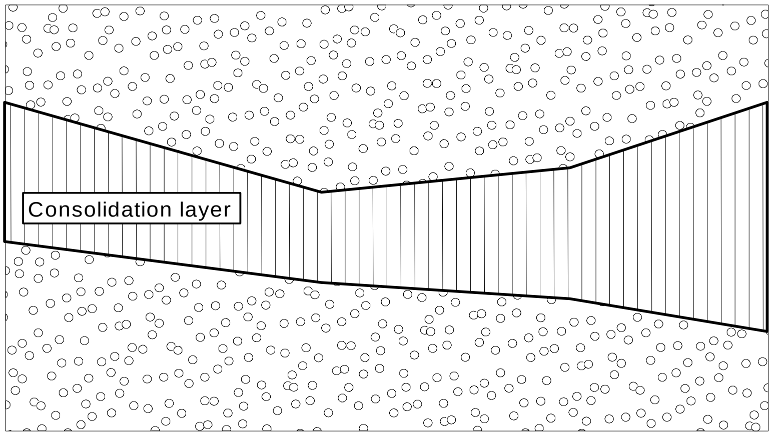


Figure 13 Consolidation layer

Consolidation layers are defined by two boundary polygons. Each polygon point must be assigned a pore water pressure. In Figure 13, a consolidation layer is shown which can drain to the top and the bottom. Due to the non-horizontal position, a two-dimensional consolidation will actually take place. The program calculates with one-dimensional consolidation in the vertical direction with sufficient precision and generally on the safe side. If a slice foot is within the layer, the layer thickness is determined from the two polygons. Together with the vertical position within the layer and the other decisive quantities, the program determines the excess pore water pressure at the user-defined time. If you calculate with consolidation layers, it may be necessary to carry out several calculations for the same system, using different times.

Besides classical consolidation theory, the program also commands cases in which the consolidation is accelerated by vertical drains (e.g. sand drains). The principles are explained in Das; Advanced Soil Mechanics; McGraw Hill.

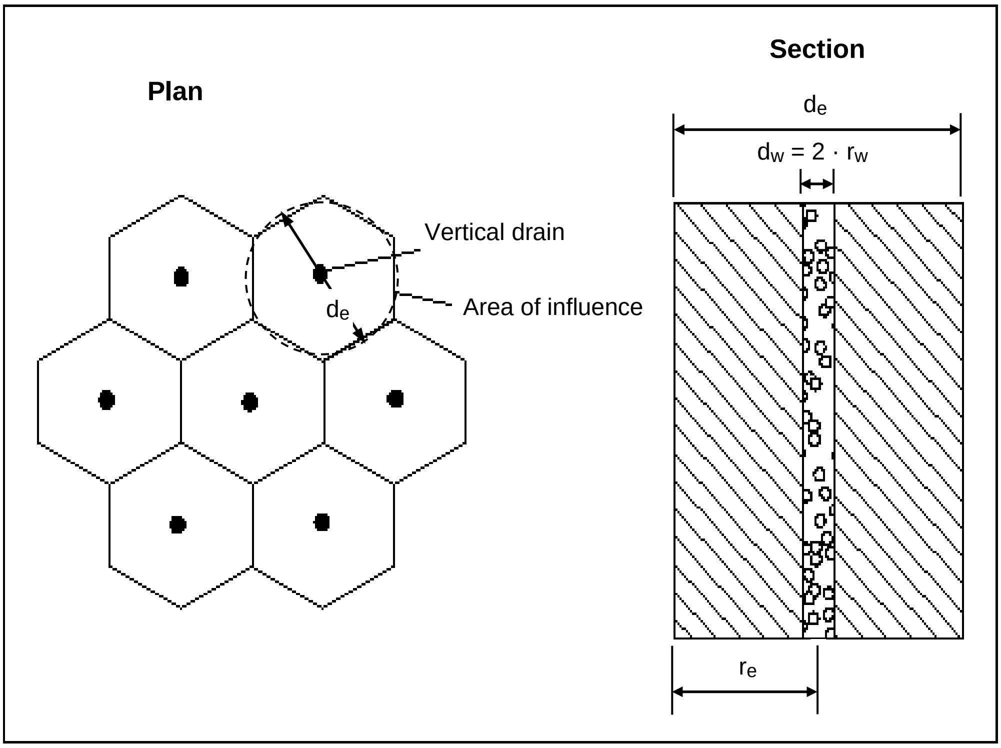


Figure 14 Vertical drains

The honeycomb structure around a drain can be converted to an equivalent circle, so that an axis-symmetrical consolidation calculation can be carried out for each drain. In this case, according to theory, dissipation of excess pore water pressure only takes place horizontal to the drains (axis-symmetrical), so the drainage conditions at the top and base of the layer need not be given. Instead, the drained distance (d_e) to one another and the radius (r_w) of the drains must be given. In consolidation layers with vertical drains the excess pore water pressure at any time is constant across the layer depth. The excess pore water pressure is, however, variable as a function of the distance r_e from the axis of the vertical drain. The program determines the average pore water pressure distribution.

The calculated pore water pressure distribution can be made visible very nicely, if you activate, from the menu item "**Safety/Utilisation factors/Preferences**", the switch

However, this does not work for calculations according to "**General wedge method**" or the "**Vertical slice method**". In these cases, you can take the values from the data protocol in "**Safety factors (Utilisation factors)/Display results**".

9.5 Safety factor definitions

In accordance with DIN 4084 (old), the program uses the safety definition after Fellenius:

$$\eta = \tan \varphi_{work.} / \tan \varphi_{req.}$$

For bearing capacity analysis according to DIN 4017, the safety factor is acquired from a comparison of the failure load V_b and the working strip footing load $V_{work.}$:

$$\eta = V_b / V_{work.}$$

Other comparisons are usual for different calculation procedures. For verification of the external stability of a soil nail wall, e.g., the nail forces are used in the safety definition.

Such safety definitions can also be created indirectly with **GGU-STABILITY**. If, for example, you would like to calculate the bearing capacity safety of a strip footing within a slope, enter the strip footing load as a permanent load and increase the actual load value by the required safety factor (e.g. η = 2.0). For the following slope failure calculation, you need only verify that the slope failure safety factor is above 1.0. If you interested in the actual safety factor value, you must vary the size of the load by hand until you get a safety factor of 1.0 according to DIN 4084. For this special case the program has a routine to do this work for you. For verification of a strip footing in a slope, the DIN 4084 (Supplement; Section 4) contains an appropriate, but very carefully formulated note. In principle, for bearing capacity analysis on the basis of the DIN 4084, you will not get the same safety factors as with DIN 4017, as the theoretical basis is not identical. In particular, the DIN 4017 assumes no shear strength above the footing base. In the DIN 4017, areas above the footing base are only considered with respect to surcharge loads.

Procedure for partial safety factor concept

As described above for the global safety factor concept, the forces V and H are modified until the utilisation factor for the slope stability analysis is "**1.0**" This gives the ultimate bearing capacity V_b and from this the utilisation factor for bearing capacity analysis = V/V_b. Using the partial safety factor concept loads must be increased by the partial factors for actions and the partial factor for bearing capacity. The partial factors for the friction angle, cohesion and permanent actions are set to "**1.0**".

In certain systems the analysis method may lead to non-converging solutions.

It is safer to deactivate the "**Bearing capacity**" check box in the "**Editor 1/Analysis options**". Then define the V and H loads as permanent actions. Using the partial safety factor concept actions must be increased by the partial factors for actions and the partial factor for bearing capacity increased. When using the global safety factor concept the loads are increased by the required bearing capacity factor. A ***traditional*** slope stability analysis can then be performed.

If utilisation factors ≤ 1.0 are achieved sufficient bearing capacity is verified.

9.6 Passive earth pressure

If the circular slip surfaces in the ***resistance area*** are steeper than 45° - φ/2, the passive earth pressure is to be used, in accordance with DIN 4084. The DIN 4084 suggests a simple earth pressure calculation for consideration. The **GGU-STABILITY** program takes a much more flexible path. In the appropriate areas, the geometry of the failure body is altered, in line with the demand 45 - φ/2 as maximum. The slice width is kept. If necessary, the user-defined number of slices is increased. If a soil change occurs within the passive earth pressure body, with a change in friction angle, the geometry will be adjusted accordingly. During numerical evaluation of such a failure body the slice relationships used in the DIN formula are kept. Additionally, however, it is to be considered that the normal force in the slice foot now creates a moment around the centre-point, which is not the case for pure slip circles. For changes in the friction angle within the slip body, a change in the lever arm of the tangential force T_i around the centre-point must also be considered. This program concept is much more flexible than the approach using a substitute earth pressure force, as complicated slope geometries can be much better assessed. Slip bodies with a passive earth pressure wedge are shown with a third ***radius***.

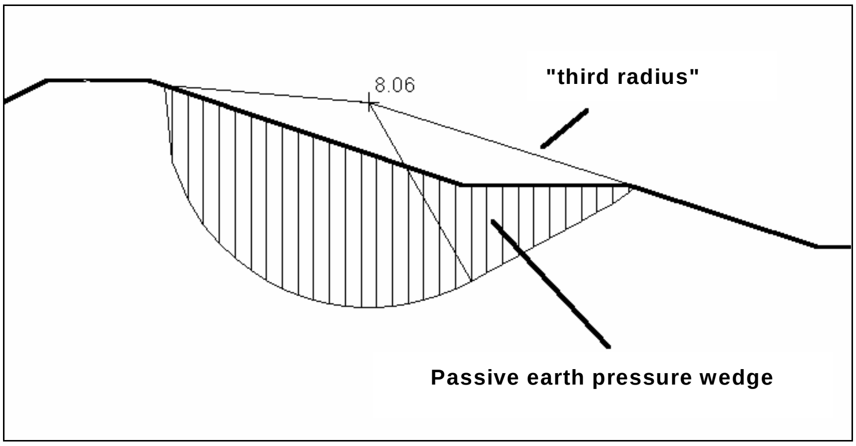


Figure 15 Passive earth pressure wedge

> **Remark**:
> The demand of the DIN 4084 to incline ***passive*** slip body areas with a maximum of 45° - φ/2, is strictly only correct for level surface.

9.7 Several slip bodies

For heavily structured slope geometries, several intersections of a slip circle with the surface can occur. In such cases the program will calculate all possible slip bodies (see "**Investigate all circle intersections (recommended)**" check box in the "**Safety/Utilisation factors/Analyse**" menu item, Section 8.5).

9.8 Nail wall or reinforced soil wall

9.8.1 General notes on nail wall analysis

A nail wall consists of tension members, geosynthetics or steel strips, allowing analysis of reinforced soil walls.

Nail wall analyses are not codified in (German) regulations. However, permits containing notes on analysis have been issued by the German Institute for Construction Technology for soil nailing systems offered by several companies.

Analysis of soil nailing generally requires six separate verifications:

- Verification of inner stability via slip body investigations, generally with a failure mechanism consisting of two bodies. The verification can be carried out using the Fellenius Rule $\eta = \tan(\text{cal } \varphi)/\tan(\text{req } \varphi)$, which is realised in the program.
- Verification of sliding safety;
- Verification of overturning stability;
  Resultant must intersect the base surface in the 1st core dimension (= b/6).
- Verification of bearing capacity safety;
- Verification of general stability ;
- Verification of the concrete shell.

9.8.2 Terms

The following simplified term definitions are used in the manual and the program:

Nail	= geosynthetics, steel strips or tension member
Nail wall/wall	= monolithic soil prism, created by nailing, or reinforced soil wall with geosynthetics
Rear face of wall	= rear face of nail wall or reinforced soil wall

9.8.3 Verification of inner stability

Verification of inner stability can be performed using the original **GGU-STABILITY** method. However, the generation of slip surfaces remains time-consuming. This process is simplified by additional functions in the "**Nail wall**" menu, which substantial quickens generation of nails in a regular array.

Following generation of the slip surfaces, the inner stability can be investigated using the menu item "**Safety/Utilisation factors/Analyse slip bodies**". The slip surfaces can be analysed according to either "**Janbu**", the "**General wedge method**" or the "**Vertical slice method**".

In accordance with current standards, the governing verification for nail design is the one producing the greatest nail forces. The following investigations are necessary for this:

- Load components in the nails arising from slip body investigation for final and construction condition
- Load components in the nails arising from earth pressure on the concrete shell or exterior skin. The earth pressure from the nailed soil prism acting on the concrete shell or exterior skin may be adopted at 0.85 x active earth pressure value, but without applying cohesion. The earth pressure distribution may be adopted in a rectangular configuration, including in stratified ground. The wall friction angle is adopted at $\delta = 0$.

9.8.4 Verification of sliding safety

Sliding safety verification is performed with the assistance of equivalent systems. Weight calculation is according to the information in the figure below.

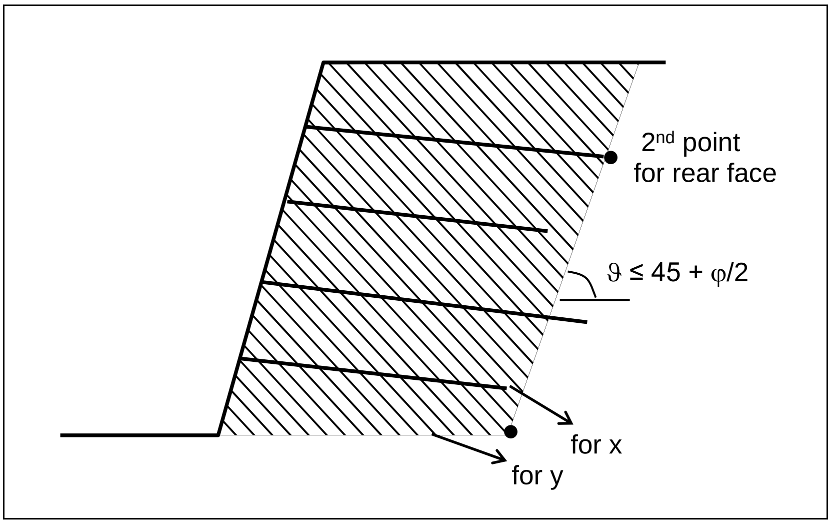


Figure 16 Equivalent system for weight calculation

The weight of the hatched area is determined. The lower right point of the body is taken as the x value of the lower end of the nail, and the y value as the lower edge of the concrete shell or exterior skin. The rear face is taken as a line running to the end of the uppermost nail (independent of the length of any other nails between these two points). The inclination of the rear wall face cannot be steeper than the inclination of the active earth pressure wedge (see further down).

The horizontal loading is taken from the active earth pressure. The vertical section for which the earth pressure is to be determined, is shown in the figure below.

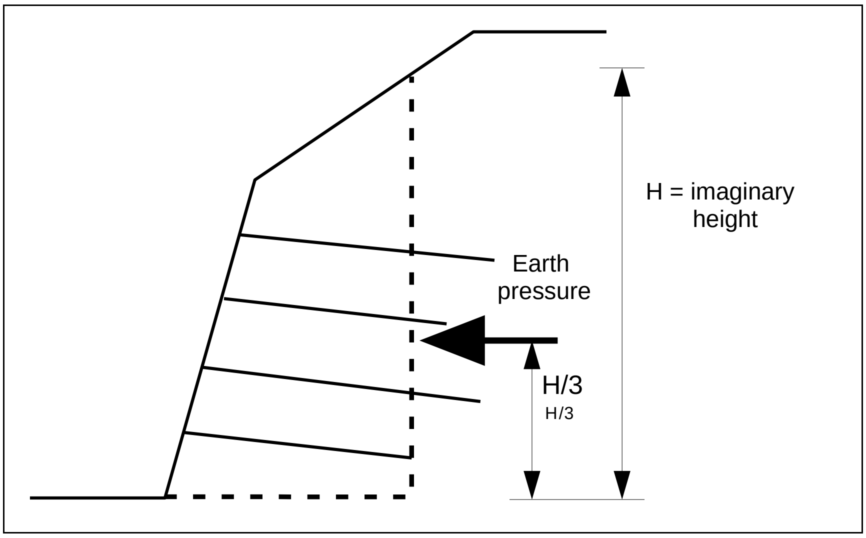


Figure 17 Vertical section for earth pressure determination

As the **GGU-STABILITY** program also allows input of inclined layers, as well as many other features (e.g. consolidation, artesian water, etc.), which do not allow a classical earth pressure calculation via kah values, the earth pressure calculation will be carried out using a variation of the earth pressure wedge.

In order to take into consideration any inclination of the rear face of the wall, a reduction is carried out via the earth pressure coefficients kah for both a vertical rear face ($\alpha = 0$) and an inclined rear face ($\alpha \neq 0$), after determining the earth pressure:

reduction factor (earth pressure) = kah(α)/kah(α = 0)

Sliding safety η_G is calculated thus:

$$\eta_G = G_k \cdot \tan \varphi / H_d$$

G_k = characteristic weight
H_d = design value of horizontal force

The friction angle φ is obtained from the mean of the soils in the base of the wall footing.

9.8.5 Verification of overturning safety

Using the information provided in the section on "**Verification of sliding safety**" the moment and the vertical force in the base are calculated and the eccentricity e determined from the result. The moment resulting from the horizontal earth pressure is determined according to Figure 17 in the section above. For example, the eccentricity in Load case 1 may not be greater than b/6.

9.8.6 Verification of bearing capacity safety

With the input from the section on slide safety, the moment, the horizontal force and the vertical force in the base is calculated, and with this the bearing capacity safety factor. The friction angle, the cohesion and the weight below the base γ_2 are taken as the mean of the soils in the foundation joint of the wall. If several layers are present, the increases or reductions in bearing capacity resulting from this are not taken into consideration. If necessary, you can determine the bearing capacity safety factor using slip circle calculation.

With regard to the γ_2 value, you can specify whether analysis is performed using the wet unit weight or the buoyant unit weight by going to the menu item "**Nail wall/Preferences**" and activating the "**gam2 as buoyant unit weight**" check box in the "**Bearing capacity and sliding analysis**" group box. If the soil properties in the ground deviate from the automatically determined mean values, it is also possible to employ user-defined soil properties (see Section 10.11.2).

9.8.7 Verification of general stability

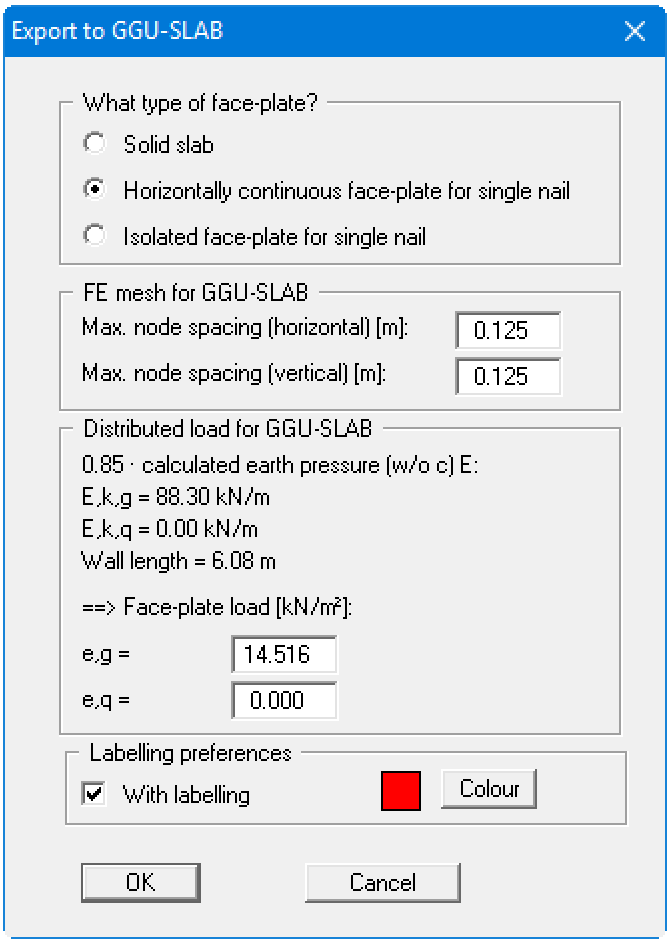
Verification of general stability can be simply performed by switching from the nail wall analysis back to slope stability analysis with slip circles. To do this, activate the appropriate option buttons in "**Editor 1/Analysis options**" menu item.

9.8.8 Verification of the concrete shell

The menu item "**Nail wall/Export to GGU-SLAB**" is available for bending design of the concrete shell. A dataset is exported which can be read by the **GGU-SLAB** program. **However, only systems with 1 slope face can be exported.** The **GGU-SLAB** program allows analysis and design of slabs using finite-element methods. In principle, there are three different methods of manufacture, and thus analysis, for concrete shells for nail walls.

- **Solid slab**
  The concrete shell is continuously reinforced and acts horizontally and vertically as a continuously solid slab. Bending moments arise between the tension members in vertical and horizontal directions and are accepted by the reinforcement.
- **Horizontally continuous face-plate for single nail**
  The concrete shell is manufactured horizontally continuous only. In the centre line between two vertical nails a gap is created in which no bending moments can be transmitted.
- **Isolated face-plate for single nail**
  The concrete shell is only reinforced in a rectangular area around the nail. In the centre line between two vertical or two horizontal nails a gap is created in which no bending moments can be transmitted. A face-plate is created for each nail, which acts as an individual structural unit.

Which of the three listed structural systems is exported to the **GGU-SLAB** program can be specified in the upper group box of the following dialog box.



The constant distributed load on the slab is generally given by the earth pressure, reduced by 15%, calculated without any cohesion components in accordance with current permits and which is divided by the length of the concrete shell. This value is given at the bottom of the dialog box and can be edited. In some publications the reduction of this value by the factor 0.85 is not mentioned. Other publications divide the maximum possible nail force by the area influenced by the nail and use this distributed load for calculations, i.e.

Distributed load = maximum nail force/(horizontal nail spacing · vertical nail spacing)

The final approach generally provides the largest moments and thus the strongest reinforcement. If you need to consider such cases, you must adjust the distributed load in the above dialog box accordingly.

9.8.9 Maximum nail forces and verification of punching

The maximum nail forces and punching safety factor are determined using the "**Nail wall/Maximum nail forces + punching**" menu item (see Sections 10.11.13 and 10.11.14).

Whilst calculating slip surfaces the program keeps a protocol of maximum forces per nail position. These forces are compared to the load components which arise from the earth pressure on the concrete shell. The largest value is decisive. The earth pressure on the concrete shell is determined without cohesion, in accordance with current permits. 0,85 times the earth pressure value is distributed rectangularly along the length of the concrete shell. The earth pressure reduction by a factor of 0,85 may not be carried out for surcharge loads. Correspondingly, the calculation of earth pressure on the concrete shell is carried out with the values from permanent and live loads, increased by 1/0.85.

After selecting this menu item a dialog box is shown with the ***distributed*** earth pressure value e,d. The value suggested for earth pressure e,d in this dialog box will also be used for punching verification. After confirming or editing of the calculated earth pressure value, the maximum nail forces, determined in accordance with the above description, are shown in a list in another dialog box. The maximum nail value is multiplied with the horizontal separation and suggested as design value for punching verification. This value can, if wished, be edited.

The pressure under the punching area will be subtracted from the nail force, in accordance with EC 2. The pressure is taken as the reduced earth pressure (see above). Besides this information the following must be given, in accordance EC 2, for punching verification:

- Effective height of concrete shell in the verification area, as % of concrete shell thickness;
- Width of nail slab (= height);
- Present bending reinforcement;
- Concrete used;
- Steel used.

After making the inputs and pressing the "**OK**" button, verification will be carried out and all decisive values for verification are shown in a dialog box.

9.8.10 Construction conditions

The inner stability of the wall must also be investigated for intermediate conditions. A construction condition exists if excavation has taken place to, or below, the level of the nail(s) but the nail has not yet been placed. These intermediate conditions can be easily calculated with the program, if in the dialog box "**Nail wall/Preferences**" the "**Investigate construction condition**" check box is activated. For calculation of slip bodies, the action of each first nail above the intersection of the polygon with the wall will be set to "**0**".

9.9 General notes on analysis with fibre cohesion

Wastes dissimilar to soil such as untreated residual waste, for example, exhibit shear behaviour considerably different to that of soil and soil-like wastes (excavated soils, sieved MBT fractions (Mechanical Biological Treatment). In their GDA recommendations (GDA - *Geotechnik der Deponien und Altlasten - Geotechnical Aspects of Landfill and Brownfield Sites*), the German Geotechnical Society (*Deutsche Gesellschaft für Geotechnik*) recommends different analysis methods to suit these classifications. Using common installation techniques, waste dissimilar to soil leads to stratified, anisotropic landfill masses. Stability analyses for anisotropic landfill masses are dealt with in GDA recommendation R 2-29.

From a waste mechanics perspective, waste dissimilar to soil is regarded as a composite material, consisting of an underlying matrix and a fibre matrix, based on the principles of fibre-reinforced soils. Based on the model describing the interactions of tensile and friction forces in a composite material, superimposing the two shear strength components generally produces a non-linear failure condition. Such failure conditions are known from reinforced soil masses (EBGEO - *Empfehlungen für Bewehrungen aus Geokunststoffen - Recommendations for Geosynthetic Reinforcements*). The cause of the non-linearity is the limited dependence of the normal stress on the reinforcing effect.

The magnitude of the allowable tensile forces depends on the fibre properties and the grain matrix, as well as the surcharge load. The strength characteristics of the fibre matrix are described by two material parameters:

- The fibre-specific tensile strength z_{max}
- The angle ζ

A surcharge-independent component of the tensile strength (z_0), which cannot be differentiated from any cohesion c of the underlying matrix, may also occur. The angle ζ describes how the normal stress is a function of the reinforcing effect. The transfer of tensile forces by the fibres is limited by the fibre-specific tensile strength z_{max}. When the fibre-specific tensile strength is reached the fibres tear, the load-bearing effect of the fibre matrix fails.

The shear strength components of the underlying and the fibre matrix are determined in order to determine the shear strength of the anisotropic waste mass. The two components are then superimposed in the analysis. The so-called fibre cohesion $\tau(z)$ is calculated from the tensile forces in the fibres. One peculiarity of waste dissimilar to soil containing fibres and films is that the material is pressed flat by the compactor and horizontally stratified when installed in thin layers. This produces the anisotropic strength behaviour. Activation of the fibre cohesion $\tau(z)$ depends on the angle ϑ between the fibre layer and the shear joint.

Stability analyses can be performed using the material parameters differentiated for the underlying matrix and the fibre matrix (φ_{GM}, c_{GM}, ζ, z_{max}). By using this separate approach, it is possible on the one hand to take the anisotropy of the fibre matrix into consideration numerically, and on the other to employ surcharge-independent material parameters despite the non-linear failure condition. The basis for slope failure analysis is the method of slices (DIN 4084:2002-11). The equation for analysis of the slice base shear resistance is extended by a term for the fibre cohesion as a function of the angle ϑ. The slice base shear resistance T is given by:

$$T = \frac{G \cdot \tan \varphi_{GM} + c_{GM} \cdot b + G \cdot \tan \varsigma \cdot \sin(1{,}5\vartheta)}{\mu \cdot \sin \vartheta \cdot \tan \varphi_{GM} + \cos \vartheta}$$

Whereby the following condition is adhered to:

$$G / b \cdot \tan \varsigma < z_{\max}$$

It must be taken into consideration when adopting the cohesion c_{GM} that surcharge-independent shear strength components resulting from the surcharge-independent tensile stresses may occur in both the underlying matrix (cohesion c_{GM}) and the fibre matrix (surcharge-independent fibre cohesion $\tau(z_0)$). Because these surcharge-independent shear strength components cannot be exactly differentiated, they may not be simultaneously adopted for analyses.

With the exception of the equation extension for analysis of the slice base shear resistance, no further changes have been made to the program's underlying analysis methods. The description given in the "**General information on Janbu and Bishop**" section applies (see Section 9.2).

Input of soil properties is described using an example for conventional analysis without fibre cohesion in "**Worked example 2: Data input via editor/Step 2: Enter system parameters (Ex. 2)/Soil properties**" (see Section 8.2.3). As described above, further parameters are required for analysis with fibre cohesion; they are entered in the corresponding input screen.

Soil	Friction angle [°]	Cohesion [kN/m²]	Unit weight [kN/m³]	PW coeff. [-]	Traction angle [°]	Degree of activation [-]	Tensile strength [kN/m²]
Residual waste	25	10	9		35	0.7 - (1.0)	210
Old waste	30	15	11		20	0.7 - (1.0)	110
MBT	35	15	12		14	0.7 - (1.0)	75
MBT < 60	35	15	13		7	0.7 - (1.0)	35

Table 2 Soil properties input screen (examples from GDA recommendation R 2-35 and Collins et al., 1997)

The degree of activation parameter describes the bonding behaviour when the tensile forces are converted to fibre cohesion. Fibre redistribution and similar effects are taken into consideration globally, beside the bonding effect. With good bonding (homogeneous mixing of underlying matrix and fibre matrix, high-friction underlying matrix, etc.) the degree of activation may be 1. Because knowledge of the bonding behaviour is limited, a conservative degree of activation of 0.7 is recommended (Kölsch, 1996). In the analysis, the degree of activation acts as an additional partial safety factor on the fibre cohesion.

10 Description of menu items

10.1 File menu

10.1.1 "New" menu item

After a security request, you can delete all previous input and thus enter a new system (see menu item "**Editor 1/Analysis options**", Section 10.2.1).

10.1.2 "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 or investigate different centre-points or slip bodies.

10.1.3 "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. The file contains all system information (including slip circle centre-points and polygonal slip bodies). Loading again later creates the same presentation as was present at the time of saving.

10.1.4 "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 "**.boe**" 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, "**.boe**" will be used automatically.

10.1.5 "Load centre-points"/"Load slip body data" menu items

You can load a file with a centre-point array or polygonal slip bodies, which was saved at a previous sitting (suffix "**.rst**" or "**.gkp**").

10.1.6 "Save centre-points"/"Save slip body data" menu items

You can create a file (suffix "**.rst**" or "**.gkp**"), which contains the current data for the centre-point array or polygonal slip bodies. As this information is also saved in the main file a separate saving of failure body geometry only makes sense if you would like to import the geometry into a different slope system.

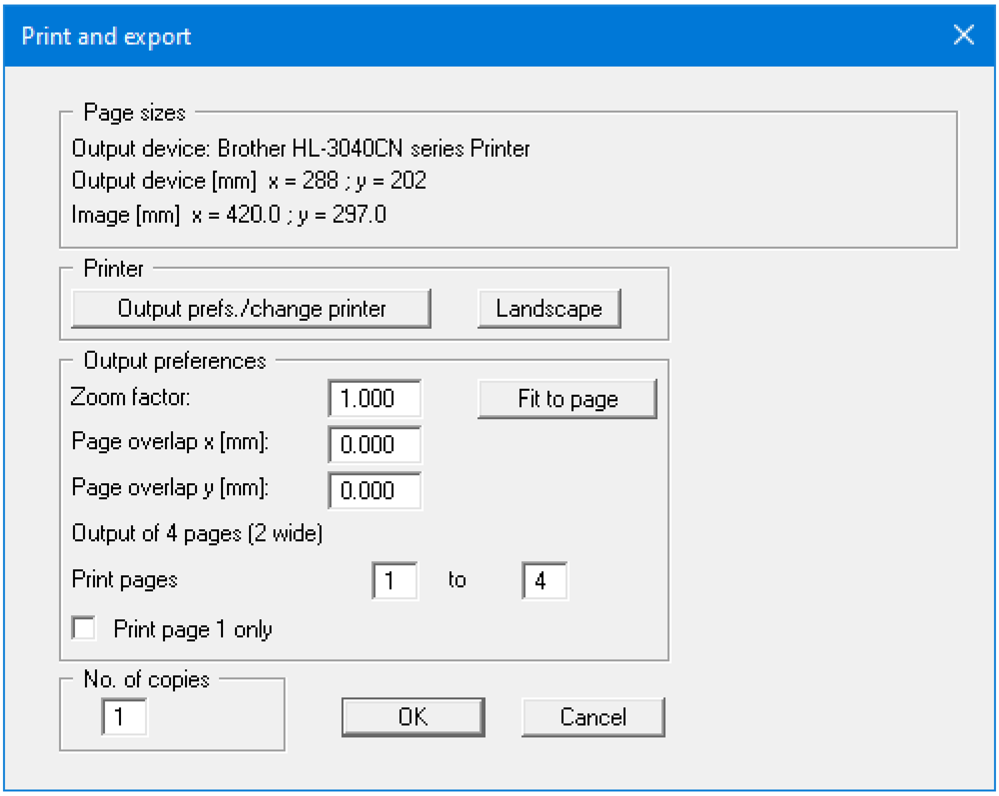
10.1.7 "Output preferences" menu item

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

10.1.8 "Print and export" menu item

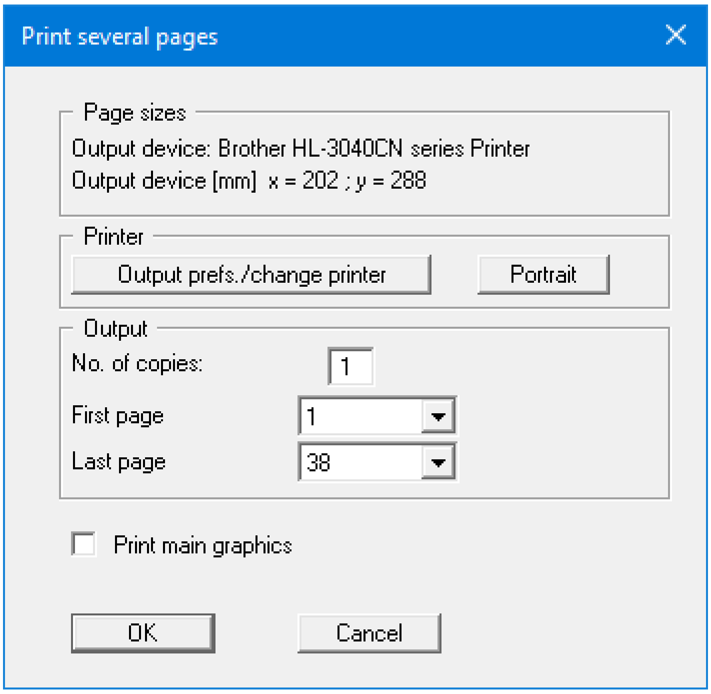
You can select your output format in a dialog box. You have the following options:

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



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

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

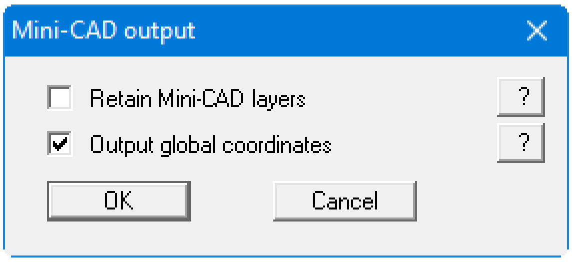


- Here, you can select the table pages to be printed. To achieve output with a zoom factor of 1, you must adjust the page format to suit the size format of the output device. To do this, use the dialog box in "**File/Print output table**" button "**Output as graphics**".
- "**DXF file**"
  allows output of the graphics to a DXF file. DXF is a common file format for transferring graphics between a variety of applications.
- "**GGU-CAD file**"
  allows output of the graphics to a file, in order to enable further processing with the **GGU-CAD** 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 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 6.4).

Using the "**Mini-CAD**" program module you can also import EMF files generated using other GGU applications into your graphics (see Section 10.9.5).

- "**Mini-CAD**"
  allows export of the graphics to a file to enable importing to different GGU applications with the **Mini-CAD** module.



If the "**Retain Mini-CAD layers**" check box is activated, the layer allocations for any existing **Mini-CAD** elements are saved. Otherwise, all **Mini-CAD** elements are saved on Layer 1 and are also inserted into Layer 1 in other **GGU** programs via the "**Load**" function in the **Mini-CAD** pop-up menu.

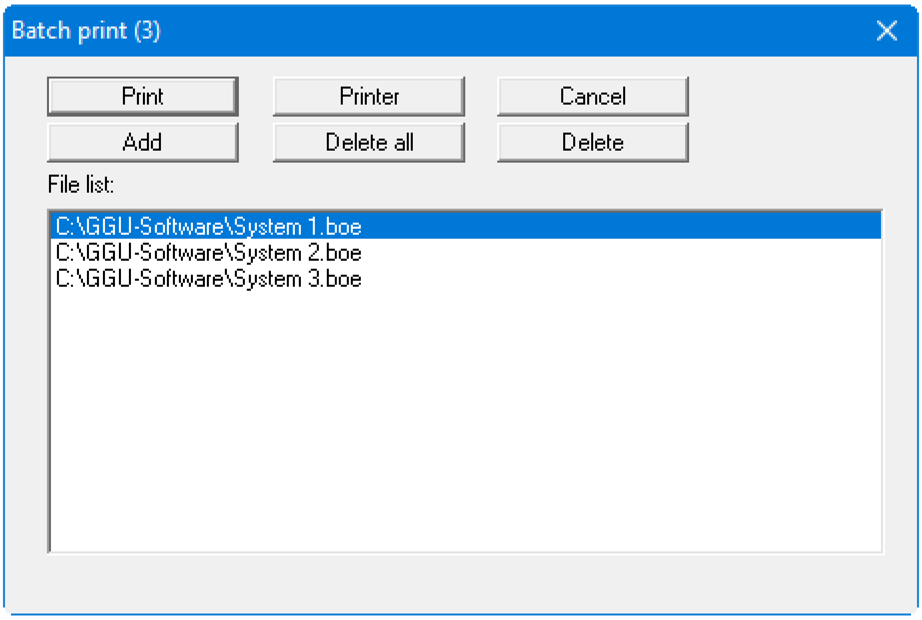
By activating the "**Output global coordinates**" check box, the present graphics are saved in the system coordinates [m]. Otherwise, they are saved in the page coordinates [mm]. If you import the **Mini-CAD** file saved using "**Global coordinates**" into a different **GGU** program, the coordinates are also transferred. If a system is transferred from **GGU-STABILITY** to **GGU-2D-SSFLOW**, for example, the system coordinates and scale are corrected compliant with the transferred global coordinates, after importing the file and pressing the function key [**F9**] (menu item "**Page size + margins/Auto-resize**").

- "**GGUMiniCAD**"
  allows export of the graphics to a file in order to enable processing in the **GGUMiniCAD** program.

- "**Cancel**"
  Printing is cancelled.

10.1.9 "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 output preferences is achieved by pressing the "**Printer**" button.

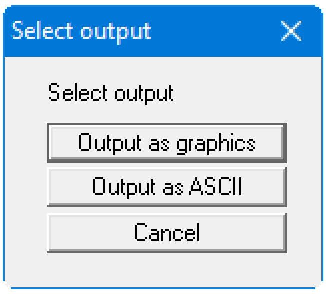
You then start printing by using the "**Print**" button. In the dialog box which then appears you can select further preferences for printer output such as, e.g., the number of copies. These preferences will be applied to all files in the list.

10.1.10 "Print output table" menu item

10.1.10.1 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.

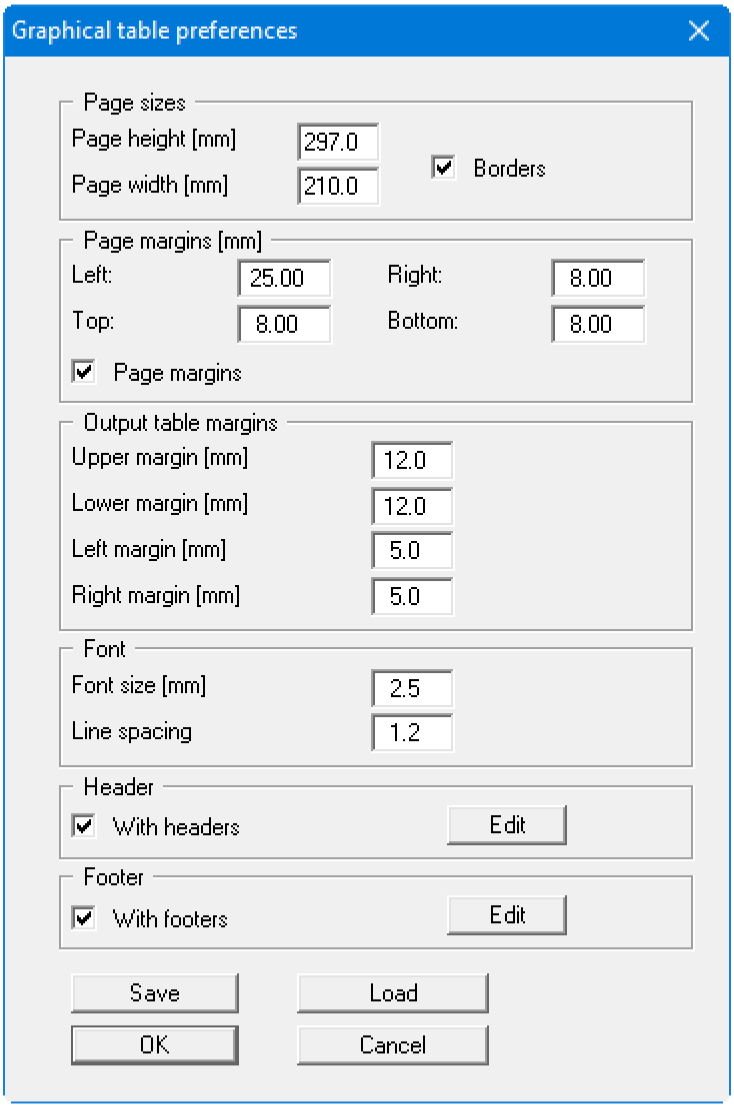
You have the option of designing and printing the output table as an annex to your report within the **GGU-STABILITY** program. To do this, select "**Output as graphics**" from the following options.



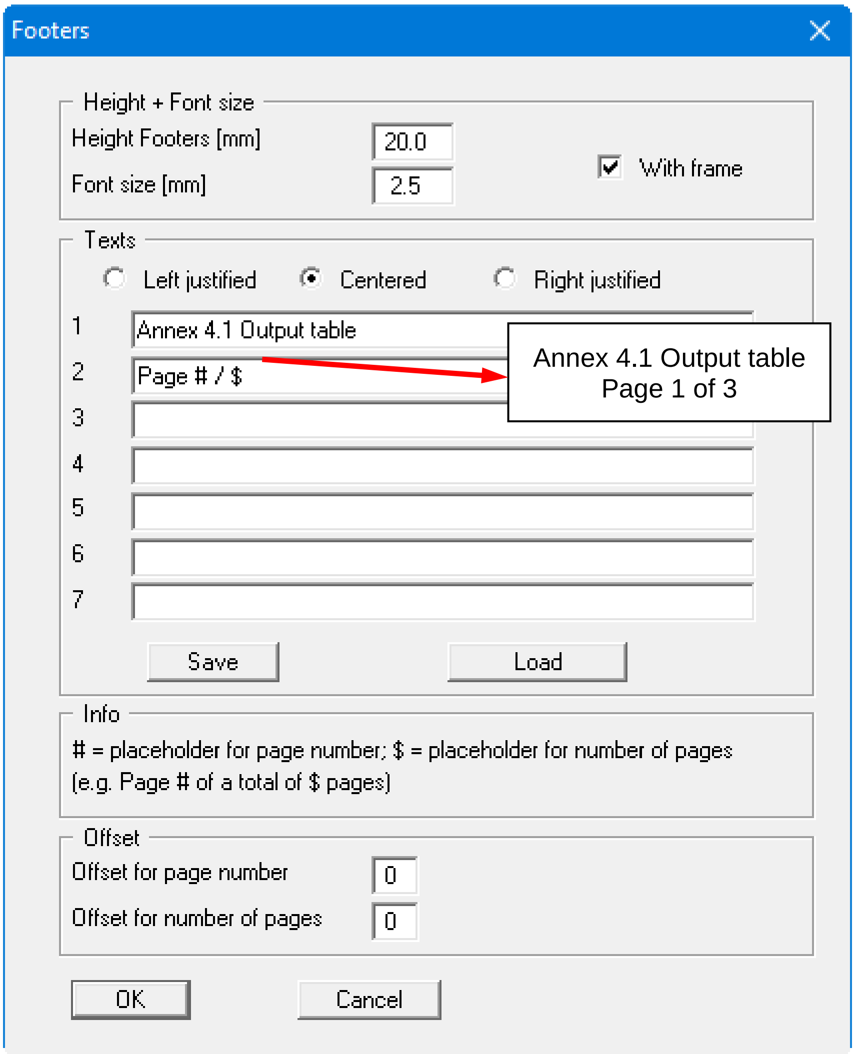
If you prefer to easily print or process the data in a different application, you have the possibility of sending them directly to the printer or to save them to a file using the "**Output as ASCII**" button.

10.1.10.2 Button "Output as graphics"

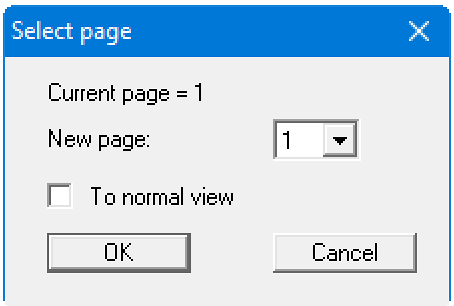
If you selected the "**Output as graphics**" button in the previous dialog box a further dialog box opens, in which you can define further preferences for result visualisation.



In the various group boxes of the dialog box, you can define preferences for the table output and layout. 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. You can save your settings for the graphical output table presentation in a "**Protokoll.pin_ggu**" file at the program level so that they are loaded when the program starts. Using the "**Load**" button, the output table settings can also be subsequently loaded into an existing file, including that of another GGU program.

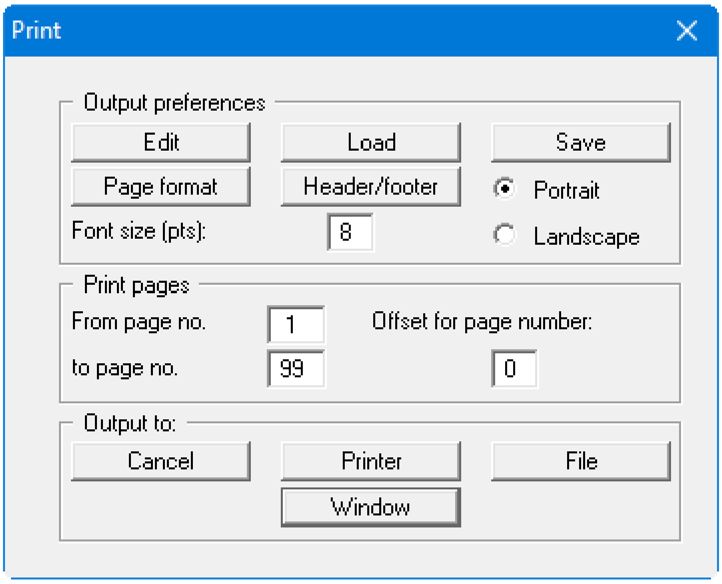


Automatic pagination can also be employed here if you work with the placeholders as described. After closing the dialog box using "**OK**" the output table is shown page by page on the screen. To navigate between the pages, use the arrow tools in the toolbar. If you need to jump to a certain page or back to the graphical visualisation, click on the tool. You will then see the following box:



10.1.10.3 Button "Output as ASCII"

You can have your calculation 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.



In the dialog box you can define output preferences.

- "**Output preferences**" group box
  Using the "**Edit**" button the current output preferencescan 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-STABILITY.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 lines 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.

10.1.11 "Exit" menu item

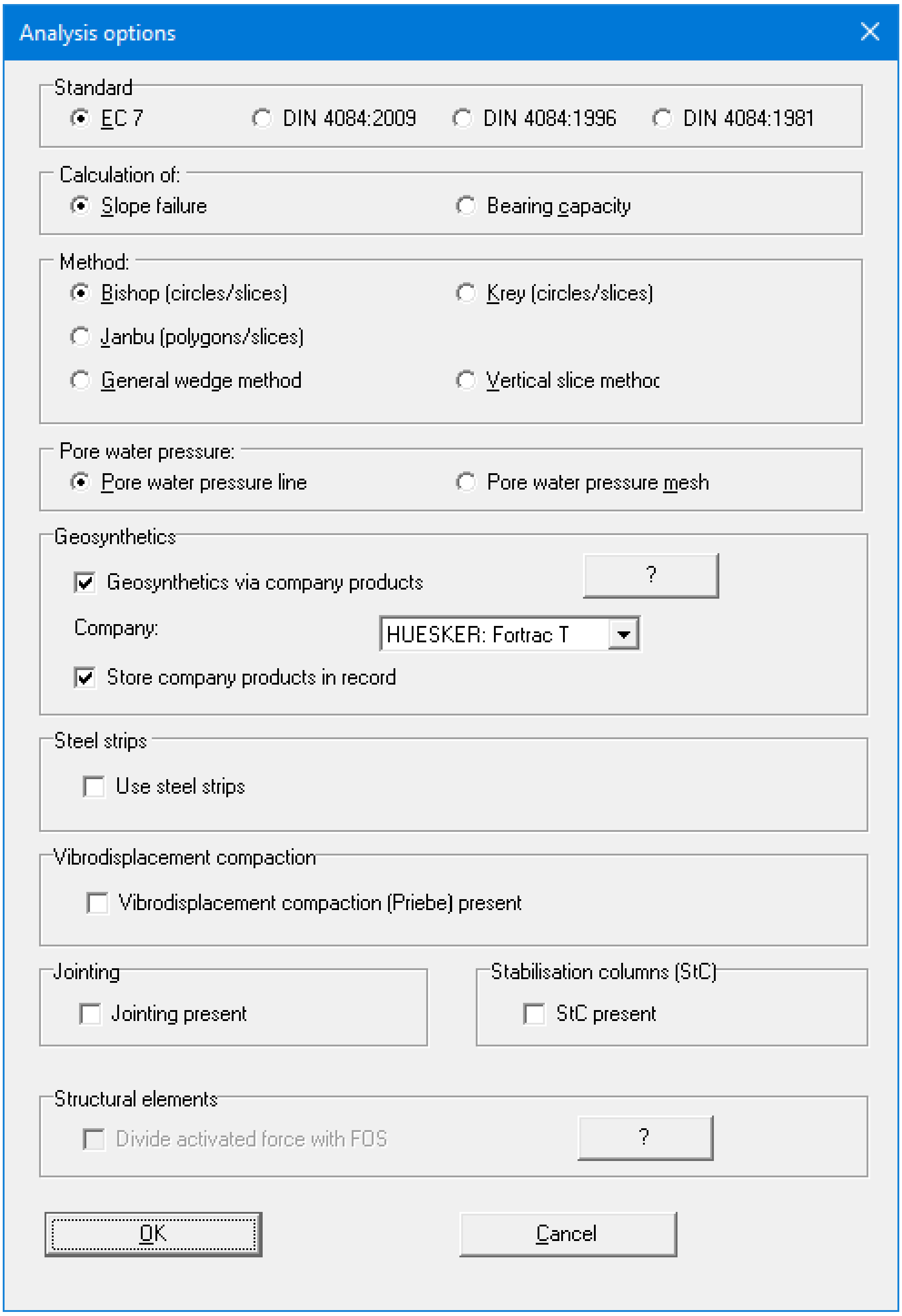
After a confirmation prompt, you can quit the program.

10.1.12 "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.

10.2.1 "Analysis options" menu item

You will see the following dialog box:



- "**Standard**" group box
  You select the standard to which the slope stability analysis is to be performed.

- "**Calculation of:**" group box
  You can then specify whether slope stability (default) or bearing capacity is to be analysed. Additional information on a strip footing is required for bearing capacity analysis (see Section 10.2.16). The safety is then acquired from a comparison of the failure load V_b and the working strip footing load $V_{work.}$ (see also Section 9.5):

  $$\eta = V_b/V_{work.}$$

  For an analysis of bearing capacity the program varies the load V_b until the slope stability FOS η = 1.0 or the utilisation factor μ = 1.0 in accordance with DIN 4084.

- "**Method:**" group box
  Select the method to be adopted. The menu bar changes according to the choice made here, showing either "**Centre-points**" or "**Slip body**".

- "**Pore water pressure:**" group box
  Generally, pore water pressures in slopes are defined using a pore water pressure line. Geotechnical applications (including **GGU-STABILITY**) compute the pore water pressure at the slice toe from the vertical distance between the slice toe and the pore water pressure line lying above it. Implicit in this procedure is the assumption that the slope is only percolated horizontally. This assumption is sufficiently precise for many slopes.

  However, this assumption is no longer justified for complex flow conditions (e.g. embankment with exterior seal). Consideration of artesian conditions is also only helpful in a few special cases. Complex flow conditions can therefore only be correctly taken into consideration if the pore water pressure is defined at every point of the slope.

  With **GGU-STABILITY** it is possible to define these pressures by means of a pore water pressure mesh (triangular mesh), which must cover the whole area of the slope under investigation. The appropriate mode is activated using the "**Pore water pressure mesh**" option button. The "**Pwp mesh**" menu, with a total of 15 menu items, is available for defining the triangular mesh.

- "**Geosythetics**" group box
  When using geosynthetics it is easiest to adopt company products. The "**Geosythetics via company products**"check box is activated when the program starts and the preferred geosynthetics manufacturer can be selected in the options menu.

  When subsequently defining the geosynthetics you can select between the products of the manufacturer defined here and their product range (see menu item "**Nail wall/Enter nails manually**" or "**Nail wall/Generate**" and the "**Geosynthetics**" button in "**Editor 1/Enter system parameters**", Section 10.2.2.11).

  Activation of the "'**Store company products in record**" check box in the above dialog box is the recommended setting. Otherwise, if the file is opened on a computer that does not have the "**.ggu-geo**" files installed, the originally selected products are deleted. The product lists of the various geosynthetics manufacturers can be selected and edited in the menu item "**Editor 1/Geosynthetics table values**" (see Section 10.2.7).

- "**Steel strips**" group box
  Activate this check box if you would like to use steel strips for slope stabilisation. In the menu item "**Editor 1/Enter system parameters**" you will then be offered the "**Steel strips**" button instead of the "**Geosynthetics**" button.

- "**Vibrodisplacement compaction**" group box
  Activate vibrodisplacement compaction adoption after Priebe (Heinz J. Priebe, *Die Bemessung von Rüttelstopfverdichtung*, Ground Engineering, December 1995) using the "**Vibrodisplacement compaction (Priebe) present**" check box. It is then possible to enter the appropriate parameters for the improved soil stratum (see menu item "**Editor 1/Enter system parameters**" button "**Soil properties**", Section 10.2.2.2)

- "**Jointing**" group box
  Using the "**Jointing present**" check box, you activate the adoption of soils with jointing, which can subsequently be defined in the menu item "**Editor 1/Enter system parameters**" button "**Soil properties**" (see Section 10.2.2.2).

- "**Stabilisation columns (StC)**" group box
  Activate analysis using stabilisation columns after Neidhart/Gömmel using the "**StC present**" check box (Gömmel, R., Neidhart, T. (2016): *Zum Ansatz von Stabilisierungssäulen beim Nachweis der Gesamtstandsicherheit*. 10. Kolloquium Bauen in Boden und Fels, S. 399–409, Technische Akademie Esslingen e.V. (Hrsg.), ISBN 978-3-943563-21-4). The stabilisation columns can then be defined in the menu item "**Editor 1/Enter system parameters**" using the "**StC**" button (see Section 10.2.2.10).

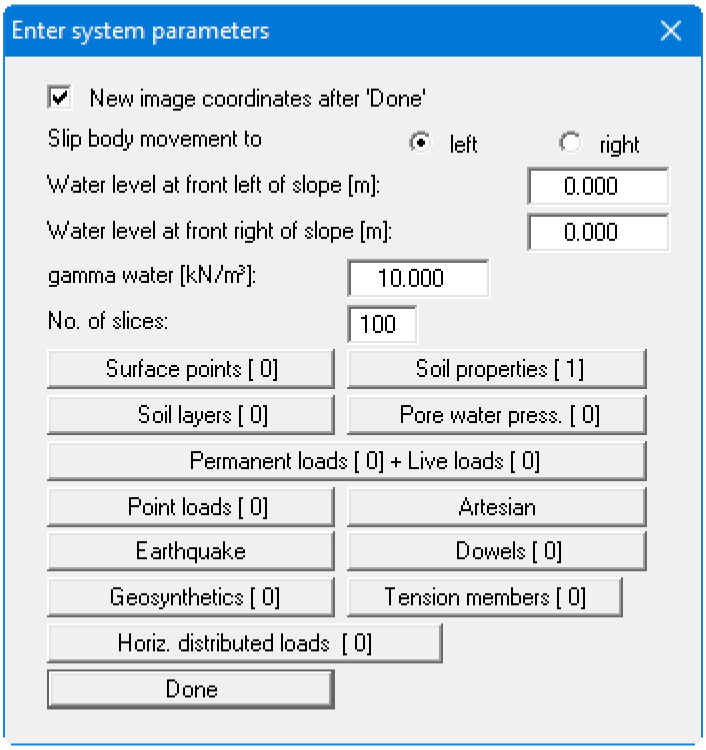
- "**Structural elements**" group box
  The literature provides several examples which divide the force of the tension member of soil dowels (e.g. DIN 4084, Example 1), geosynthetics and soil nails by the current FOS and then consider this value in the numerator of the relationships after Bishop or Janbu. Normally the check box "**Divide activated force with FOS**" can be deactivated. The forces from structural elements are then incorporated into the equation without reductions.

If you have selected the ***partial safety factor concept*** to DIN 4084:1996 or DIN 4084:2009 and then exit the dialog box using "**OK**", you first get the dialog box for defining the partial safety factors (see menu item "**Editor 1/Partial factors, ...**", Section 10.2.5).

10.2.2 "Enter system parameters" menu item

Using this menu item system input can be carried out manually using direct numerical input. Experience has shown that data input via editor is usually only used to correct geometry entered using the mouse.

You will see the central dialog box for the system to be processed:



For more inexperienced users it makes sense to have the input and settings displayed on the screen immediately after closing the above dialog box. The "**New image coordinates after 'Done'**" check box is therefore activated in the program's default settings. After an introductory phase this constant screen refresh may become onerous to experienced users, who can then deactivate this check box. The **GGU-STABILITY** program uses the latest settings next time it starts, regardless of whether an existing file is opened, or new input is made.

For wall analyses (see "**Nail wall**" menu"), only slopes failing to the left can currently be considered.

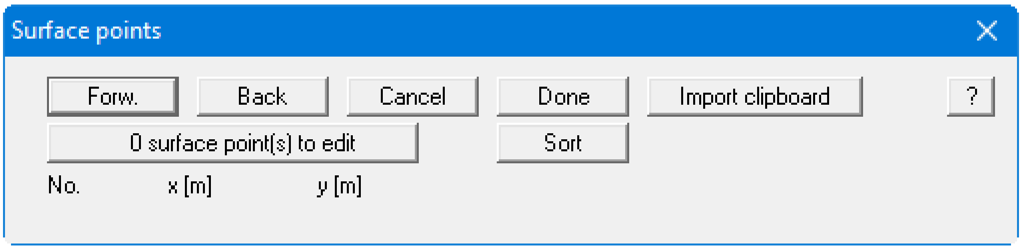
Note on water levels:
From the water levels the program calculates the water load in the area of a slice and the horizontal loading of the slope due to water pressure. If the water level is below surface level, it has no meaning for calculations.

Note on number of slices:
Small numbers of slices mean low precision and shorter calculation times. Large slice numbers mean a correspondingly longer calculation time and higher precision. The minimum number of slices is also dependent upon the complexity of the slope. A slope which is heavily layered will require a larger number of slices than one which is homogenous. We recommend analysing using at least 50 slices.

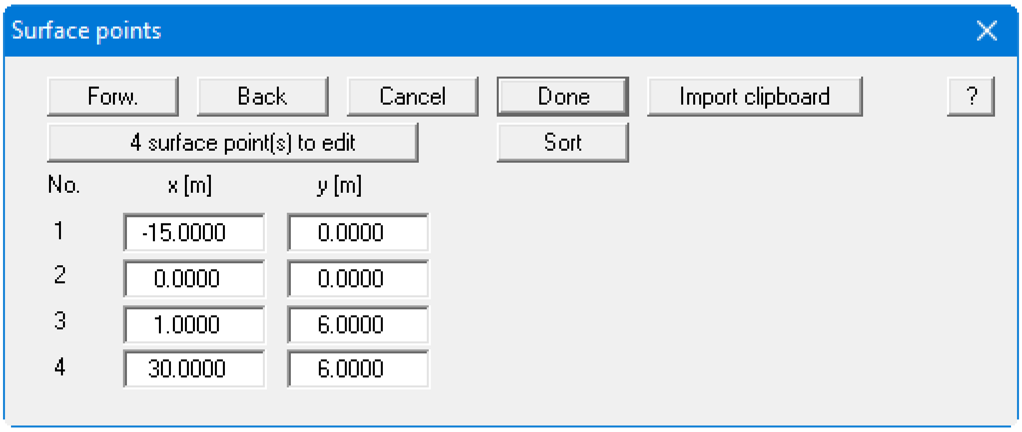
Further system input is made by pressing the relevant buttons and entering the data in the subsequent dialog boxes. The following sections describe the individual buttons in more detail. After finishing data input, select the "**Done**" button.

10.2.2.1 "Surface points" button

Using this button, the coordinates of the surface points can be specified.



To edit the number of surface points, select "**0 surface point(s) to edit**". Then enter e.g. **4** as new number and confirm with "**OK**". You can now enter 4 coordinate pairs (x/y):



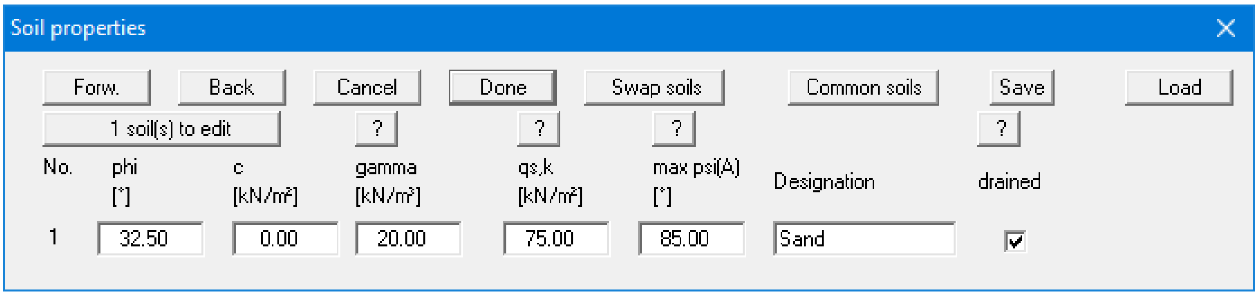
Note on surface points:
After leaving the dialog box (via the "**Done**" button), the coordinate pairs will be sorted according to increasing x-values. This allows simple deleting and inserting of surface points. You can delete a point by entering a "large" x-value and then reducing the number of points by 1. Inserting a point is achieved by increasing the number of points by 1 and then entering the coordinate pair at the end of the table. You can edit ***superfluous*** surface points even simpler using the mouse (see Section 7.5).

In the dialog box, 32 value pairs maximum are displayed at once. If the system has more than 32 value pairs, you can move through the table with the "**Forw.**" and "**Back**" buttons.

It is even easier to import surface points via the Windows clipboard. For example, if the x/y coordinates of the surface points 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 shown above by pressing "**Import clipboard**".

10.2.2.2 "Soil properties" button

Using this button, the system soil properties can be specified. The following dialog box opens:



For each soil information on friction angle, cohesion, unit weight and pore water pressure coefficient are needed. Using the "**Common soils**" button, you can easily select the soil properties of many common soils from a database or determine intermediate values. In the dialog box, which you open by pressing the "**Common soils**" button, open the "**Soils_*english*.gng_ggu**" file when first starting the program in English ("**Edit table**"/"**Load**" buttons). Then save the data set in the "**Soils.gng_ggu**" file on the program level in order to open your modified database file when the program starts. You can also enter your own data ("**Edit table**"/"**x soils to edit**" button) and save it in the "**Soils.gng_ggu**" file. You can also use your adapted file in other GGU programs by means of the "**Common soils**" function if you copy the file into the appropriate GGU program folder. The soil properties can also be saved in a "**.ggu_bkw**" file and reloaded in another project.

> In saturated zones you enter unit weight of saturated soil. In unsaturated zones you enter unit weight of wet soil.
> Under no circumstances may, for the unit weight, the buoyant unit weight be entered, as the program considers vertical water pressures via pore water pressures, and thus achieves the most exact approach.

The pore water pressure coefficient has been taken into the data input because the DIN 4084 provides for this possibility of considering consolidation processes. This method is now regarded as obsolete, as it is known to give erroneous results, especially for flat slopes. A very much more exact consideration of consolidation processes is possible via the so-called $\varphi_u = 0$ - analysis. For the shear parameters, the soil properties in the initial condition (UU-test) φ_u and c_u are used in calculations. The pore water pressure coefficient is then set to zero. Alternatively, you have the possibility of calculating with consolidation layers (see Section 9.4). In this case you enter the shear parameters of the final condition (φ', c').

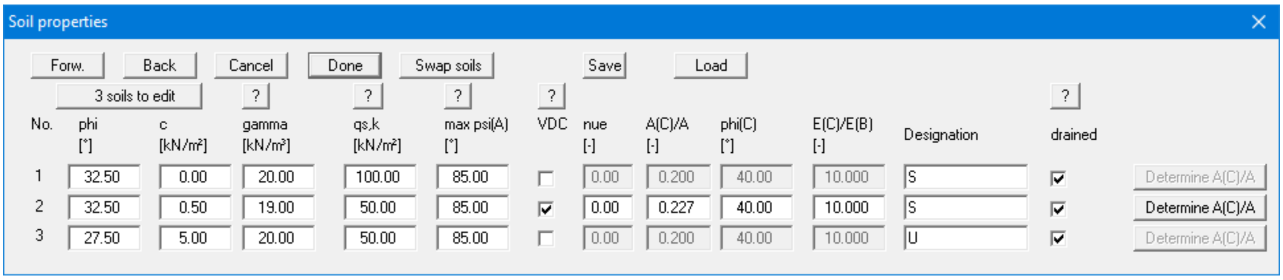
If working with tension members, you can enter the soil parameter qs,k (= skin friction) into the above dialog box. Otherwise the soil parameter qs,k is not taken into consideration and does not appear in the soil properties legend.

Max. psi(A) can be entered in the dialog box as an additional soil property but is only adopted if analyses are performed to DIN 4084:2009/EC 7. DIN 4084:2009 introduces the term 'passive tension member'. A tension member (anchor) is regarded as passive if the slip body containing the head of the tension member moves approximately like a rigid body on a slip surface and the angle max. psi(A) between the tension member axis and the slip surface achieve the following maximum values:

- for loose, cohesionless soils or soft, cohesive soils = 75°
- for firm, cohesive soils = 80°
- for medium dense, cohesionless soils and stiff, cohesive soils = 85°
- for dense, cohesionless soils = 90°

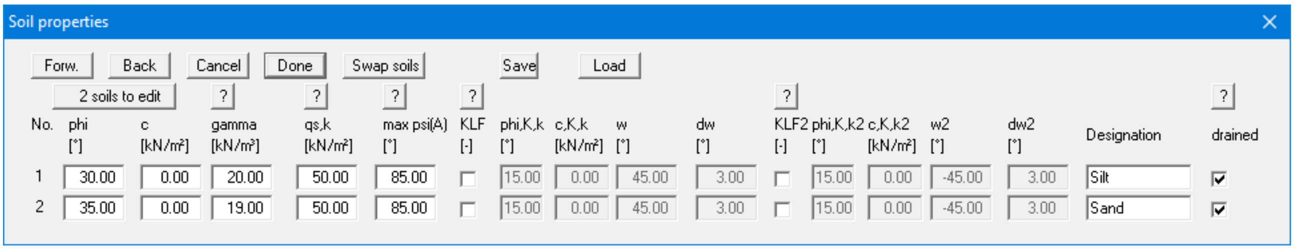
If the Kölsch method is activated (see Section 10.12.7), the dialog box for input of soil properties is expanded correspondingly (see "**Theoretical principles**", Section 9.9).

If you are working with ***vibrodisplacement compaction*** the following dialog box opens after clicking this menu item:



Define the soil layer improved by vibrodisplacement compaction by activating the "**VDC**" check box. Enter the corresponding parameters for this layer (see "**?**" button). The ratio of the areas "**A(C)/A**" can be determined and adopted in the table by pressing the far-right button. The ratio "**E(C)/E(B)**" should not be greater than 15 to a maximum of 20.

If the soil strata are ***jointed***, activate the "**Jointing present**" check box in the "**Editor 1/Analysis options**" menu item dialog box (see Section 10.2.1). You will see the following, adapted, dialog box for entering the "**Soil properties**":

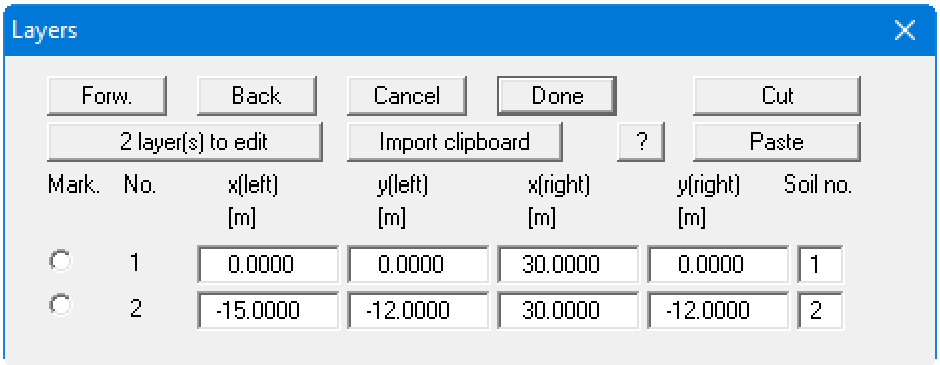


Activate the "**KLF**" check box for the appropriate soil stratum and then enter the friction angle, cohesion, and the angle and delta angle for the joint. The joint's shear parameters are adopted given a suitable slice angle. Please also review the information given by pressing the "**?**" button. In line with the joint angle given, marked soils are shown with hatching.

10.2.2.3 "Soil layers" button

The positions of the soil layers are now entered. The layering of the slope is described by the soil layers. A soil layer always consists of two x/y coordinate pairs. The area above these two coordinate pairs is assigned the properties of the soil, which is also to be entered (as a number in accordance with the numbering in the "**Soil properties**" dialog box). The soil layers are upwardly valid as far as surface level or to the lower edge of a higher soil layer. Soil layer input must be in such a manner that in all possible vertical sections the soil layer which is above has the lower layer number (do not confuse the layer number with the soil number!). This condition is checked by the program. If this condition is not adhered to, an error message will be issued, with information on the layer number and correction possibilities. Erroneous input is therefore not possible.

To edit the number of soil layers, select "**x layer(s) to edit**". Then enter **2** as the new number and confirm with "**OK**". Enter the coordinates and soil numbers of your layers.



The dialog box for the soil layers has two additional buttons in comparison to the corresponding boxes for surface points and soil properties

- "**Cut**" and
- "**Paste**".

Further, small boxes are placed in front of each of the layer values. To delete a layer you must click on the box of the layer to be deleted and then select the "**Cut**" button. The layer will then be deleted. The layer values are copied to an internal buffer and used as the new values for a layer if a "**Paste**" is necessary. This makes it easy to reorganise layers. Pasting of layer is analogous to cutting. A new layer will be pasted in front of the marked layer. Correct sorting can also be left to the program by selecting the "**Test system**" menu item from the "**Editor 1**" menu. The program will find any irregularities and offer correction possibilities.

It is even easier to import soil layers via the Windows clipboard (see Worked example 2, Section 8.2.4). For example, if the soil strata information is available in an Excel table, it is possible to copy the respective columns containing the data into the clipboard ("*Edit/Copy*") and then to paste them into the dialog box shown above by pressing "**Import clipboard**". Before importing the information, select the soil number of the strata layer being pasted.

10.2.2.4 "Pore water pressure" button

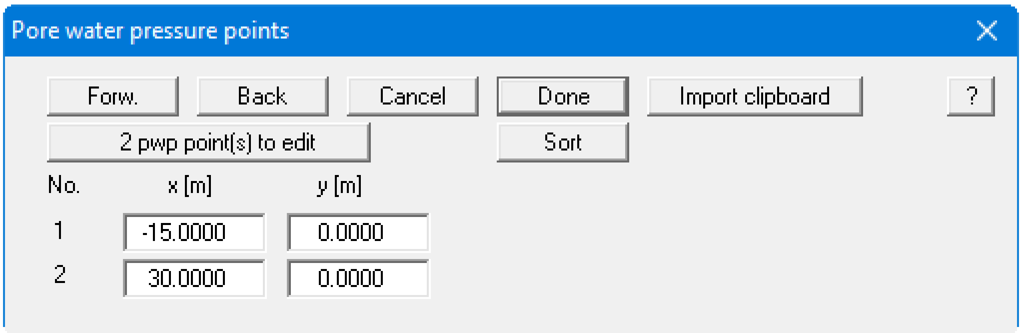
The coordinates of the pore water pressure line are entered as a polygon course. The x-values of the polygon course must increase from left to right. From the vertical distance between the slice foot and the point on the pore water pressure line above this, the program calculates the valid pore water pressure (u) for the slice. If the pore water pressure line lies below the slice foot, the pore water pressure will be set to zero.

The pore water pressure line must cover the whole of the area to be investigated in the calculations. If no pore water pressures are present you can simply define a pore water pressure line which consists of two points, and which runs below any possible y-values of the slice feet.

If a phreatic line is present within the slope, this line is generally a pore water pressure line.

First, select the "**Pore water press.**" button in the central dialog box and then, of

To enter pore water pressure points, alter the number to **2** e.g. using the "**x pwp point(s) to edit**" button and enter the values in the following dialog box:



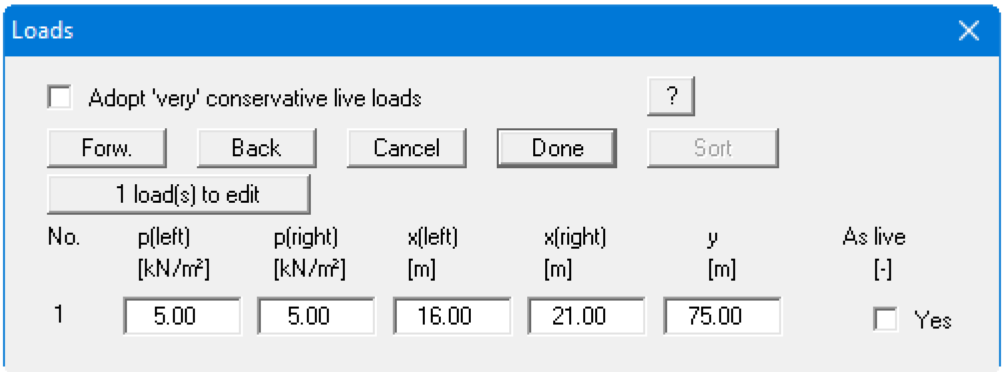
For deleting and pasting of pore water pressure points see the note in "**Surface points**" (Section 10.2.2.1).

The pore water pressure points can also be imported via the Windows clipboard. For example, if the x/y coordinates of the pore water pressure points are available in an Excel table, it is possible to copy the two columns containing the data into the Windows clipboard ("*Edit/Copy*") and then to paste them into the dialog box shown above by pressing "**Import clipboard**".

10.2.2.5 "Permanent loads + live loads" button

If you have a system e.g. with a permanent load, first, select the "**Permanent loads + Live loads**" button. To edit the number of permanent loads, select the "**x load(s) to edit**" button. Then enter **1** as the new number and confirm with "**OK**".

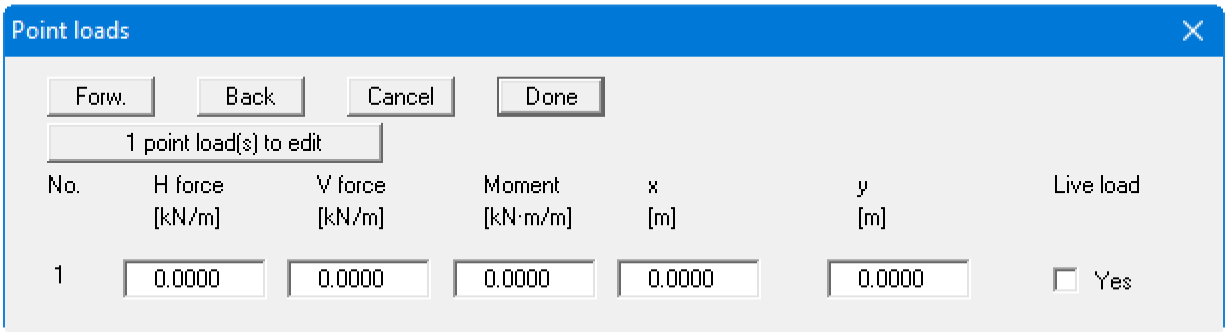
For the load definition the size of the load (as a line load) and the two x coordinates of the load must be entered. The value "**y**" designates the height of the point where the load acts. For the example the following values are to be entered:



Input of live loads is analogous to that of permanent loads; you must additionally activate the "**As live**" check box. In contrast to the permanent loads to DIN 4084, the live loads are only adopted if the resultant friction force at the foot of the slice does not possess a resisting component. If the mobilising component is always to be adopted, activate the "**Adopt 'very' conservative live loads**" (also see the "**Info**" button).

10.2.2.6 "Point loads" button

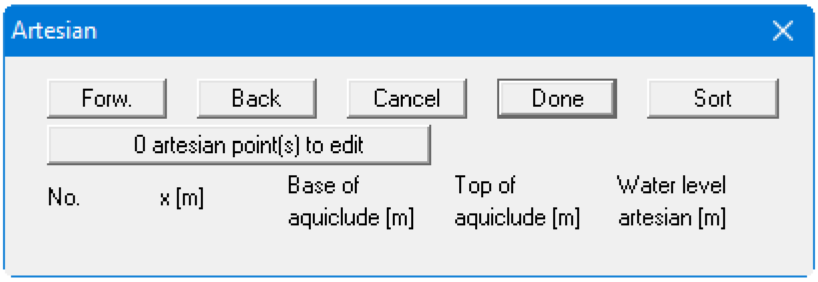
Using this button, you can enter any point loads. For input alter the number of point loads in accordance with the previous descriptions.



Then, the size of the point load (horizontal force, vertical force and moment) and the coordinates of the point of acting must be entered. The horizontal components of point loads and the moments will naturally only be considered if the point of acting lies within the slip body. The vertical component is handled corresponding to the permanent or changeable distributed loads.

10.2.2.7 "Artesian" button

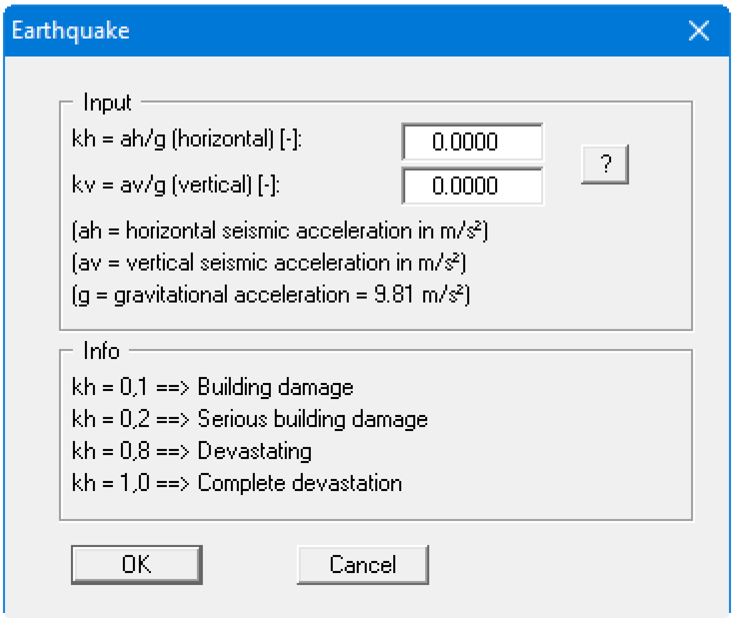
If you enter an artesian, you can thus consider confined groundwater systems. For input, select the "**Artesian**" button and enter the number of polygon points for the confined aquiclude in accordance with the previous description. Then, you must enter the x coordinates of the confined aquifer and the corresponding values for base and top of aquiclude and the water level below the base of the aquiclude.



If a slice foot is above the top of the aquiclude, the pore water pressure will be calculated from the pore water pressure line (see Section 8.2.5). If a slice foot is below the base of the aquiclude, the pore water pressure will be calculated from the water level below the base. If a slice foot is between the base and top of the aquiclude, linear interpolation will be carried out.

10.2.2.8 "Earthquake" button

After selecting the "**Earthquake**" button you can specify earthquake loads. Consideration is done using acceleration values in the horizontal and vertical direction in multiples of gravitational acceleration.



10.2.2.9 "Dowels" button

The action of soil dowels is shown in the following figure:

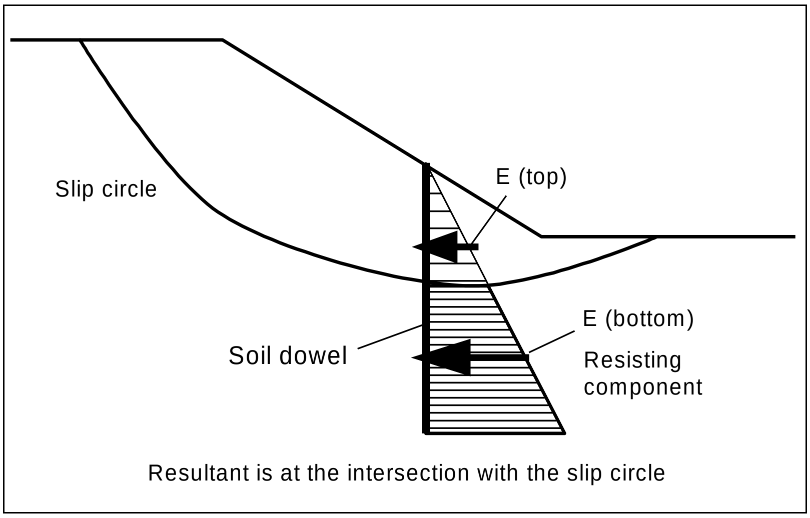


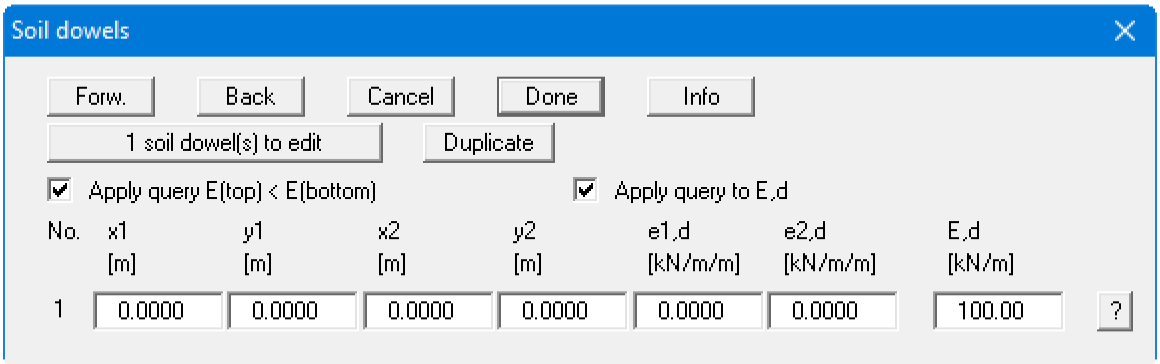
Figure 18 Soil dowels

For input, select the "**Dowels**" button and enter the number of soil dowels in accordance with the previous description. Then, the coordinates of the soil dowel and the values of the earth pressures e1,d and e2,d to be transmitted by the soil dowel must be entered. If the soil dowel consists of, e.g., a 0.75 m diameter pile which can activate a lateral earth pressure e,d of 25 kN/m² constantly along its length, and has a pile spacing perpendicular to the observation plane of 2.1 m, the earth pressure force to be entered is

$$\mathbf{25 \cdot 0{,}75/2{,}1 = 8{,}9\ kN/m^2 \cdot m/m = 8{,}9\ kN/m/m}$$

> Soil dowels will naturally only then be considered when the dowel intersects the slip surface.

The earth pressure force component which lies outside of the slip body is determined, and added to the lever arm of the intersection of the soil dowel with the slip circle centre-point. For polygonal slip surfaces, only the horizontal component of the earth pressure force is considered. If the start and end points lie outside of the slip body and two intersections with the slip body are present, the soil dowel force will not be considered. After calculations, that part of the soil dowel which was taken into account will be colour-filled, so that simple checking is possible.



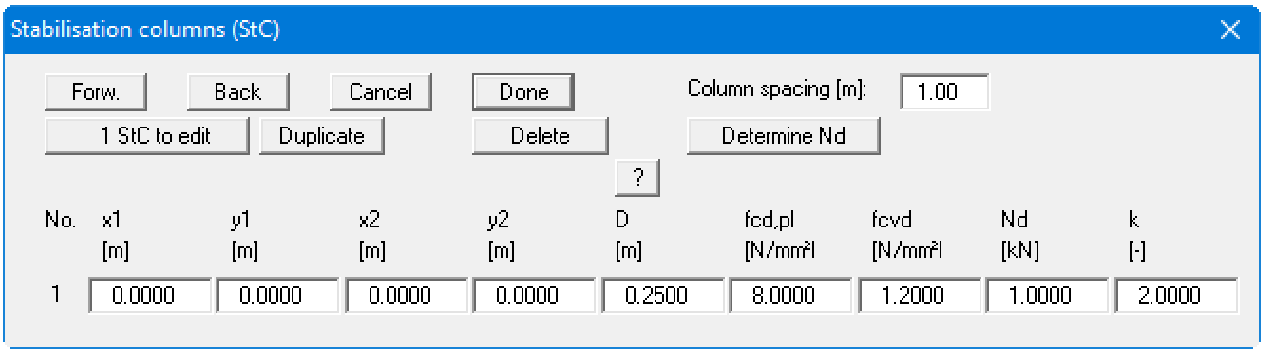
If you activate the "**Apply query E(top) < E(bottom)**" check box the earth pressures ranges are investigated (see above Figure 18). The smaller of the earth pressures is adopted for the analysis.

e1,d and e2,d are design values from the new standard and describe the magnitude of the axial forces in kN/m/m. E,d is the design force that can be accepted by the soil dowel. If the "**Apply query to E,d**" check box is activated, the computed dowel force cannot become larger than E,d (see also "**Info**" button).

Example 1 of Supplement 2 of DIN 4084 contains an H pile system, which can be considered as soil dowels.

10.2.2.10 "StC" button

If, in the "**Editor 1/Analysis options**" menu item, you have activated the "**StC present**" check box, the "**StC**" button appears instead of the soil dowel input button, to allow input of stabilisation columns after Neidhart/Gömmel. The following dialog box opens:



To enter the stabilisation columns first change the number using the "**0 StC to edit**" button. You then enter the appropriate data. An explanation of the input data can be found by pressing "**?**". Further literature can be found in Section 11.

10.2.2.11 "Geosynthetics" button

Geosynthetics are a special case of soil nails.

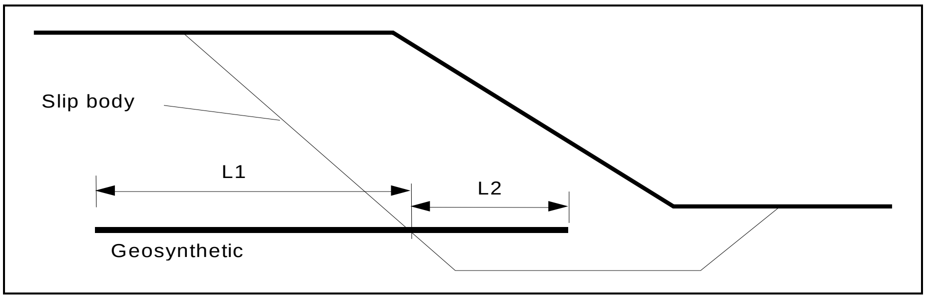


Figure 19 Geosynthetic

For each geosynthetics, you define the start and the end point. The adhesive stress (= bonding stress) is automatically calculated. Using this bonding stress t and the lengths within and outside of the slip body, the program calculates the resultant forces F_1 and F_2 from:

$$\mathbf{F_1 = f \cdot L_1 \text{ and } F_2 = f \cdot L_2}$$

The smaller of the two is the governing value. If the smaller value is larger than the maximum acceptable force R,d, only this will be used in the calculations.

If the geosynthetics intersect the slip body twice, the values for L_1 and L_2 result as follows:

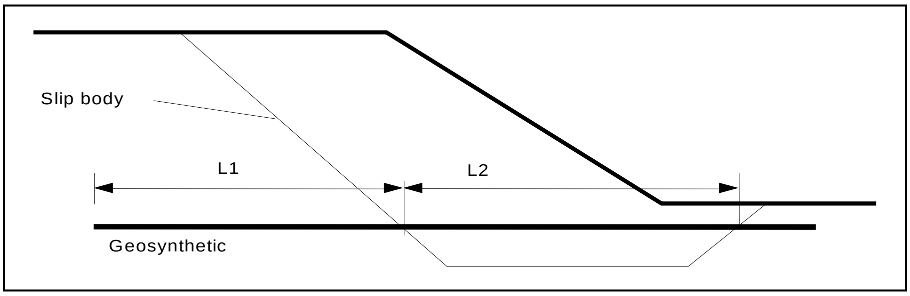
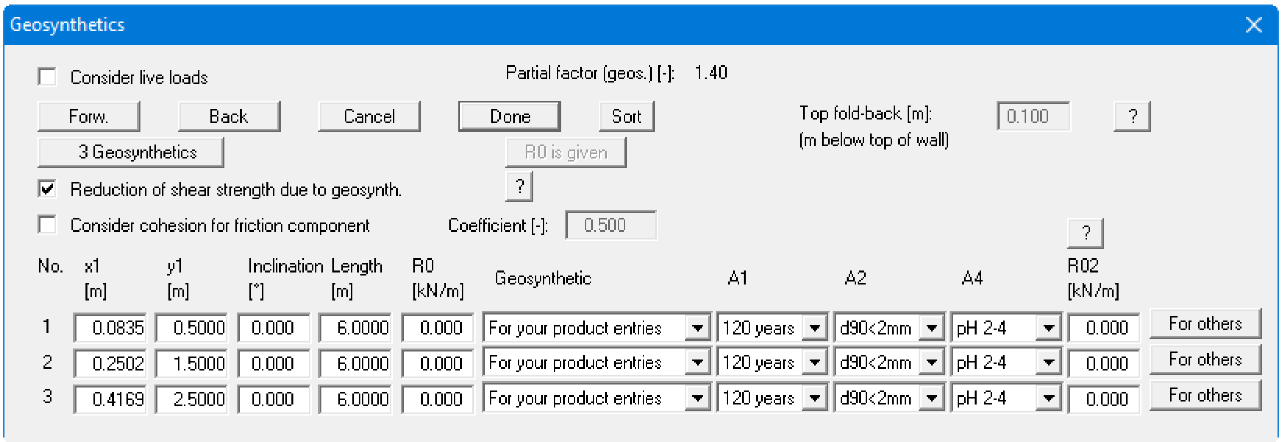


Figure 20 Geosynthetics with two intersections

By starting the program, the "**Geosynthetics via company products**" check box in the "**Editor 1/Analysis options**" dialog box is activated by default (see Section 10.2.1). When using geosynthetics you directly adopt the products of the geosynthetics manufacturers or the product range you have chosen in the beginning. The following dialog box will open, where the number of geosynthetics must be defined using the "**0 Geosynthetics**" button.

The required company product can then be selected for each of the geosynthetics. The coefficients defined for a geosynthetics, the length and the inclination can be adopted for the remaining geosynthetics by using the "**For others**" button.

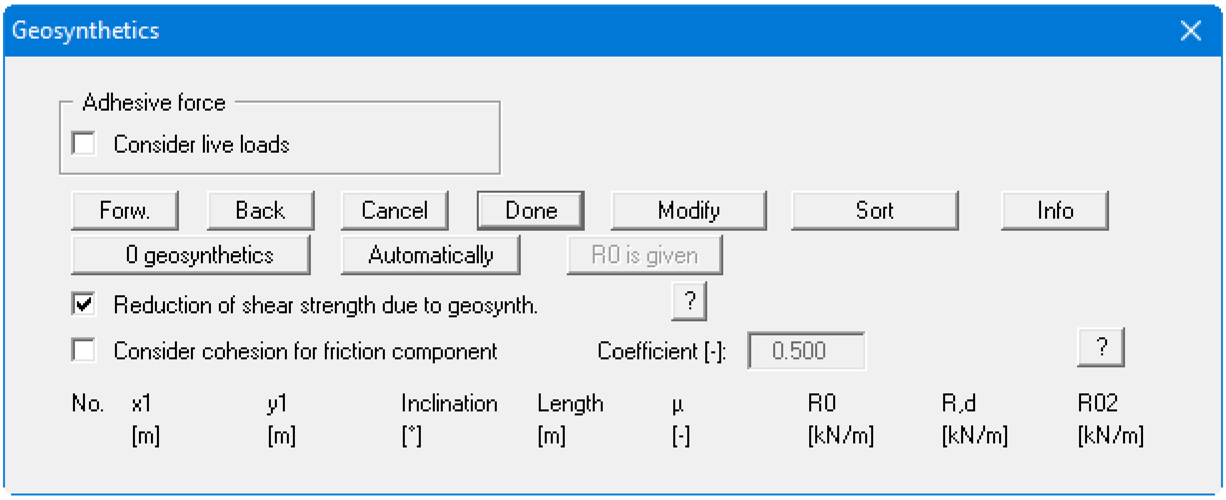


The force "**R0**" designates an anchorage at the head of the geosynthetics which can for instance be generated by folding over the geosynthetics. The resulting force cannot be larger than "**R,d**". After calculations are complete, that part of the resultant force which was taken into account will be colour-filled, so that simple checking is possible.

On exiting the dialog box, the geosynthetics are automatically sorted descending from top to bottom based on the values of y entered.

If geosynthetics by a different manufacturer are required, you select again the menu item "**Editor 1/Analysis options**" (see Section 10.2.1). New product lists of different manufacturers can be added using the menu item "**Editor 1/Geosynthetics table values**" (see Section 10.2.7).

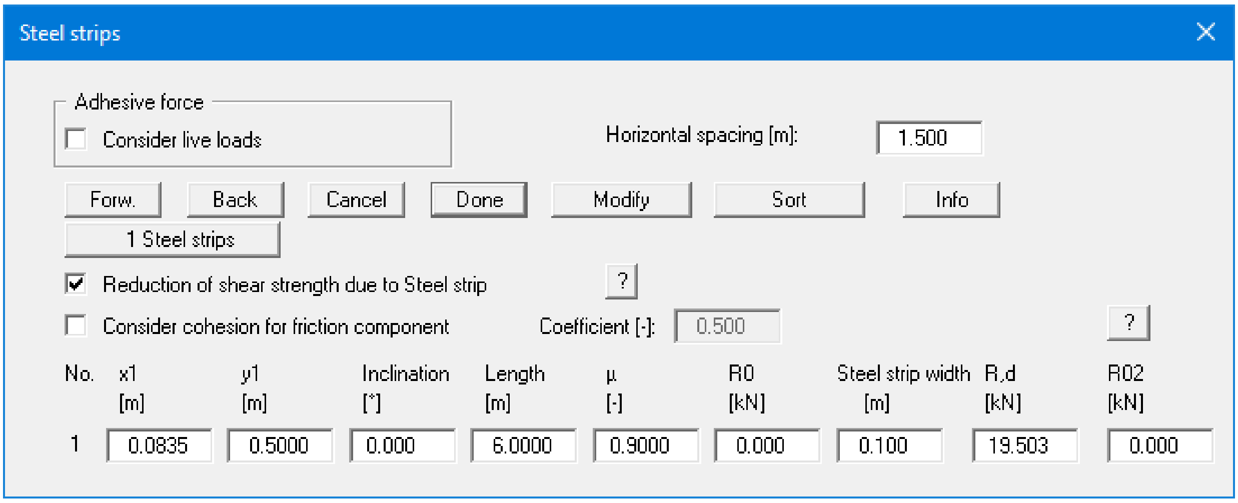
If the "**Geosynthetics via company products**" check box is <u>not</u> activated in the "**Editor 1/Analysis options**" dialog box, the following dialog box opens after clicking the "**Geosynthetics**" button in the central dialog box. The "**Info**" button provides additional information on the forces involved.



The required number of geosynthetics can be specified via the "**0 geosynthetics**" button and then the appropriate values entered. The process can be simplified using the "**Automatically**" button. A further dialog box opens for automatically generating the geosynthetics. If the "**Values from slope face**" button is used, the program automatically determines and adopts the x- and y-coordinates of the required slope face.

10.2.2.12 "Steel strips" button

If you have activated the "**Use steel strips**" check box in the "**Editor 1/Analysis options**" menu item, the "**Steel strips**" button appears instead of the button for entering geosynthetics. You will see a dialog box that is almost identical to the dialog box for geosynthetics:

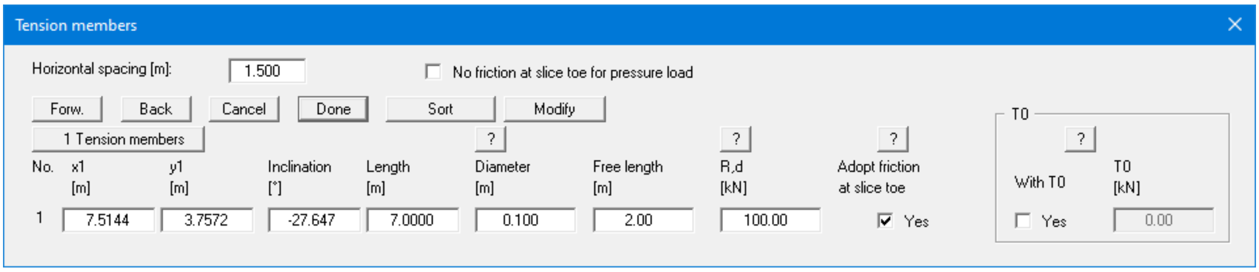


In addition, enter the horizontal distance and the steel width.

10.2.2.13 "Tension members" button

To emphasise the universal usefulness of this structural element the original "**Injection pile**" designation was changed to "**Tension member**". You can modify designation at will (e.g. to "**Soil nails**") in the "**Editor 1/Tension member designations**" menu item to make program output more legible to third parties.

When defining tension members, the acceptable friction forces are determined using the shaft friction $q_{s,k}$ defined for each soil. The following example can be found as "**Example manual tension member.boe**" file in the program's example folder.

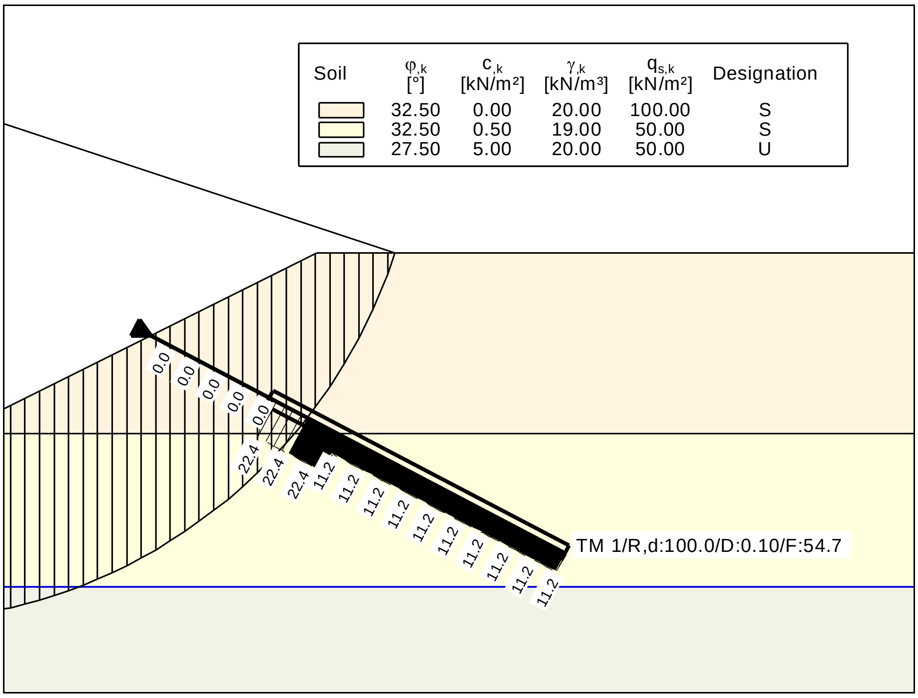


The following data are required for tension members:

- x1, y1 = tension member head coordinates
- Inclination = tension member inclination
- Length = tension member length
- Diameter = diameter D of tension member
- Free length = length from tension member head excluding grouted section
- R,d = design value of material resistance
- T0 = anchoring force at the head

If the "**Adopt friction at slice toe**" check box is activated, the friction generated by the friction force in the corresponding slice is adopted.

The following tension member is given by the input in the above dialog box:



A characteristic shaft friction $q_{s,k}$ of 100.0 kN/m² has been entered for Soil 1 and 50.0 kN/m² for Soil 2. The program computes the bonding stress τ using:

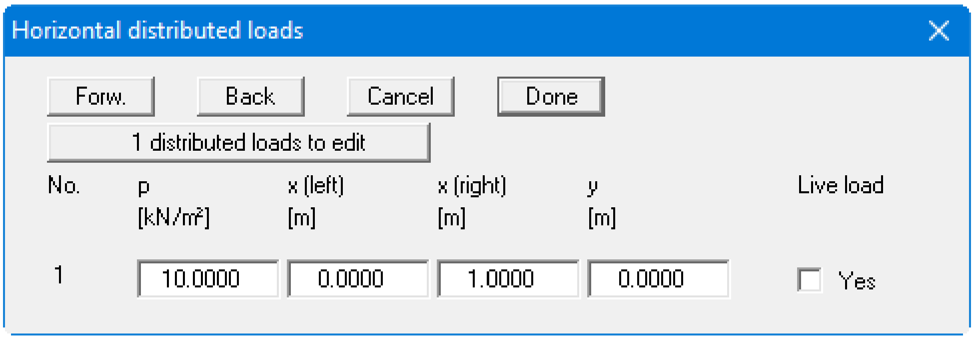
$$\boldsymbol{\tau = q_{s,k} \cdot \pi \cdot D / \gamma_N}$$

- D = diameter = 0.2 m
- γ_N = 1.40 = pull-out resistance partial factor
  (entered in "**Editor 1/Partial factors, …**", see Section 10.2.5)
- τ (Soil 1) = 22.4 kN / m²
- τ (Soil 2) = 11.2 kN / m²

This allows the pull-out forces to be computed by integration, see the blackened region in the above graphics. If the integral value becomes greater than R,d integration is aborted.

10.2.2.14 "Hor. distributed loads" button

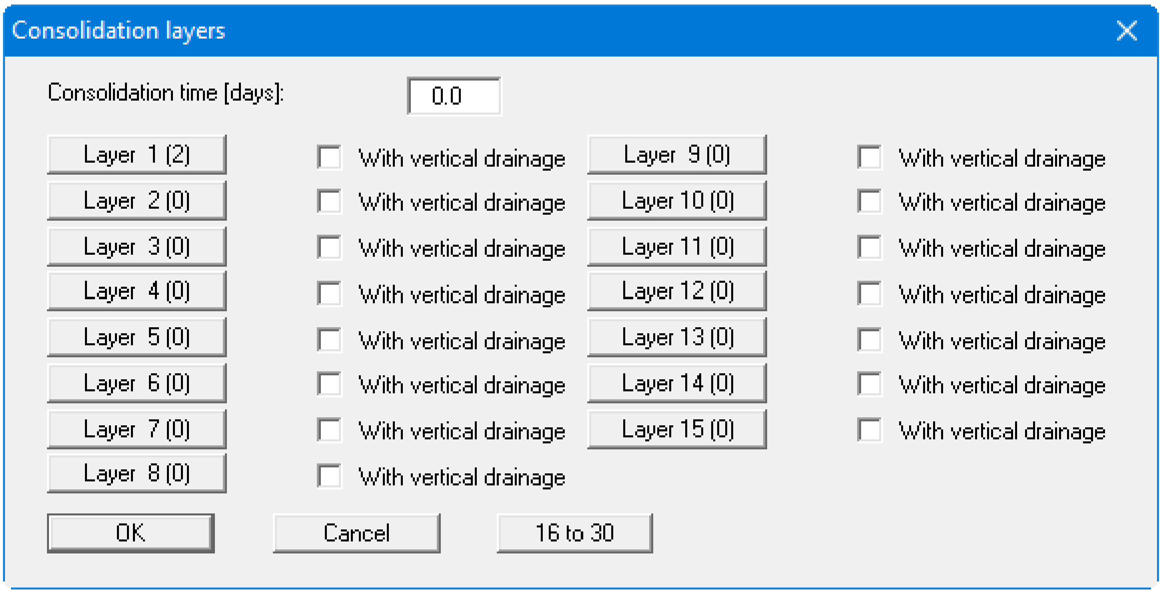
This button allows horizontal distributed loads to be defined. To change the number of horizontal distributed loads press "**x distributed loads to edit**" and enter the new number required.



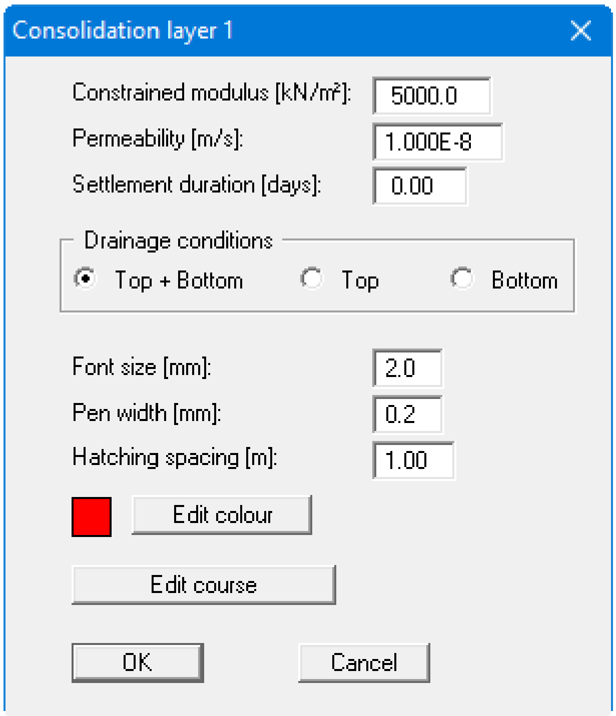
The size of the load and the two x-coordinates are required to define the load. The value in "**y**" designates the elevation of the point where the load acts. If the horizontal distributed load is a live load, also activate the "**Live load**" check box.

10.2.3 "Consolidation layers" menu item

The program can calculate excess pore water pressures due to consolidation processes in accordance with classical consolidation theory (one-dimensional) (see "**Theoretical principles**", Section 9.4). A layer can also be equipped with vertical drains. Input of consolidation layers is almost completely analogous to that of artesian water. Consolidation layers are independent of the layers entered using the menu item "**Editor 1/Enter system parameters**".



A total of 30 consolidation layers can be defined. You must first enter the "**Consolidation time [days]**", for which the excess pore water pressures are to be determined. You then select the "**Layer x**" button. The number in brackets after the layer name indicates how many polygon course points have been defined for this layer. If the "**With vertical drainage**" check box is not activated, the following dialog box opens:



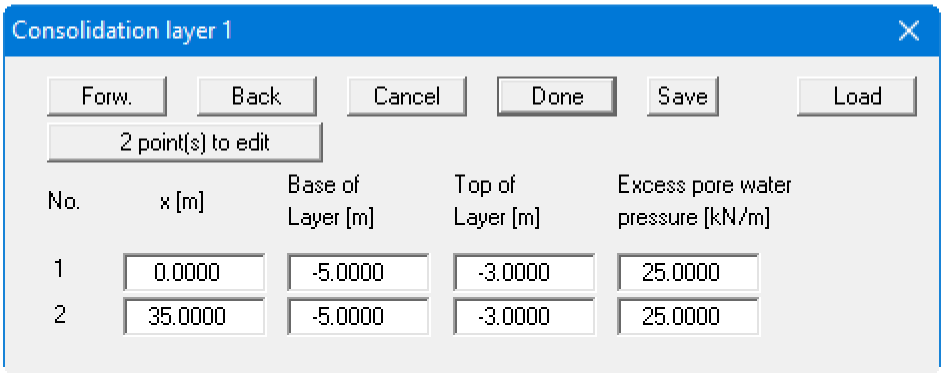
You must then define the "**Constrained modulus**", the "**Permeability**" and the "**Settlement duration**" of the respective layer. If you enter a value for "**Settlement duration**" ≠ 0, the program calculates the pore water pressure distribution within the layer at the time:

Settlement duration + t .

Then enter the "**Drainage conditions**". Subsequent input affects graphical representation only.

If you have defined a consolidation layer "**With vertical drainage**", a slightly different dialog box appears. Additionally, you must enter the "**Drained distance de**" and the "**Radius of drain rw**" of the vertical drain. An input of drainage conditions is not necessary, as only horizontal drainage to the drains is assumed for vertical drains.

The polygon course, which describes the consolidation layer, is entered after pressing the "**Edit course**" button.



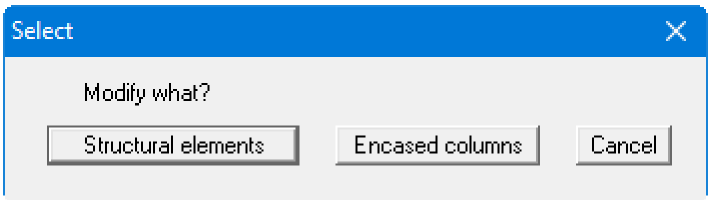
You must enter for each polygon course point:

- x-ordinate,
- base of the layer,
- top of the layer and
- excess the pore water pressure at time t = 0.0

If a slice foot is within the consolidation layer, the program can, using the above described input, determine all necessary values for a consolidation calculation at his point.

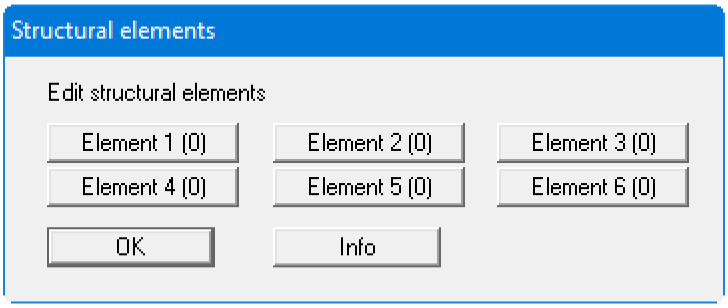
10.2.4 "Structural elements/Encased columns" menu item

Structural elements or encased columns can be defined using this menu item. First, select the required object from an option box:

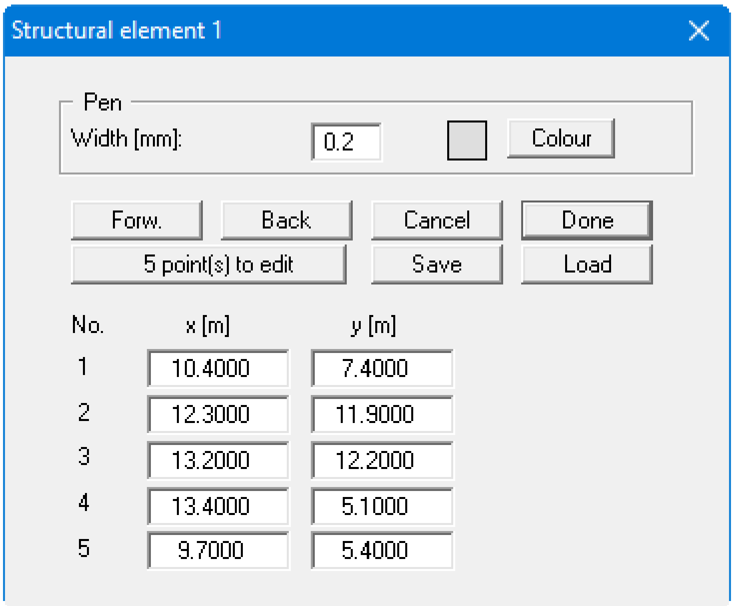


10.2.4.1 Structural elements definition

In some cases, structural elements are within the slope, the shear strength of which is so great that no safety calculation is necessary, if the slip body cuts the structural element. In principle, help can be had by defining layers with high cohesion. However, it is more elegant to define the structural element as an enveloping polygon course. Structural elements have no material properties (in particular no unit weight !!!). If, however, a polygonal section of the structural element has an intersection with the currently investigated slip body, this slip body will be assigned a safety factor of 20 000.0 and no calculation will take place.



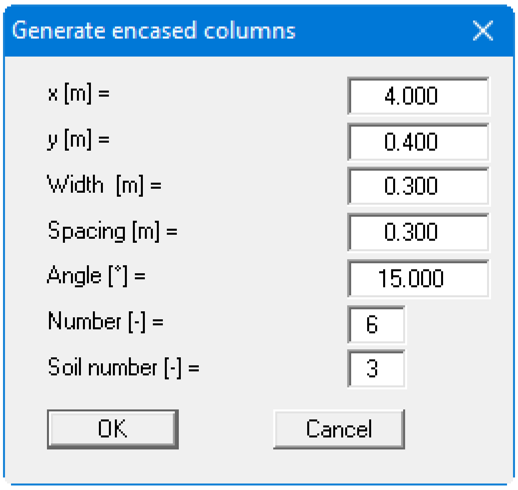
A total of 6 structural elements can be defined. The numbers in brackets on the "**Element x**" buttons indicate how many polygon course points each structural element contains. To define an enveloping polygon course, select the "**Element 1 (0)**" button, for example.



Using the "**x point(s) to edit**" button, define the number of polygon course points and enter the coordinates for the individual points. You do not need to close the polygon; the program does this automatically. The pen width and colour of the structural element can be edited at the top of the dialog box.

10.2.4.2 Encased columns definition

Define soil columns analogous to the system soil strata by specifying the x- and y-coordinates of the base and entering a soil number. The required number of soil columns (but at least two) can be simultaneously generated using the dialog box.



A separate number should be awarded for each new soil to allow the generated soil columns to be better differentiated from other soil types by their colours. The angle corresponds to the increase in elevation of the bases of the soil columns (see drawing below):

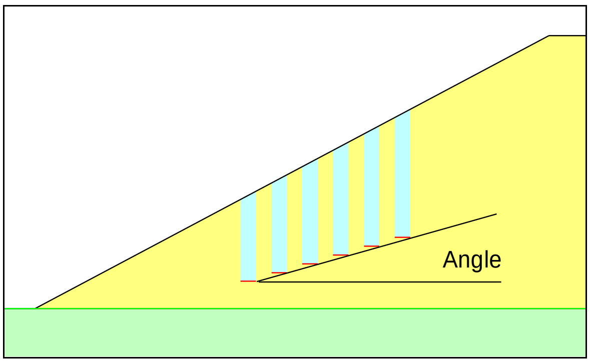


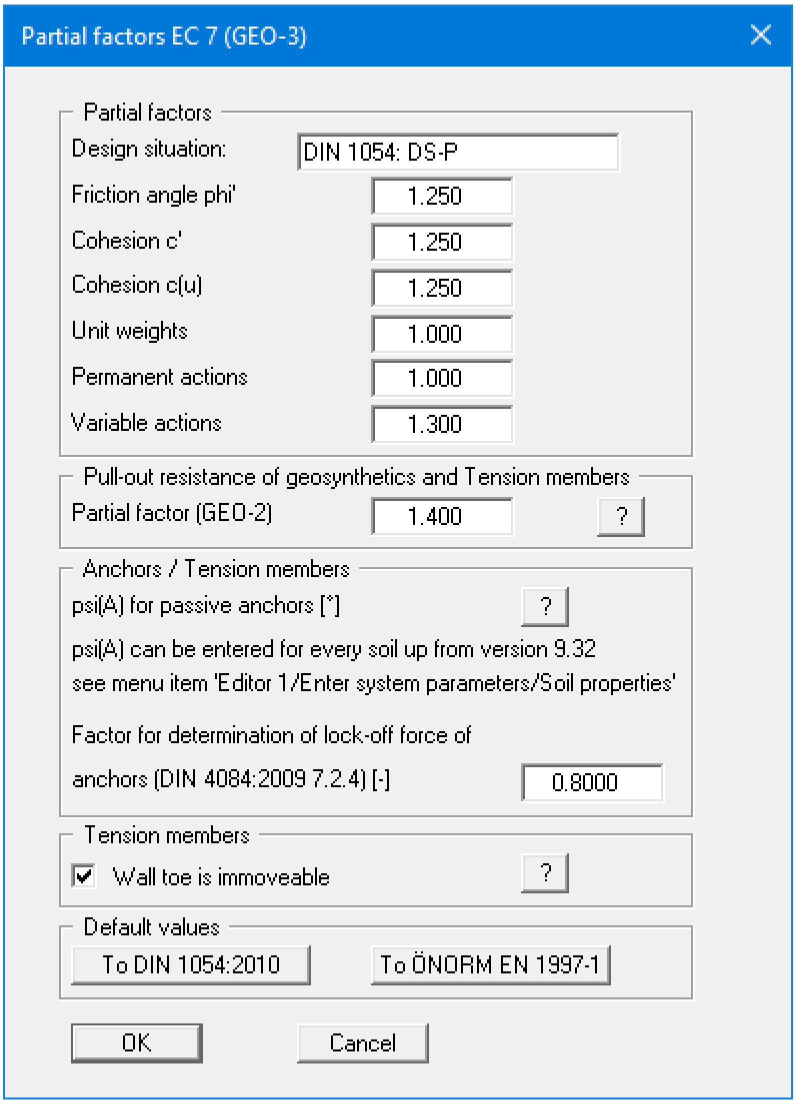
Figure 21 Encased columns - angle input

After confirming your input, a message is shown with the number of newly generated strata. It is then possible to test the system. Soil columns protruding into overlying strata are identified and, following a query, corrected.

If it is necessary to delete the last created soil columns, simply use the undo function. Click the  icon in the toolbar (also see Section 10.9.6). The soil columns generated are added to the list of soil strata and can also be edited or deleted via the list (see Section 8.2.4, "**Example 2: Data input via editor/Step 2: Enter system parameters/Soil layers**").

10.2.5 "Partial factors, ..." menu item

If, using the "**Editor 1/Analysis options**" menu item, you have selected "**EC 7**" as applicable standard, the following dialog box appears:



Here you enter the partial factors, the values for psi(A) and the factor for the lock-off force of tension members (see Section 9.1).

In the "**Default values**" group box you can use the "**To DIN 1054:2010**" button to apply the partial safety factors of DIN 1054:2010 or EC 7 for the various load cases. Depending on the selection, the design situation is entered in the upper box and can be displayed in the General legend, if required ("**Design situation text**" check box, in the menu item "**Editor 1/General legend**", see Section 10.2.12). The load case designations were altered for the EC 7 partial safety factor concept:

- Load Case 1 is now DS-P: Persistent Design Situation
- Load Case 2 is now DS-T: Transient Design Situation
- Load Case 3 is now DS-A: Accidental Design Situation

In addition, there is a seismic design situation (DS-E). In the DS-E design situation all partial factors = '**1,0**'.

It is also possible to select the partial safety factors compliant with Austrian standards using the "**To ÖNORM EN 1997-1**" button.

The partial factors for the design of structural elements (e.g. the concrete shell of a nail wall) can be entered using the menu item "**Nail wall/Verifications/Safety**" (see Section 10.11.4).

10.2.6 "Project identification" menu item

You can enter a description of the problem being processed, which will then be used in the "**General legend**" (see Section 10.2.12).

10.2.7 "Geosynthetics table values" menu item

This menu item's dialog box contains your selected geosynthetics manufacturers products or the selected product range. **Editing the data is not recommended!** Generally, files that can be imported into **GGU-STABILITY** can be requested from the geosynthetics manufacturers. During program installation a variety of these manufacturer files are stored in the program folder. Normally, because you change between different geosynthetics using the "**Editor 1/Analysis options**" menu item, this menu item is not required (see Section 10.2.1).

If you would like to create your own products or have opened a different manufacturer's products, make the appropriate entry in the "**Company name:**" box. If you save your new product list at the program level, e.g. using the file name "**GGU-STABILITY9.ggu_geo**", your new list will appear with the company name entered above in the "**Editor 1/Analysis options**" menu item option box.

The **GGU-STABILITY** program automatically loads up to 10 geosynthetics files ("**GGU-STABILITY.ggu_geo**" to "**GGU-STABILITY10.ggu_geo**") when the program is started.

10.2.8 "Test system" menu item

After data input via mouse or *by hand* you can have the program check your input using this menu item. For almost all (!!!?) erroneous input, the program offers correction possibilities, which you may also reject. You should, nevertheless, critically check your system after making corrections.

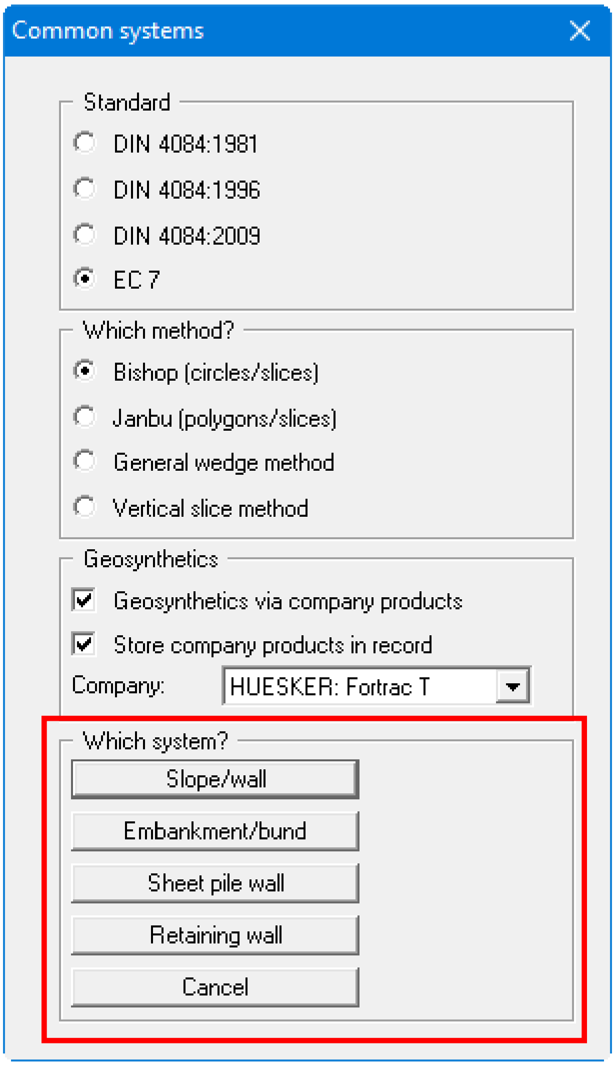
> This check will **ALWAYS** be carried out, even if you don't select this menu item, when you start calculations. If the program recognises erroneous input, calculation of safety factors will be refused.

10.2.9 "Mirror system" menu item

You can have the data mirrored around the vertical axis.

10.2.10 "Common systems" menu item

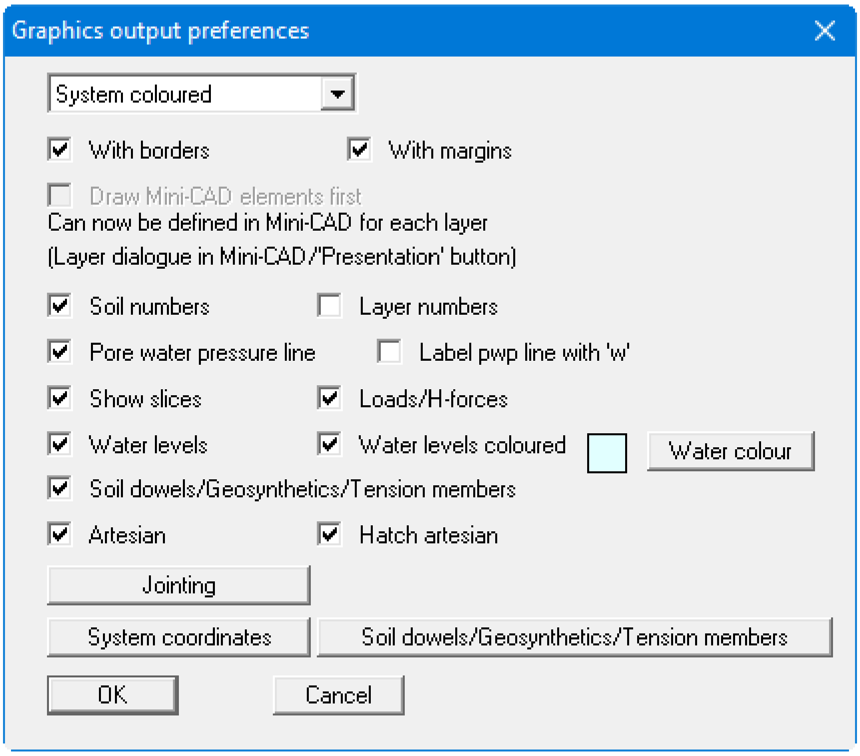
Often, only a simple system needs to be investigated. A variety of systems can be generated in a matter of seconds using this menu item. If the "**Common systems**" dialog box is selected directly after the program starts, the dialog box shown below opens. If this menu item is opened, you immediately see only the lower "**Which system?**" area. The standard and procedure to be used have already been selected using the "**Editor 1/Analysis options**" menu item.



After input of the required data the system is displayed on the screen. You now need only define the soil properties, the centre-point array and search grid or polygonal slip body. You are free to use this system as a data basis for further refinements after generation.

10.2.11 "Graphics output preferences" menu item

With this menu item you can specify in which form the system is to be presented on the screen, in order to carry out an optical check of the details. The corresponding dialog box is, on the whole, self-explanatory.



Use the "**System coloured**" combo box to activate coloured system visualisation, i.e. the soil colours defined in "**Editor 1/Soil properties**" are adopted for use in the soil properties legend and in system visualisation. It is also possible to define hatching for the various soil types, or turn off colour and hatching completely (see also Section 10.2.13).

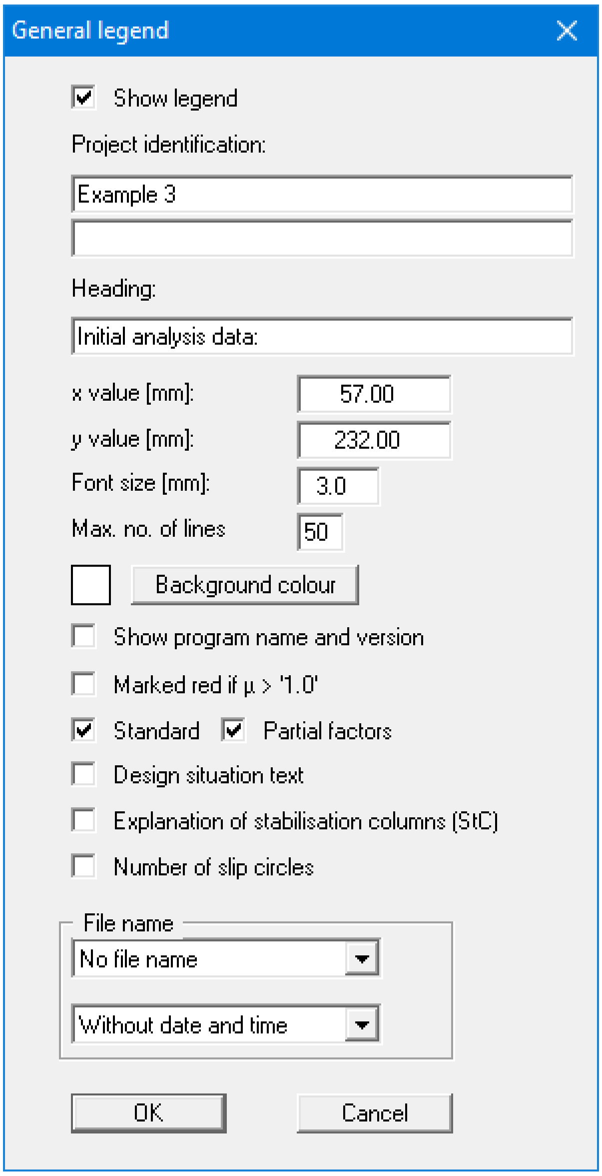
Using the upper two check boxes the borders and page margins can be switched off (also see menu item "**Page size + margins/Page size and margins**", Section 10.10.7).

Using the "**Jointing**" and "**System coordinates**" buttons you open a dialog box in which the visualisation format in the graphics can be influenced.

With the "**Soil dowels/Geosynthetics/Tension members**" button you can call up a dialog box, in which you can influence the visualisation of these slope elements (presentation width, height, labelling). If, in the "**Editor 1/Analysis options**" menu item, you have activated the "**StC present**" check box, "**StC**" appears for stabilisation columns after Neidhart/Gömmel on the button instead of the soil dowels. Instead of "**Geosynthetics**", if the use of steel strips is activated, "**Steel strips**" is then displayed here.

10.2.12 "General legend" menu item

A legend with general information about the system is shown, if the button "**Show legend**" is activated. In particular, important information on the input values for consolidation layers is given.



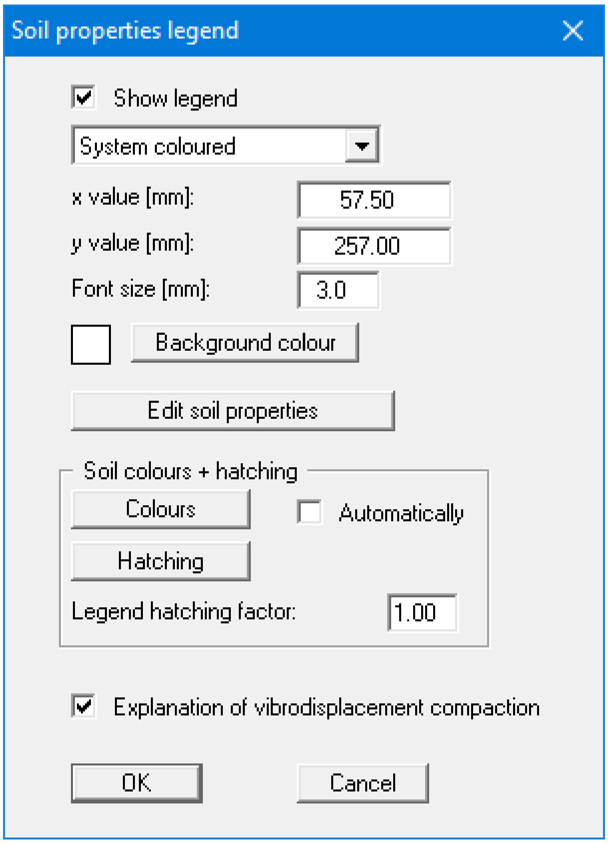
Any description previously entered using the menu item "**Editor 1/Project identification**" is used in the above dialog box. In addition to the project identification it is also possible to enter a heading for the general legend here. Activate the corresponding check boxes to also display the adopted standards and the partial factors. The further explanations are adapted to your settings in each case, for example, explanation StS for stabilisation. The number of slip circles can be entered for the slip circle method.

You can define and edit the position of the legend using the values "**x value**" and "**y value**". 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. The current file name can be shown in the legend, either with or without the path. Date and/or time can be displayed, too.

10.2.13 "Soil properties legend" menu item

If the "**Show legend**" check box is activated, a legend is displayed on the output sheet consisting of the current soil layers and the corresponding soil properties.



You can define and edit the position of the legend using the values "**x value**" and "**y value**". 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.

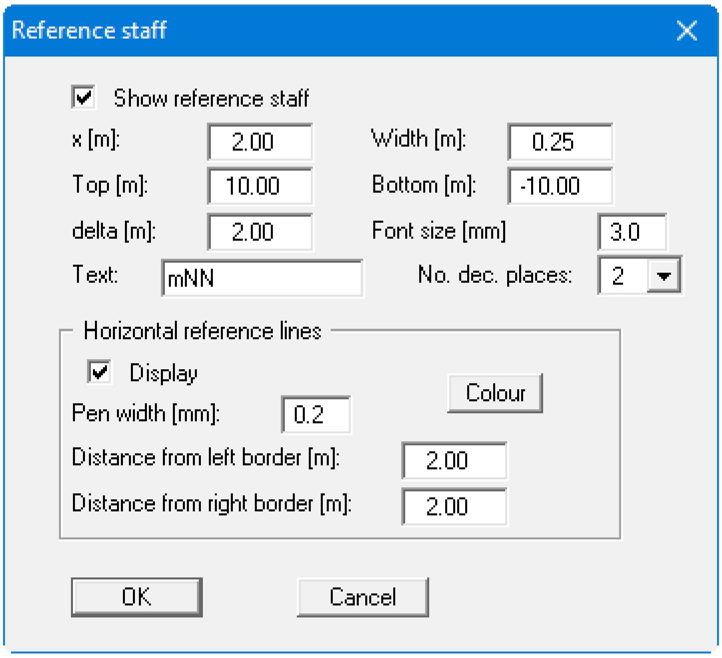
It is possible to edit the soil properties in the soil legend box by clicking the "**Edit soil properties**" button. A dialog box opens; this is normally reached via the menu item "**Editor 1/Enter system parameters**", "**Soil properties**" button (see Section 8.2.3).

If "**System coloured**" is activated in the combo box, the soils will be displayed coloured both in the soil properties legend and in the system graphics. You can also define either hatching or colour fill and hatching for the different soil types in the combo box. If you select "**System without all**", the soils are merely numbered. The required settings can be made in the "**Soil colours + hatching**" group box:

- "**Automatically**"
  The soils are assigned soils colours automatically by the program. If the check box is not selected, the soil colours individually defined using the "**Colours**" button will be adopted.
- "**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.
- "**Hatching**"
  Opens a dialog box in which you can define different hatching for each soil.
- "**Legend hatching factor**"
  Input here allows tighter hatching in the soil legend. Input **< 1.00** can be useful if the hatching spacing is so large that the differences in the hatching of individual soils can no longer be properly discerned in the relatively small boxes used in the legend.
- "**Explanation of vibrodisplacement compaction**"
  The explanation of vibrodisplacement compaction can be displayed in legend by activating this check box.
- "**Explanation of jointing**"
  If this check box is activated, the soil properties shown for the joint layers are explained.

10.2.14 "Reference staff" menu item

Representation of a reference staff including horizontal auxiliary lines provides improved clarity with regard to terrain heights.



The position of the reference staff is entered as the distance from the left page margin in "**x[m]**". The height range in y-direction is given as "**Top [m]**" and "**Bottom [m]**". All values are in metres in the scale selected (see menu items in the "**Page size + margins**" menu).

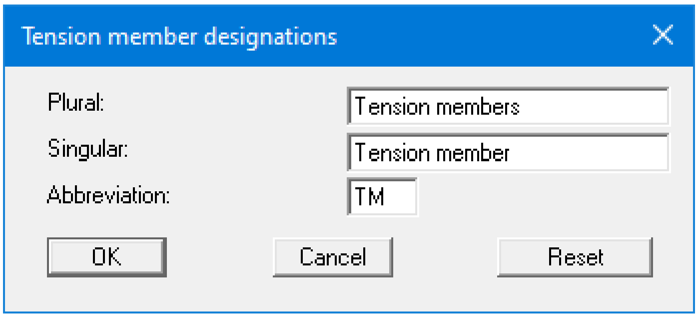
10.2.15 "Move objects" menu item

Select this menu item in order to position legends, diagrams and 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.

10.2.16 "Designations for tension member etc." menu item

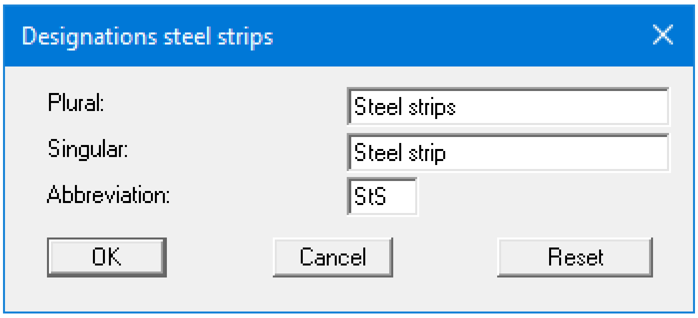
In this menu item you can change the designations for tension members and steel straps according to your ideas.

The following dialog box opens when you press the "**Tension members**" button:



You can define here whether you want to name the tension member differently. The designations entered are used in the further dialog boxes under the menu "**Nail wall**" and in the diagrams and legends displayed on the screen.

Similarly, the following dialog box opens by pressing the "**Steel strips**" button:

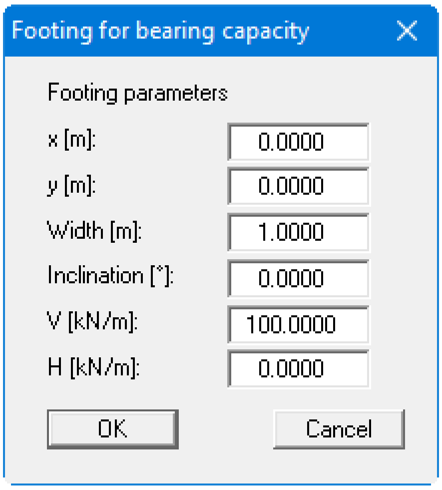


Here, too, you can change the designations for the diagrams and legends. The "**Reset**" button allows the program to re-enter the standard terms at any time.

If you change the language while editing the file, you can use the "**Reset**" button to have the program insert the respective standard terms in the set language.

10.2.17 "Footing" menu item

If you activated the "**Bearing capacity**" option button in the "**Editor 1/Analysis options**" menu item, a further menu item, "**Footing**", becomes visible.



Here, you enter the footing parameters required for bearing capacity analysis.

- "**x**", "**y**" = position of left footing edge
- "**Width**" = footing width
- "**Inclination**" = inclination of footing base
- "**V**" = vertical load on footing
- "**H**" = horizontal load on footing

For a bearing capacity analysis the safety is then acquired from a comparison of the failure load V_b and the working strip footing load $V_{work.}$ (also see Section 9.5):

$$\eta = V_b/V_{work.}$$

For an analysis of bearing capacity the program varies the load V_b until the slope stability FOS η = 1.0 using the global safety factors or the utilisation factor μ = 1.0 using the partial factors. When using variation, the horizontal load is increased or decreased corresponding to the vertical load.

10.3.1 "Array" menu item

This menu item was previously explained in "**Worked example 1: Data input with mouse**" (see Section 7.4). When entering system data using the mouse it may often be useful to lock on to certain array points.

Using this menu item you can define the array to be adopted. The default array colour is a light grey which does not ***overwhelm*** the graphics. If you would prefer to use a different colour, use the "**Edit array colour**" button. When the "**Use array**" check box is activated the defined array is used in conjunction with the subsequent menu items for ***mouse input*** of system data and for the definition of polygonal slip bodies.

10.3.2 "Surface" menu item

With this menu item you can enter the system surface line via the mouse (see **Worked example 1**, Section 7.5). The dialog box which appears explains the possibilities. This function can also be initialized pressing the [**F3**] function key.

If you activate the "**Snap to current line ends**" check box, crosshairs will be shown instead of the mouse cursor or, if you prefer, large crosshairs with an enveloping rectangle. If you have already made some input or Mini-CAD data (including those created with DXF import) are present, the program will snap on to the nearest point (if the point is within the rectangle) when you click the mouse. If, e.g., the system to be calculated exists as a DXF planning file, you can greatly simplify input.

This information is also valid for all the following sections which are concerned with system input with the mouse.

10.3.3 "Pore water pressure" menu item

You can define a pore water pressure line, in complete analogy to surface points (see **Worked example 1**, Section 7.5).

> Input is rejected if you selected the pore water pressure mesh for pore water pressure definition in the "**Editor 1/Analysis options**" menu item.

10.3.4 "Layers" menu item

You can define layers, in complete analogy to surface points. Here, however, you must click on two points for each layer points (see **Worked example 1**, Section 7.6). This function can also be initialized pressing the [**F6**] function key.

10.3.5 "Loads/point loads" menu item

You can define "**Permanent loads**", "**Live loads**", "**Distributed loads**" or "**Point loads**", in complete analogy to surface points. After input of two x-coordinates via the mouse, you must specify the size of the load (e.g. in kN/m²) in a dialog box. For point loads, only input of the single force (e.g. in kN/m) is required. The direction of action is clearly marked in the following graphics.

Permanent loads and live loads can be edited in a dialog box by pressing [**Shift**] and the right mouse button simultaneously in the centre of the load. These loads can be deleted by pressing [**Ctrl**] and the right mouse button. Point loads and distributed loads are edited by double-clicking the loads. The dialog boxes for these elements, which are otherwise accessed via "**Editor 1/Enter system parameters**", then open (see Sections 10.2.2.5 and 10.2.2.6). If these loads need to be deleted, specify the number in the dialog box as "**0**".

10.3.6 "Soil dowels/StC/Geosynthetics/Steel strips/Tension members" menu item

In complete analogy to surface points, you can define the locations of soil dowels, stabilisation columns after Neidhart/Gömmel, geosynthetics, steel strips or tension members. First, select the required element from the dialog box. After defining two x-/y-coordinates with the mouse you will see a dialog box in which you can enter further parameters for the respective element.

- "**Soil dowels**"
  You specify the acceptable earth pressure e,d (acts perpendicular to dowel axis; also see Section 10.2.2.9) at the two points clicked (in kN/m/m). The direction of action is clearly indicated in the subsequent graphics. Negative values must be entered where required.
- "**StC**"
  If, in the menu item "**Editor 1/Analysis options**" menu item, you have activated the "**StC present**" check box, you can define stabilisation columns by means of the displayed "**StC**" button instead of soil dowels using the mouse (see Section 10.2.2.10).
- "**Geosynthetics**"
  A dialog box opens, from which you select the required product from the list of geosynthetics manufacturers or the product range you selected in the "**Editor 1/Analysis options**" menu item.
- "**Steel strips**"
  A dialog box appears in which you enter μ, steel strip width, R0 and R,d.
- "**Tension members**"
  Enter the design value of material resistance R,d, the diameter and the free length of the tension member (also see Section 10.2.2.13).

10.3.7 "Artesian" menu item

You can define the position of an artesian aquiclude, in complete analogy to surface points. After entering an x and y coordinate pair via the mouse, you must specify the thickness of the aquiclude ("**Thickness of the aquiclude**") and the artesian water pressure ("**Water level**").

10.3.8 "Water levels" menu item

You can specify the water level at the front "**left**" of the slope by pressing the left mouse button, at the front "**right**" by pressing the right mouse button.

10.3.9 "Consolidation layers" menu item

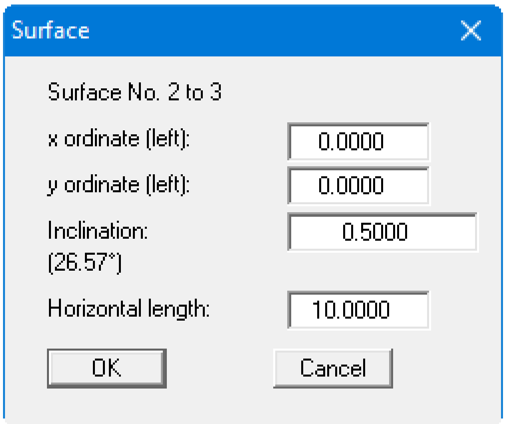
Consolidation layer input is carried out in complete analogy to artesian input (see Section 10.3.7). After selecting this menu item, however, you must first enter the number of the consolidation layer to be edited.

10.3.10 "Structural elements/Encased columns" menu item

First, decide whether to define structural elements or encased columns. For structural element input first enter the number of the structural element to be edited. Then define your points using the left mouse button. For a soil column, define the x- and y-coordinates of the base of the soil columns using mouse clicks. The same dialog box opens for generating encased columns opens as that in the menu item "**Editor 1/Structural elements/Encased columns**" (see Section 10.2.4.2).

10.3.11 "Inclinations" menu item

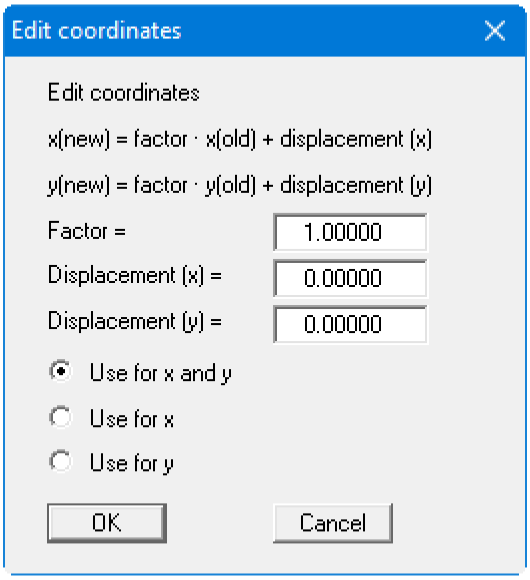
In some cases, it is desirable to be able to edit slope geometry via inclinations. After selecting this menu item, click in the centre of the surface line (layer line, etc.) to be edited.



Among other things, you can edit the inclination in the dialog box that opens here.

10.3.12 "Coordinates" menu item

System input via the mouse is comfortable. You can ease your work further by getting hold of a Bitmap file of the system using a scanner. This Bitmap file can be displayed on the screen using the "**Mini-CAD**" program module (see Section 10.9.5 and supplied "**Mini-CAD**" manual). You need then only click on decisive system points. As the exact scale adjustment of the Bitmap file can be somewhat time-consuming, you can edit and move the coordinates of the slope geometry using this menu item.



10.3.13 "Stresses" menu item

After selecting this menu item, you can have the governing stresses displayed at any point in your system. You will see an information box containing the values and a "**Soil properties**" button. If you then click on this button, a further information box containing the soil property values for that location is displayed.

10.3.14 "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 "**Editor 1/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**] + [**Backspace**] or the appropriate tool in the toolbar (see Section 10.9.6).

10.3.15 "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 "**Editor 2/Undo**" will be restored. This function can also be reached by using the key combination [**Ctrl**] + [**Backspace**] or the appropriate tool in the toolbar (see Section 10.9.6).

10.3.16 "Preferences" menu item

You can activate or deactivate the undo functions.

10.4.1 Principles

Generally, pore water pressures in slopes are defined using a pore water pressure line. The appropriate programs (including **GGU-STABILITY**) calculate the pore water pressure at the slice foot from the vertical distance between the slice foot and the pore water pressure line directly above it. This procedure implies the supposition that percolation through the slope is exclusively horizontal. This supposition is sufficiently exact for a large number of slopes. With complicated flow conditions (e.g. embankment with exterior seal), this supposition is no longer justified. Consideration of artesian conditions is also only helpful in a few special cases. A correct consideration of complicated flow conditions can therefore only be done if the pore water pressure is defined at every point of the slope. The **GGU-STABILITY** program allows such a definition via a pore water pressure mesh (triangular mesh), which must cover the whole area of the slope to be investigated. The potential h is defined at each triangle point

$$\mathbf{h = u/\gamma_w + y}$$

with

- u = pore water pressure, e.g. in kN/m²
- γ_w = unit weight of water
- y = elevation head

With the help of this triangular mesh the program can determine the pore water pressure u at every point, using linear interpolation within the triangular mesh. You can define this pore water pressure mesh completely with the mouse. This can, however, take some time for complicated flow conditions. If you are in possession of a groundwater program, you can read-in these data as an ASCII file (x, y and h). Reading-in a pore water pressure mesh is especially comfortable if you own the groundwater modeller **GGU-2D-SSFLOW**. Post-processing is then no longer required.

Example 2 from the Supplement to DIN 4084 (see file DIN4084_Bishop_pwp_02-e.BOE) contains, e.g., the following potential line mesh, which was calculated with **GGU-2D-SSFLOW**:

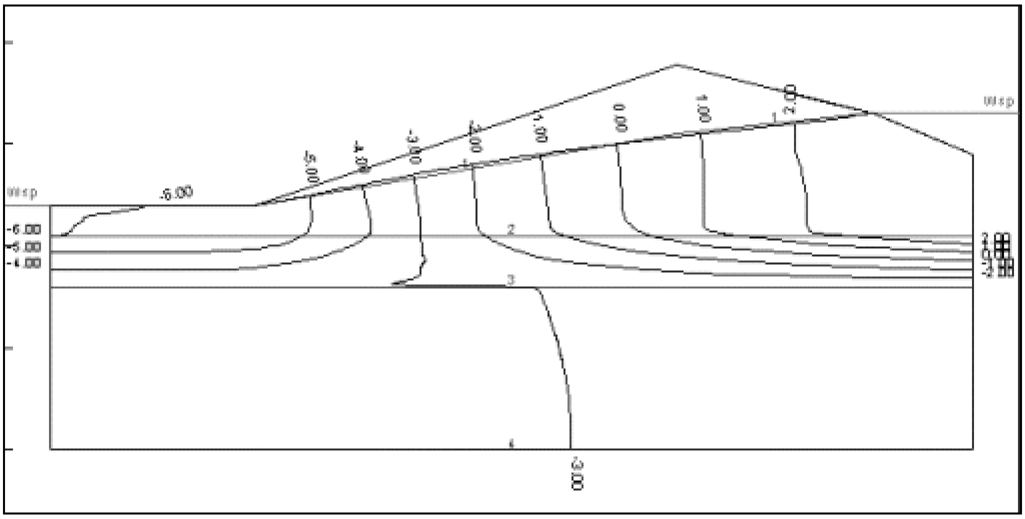


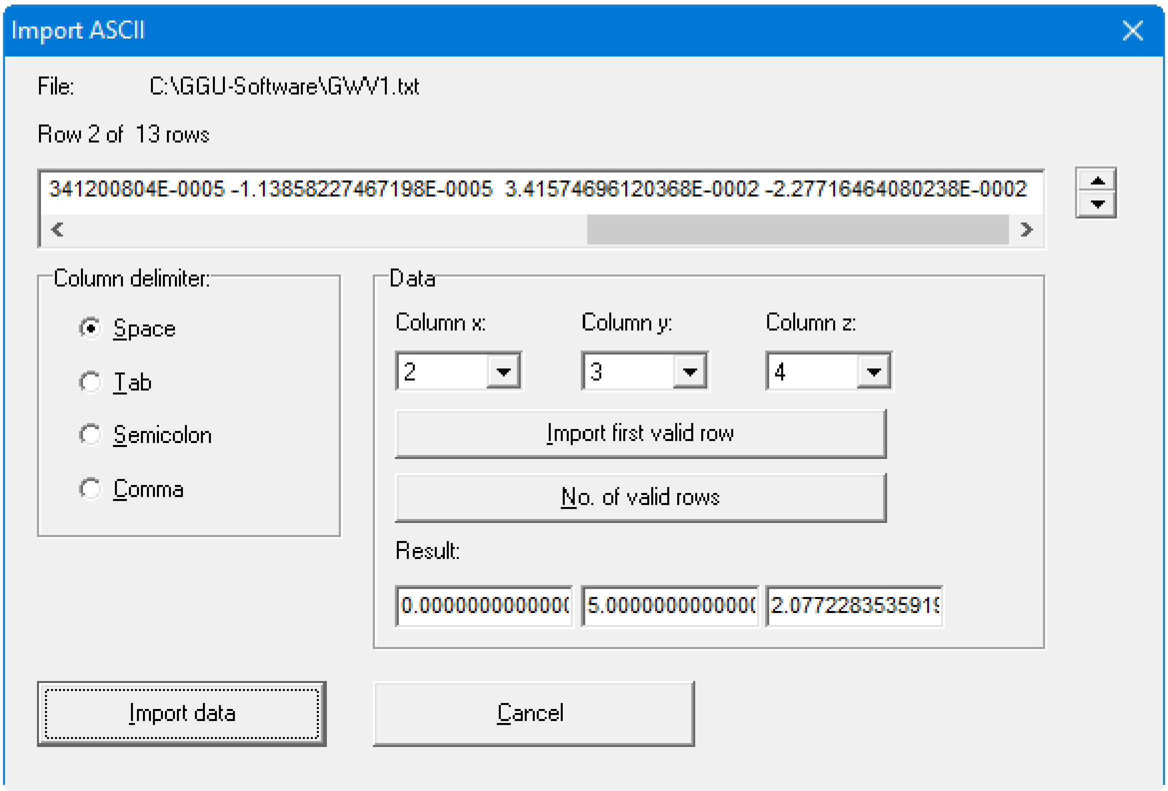
Figure 22 Potential lines

If the system does not include complicated flow conditions and defining the pore water pressure via a pore water pressure line is sufficient, you can forego study of the following sections on pwp mesh.

10.4.2 "ASCII file" menu item

You can read-in the coordinates (x, y) and the corresponding potentials (h) from an ASCII file created with a different program (e.g. **GGU-2D-SSFLOW**). Alternatively, you can save an existing pw mesh as an ASCII file by pressing the "**Save**" button in the dialog box.

After selecting the "**Read**" button and then a file, the following dialog box appears:



The current line of the ASCII file is shown at the top of the window. Using the arrows at the right you can move through the file. If all input is correct, the result for this line will appear in the box below the columns. Otherwise, "**Error**" will appear. You may then have to alter the column delimiter. If the file contains invalid as well as valid lines, these will be simply skipped when reading. Finally, select the "**Import data**" button. You can then further process the read-in coordinates.

The coordinates and the potential alone do not supply the program with the necessary information. After loading an ASCII file, you must additionally specify how the individual nodes are linked to one another. Further explanations can be found in the other menu items of the "**Pwp mesh**" menu.

ASCII files generated for use with **GGU-STABILITY** by the **GGU-2D-SSFLOW** or **GGU-3D-SSFLOW** programs can be imported directly. The node relationships are also imported with these files. Further processing within this menu are then no longer necessary.

10.4.3 "Points to mesh" menu item

Using this menu item, if you have already entered a slope system, you can create a pwp mesh from this system, which must then generally be further refined.

10.4.4 "Contours" menu item

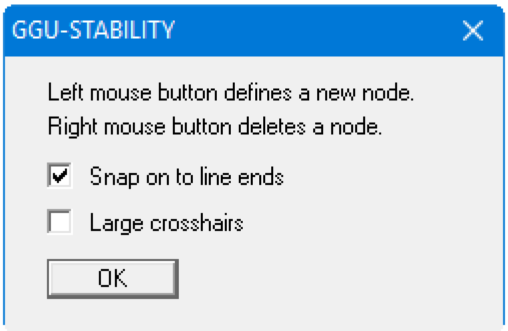
If you have read-in a pwp mesh or have defined a pwp mesh by hand, you can have lines of equal potential (contours) displayed using this menu item. It is also possible to have normal contours displayed or colour-filled contours. The dialog boxes which appear are described more closely in Section 10.7.8 ("**Safety/Utilisation factors (slip circles)/Contours**" menu item) and in Section 10.7.9 ("**Safety/Utilisation factors (slip circles)/Coloured**" menu item).

10.4.5 "Determine pwp" menu item

For better input control you can determine the pore water pressure at any point within the system with a mouse click using this menu item.

10.4.6 "Define nodes" menu item

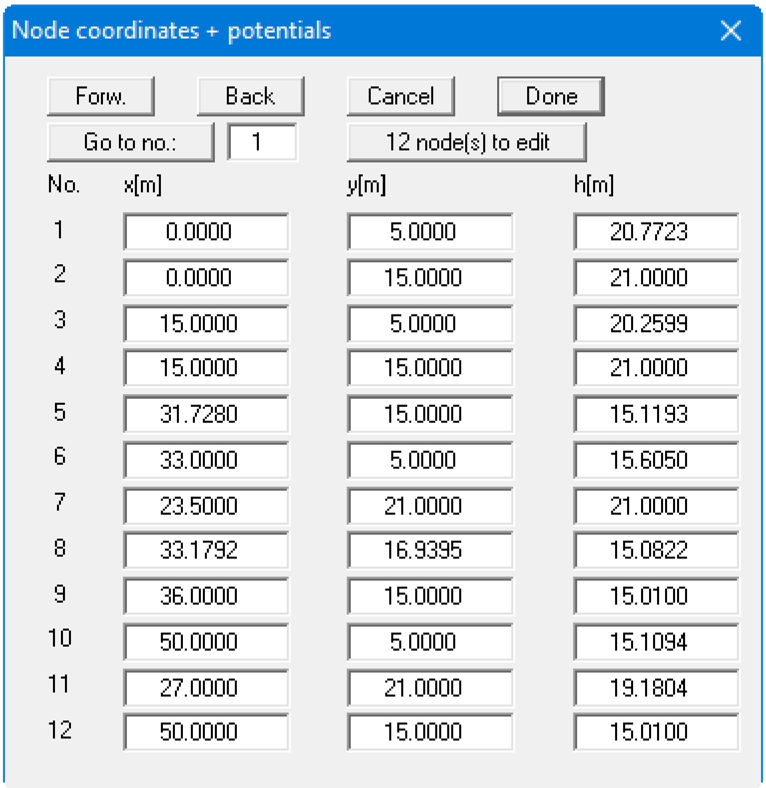
With this menu item you specify the position of triangle nodes using the mouse. Use of the mouse is in accordance with the description in the box.



After clicking on a node, you must enter the corresponding potential.

10.4.7 "Change" menu item

After clicking on this menu item the following dialog box appears:



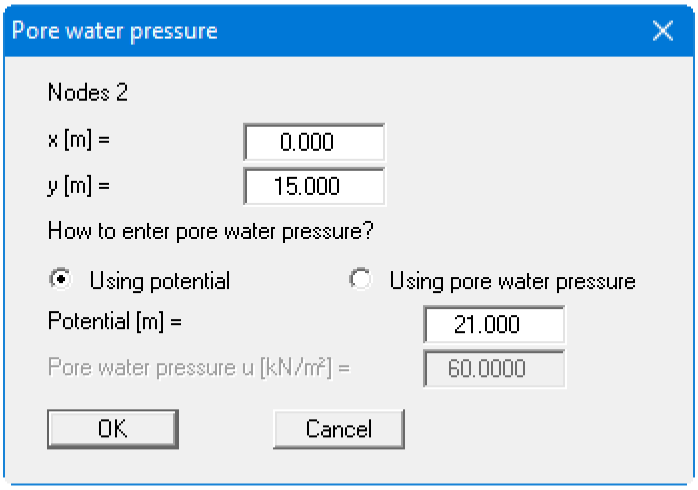
Within this box it is possible to change the x and y coordinates and the potential via the keyboard. The use of the buttons corresponds to the explanations in Section 8.2.2 ("**Worked example 2: Data input via editor/Step 2: Enter System parameters (Ex. 2)/Surface points**").

10.4.8 "Move" menu item

Using this menu item it is possible to displace current triangle nodes with the mouse and thus alter the x and y coordinates. Use of the mouse is in accordance with the description in the box.

10.4.9 "Edit" menu item

This menu item allows editing of coordinates and potentials of the individual triangle nodes. Use is in accordance with the description in the box. Double-clicking with the mouse on node number 12 of an example pwp mesh would call up the following box:



Here you can edit the x and y coordinates and the potential h.

10.4.10 "Manual meshing" menu item

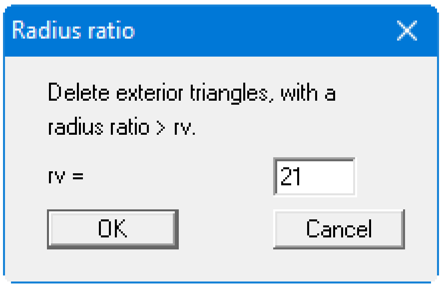
You can create the pore water pressure mesh manually. Triangulation (amalgamation of 3 mesh nodes to a triangle) of the pore water mesh nodes by hand must be carried out as instructed in the dialog box.

10.4.11 "Automatic meshing" menu item

This menu item runs a program controlled mesh generation routine (Delauney triangulation). You have the possibility of deleting or supplementing a current triangular mesh. Only in exceptional cases should you select "**Supplement**", as Delauney triangulation follows certain laws which may not allow sensible complementing of a current partial grid which has been entered by hand.

10.4.12 "Round off" menu item

With Delauney triangulation a triangular mesh is created which envelopes all nodes. In this way triangular elements can occur which, in the boundary areas, can be very acutely angled. Using this menu item you can remove such triangles from the mesh.



The radius ratio describes the relationship between exterior radius and interior radius of a triangle. For an equilateral triangle this ratio equals 2.0 (optimum). Before you arrive at the dialog box above, the maximum and the average radius ratio of the mesh appear. In the example above, all exterior triangles with a radius ratio larger than 21 will be removed.

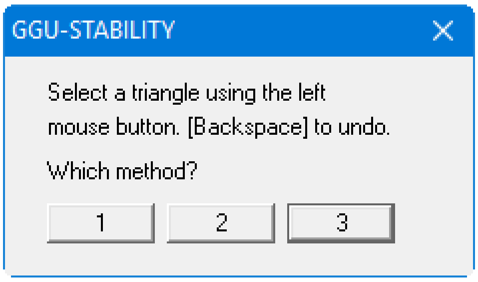
In order to avoid ***interpolation holes*** in the triangle system only triangles at the boundaries are deleted.

10.4.13 "Delete" menu item

With this menu item you can delete selected system triangles. You must click on four points in an anti-clockwise direction. All triangles with a centroid within the quadrilateral will be deleted. Alternatively, you have the possibility of completely deleting a pwp mesh.

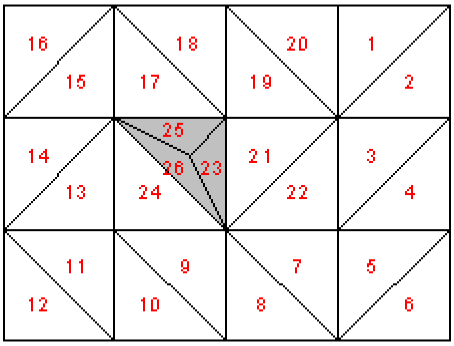
10.4.14 "Refine individually" menu item

Three menu items are available for mesh refinement. After selecting this first menu item the following dialog box opens:

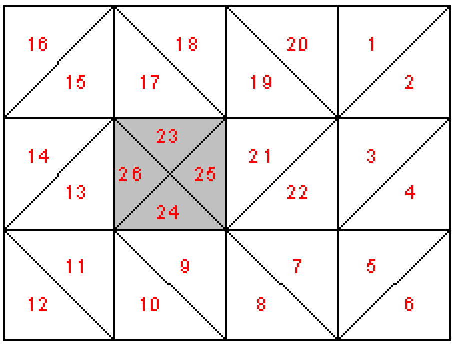


For refinement of the pwp mesh, three differing refinement procedures are available:

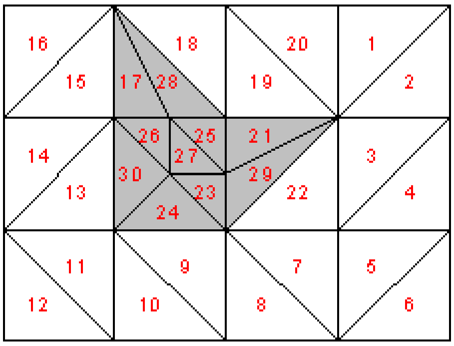
- **Method 1**:
  In the centroid of the selected triangle an additional node is created.



- **Method 2**:
  The selected triangular element and the neighbouring element are halved.



- **Method 3**:
  A new triangular element will be inserted at the median of the clicked triangular element. The neighbouring triangular elements will be halved.



When refining the mesh, new nodes are assigned potential, which result from the values of the neighbouring nodes.

10.4.15 "Section" menu item

A region of the triangular mesh can be selected for refinement by clicking the four points of a polygon in an anti-clockwise direction, thus enclosing the region to be refined. The refinement procedures described in "**Pwp mesh/Refine individually**" are also available for mesh refinement in section (see Section 10.4.14).

10.4.16 "All" menu item

All triangular elements can be refined using the refinement procedures described in "**Pwp mesh/Refine individually**" (see Section 10.4.14).

10.5.1 "Define in quadrilateral" menu item

You can define the centre-points of the slip circles to be calculated via a quadrilateral array on the screen.

The system is displayed on the screen. With the mouse pointer you can specify the corner points of a centre-point array. To do this, you must give the four corner points of a quadrilateral. The first corner point is set with the left mouse button. Then, position the mouse at the second point and press the left mouse button a second time. If the first corner point is not according to your wishes, you can set it again after pressing the right mouse button (press the left mouse button). After specifying the corners of the quadrilateral, you must enter the number of subdivisions (in x and y direction).

By repeating the above procedure you can define several centre-points arrays. You can also carry out this action after centre-points have been calculated.

10.5.2 "In rectangle" menu item

The proceeding corresponds to the definition of centre-points via a quadrilateral array. You must give the two diametrically opposed corners of a rectangle. The first corner point is set with the left mouse button. Then, position the mouse at the second point and press the left mouse button a second time. If the first corner point is not according to your wishes, you can set it again after pressing the right mouse button (press the left mouse button). After specifying the corners of the rectangle, you must enter the number of subdivisions (in x and y direction).

By repeating the above procedure you can define several centre-points arrays. You can also carry out this action after centre-points have been calculated.

10.5.3 "Individually (graphically)" menu item

Individual centre-points can be defined by clicking with the left mouse button.

10.5.4 "Individually (editor)" menu item

You can define individual centre-points entering the x/y coordinates in an editor box. This menu item is useful if, e.g., you would like to check a ***third party*** bank for correct static using specific slip circle centre-points.

10.5.5 "Refine" menu item

The program carries out a triangulation with the current centre-points. An additional centre-point is placed at the centre of the thus determined triangles and the current array is refined.

10.5.6 "Info" menu item

You will see information on the current centre-point array in a dialog box.

10.5.7 "Define search grid" menu item

A description of this menu item can be found in Section 8.4.2 under "**Worked example 2: Data input via editor/Step 4: Define slip circles (Ex. 2)/Define search grid**". Further examples below will clarify procedure for defining the search grid.

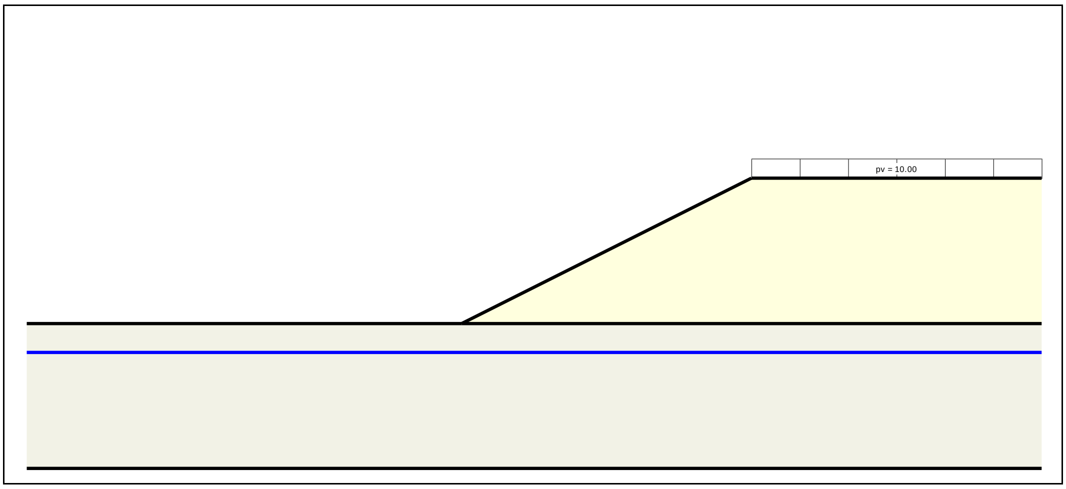


Figure 23 Example system for defining the search grid

Select a centre-point array by going to the menu item "**Centre-points/In rectangle**" (see Section 10.5.2).

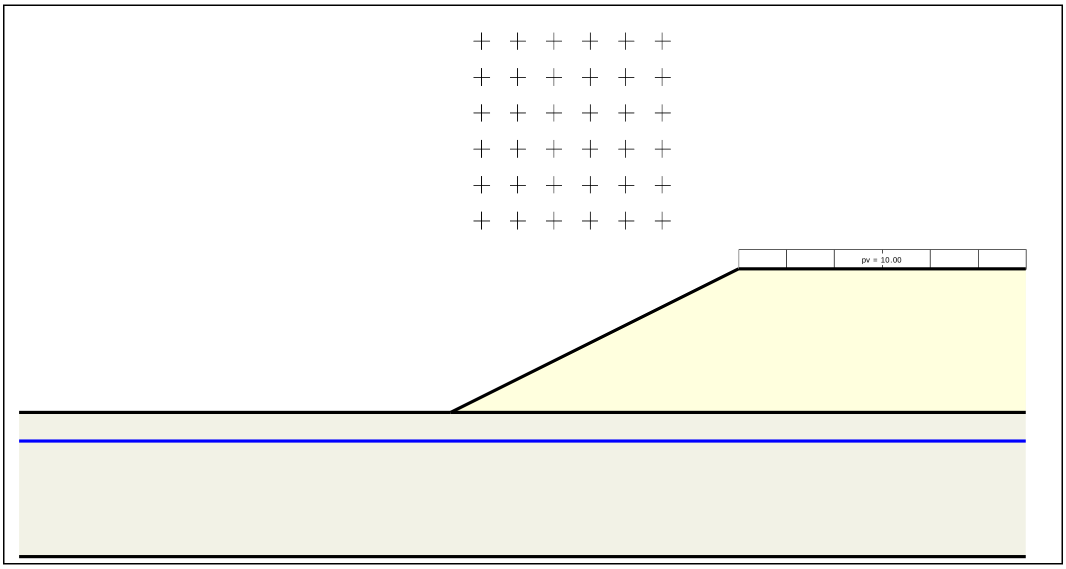
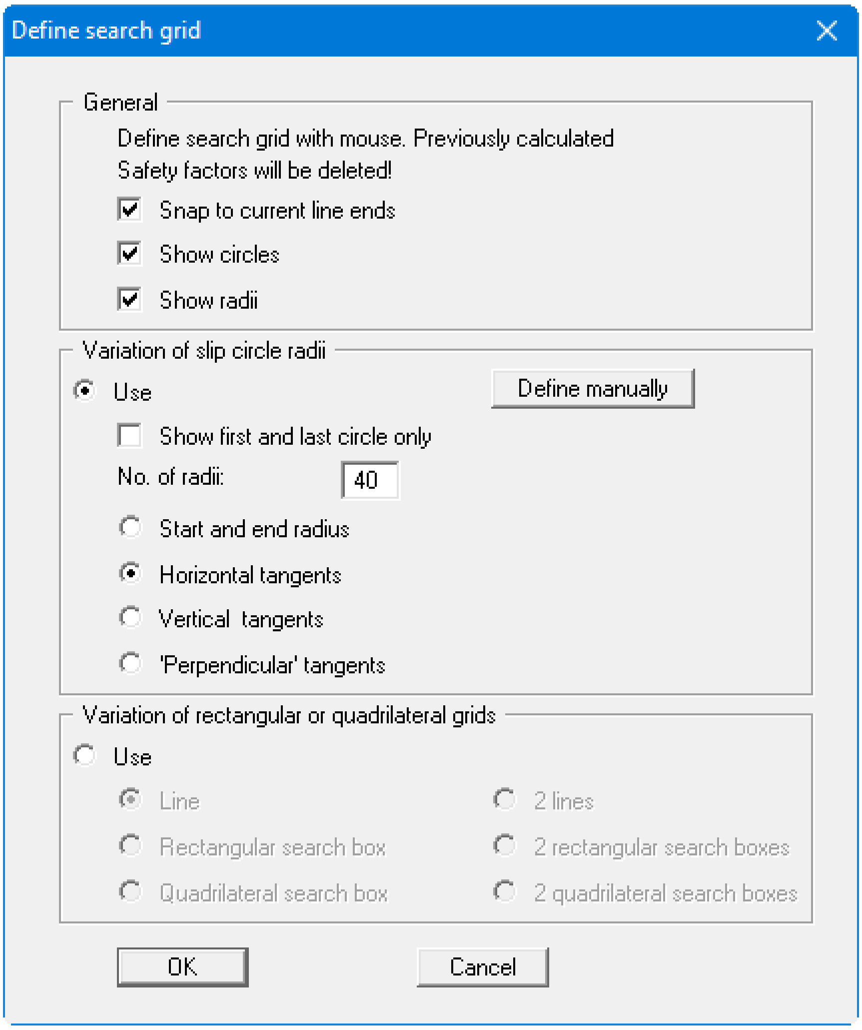


Figure 24 Centre-point array 1

Using this menu item to define the search grid the following dialog box opens. If there is any uncertainty in terms of the search, always select the "**Horizontal tangents**" search grid and a larger number of radii (40 or more), depending on your safety requirements.



After leaving the dialog box by pressing "**OK**", define "**Top of slope**" as the uppermost point by clicking with the mouse, and "**Bottom of system**" as the lowest point.

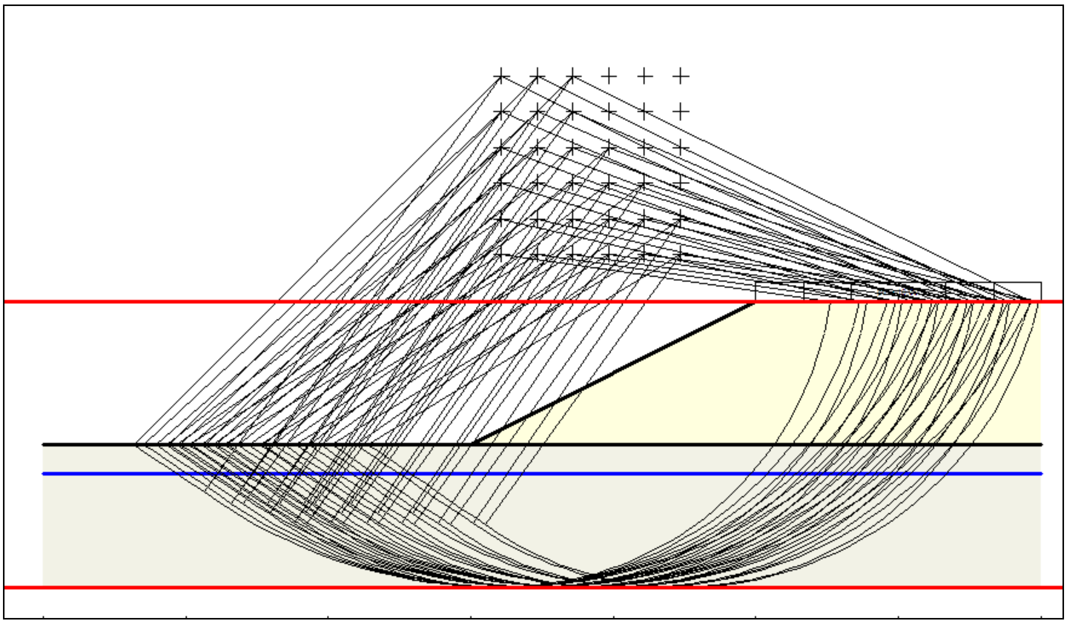


Figure 25 Selected search grid

Then start the analysis and the following results are acquired:

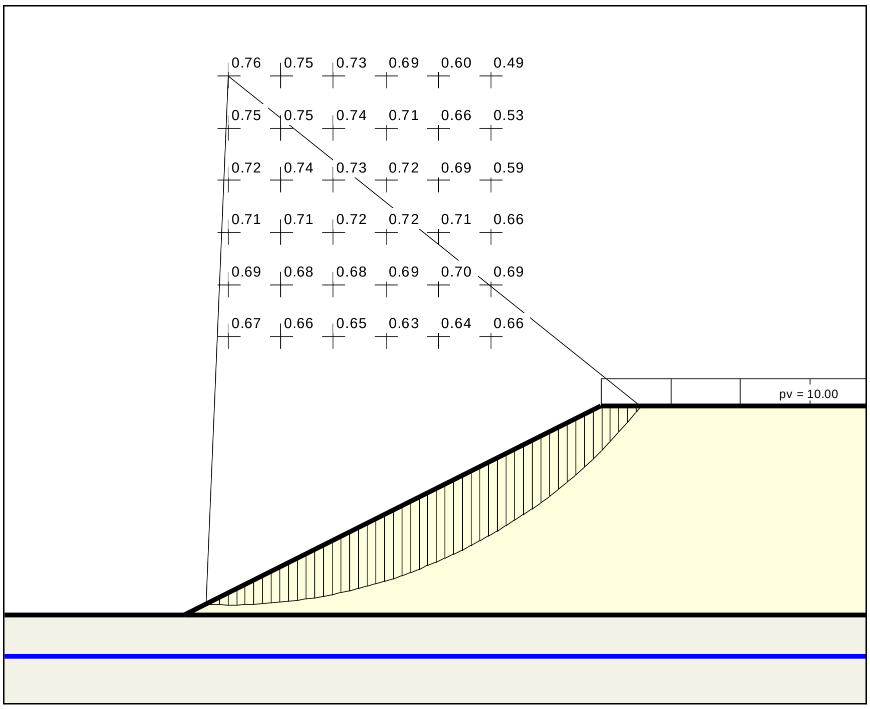


Figure 26 Results for centre-point array 1

The slip circle with the highest utilisation factor is located at the top left edge of the selected array. The array must be expanded towards the upper left. Alternatively, a new, appropriately displaced array can be selected.

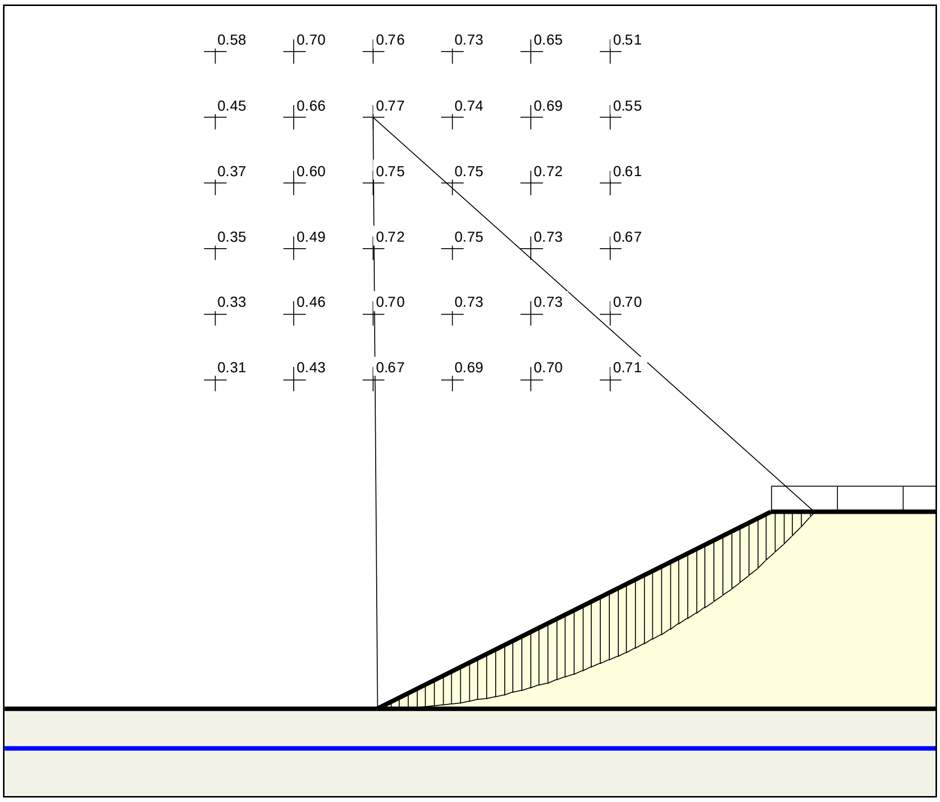


Figure 27 Results for centre-point array 2

The slip circle with the highest utilisation factor now lies within the selected array. The stability of the slope is verified. Further arrays may need to be investigated for slopes with numerous layers. In case of uncertainty, the region above the slope face can be ***plastered*** with centre-points.

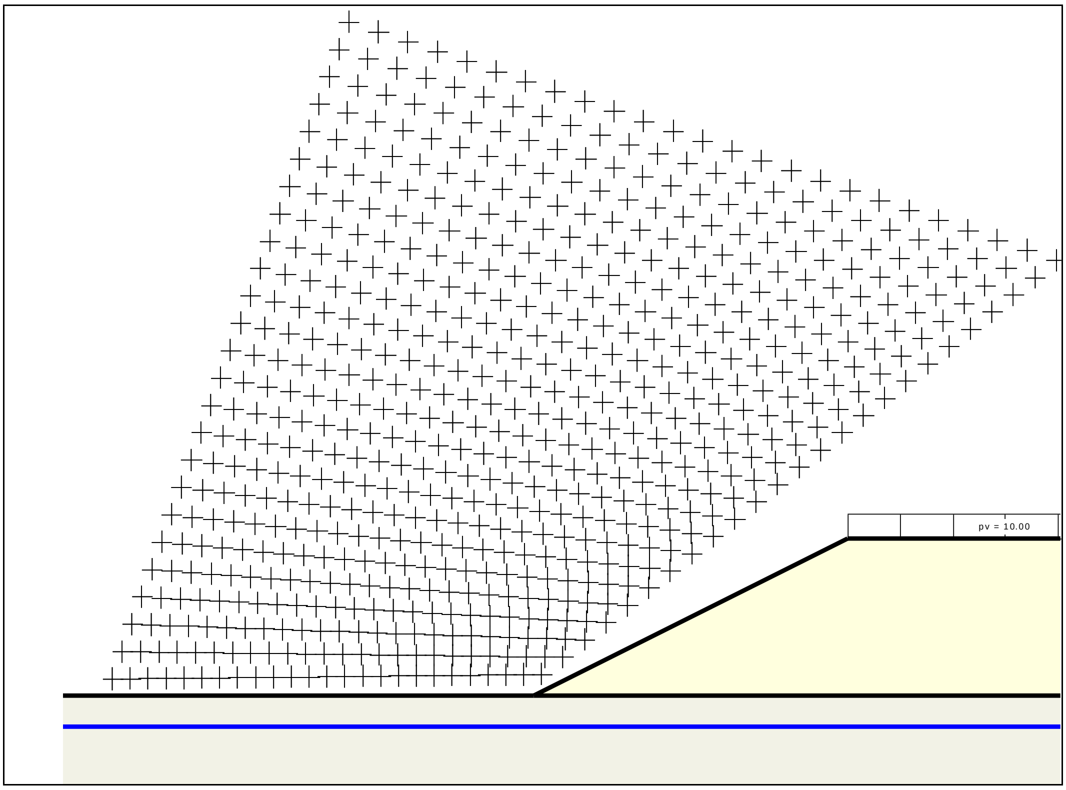
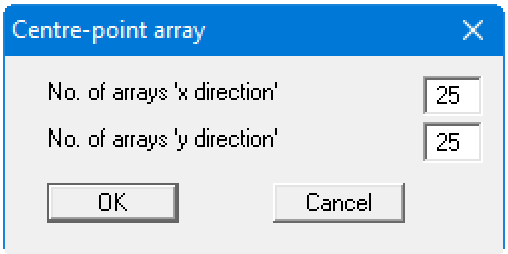


Figure 28 Centre-point array 3

Centre-point array 3 was generated using the menu item "**Centre-points/Define in quadrilateral**". The following array subdivisions were selected:



The following result is acquired using the selected centre-point array 3:

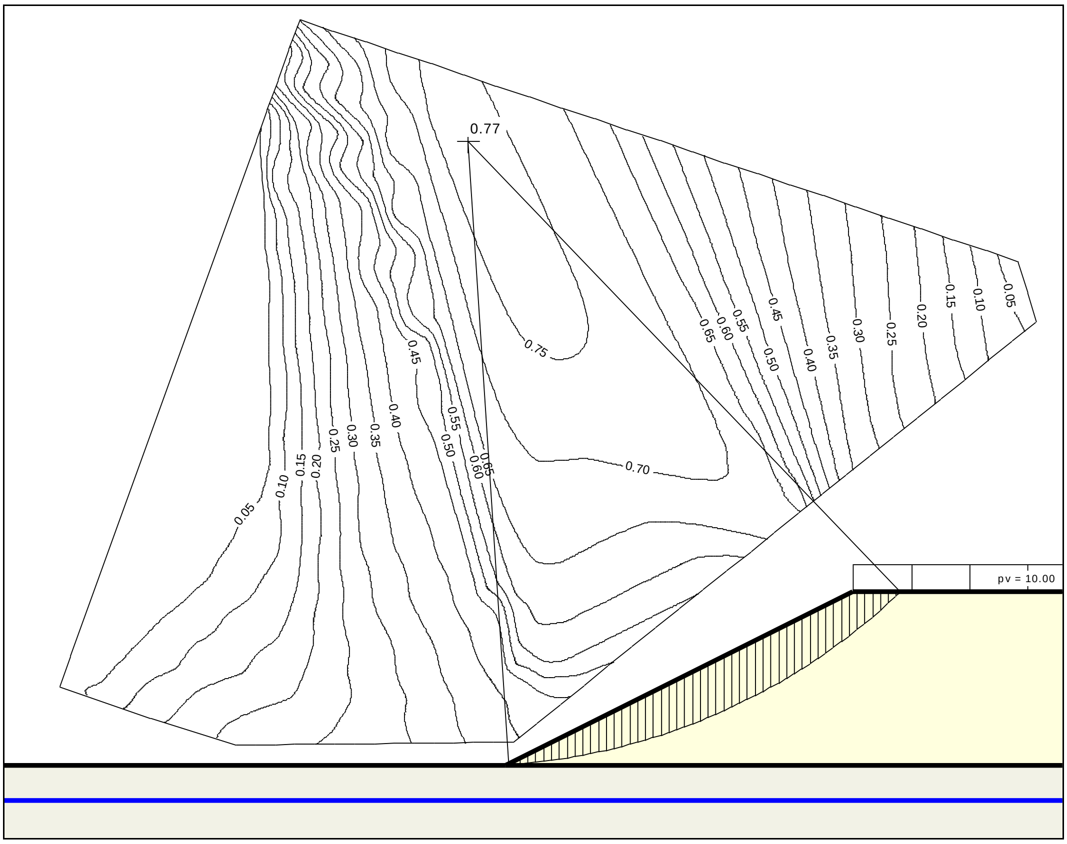


Figure 29 Results for centre-point array 3

The result confirms the previous investigations. Defining the search grid using "**Horizontal tangents**" works almost every time. However, computation may be optimised by using a different definition.

Defining the search grid by selecting the "**Start and end radius**" is useful for investigating the global stability of a cantilever wall, for example. This allows slip circles passing through the cantilever wall to be excluded from the analysis.

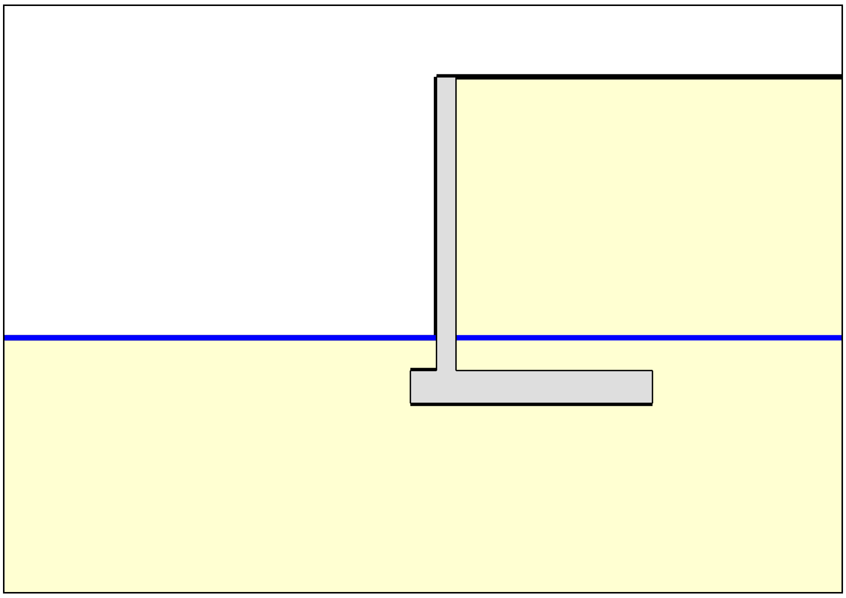


Figure 30 Cantilever Wall

A rational search grid is shown below:

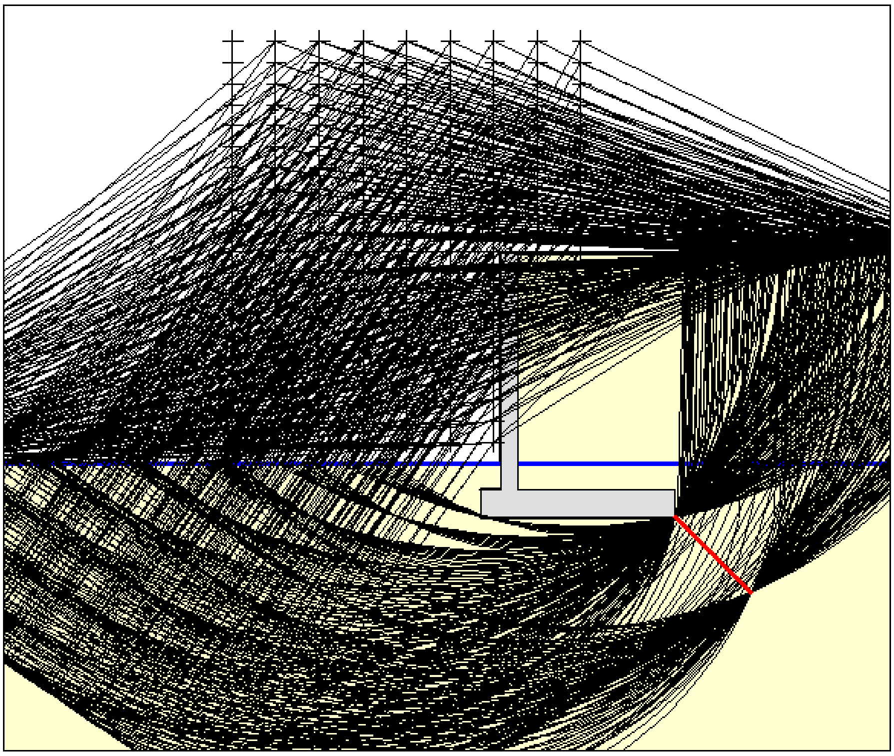


Figure 31 Search grid for cantilever wall

The following analysis result is acquired using this search grid.

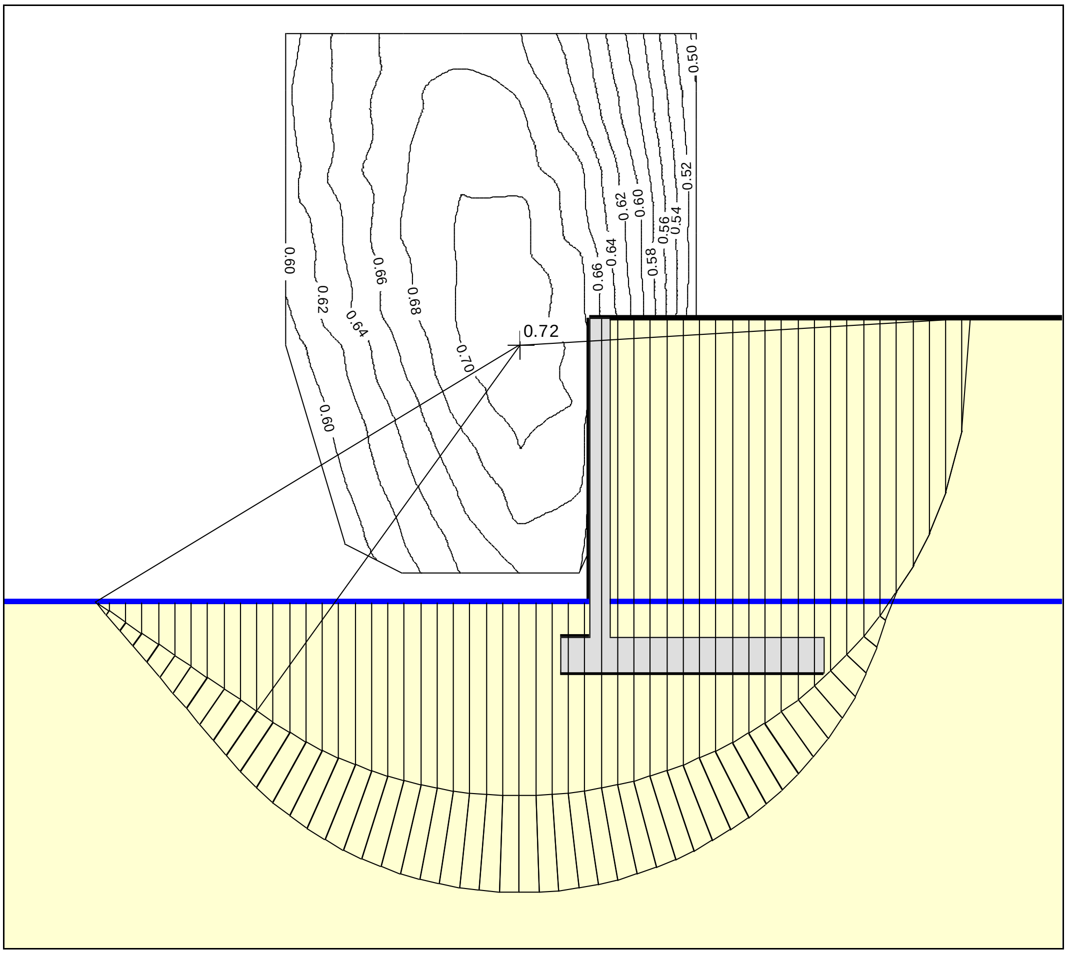


Figure 32 Result for cantilever wall

10.5.8 "Semi-automatic" menu item

You can enter the centre-points of the slip circles to be analysed via the keyboard by entering the start and end coordinates, and the maximum radius.

10.5.9 "Delete individually" menu item

You can delete individual centre-points by clicking on them with the left mouse button.

10.5.10 "Delete all" menu item

After a security request all specified centre-points and any calculated safety/utilisation factors will be deleted.

10.5.11 "Display" menu item

All current centre-points can be displayed. If you have already defined a search grid, you can also display the boundary search radii.

10.6 Slip body menu (for polygonal slip surfaces only)

10.6.1 "Info" menu item

You will see information on input and editing of slip bodies.

10.6.2 "Define new" menu item

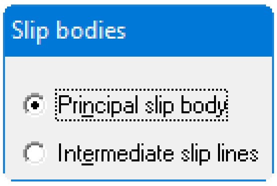
You can define the positions of the individual slip body polygon points with the left mouse button. After selecting this menu item you can activate a dialog box with [**Shift**] and the [**F4**] function key, which will inform you as to the position of the mouse pointer (x and y coordinates; soil type. [**Shift**] and the [**F5**] function key open a dialog box which offers information, amongst other things, on the inclination of the nearest slip line. If the shape of the slip body is according to your wishes, you must press the [**Return**] key to accept. You can then, if so wished, define further slip bodies.

You can also carry out this action after slip bodies have been calculated.

If you have mistakenly carried out a wrong mouse click, which has ***ruined*** your slip body, simply press the [**Backspace**] key and you will return to the old condition.

If you have selected "**Vertical slice method**" as calculation procedure, input is completely analogous. You may even switch to the "**Vertical slice method**" after having specified polygonal slip bodies after "**Janbu**" in the menu item "**Editor 1/Analysis options**", and carry out the calculations immediately after.

If you have selected "**General wedge method**" as calculation procedure, input is completely analogous, except that then vertical intermediate slip surfaces will be assumed. Therefore, a dialog box appears at the upper left of the program window:

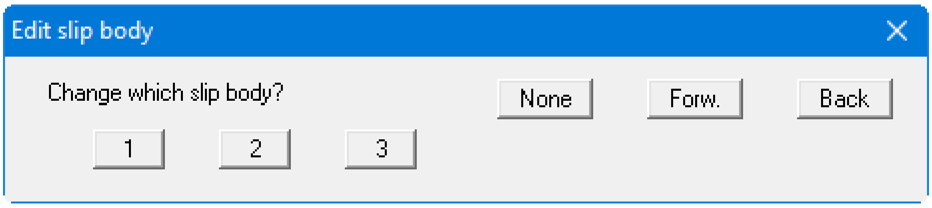


Activate the "**Principal slip body**" switch if you would like to define the main slip surfaces of the failure body. You then proceed in complete analogy to the above described actions. For each new main slip body point, the program automatically assign a vertical intermediate slip surface. Then activate the "**Intermediate slip lines**" switch in order to define the position of the intermediate slip line with the mouse.

You may even, after having specified polygonal slip bodies after "**General wedge method**", switch to the "**Vertical slice method**" or "**Janbu**" in the menu item "**Editor 1/Analysis options**", and carry out the calculations immediately after. The defined intermediate slip lines will then simply not be considered.

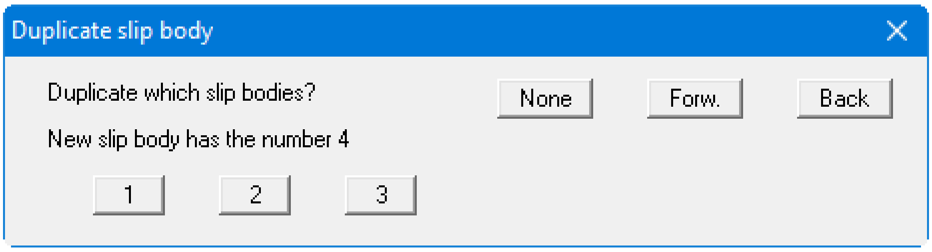
10.6.3 "Edit old" menu item

You can edit an existing slip body.



10.6.4 "Duplicate" menu item

You can duplicate an existing slip body, in order to use it as the basis for editing.



Click on the button of the slip body to be duplicated. The slip body is then represented in the system and can be modified to suit your needs by moving the polygon points using the mouse. Then press the [**Return**] key to accept the new slip body and confirm the "**Use slip body?**" query with "**Yes**".

10.6.5 "Edit" menu item

You can define new slip bodies or edit existing ones by entering the coordinates. Accordingly, you should click on either the "**New**" or the "**Old**" button in the box.

If you selected "**New**", you must now specify the "**No. of polygon points**". In the following coordinate input box the number of polygon points can be edited again by pressing the "**x point(s) to edit**" button. If "**Old**" was selected, open the coordinates input box by clicking the button with the appropriate slip body number.

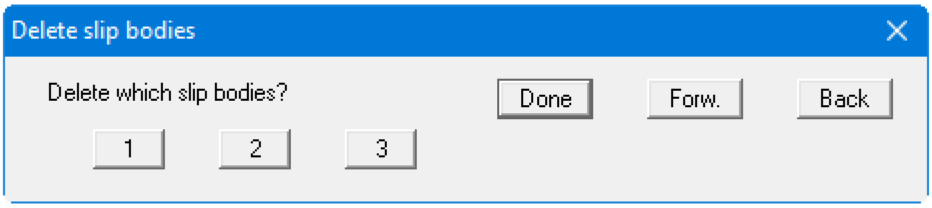
If you are calculating with "**General wedge method**" you must additionally enter the intersection of the intermediate slip surface with the surface as "**xzw**" and "**yzw**". You need not hit the intersection exactly, as the correct intersection is calculated by the program automatically before calculating the safety of the slip body.

10.6.6 "Display" menu item

You can select the slip bodies to be displayed.

10.6.7 "Delete individually" menu item

Previously specified individual slip bodies can be deleted after a security request.

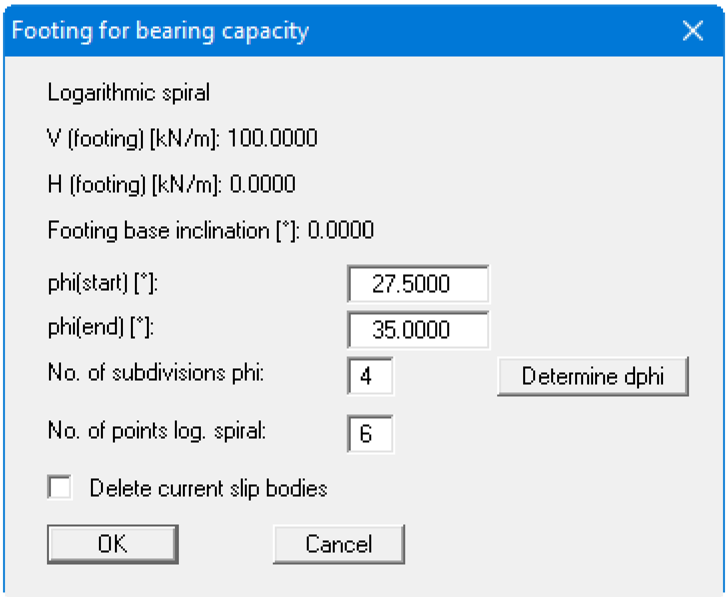


10.6.8 "Delete all" menu item

After a security request all specified slip bodies and any calculated safety factors will be deleted.

10.6.9 "Logarithmic spiral" menu item

If you activated the "**Bearing capacity**" option button in the "**Editor 1/Analysis options**" menu item, you can generate polygonal slip surfaces for the strip footing which, in accordance with DIN 4017, consist of a logarithmic spiral in the central area and straight lines at the ends.

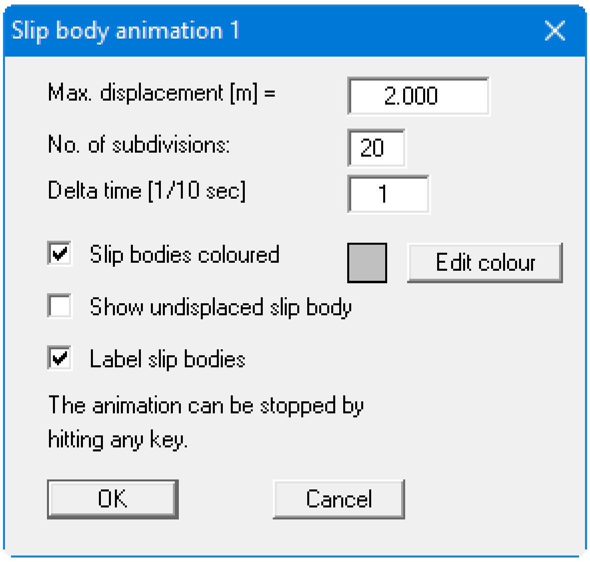


You must first define two friction angles and then a number of subdivisions. The "**Determine dphi**" buttons allows determination of the increase in friction angle resulting from the three values.

The "**No. of points log. spiral**" specifies the number of points with which the logarithmic spiral is generated. After confirming with "**OK**" the polygonal slip surfaces are generated in accordance with your preferences. You can then analyse these polygonal slip surfaces.

10.6.10 "Animation" menu item (General wedge method and Vertical slice method only)

If you investigate with the above-named procedures, you can have the failure mechanism for a selected slip body displayed as a ***cartoon***.



First select the "**Max. displacement**". With "**No. of subdivisions**" you can specify the resolution with which the film is to be shown. All further preferences are self-explanatory. After pressing "**OK**" the ***cartoon*** will run.

10.6.11 "Move slip body" menu item (General wedge method and Vertical slice method only)

As impressive as the animation of the slip body is, there is no possibility of getting this across to the client within the framework of a geotechnical report. In order to offer some assistance, this menu item was introduced in the program. You can thus display and print individual ***frames***. Usage is similar to that described in Section "**Slip body/Animation**".

10.7.1 General notes

This menu appears as "**Safety factors**" if the setting in "**Editor 1/Analysis options**" is "**DIN 4084 1981**" or as "**Utilisation factors**" for "**DIN 4084:1996**", "**DIN 4084:2009**" and "**EC 7**" (see also the "**Theoretical principles**" section).

The following menu items are only active with a method employing slip circles. The menu items for methods employing polygonal slip surfaces can be found in Section 10.8.

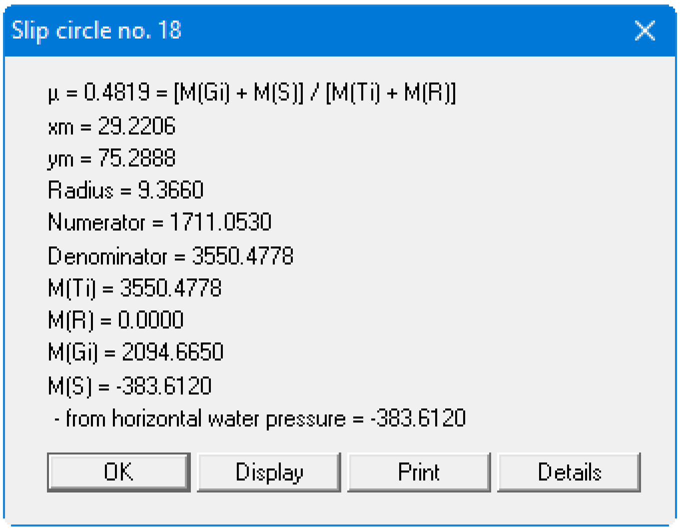
10.7.2 "Analyse" menu item

Once you have defined a centre-point array and a search grid (see the description for the "**Centre-points**" menu), computation of the safety/utilisation factors can begin. You can also initiate the analysis using the [**F5**] function key.

An explanation of the dialog box can be found in the "**Worked example 2: Data input via editor/Step 5: Analyse slope with circular slip surfaces**" in Section 8.5.

10.7.3 "Display/details" menu item

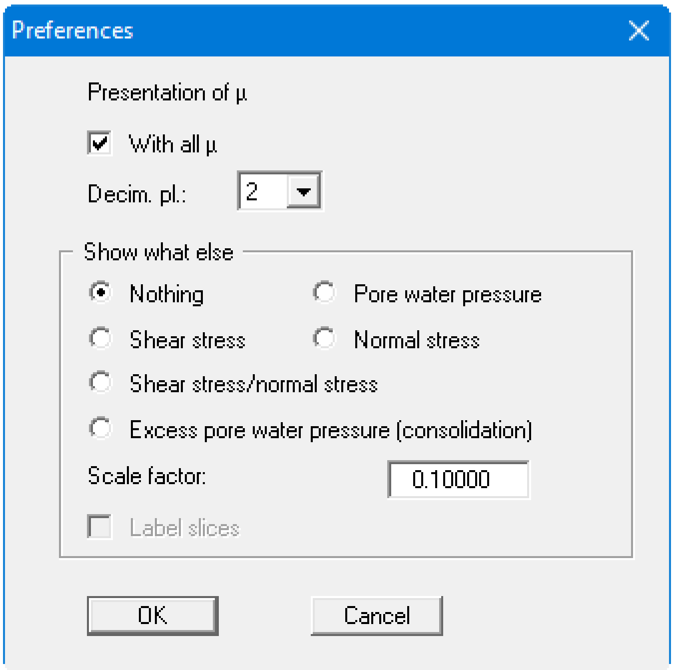
You click on the centre-point for which you would like to see all important analysis results. The designations used correspond to those used in the DIN 4084.



If you select the "**Details**" button, you will see a precise output table of the selected slip circle with all analysis results (see Section 10.1.10.3, "**File/Print output table**" button "**Output as ASCII**").

10.7.4 "Preferences" menu item

In a dialog box you can specify whether, besides the most unfavourable and a specific slip circle, the other slip circles and safety/utilisation factors are to be displayed, and whether "**Pore water pressure**", the "**Shear stress**" or the "**Normal stress**" on the slip surface is to be displayed.



With the "**Scale factor**" you specify the graphical presentation length for the shear stress etc. A value of 0.02 means e.g., that a shear stress of 120 kN/m² will be presented with a length of $120 \cdot 0.02 = 2.4$ m (!).

10.7.5 "Show most unfavourable slip circle" menu item

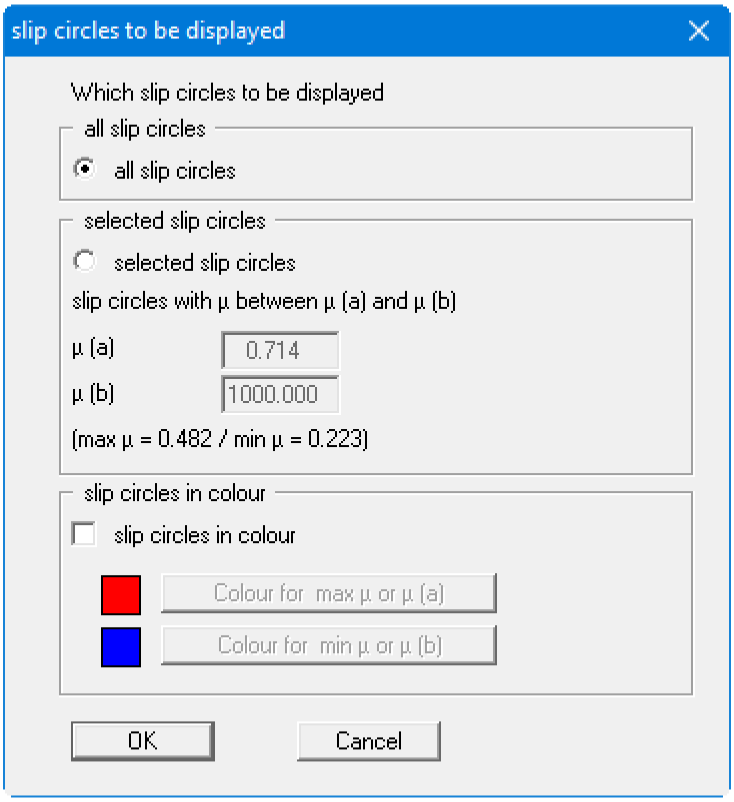
After computation of the safety/utilisation factors, they can be graphically displayed together with the most unfavourable slip circle.

10.7.6 "Specific" menu item

After computation of the safety/utilisation factors, they can be graphically displayed together with a specific slip circle. Just click the desired centre-point with the left mouse button.

10.7.7 "All" menu item

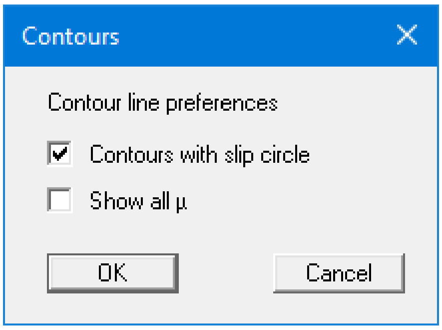
After computation of the safety/utilisation factors, they can be graphically displayed together with the all slip circles. You may also apply limitations by entering a boundary value in the following dialog box:



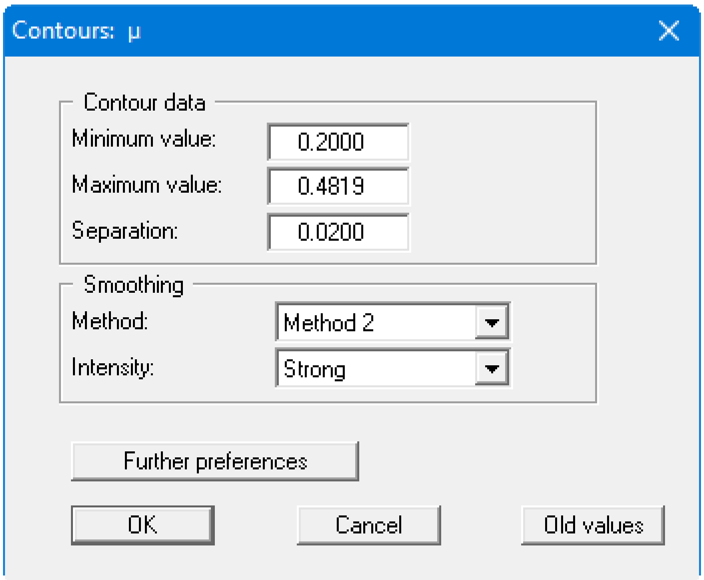
The slip circle lines may also be displayed in colour.

10.7.8 "Contours" menu item

If safety/utilisation factors have been calculated, you can have a contour plan of equal safety/utilisation factors displayed above the slope. For this, the program triangulates the calculated centre-points. The thus determined triangular mesh serves as an interpolation mesh for the presentation. After selecting this menu item, the following dialog box appears:



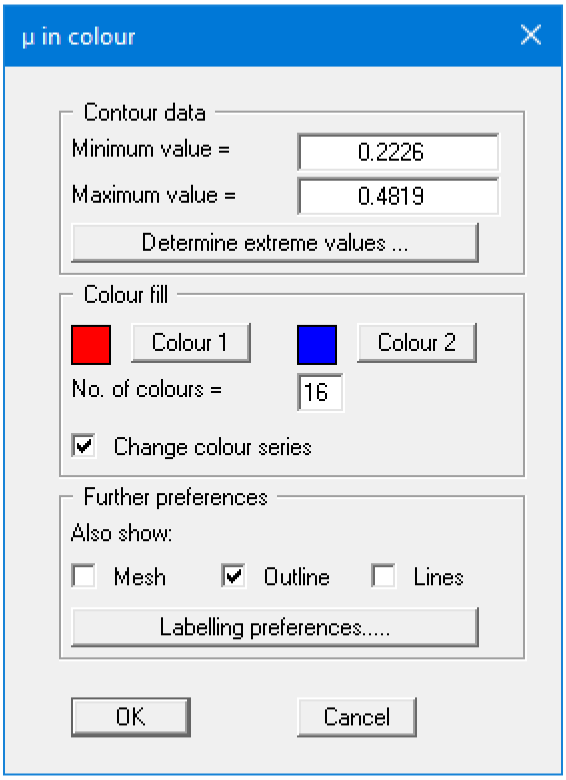
Here you can determine what is to be displayed additional to the contours. For a better overview, you should deactivate the lower check box. Only the most unfavourable utilisation factor is then entered. You can then specify the type of presentation of the contours.



At the top the minimum (rounded down) and maximum safety/utilisation factors are shown. Below this is a contour separation calculated by the program. All three values can be edited to suit your wishes. Further down still, you can select from three smoothing out procedures, whereby method 2 generates the ***roundest*** contour lines but, on the other hand, cannot exactly model the safety/utilisation factor course for abrupt safety/utilisation factor jumps. With the "**Further preferences**" button you can set further preferences for the type of presentation. If you have previously selected this menu item, you can restore the old values with the button "**Old values**". If you leave the box via "**OK**", the contours will be displayed.

10.7.9 "Coloured" menu item

In analogy to the previous menu item, colour-filled contour lines can also be created. The following dialog box appears:



- "**Contour data**" group box
  The "**Determine extreme values**" button computes the minimum and maximum safety/utilisation factors and enters them into the appropriate input boxes. However, you need not retain these values, but can also define your own. No smoothing of the contour lines can be performed here.
- "**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. Here you can also adjust the font size of the colour bar.
- "**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 10.10.7).

10.8 Safety/Utilisation factors menu (for polygonal slip surfaces only)

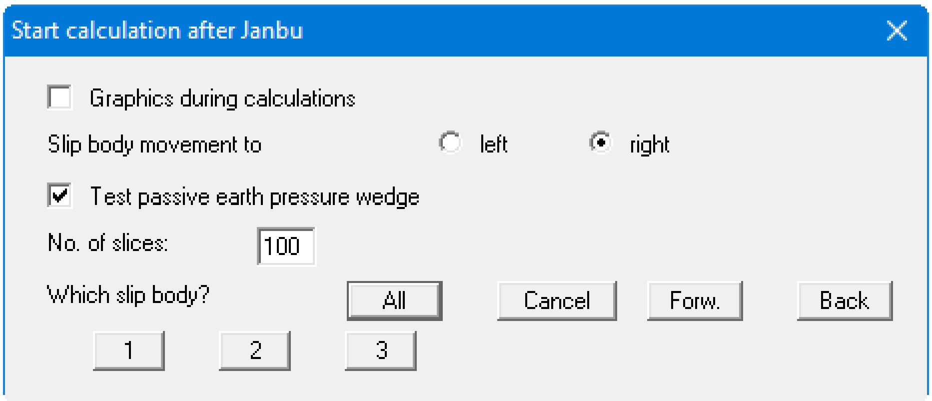
10.8.1 General notes

This menu appears as "**Safety factors**" if the setting in "**Editor 1/Analysis options**" is "**DIN 4084 1981**" or as "**Utilisation factors**" for "**DIN 4084:1996**", "**DIN 4084:2009**" and "**EC 7**" (see also the "**Theoretical principles**" section).

The following menu items are only active with a method employing polygonal slip surfaces. The menu items for methods employing slip circles can be found in Section 10.7.

10.8.2 "Analyse slip bodies" menu item

Once you have specified the slip bodies, computation of safety/utilisation factors can begin.



Adjust the number of slices to suit your wishes. If the "**Test passive earth pressure wedge**" check box is activated, the program will examine whether gradients greater than 45° - φ/2 occur in the passive earth pressure region of the respective slip body. These slip bodies will not be investigated. Slip body movement should to the "**left**". Begin calculation of all slip bodies using the "**All**" button.

If you chose to view the graphical representation during computation you will now see the course of the calculations, together with safety factor data, on the screen. A large number of slip bodies can greatly extend computation time. Deactivate the "**Graphics during calculations**" check box.

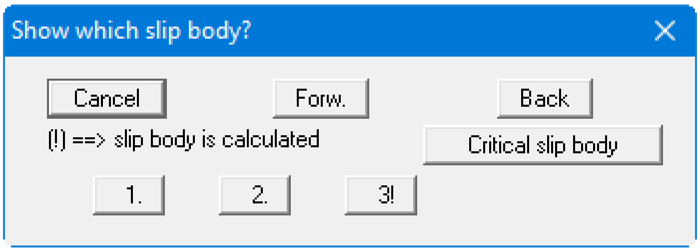
Once calculations are complete you will be shown some statistics. Beside stability, the maximum geosynthetics forces resulting from slip body variation are determined.

After confirming with "**OK**", the slip body with the lowest FOS is displayed.

The initial dialog box is slightly different for calculations using the "**General wedge method**" or the "**Vertical slice method**". Although these calculation methods are not slice methods, the program computes the weights, pore water pressures, etc. of the individual slip bodies internally by dividing the system into vertical slices. The "**Max. slice width [m]**" can be specified in the dialog box. However, a minimum of 10 slices is used for each slip body, regardless of the value specified. A maximum of 500 slices per slip body will be considered. The default value of "**0.2**" normally provides sufficiently precise results. If you deactivate the "**Shear strength on interm. slip lines**" switch, the shear strengths on the intermediate slip lines will not be taken into consideration.

10.8.3 "Show individual slip bodies" menu item

After computation of the safety/utilisation factors, they can be graphically displayed for individual slip bodies.



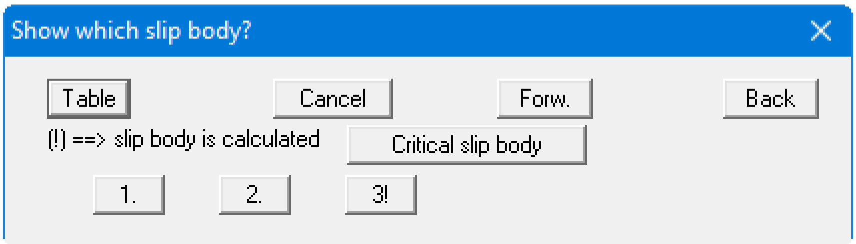
You can search for and display the "**Critical slip body**" using the button provided.

10.8.4 "All" menu item

After computation of the safety/utilisation factors, they can be graphically displayed together with the all slip bodies. You may also apply limitations by entering a boundary value. The slip body lines may also be displayed in colour.

10.8.5 "Display results" menu item

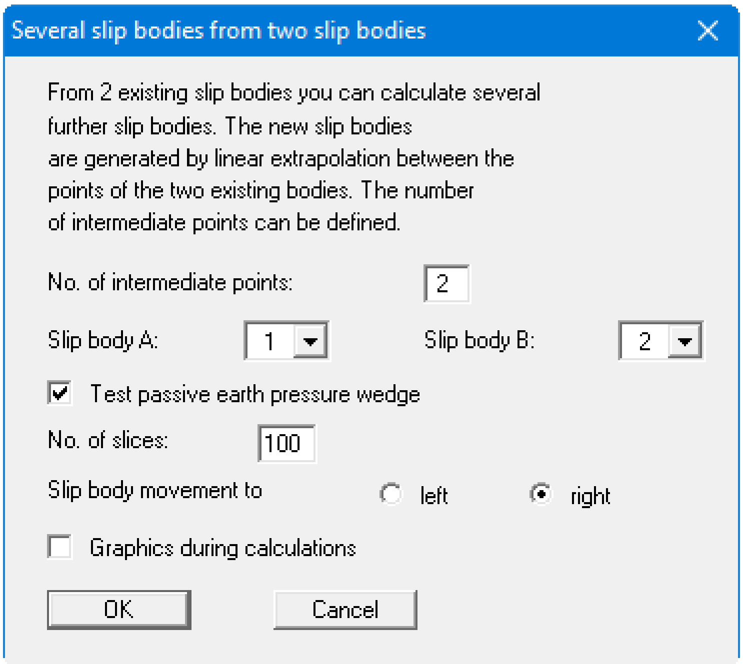
You will see a dialog box. By clicking on the button for the required slip body you open a further dialog box containing all the principal analysis results for the selected slip body. The designations correspond to those used in the equation in DIN 4084.



In this slip body result box you can view and print a table of individual slip body results using the "**Table**" button (see Section 10.1.10.3, "**File/Print output table**" button "**Output as ASCII**").

10.8.6 "Analyse intermediate slip bodies" menu item

If you have defined a minimum of two polygonal slip bodies, you can have further slip bodies (intermediate slip bodies) calculated from these. The only condition is that both slip bodies have the same amount of slip body points.



You must first specify the number of intermediate points and the number of both slip bodies for which ***intermediate slip bodies*** are to be calculated. Using this value, the positions of intermediate points between the x and y coordinates of both boundary slip bodies are determined. The higher this value is, the more ***intermediate slip bodies*** will be calculated. If both points of the ***boundary slip bodies*** concerned are equal, the program reacts intelligently and sensibly reduces the number of ***intermediate slip bodies***. After pressing the "**OK**" button calculations will begin. During calculations you will see the slip body which is currently being calculated. You can also prematurely cancel a calculation by pressing the "**Cancel**" button in the dialog box which appears after calculations have commenced. The lowest safety factor which has been calculated at the time of cancellation, and the number of slip bodies calculated so far will be displayed. After calculations are complete you have transfer the slip body with the lowest safety factor to the list of current slip bodies.

When analysing using the "**General wedge method**" and the "**Vertical slice method**", the shear strengths on the intermediate slip lines may be taken into consideration.

The following procedure, e.g., is sensible:

- Define a slip body with, e.g., extremely steep inclinations in the active and passive earth pressure areas.
- Then duplicate this slip body and change the inclinations to extremely flat. If necessary, alter the positions of slip body points. Confirm the alterations to the duplicated slip body with the [**Return**] key.
- By duplicating the first slip body you ensure that slip body points which are the same for both slip body really are the same and the program therefore does not carry out a variation for such points.
- Now select the "**Analyse intermediate slip bodies**" menu item.

10.8.7 "Show" menu item

You can view the intermediate slip bodies and, if wished, send them to the printer.

10.8.8 "Define slip bodies via lines, rectangles/quadrilaterals" menu item

Using this menu item a family of slip bodies can be generated using the mouse, with relatively little effort. You can select from a variety of methods in a dialog box. After selecting the method to be adopted and entering the subdivisions in a dialog box, the selected element (line, rectangle, etc.) is displayed on the screen. In this manner you have an example for application of the individual procedures and, in analogy, can subsequently try out your own definitions.

For example, define slip bodies by means of "**1 rectangular search box**". The number of the polygons is varied according to the defined array subdivision and the size of the box subsequently represented on the screen. The array points then represent the slip body polygon points.

10.8.9 "Slip bodies ... analyse" menu item

You can analyse the slip bodies defined using the menu item "**Define slip bodies via lines, rectangles/quadrilaterals**" (Section 10.8.8).

10.8.10 "Slip bodies ... show" menu item

You can display the slip bodies defined using the menu item "**Define slip bodies via lines, rectangles/quadrilaterals**" (Section 10.8.8).

10.8.11 "Slip bodies ... load/save" menu item

The slip bodies defined using the "**Define slip bodies via lines, rectangles/quadrilaterals**" menu item can be saved to a separate file, or previously saved slip bodies loaded. In this way you can analyse a number of variants.

10.8.12 "Preferences" menu item

You can specify the method of display of certain stresses, etc. on the slip surfaces. You will see differing dialog boxes, depending on the method.

Janbu:
You can select a representation including the pore water pressure, the shear stress or the normal stress, etc. on the slip surfaces.

General wedge method and **vertical slice method:**
Among other things, you can specify which forces of the force polygon are displayed in the graphics. Using "**Forces factor**" you enter the scale factor for force representation. A value of 0.02 means, for example, that a force of 120 kN/m is represented with a length of 120 · 0.02 = 2.4 m (!). The forces can be equipped with arrow points, which can be colour-filled ("**Colour**" button) and may also be framed in the standard colour. The force arrows can be labelled (e.g. with the letter "**C**" only, for cohesion force). Additionally, the force value can be shown. Experiment with the preferences to discover your own ideal layout. Furthermore, the pressure line can be displayed when working with the vertical slice method.

10.8.13 "Force polygon" menu item (General wedge method and Vertical slice method only)

After calculating a slip body you have the failure body force polygon displayed.

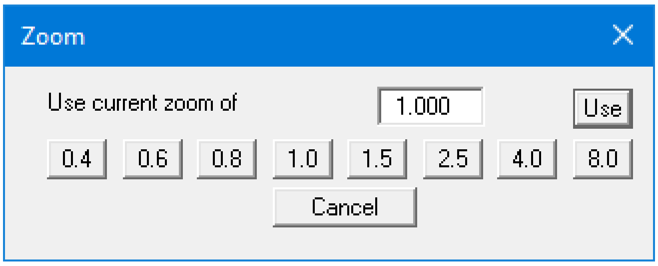
10.8.14 "Displacement diagram" menu item (General wedge method only)

After calculating a slip body you can have the displacement diagram for a failure body displayed.

10.9.1 "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.

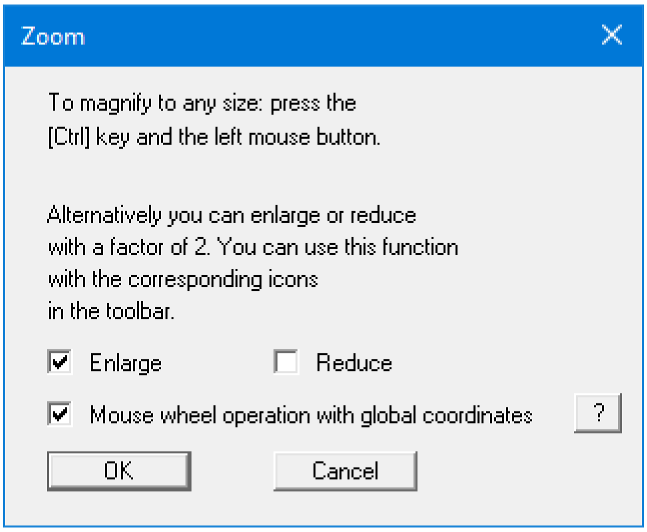


A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "**Use**" to exit the box the current factor is accepted. By clicking on the "**0.4**", "**0.6**", etc. buttons, the selected factor is used directly, and the dialog box closed.

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.

10.9.2 "Zoom info" menu item

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



When the program is started for the first time, the setting for mouse wheel operation according to Windows conventions is activated as standard. If you want to change the system coordinates and the scale of your system via the mouse wheel, activate the check box "**Mouse wheel operation with global coordinates**". If you close the program with this setting, the setting is still activated the next time you start it.

10.9.3 "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. ground line, soil layers 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.

10.9.4 "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.

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

Using these two menu items you can add free text, lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.) to the main program graphics. PDF files can also be imported as images. The same pop-up menu opens for both menu items, the icons and functions used are described in more detail in the **Mini-CAD** manual saved in the '**C:\Program Files (x86)\GGU-Software\ Manuals**' folder during installation. 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 footings).
- Objects created with the **Header CAD** are based on the page format (in mm). This makes them independent of the coordinate system and keeps them 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 (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.

10.9.6 "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.

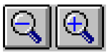
At the bottom of the program window, you find a status bar with further information. You can also activate or switch off the status bar here. The preferences will be saved in the "**GGU-STABILITY.alg**" file (see menu item "**Graphics preferences/Save graphics preferences**") and will be active at the 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 these tools, you can navigate between the individual pages in the ***tabular representation***.

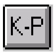
 "**Select page**"
If you are in the ***tabular representation***, you can use this tool to jump to a specific page or to return to the ***normal representation***, that is, to the graphics.

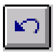
 "**Zoom out**"
If you have previously ***zoomed in***, this tool returns to a full screen display.

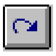
 "**Zoom (-)**"/"**Zoom (+)**"
With the zoom functions you can zoom in or out of parts of the image, by clicking the left mouse button.

 "**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 6.4).

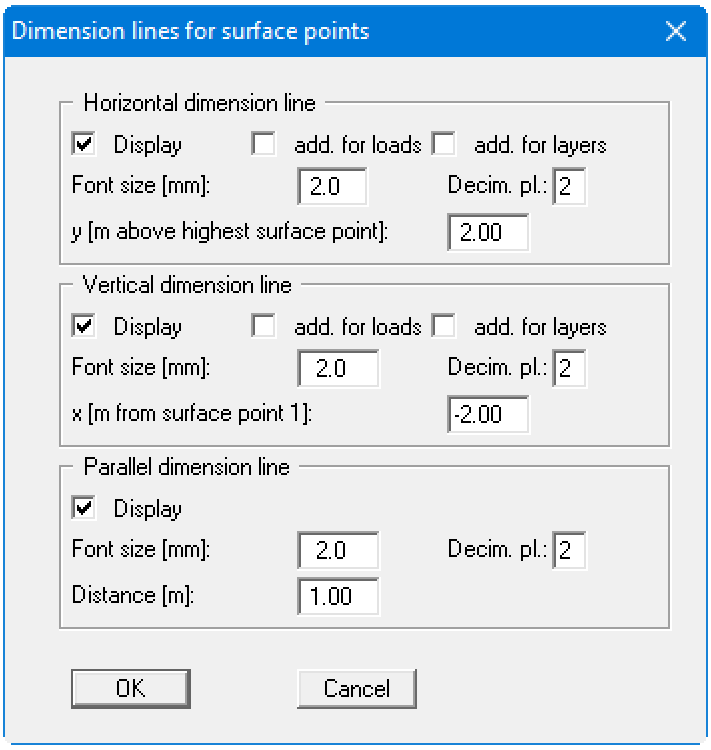
 "**Change method**"
By clicking on this tool you switch to the next method (Janbu, General wedge method, etc.). The method and selected standard are displayed at the right of the status bar (lower screen boundary).

 "**Undo move object**"
By clicking this tool, the last alteration in your system or the last performed movement of graphical elements made using the [**F11**] function key or the menu item "**Editor 1/Move objects**" can be undone. This only works if the undo function is activated in the menu item "**Editor 2/Preferences**".

 "**Restore move object**"
By clicking this tool, the last alteration in your system or object movement undo carried out using "**Undo move object**" can be restored. This only works if the undo function is activated in the menu item "**Editor 2/Preferences**".

10.9.7 "Dimension lines" menu item

You can define a vertical and/or horizontal dimension line for the graphics in order to emphasise and clarify the system dimensions.



The distance to the surface points is defined by means of "**y**" (horizontal), "**x**" (vertical) and "**Distance**" (parallel). Negative values define a position below or to the left of the surface points. All values are in metres in the scale selected (see the menu item "**Page size + margins/Manual resize (editor)**" in Section 10.10.2).

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

10.9.8 "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-STABILITY.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**".

10.9.9 "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.

10.10 Page size + margins menu

10.10.1 "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 using the mouse 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.

10.10.2 "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 and margins**" menu item by means of the plot margins (see Section 10.10.7).

10.10.3 "Zoom" menu item

You can linearly enlarge or reduce the image coordinates. This menu item is useful for specifying a centre-points array which lies outside of the current screen section.

10.10.4 "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.

10.10.5 "Save coordinates" menu item

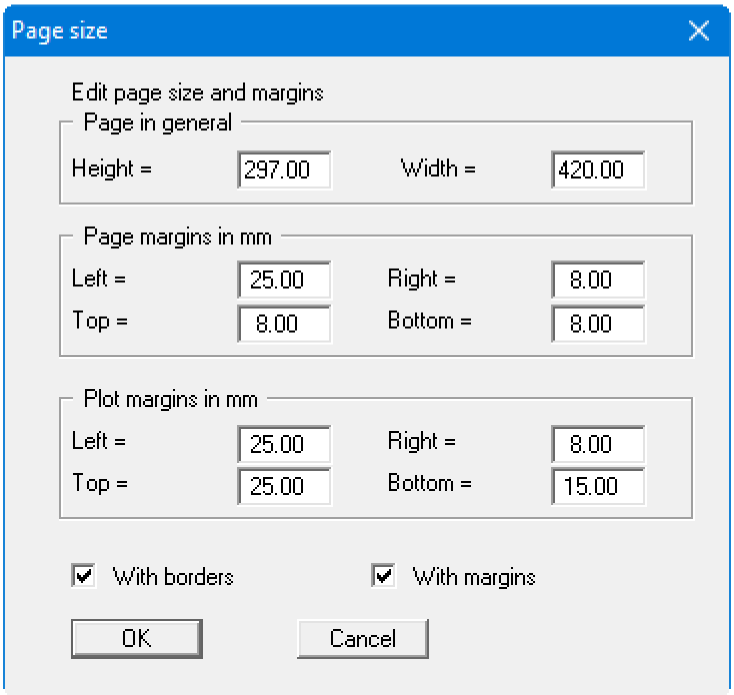
The current image coordinates can be saved in a file with the extension "**.bxy**" and be reloaded later for the same file or for different files

10.10.6 "Load coordinates" menu item

You can reload the coordinates which you saved earlier.

10.10.7 "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.
- "**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.
- 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.

10.10.8 "Font size selection" menu item

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

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

10.11 Nail wall menu

10.11.1 General notes on nail wall input

Nail walls are analysed using polygon slip surfaces in line with applicable approvals. The analysis of circular slip surfaces (Bishop) serves to investigate the overall stability, i.e. slip circles that do not intersect either the nails or the wall.

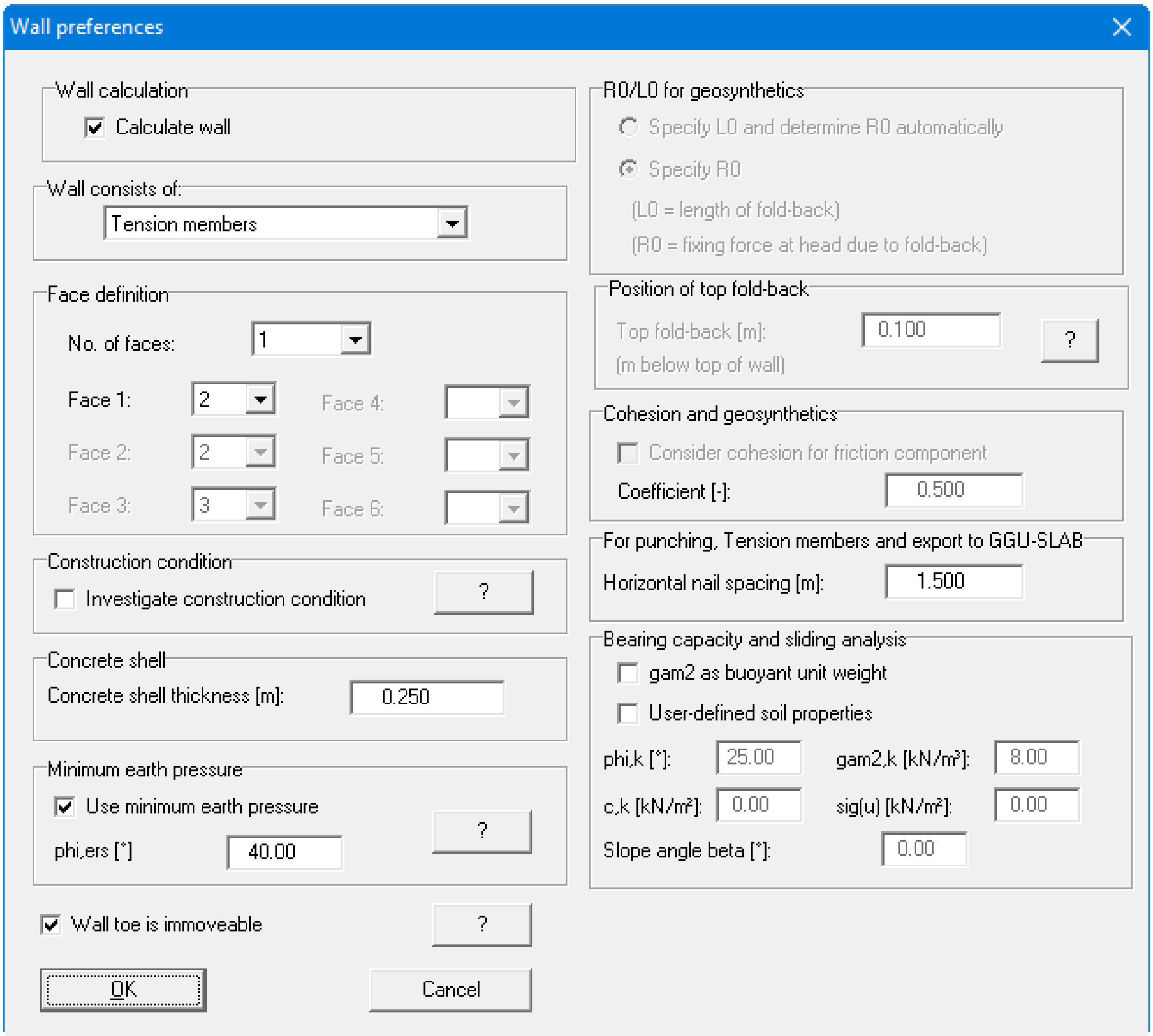
Input of system geometry follows the course described in the previous menu items. The nail wall preferences are specified in the "**Nail wall**" menu. The menu items can only be selected if you activate the "**Calculate wall**" box in the "**Nail wall/Preferences**" menu item.

For the analysis, you must specify the polygon course section in which the nail wall is located. The specified polygon course section (= nailed slope face) governs all necessary verifications.

An investigation of nail walls can only be performed for ***slopes failing to the left***.

10.11.2 "Preferences" menu item

Using this menu item, you make basic settings for the nail wall. The "**Calculate wall**" check box must be activated for this. First select the elements making up the wall.



The faces are sections of the defined terrain polygon course. They are continuously numbered from left to right. In the **GGU-STABILITY** program it is possible to define up to a maximum of six nailed slope faces. To do this, enter the number of ***nailed*** slope faces and assign these faces the corresponding face numbers (see Figure 33). The thus defined slope faces are highlighted in the graphics with a line wall. The thickness of the 'wall' can be altered using the menu item "**Nail wall/Graphics**" (see Section 10.11.3).

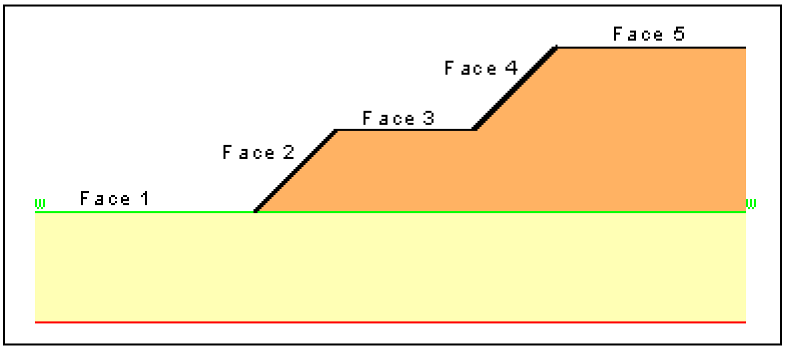


Figure 33 Face numbering (face 2 and face 4 are nailed)

Corresponding to the selected element various input areas will be activated. Explanations can be found by clicking on the "**?**" buttons.

The force R0 designates an anchorage at the head of the geosynthetics, e.g. in a gabion wall. If you have selected "**Geosynthetics**" as nailing element and "**Specify L0 and determine R0 automatically**" you enter the fold-over length L0 when generating the geosynthetics. The program will then automatically calculate the force R0 from the fold-over length.

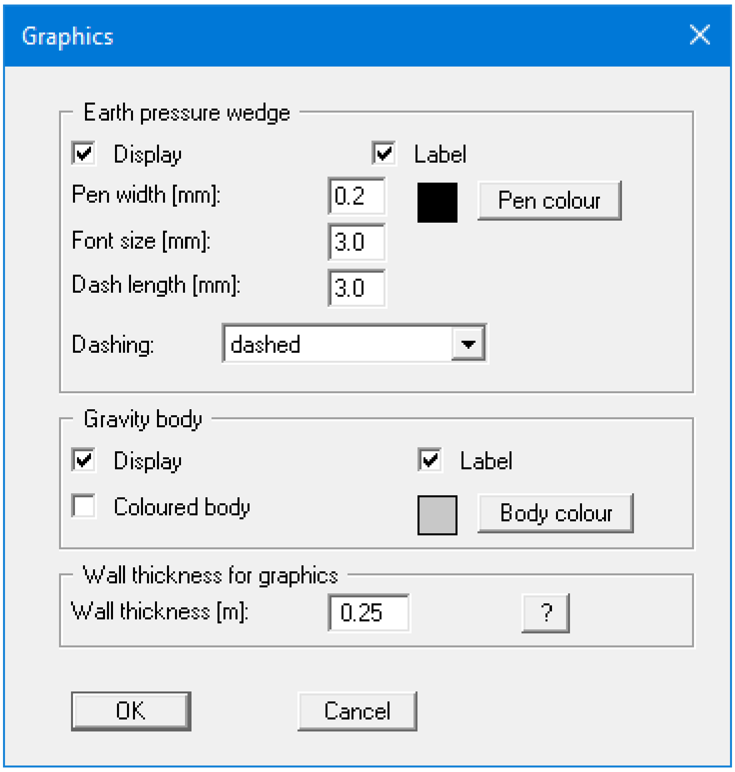
The "**Concrete shell thickness**" and the "**Horizontal nail centres**" are important for punching verification, tension members and for data export to **GGU-SLAB**.

Further information on the "**Investigate construction condition**" check box can be read by clicking the "**?**" button.

If the "**gam2 as buoyant unit weight**" check box is activated, this is taken into consideration accordingly when verifying bearing capacity (see "**Theoretical principles**", Section 9.8.6). If the soil properties in the ground deviate from the automatically determined mean values, activate the "**User-defined soil properties**" check box. Then, it is also possible to employ user-defined soil properties in the following input boxes.

10.11.3 "Graphics" menu item

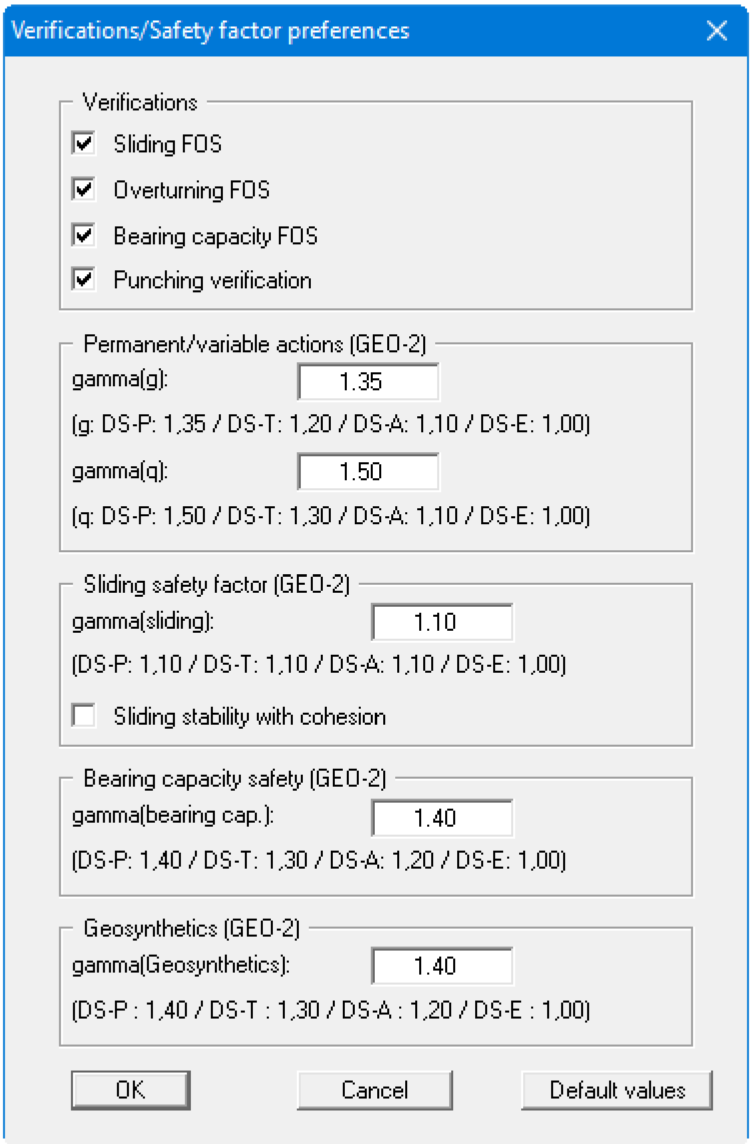
The appearance of the earth pressure wedge and the gravity body can be edited. This menu item cannot be selected if more than 1 nailed slope face exists in the system.



The dialog box inputs are self-explanatory and without meaning for the calculation results. Try out the possibilities which the dialog box presents You can use the wall thickness to make the slope face more accentuated for geosynthetics or steel strips. When using tension members, the wall thickness corresponds to the value you entered as the thickness of the concrete shell in the menu item "**Nail wall/Preferences**". Try out the possibilities of the dialog box.

10.11.4 "Verifications/Safety" menu item

With this menu item you can influence entries in the legend and in the print output table.

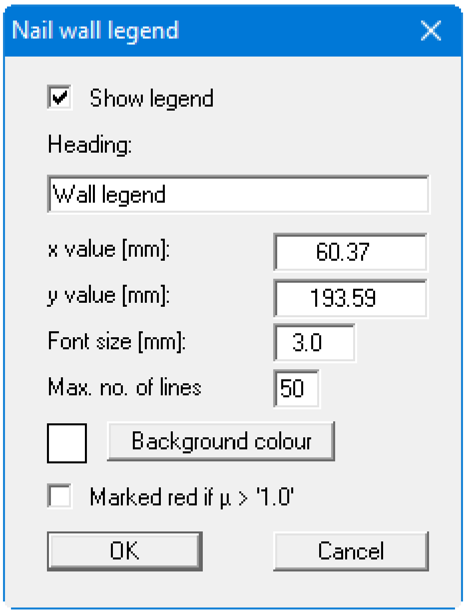


If, e.g., you do not wish to calculate bearing capacity with the simplified procedure, but would rather do an exact calculation using polygonal or circular slip surfaces, you may stop output of bearing capacity with the simplified procedure completely.

By pressing the "**Default values**" button the partial factors given in DIN 1054:2010 and EC 7 for the various load cases can be adopted (see menu item "**Editor 1/Partial factors, …**", Section 10.2.5).

10.11.5 "Nail wall legend" menu item

When analysing a nail wall a legend is displayed in the diagram if the "**Show legend**" check box is activated.

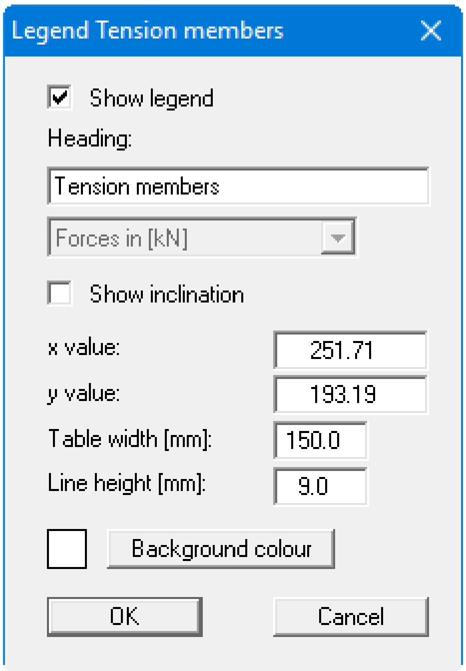


You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**". If necessary, presentation will be in several columns.

> 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.

10.11.6 "Nail force legend" menu item

If you activate the "**Show legend**" check box in the dialog box, a table of forces for the selected nailing elements will be displayed in the diagram.



The position and dimensions of the legend can be defined here. The font size is adjusted to suit the defined table width.

> 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

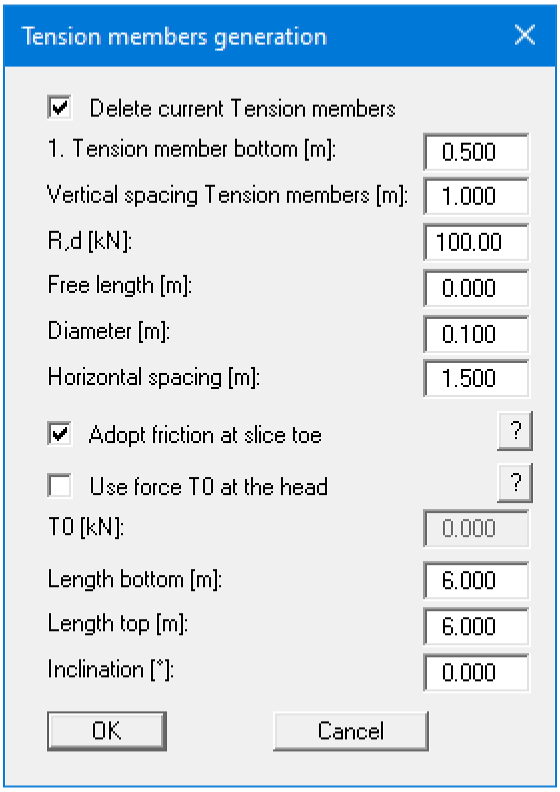
10.11.7 "Enter nails manually" menu item

This menu item is identical to the "**Geosynthetics**", "**Steel strips**" and "**Tension members**" buttons that appear in the "**Editor 1/Enter system parameters**" menu item dialog box. It is integrated here for users comfort only.

You can find a description for the input of geosynthetics in Section 10.2.2.11.
You can find a description for the input of stell strips in Section 10.2.2.12.
You can find a description for the input of tension members in Section 10.2.2.13.

10.11.8 "Generate" menu item

With this menu item you can define all the nails in a wall with very little input. If the wall consists of tension members, the following dialog box will appear:



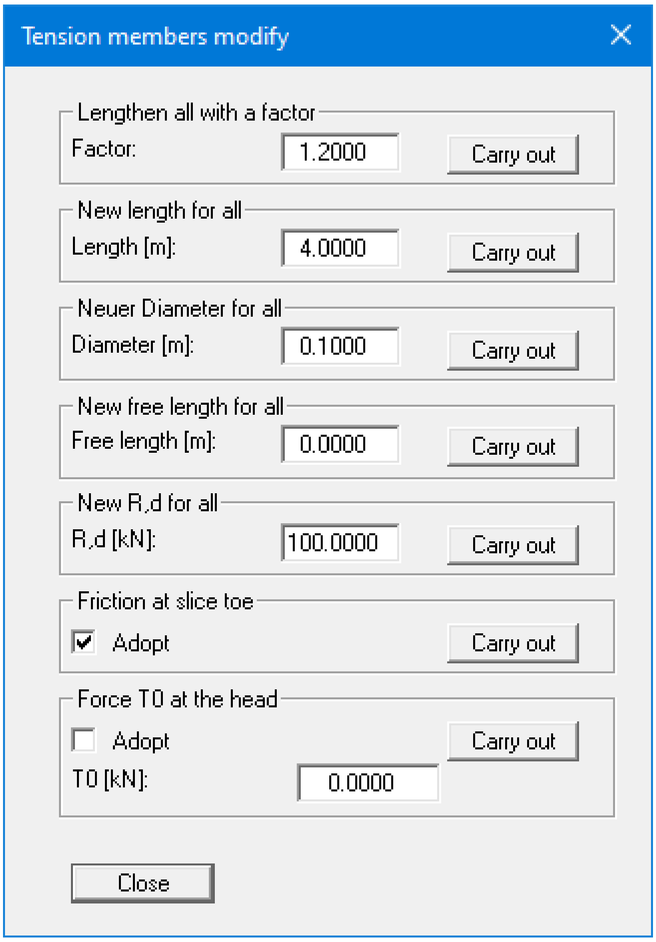
After leaving the dialog box the tension members will be displayed. If the position and length of the tension members still do not meet your wishes, select this menu item once again, activate the "**Current tension members to be deleted**" check box, and correct your input.

If the nail wall consists of geosynthetics,or steel strips, you will see an adapted generation dialog box.

Directly after geosynthetics, steel strips or tension members generation a dialog box opens in which you can initiate regeneration of slip surfaces (see "**Nail wall/Generate slip surfaces**" menu item in Section 10.11.11).

10.11.9 "Modify" menu item

If you have defined tension members and calculations have shown that other tension members would suffice, you can edit length and adhesive force easily using this menu item.

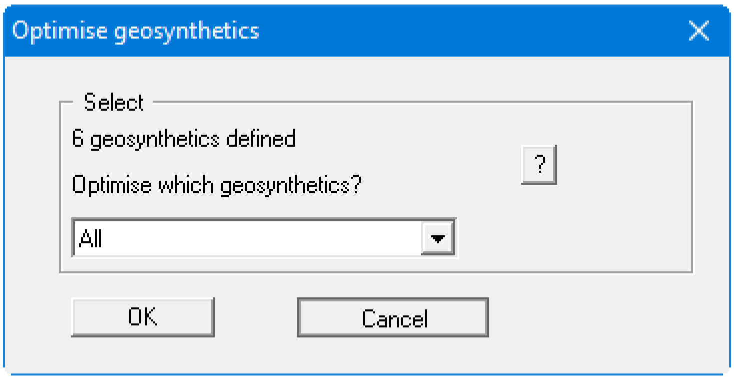


If you then select one of the "**Carry out**" buttons, the corresponding changes will be made.

If the nail wall consists of geosynthetics or steel strips, you will see an adapted modification dialog box.

10.11.10 "Optimise" menu item

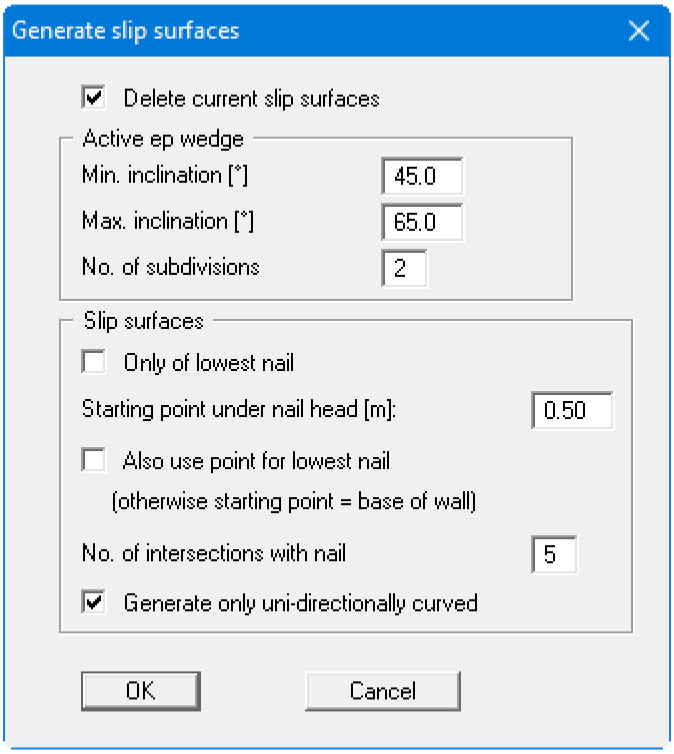
If the used geosynthetics will lead to an unstable slope you can use this menu item to let the program search for the optimum geosynthetics. The following dialog box opens:



Adopt the preferences and start optimisation of all geosynthetics. The result with optimum geosynthetics will be shown in a message box. Adopt the new geosynthetics with "**OK**". Then, restart the slope analysis using "**Utilisation factors/Analyse slip bodies**" menu item.

10.11.11 "Generate slip surfaces" menu item

Using this menu item, you can create a variety of slip surfaces with just a few entries. Slip surfaces with 2 polygonal sections are generated (two-wedge failure mechanism).



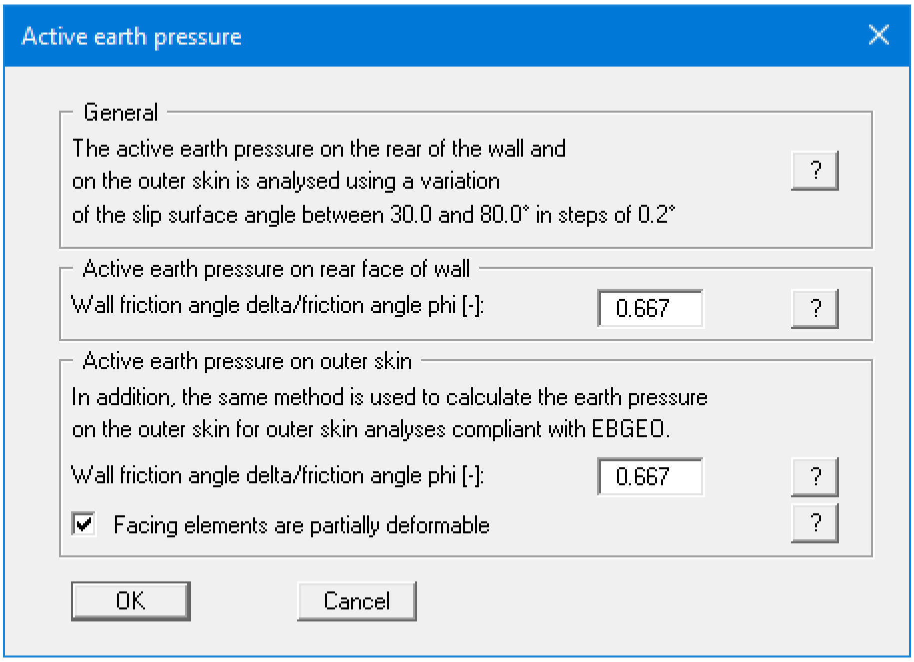
In the upper group box of the dialog box, define the inclinations of the rear slip line, ***responsible*** for the active earth pressure acting on the system. The two inclinations given are used as limit values. The number of subdivisions used is given by "**No. of subdivisions**".

Define the location of the front slip line in the lower group box. The lowest slip lines always begin at the base of the wall and end at the tips of the nails. If the "**only of lowest nail**" check box is deactivated, additional slip lines are also generated from the soil nails above. These lines will begin at the head of the respective nails or at a given distance below this; you can specify this distance with "**Starting point under nail head [m]**". The slip lines end at the nail foot if the slip line has a positive inclination, otherwise the slip line is rejected (except for the lowest nail). If the "**No. of intersections with nail**" value is > 0, additional slip surfaces are generated, which intersect the respective nail.

Strictly speaking, an analysis utilising the "**General wedge method**" must also vary the intermediate slip line. The inclination of the intermediate slip line is adopted by the program at the same value as the rear principal slip line but with the opposite sign. Generally, this is not the most unfavourable location for the intermediate slip line only for highly variable stratification in the region of the intermediate and principal slip lines.

10.11.12 "Sliding, overturning, bearing failure" menu item

Verification is in accordance with the notes in the "**Theoretical principles**" section. You will receive the following box when using geosynthetics or steel strips.



You will see the results in message boxes. After exiting the message boxes, the principal design values are displayed on the screen in the "**Nail wall legend**" (see Section 10.11.5), if the corresponding check boxes are activated in the "**Nail wall/Verifications/Safety**" menu item (see Section 10.11.4).

This function can also be accessed by pressing the [**F7**] function key. This menu item cannot be selected if more than 1 nailed slope face exists in the system.

10.11.13 "Maximum nail forces" menu item

This menu item is only activated for analysis of a nail wall with geosynthetics or steel strips.

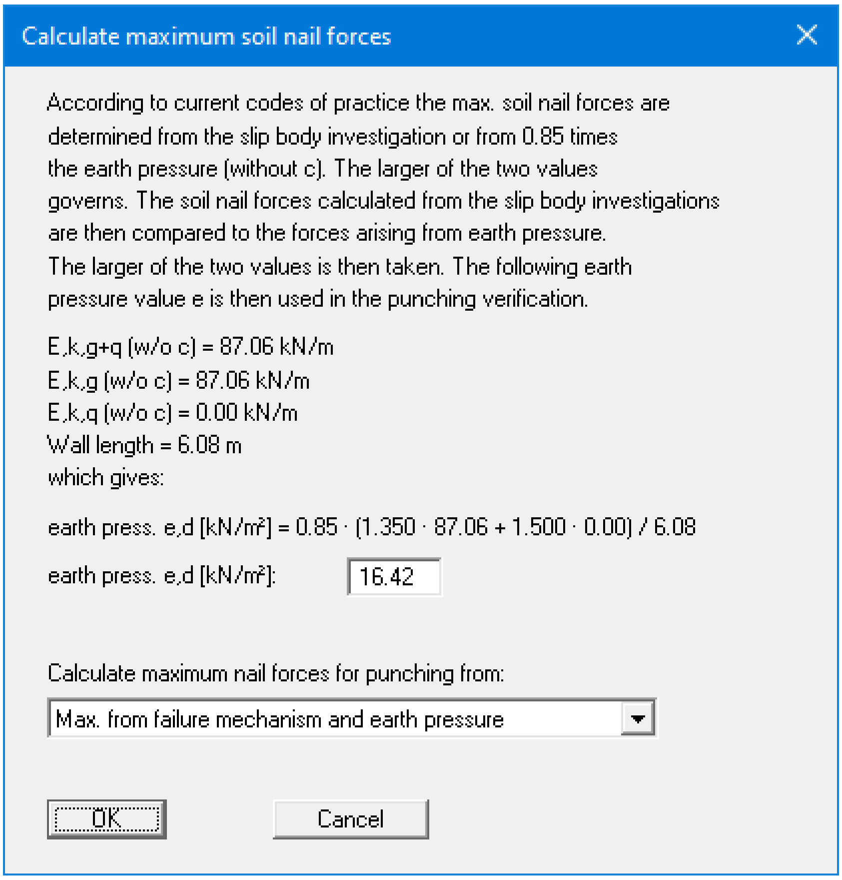
In accordance with current standards, the governing verification for design of the nails is the one producing the greatest nail forces. The following investigations are necessary for this:

- Load components in the nails arising from slip body investigation for final and construction conditions
- Load components in the nails arising from earth pressure on the concrete shell or exterior skin. The earth pressure from the nailed soil prism acting on the concrete shell or exterior skin may be adopted at 0.85 x active earth pressure value, but without applying cohesion. The earth pressure distribution may be adopted in a rectangular configuration, including in stratified ground. The wall friction angle is adopted at $\delta = 0$.

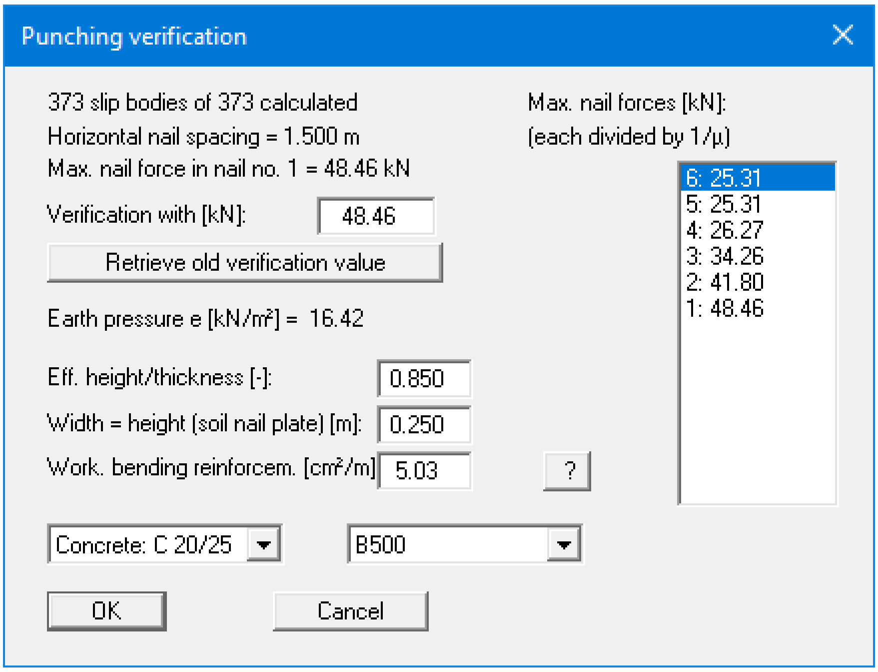
The "**Maximum nail forces**" menu item can only be selected if slip body investigations have already been carried out. This menu item determines the earth pressure on the outer skin and the stability verifications required by EBGEO. Once the analysis is complete the maximum nail forces are displayed in a message box.

10.11.14 "Maximum nail forces + punching" menu item

Calculation of the maximum nail forces using tension members is in analogy to the description given in the "**Maximum nail forces**" menu item in Section 10.11.13.



The value proposed for the earth pressure e,d in the "**Calculate maximum nail forces**" dialog box is also adopted for the punching verification. After confirming or editing of the calculated earth pressure value, the following dialog box will appear:

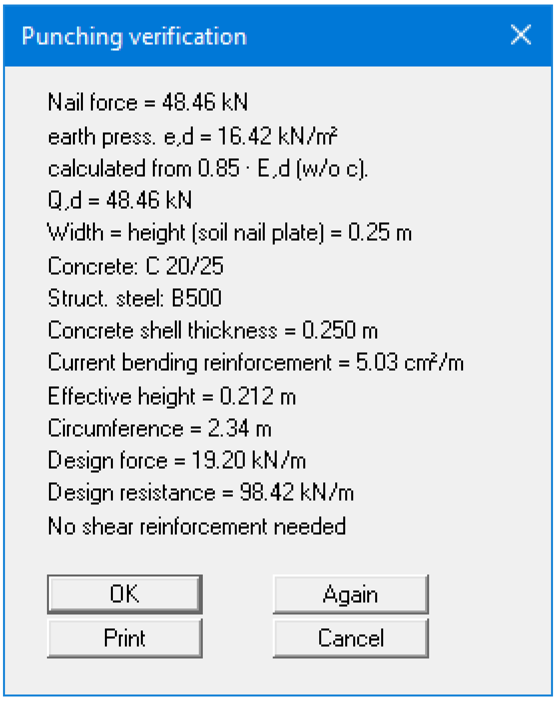


The maximum nail forces, determined in accordance with the descriptions of Section 10.11.13, are shown in the list in the dialog box. The maximum nail value is multiplied with the horizontal separation and suggested as design value for punching verification (see entry after "**Verification with [kN]:**"). This value can, if wished, be edited. If you are carrying out several calculations with slip bodies but want to calculate the punching safety factor with a different value to that suggested, you can use the "**Retrieve old verification value**" button to bring back the last entered value into the input box.

After this, punching verification is done. The pressure under the punching area will be subtracted from the nail force, in accordance with EC 2. The pressure is taken as the reduced earth pressure (see above). Besides this information the following must be given, in accordance with EC 2, for punching verification:

- Effective height of concrete shell in the verification area, as % of concrete shell thickness;
- Width of nail slab (= height):
- Present bending reinforcement;
- Concrete used;
- Steel used.

These inputs can be made in the lower part of the dialog box. Verification will be carried out after pressing the "**OK**" button and the results are displayed.



The dialog box contains all decisive values for verification. Using the "**Again**" button, you can repeat the verification with new input values.

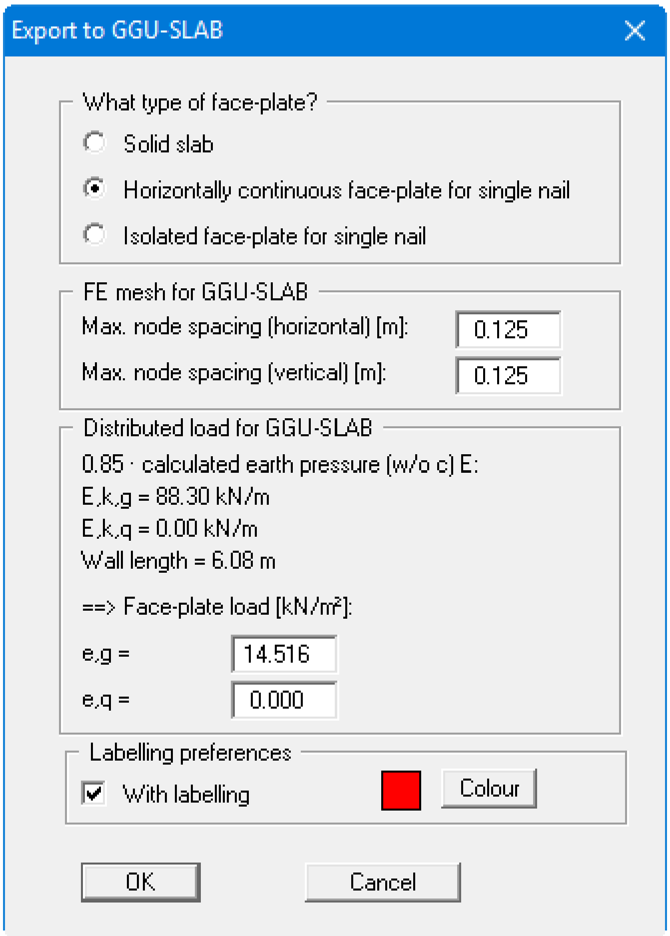
If, in the menu item "**Nail wall/Verifications**", you have activated the "**Punching verification**" check box, the most important design values will be presented in the "**Nail wall legend**" on the screen.

10.11.15 "Calculate earth pressure + weight" menu item

You can calculate earth pressure and weight independent of previous menu items. The results are shown in dialog boxes.

10.11.16 "Export to GGU-SLAB" menu item

This menu item is available for bending design of the concrete shell. A dataset is exported which can be read by the **GGU-SLAB** program. **Only systems with 1 slope face can be exported.** The **GGU-SLAB** program allows analysis and design of slabs using finite-element methods.



For more information on the upper dialog group box, "**What type of face plate?**" please refer to "**Theoretical principles**" (Section 9.8.8). If you have not selected a "**Solid slab**" the following dialog box opens after confirming with "**OK**":

The program calculates the maximum value from the existing vertical nail centres and proposes this as the value for the equivalent plate for the **GGU-SLAB** analysis. This value can be edited by the user.

The two values in the middle group box of the "**Export to GGU-SLAB**" dialog box define the fineness of the finite-element array exported to **GGU-SLAB**. However, refinement of the finite-element array can also be very easily performed in the **GGU-SLAB** program.

For more information on the constant distributed load on the slab please refer to Section 9.8.8.

The structural system exported to **GGU-SLAB** utilises the symmetry of soil nailing. The three possible structural systems for export to the **GGU-SLAB** program are shown in the following figures.

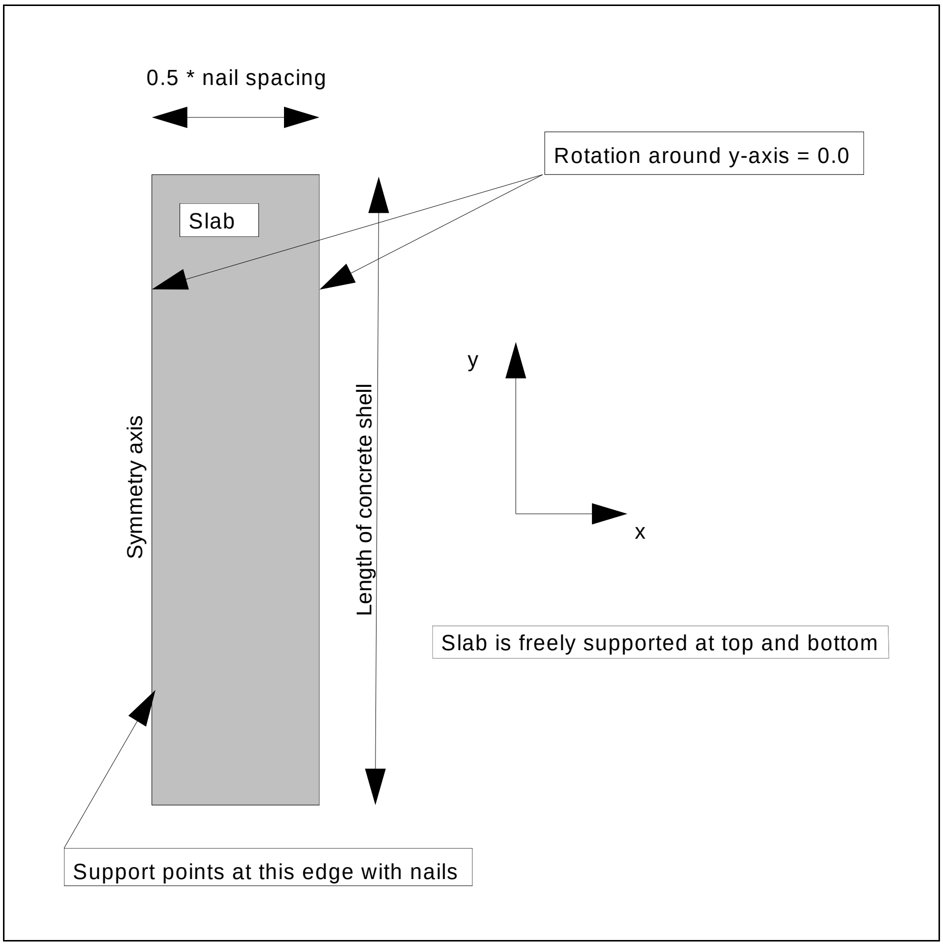


Figure 34 Structural system for a solid slab

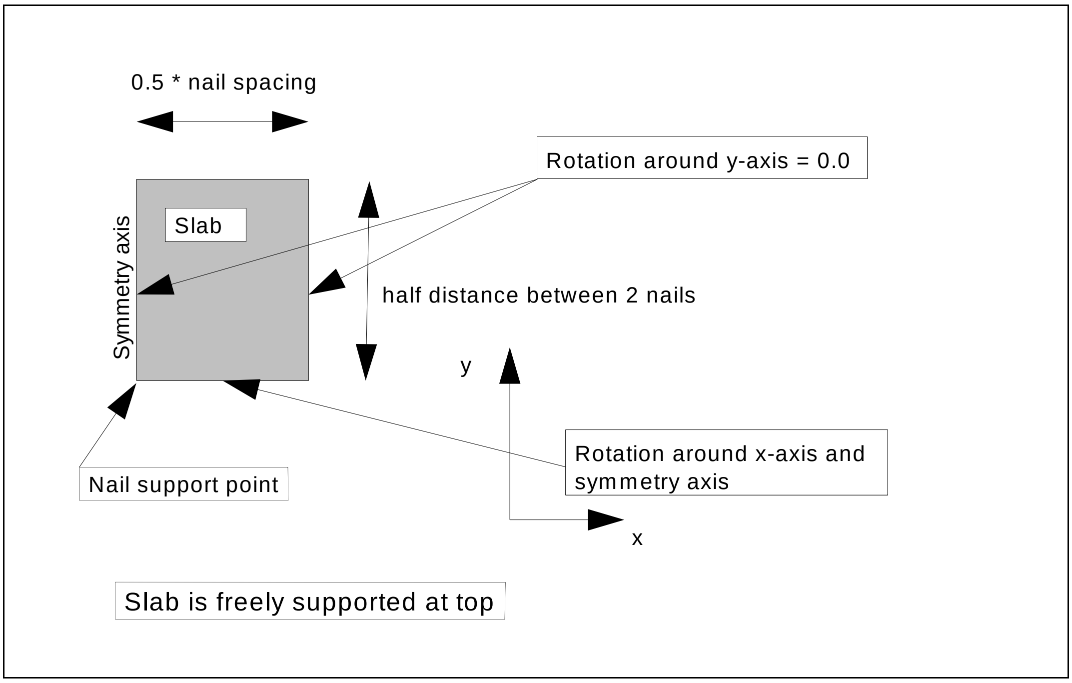


Figure 35 Structural system for a horizontally continuous slab

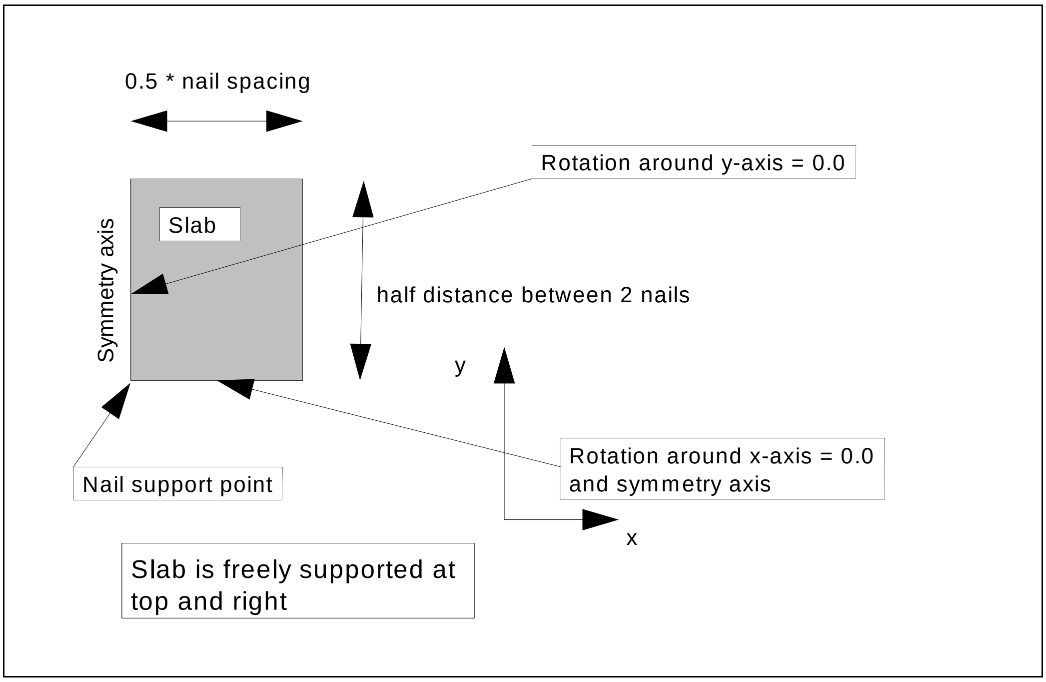
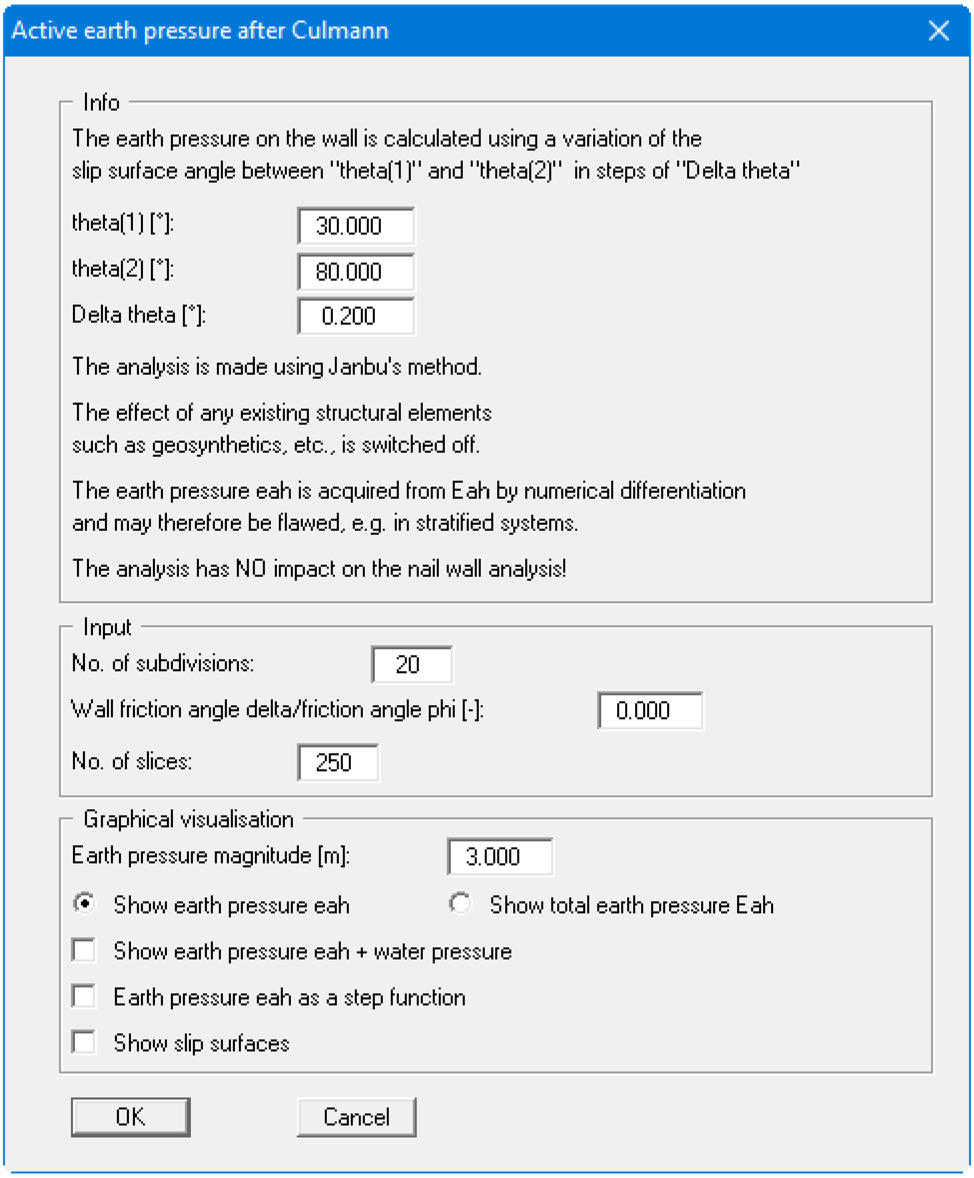


Figure 36 Structural system for a "solo" slab

10.11.17 "Active earth pressure after Culmann" menu item

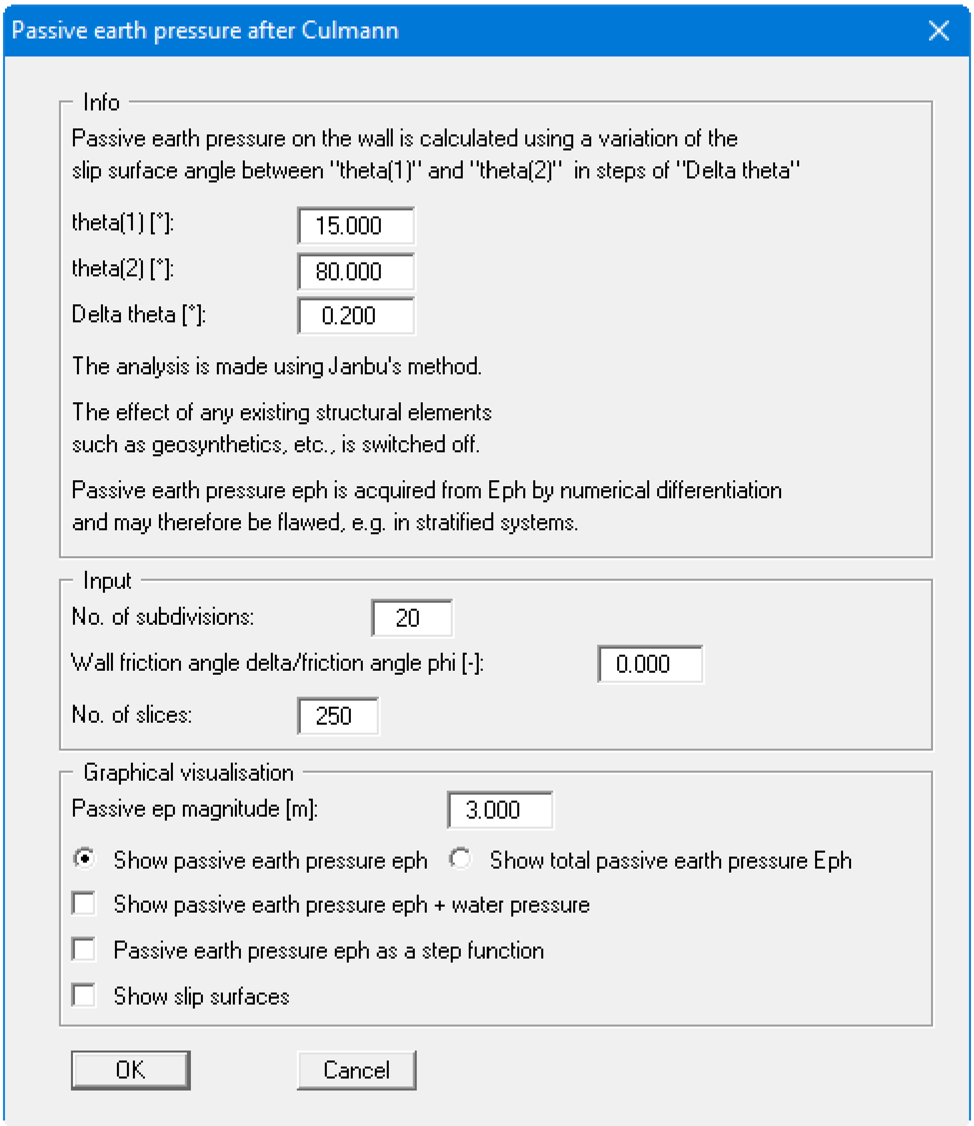
The active earth pressure after Culmann can be determined using this menu item. The following dialog box opens:



A number of preferences can be specified for subsequent graphical visualisation. The analysis results are initially presented in a message box. Using the "**Copy to clipboard**" button in the message box the data can be copied to the Windows clipboard, e.g. to paste into a Word document. After exiting the message box, the computed earth pressure is displayed in the system in line with the selected settings.

10.11.18 "Passive earth pressure after Culmann" menu item

The passive earth pressure after Culmann can be determined using this menu item. The following dialog box opens:



A number of preferences can be specified for subsequent graphical visualisation. The analysis results are initially presented in a message box. Using the "**Copy to clipboard**" button in the message box the data can be copied to the Windows clipboard, e.g. to paste into a Word document. After exiting the message box, the computed earth pressure is displayed in the system in line with the selected settings.

10.12.1 "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-STABILITY**.

10.12.2 "Help" menu item

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

10.12.3 "GGU on the web" menu item

Using this menu item, you can access the GGU Software website: www.ggu-software.com.

Get information on updates and modifications on a regular basis from your program module page. You can also subscribe to email notifications, which provide information on all modifications on a monthly basis.

10.12.4 "GGU support" menu item

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

10.12.5 "What's new?" menu item

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

10.12.6 "Language preferences" menu item

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

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

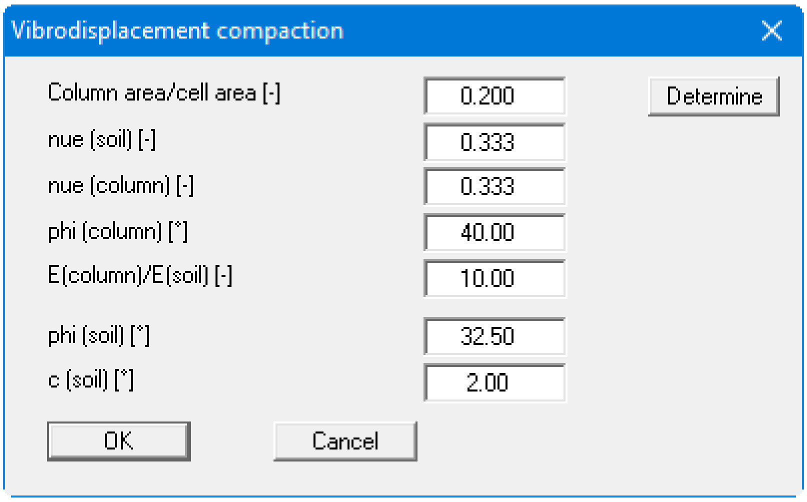
10.12.7 "Kölsch method preferences" menu item

Kölsch developed a constitutive equation for the analysis of waste slopes in 1996 (see '*Der Einfluss der Faserbestandteile auf die Scherfestigkeit von Siedlungsabfall'. Mitteilungen des Leichtweiß-Institutes für Wasserbau der TU Braunschweig*, Bulletin 133/96). This constitutive equation is implemented in **GGU-STABILITY** as the "**Kölsch method**" and can be activated for the corresponding analysis using this menu item. The data columns required to perform the analysis are then included in the soil properties dialog box (see Section 10.2.2).

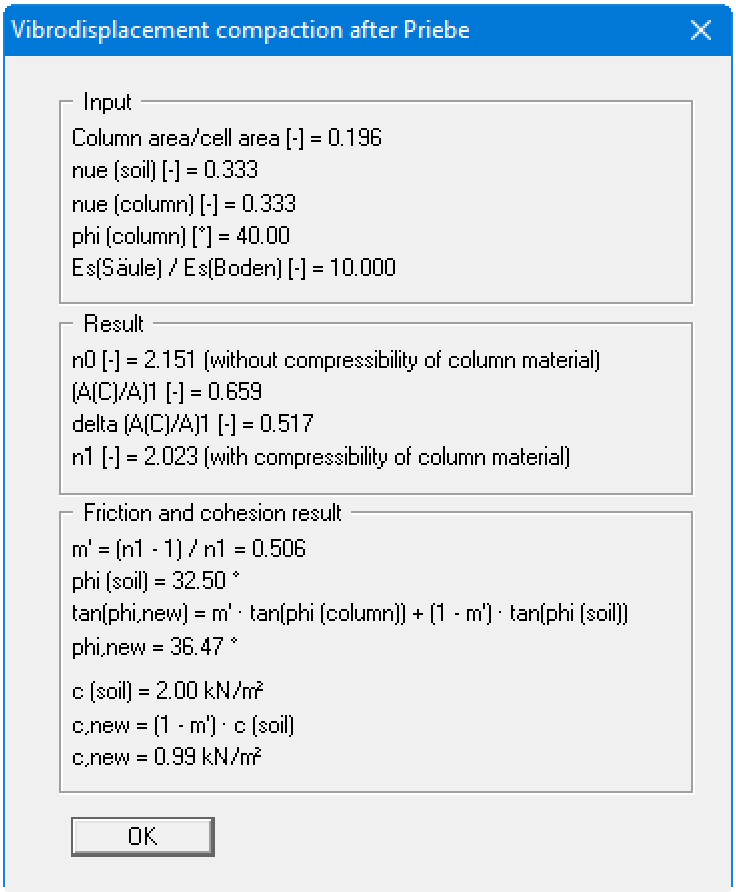
The theoretical principles are described in the "**General notes on analysis with fibre cohesion**" section (see Section 9.9), further literature can be found in Section 11.

10.12.8 "Test vibrodisplacement compaction" menu item

The improvement brought about by vibrodisplacement compaction can be tested by varying the soil properties using this menu item.



Using the "**Determine**" button the column/cell area ratio "**A(C)/A**" can be determined using a variety of array procedures. After confirming your input values using "**OK**", the results are presented in a message box:



Click "**Cancel**" in the above dialog box to end the menu item.

11 Literature

Collins, Kölsch, Ziehmann (1997): *Veränderung des Tragverhaltens und der mechanischen Eigenschaften von Abfällen durch Alterung und Abbau*. DFG final report. Ref no. Co 76/26-1 to -5. Leichtweiß Institute for Hydraulic Engineering, Technical University of Braunschweig

Kölsch (1996): *Der Einfluss der Faserbestandteile auf die Scherfestigkeit von Siedlungsabfall. Mitteilungen des Leichtweiß-Institutes für Wasserbau der TU Braunschweig*, Bulletin 133/96.

Deutsche Gesellschaft für Geotechnik (1997): *Empfehlungen der Arbeitskreise zur 'Geotechnik der Deponien und Altlasten' GDA*, (Recommendations of the Working Group on 'Geotechnical Aspects of Landfill and Brownfield Sites', GDA), Berlin: Ernst, ISBN 3-433-01318-7

Gömmel, R., Neidhart, T. (2016): *Zum Ansatz von Stabilisierungssäulen beim Nachweis der Gesamtstandsicherheit.* 10. Kolloquium Bauen in Boden und Fels, S. 399–409, Technische Akademie Esslingen e.V. (Hrsg.), ISBN 978-3-943563-21-4

12 Index

E

F

G

H

I

J

K

L

M

N

O

P

R

S

T

U

V

W

Z