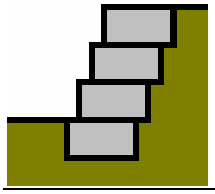
Analysis of retaining structures manufactured using concrete elements, stacked blocks and gabions

GGU-GABION

VERSION 7



Last revision: June 2020
Copyright: Prof. Dr. Johann Buß
Technical implementation, layout and sales: Civilserve GmbH, Steinfeld

Contents:

1 Preface 7

2 Capabilities 7

3 Licence protection 9

4 Language selection 9

5 Starting the program 10

6 First steps 11

6.1 Worked example 1: Gabion wall 11

6.1.1 System description (Ex. 1) 11

6.1.2 Step 1: Select analysis options (Ex. 1) 12

6.1.3 Step 2: Define the system (Ex. 1) 13

6.1.4 Step 3: Define body loads (Ex. 1) 14

6.1.5 Step 4: Define body geometry (Ex. 1) 15

6.1.6 Step 5: Define material of gabion (Ex. 1) 17

6.1.7 Step 6: Define berm (Ex. 1) 18

6.1.8 Step 7: Define soils (Ex. 1) 19

6.1.9 Step 8: Define type of earth pressure (ex. 1) 20

6.1.10 Step 9: Analyse system (Ex. 1) 21

6.1.11 Step 10: Evaluate and visualise the results (Ex. 1) 22

6.2 Worked example 2: Noise abatement wall 23

6.2.1 System description (Ex. 2) 23

6.2.2 Step 1: Select analysis options and generate noise abatement wall (Ex. 2) 24

6.2.3 Step 2: Define material of gabion (Ex. 2) 25

6.2.4 Step 3: Analyse system (Ex. 2) 26

7 Theoretical principles 27

7.1 Body 27

7.2 Soil properties 28

7.3 Active earth pressure 28

7.4 At-rest earth pressure 28

7.5 Increased active earth pressure 28

7.6 Passive earth pressure 29

7.7 Berms 29

7.8 Area loads 30

7.9 Line loads 31

7.10 Bounded surcharges (active side) 32

7.11 Double-bounded surcharges (active side) 33

7.12 Bounded surcharges (passive side) 34

7.13 Double-bounded surcharges (passive side) 34

7.14 Structural system 35

7.15 Base inclination 36

7.16 Bedding conditions of wall toe 36

7.17 Action and displacement boundary conditions 37

7.18 Earth pressure redistribution 37

7.19 Verification concept 38
7.19.1 Verifications 38
7.19.2 Systems without geogrids 38
7.19.3 Systems with geogrids 39
7.20 Sliding safety to EC 7 40
7.20.1 Sliding safety in the base plane 40
7.20.2 Sliding safety in the tier joint planes 40
7.21 Verifications reinforced soil system 41
7.21.1 Internal stability 41
7.21.2 External stability 43
7.21.3 Overturning safety 44
7.21.4 Distribution of earth pressure on the gabion wall 45
7.22 Bearing capacity safety 47
7.22.1 Analysis principles 47
7.22.2 Determining the mean governing soil properties 48
7.23 Settlements 49
7.24 General stability 49
7.25 Analysis of internal stability 50
7.25.1 Analysis to DIN 1045 (old) 50
7.25.2 Analysis to EC 2/DIN 1045-1 51
7.25.3 Analysis to DIN 4093 52
7.25.3.1 Ultimate limit state 52
7.25.3.2 Serviceability limit state 53
7.25.3.3 Slenderness 53
8 Description of menu items 54
8.1 File menu 54
8.1.1 "New" menu item 54
8.1.2 "Load" menu item 55
8.1.3 "Save" menu item 55
8.1.4 "Save as" menu item 56
8.1.5 "Print output table" menu item 56
8.1.5.1 Selecting the output format 56
8.1.5.2 "Output as graphics" button 57
8.1.5.3 "Output as ASCII" button 59
8.1.6 "Export" menu item 60
8.1.7 "Output preferences" menu item 60
8.1.8 "Print and export" menu item 60
8.1.9 "Batch print" menu item 63
8.1.10 "Exit" menu item 63
8.1.11 "1, 2, 3, 4" menu items 63
8.2 Editor 1 menu 64
8.2.1 "Analysis options" menu item 64
8.2.2 "System" menu item 64
8.2.3 "Body (general)" menu item 65
8.2.4 "Body (geometry)" menu item 65
8.2.5 "Material" menu item 66

8.2.6 "Berms (active side)" menu item 67
8.2.7 "Berms (passive side)" menu item 67
8.2.8 "Soils" menu item 68
8.2.9 "Type of earth pressure" menu item 69
8.2.10 "Active earth pressure" menu item 70
8.2.11 "Passive earth pressure" menu item 71
8.2.12 "At-rest earth pressure" menu item 72
8.2.13 "User-defined earth pressure coefficients" menu item 72
8.2.14 "Verifications/Safety factors" menu item 73
8.2.15 "Verifications/Partial factors" menu item 74
8.2.16 "Seismic acceleration" menu item 75
8.3 Editor 2 menu 76
8.3.1 "Area and line loads" menu item 76
8.3.2 "Bounded surcharges" menu item 76
8.3.3 "Double-bounded surcharges" menu item 77
8.3.4 "Lateral pressures" menu item 78
8.3.5 "Action boundary conditions" menu item 78
8.3.6 "Displacement boundary conditions" menu item 79
8.3.7 "Settlements" menu item 80
8.4 Reinforced soil system menu 81
8.4.1 "Graphic preferences" menu item 81
8.4.2 "Verifications" menu item 82
8.4.3 "Enter geogrids manually" menu item 83
8.4.4 "Generate geogrids" menu item 85
8.4.5 "Modify geogrids" menu item 86
8.4.6 "Generate slip surfaces" menu item 87
8.4.7 "Geogrid forces" menu item 87
8.4.8 "Sliding, overturning, bearing capacity" menu item 88
8.4.9 "Calculate earth pressure + weight" menu item 88
8.4.10 "Geogrids table values" menu item 88
8.5 System menu 89
8.5.1 "Info" menu item 89
8.5.2 "Special preferences" menu item 89
8.5.3 "Depth subdivisions" menu item 89
8.5.4 "Analyse" menu item 90
8.5.4.1 Start dialog box 90
8.5.4.2 "Type of toe pivot" group box 90
8.5.4.3 "Special preferences" group box 91
8.5.4.4 "Type of redistribution" group box 92
8.5.5 "Graph positioning preferences" menu item 95
8.5.6 "Graphics output preferences" menu item 96
8.5.7 "Labelling preferences" menu item 97
8.5.8 "Body pens + colours" menu item 98
8.5.9 "Dimension lines" menu item 99
8.5.10 "Display system" menu item 99
8.5.11 "Display results" menu item 99

8.6 Evaluation menu ... 100
8.6.1 General note ... 100
8.6.2 "Earth pressure redistribution" menu item ... 100
8.6.3 "Main output summary" menu item ... 100
8.6.4 "Bearing capacity" menu item ... 100
8.6.5 "Settlements" menu item ... 100
8.6.6 "Sliding safety" menu item ... 100
8.6.7 "Static equilibrium (DIN 1055)" menu item ... 100
8.6.8 "EQU analysis" menu item ... 100
8.7 Graphics preferences menu ... 101
8.7.1 "Refresh and zoom" menu item ... 101
8.7.2 "Zoom info" menu item ... 101
8.7.3 "Legend font selection" menu item ... 101
8.7.4 "Pen colour and width" menu item ... 101
8.7.5 "Mini-CAD toolbar" and "Header toolbar" menu items ... 102
8.7.6 "Toolbar preferences" menu item ... 102
8.7.7 "Soil properties legend" menu item ... 104
8.7.8 "General legend" menu item ... 105
8.7.9 "Design legend" menu item ... 106
8.7.10 "Settlement legend" menu item ... 107
8.7.11 "Reinforced soil system legend" menu item ... 107
8.7.12 "Geogrid legend" menu item ... 108
8.7.13 "Quick header" menu item ... 109
8.7.14 "Move objects" menu item ... 109
8.7.15 "Save graphics preferences" menu item ... 110
8.7.16 "Load graphics preferences" menu item ... 110
8.8 Page size + margins menu ... 111
8.8.1 "Auto-resize" menu item ... 111
8.8.2 "Manual resize (mouse)" menu item ... 111
8.8.3 "Manual resize (editor)" menu item ... 111
8.8.4 "Font size selection" menu item ... 111
8.8.5 "Page size and margins" menu item ... 112
8.8.6 "Undo" menu item ... 113
8.8.7 "Restore" menu item ... 113
8.8.8 "Preferences" menu item ... 113
8.9 Info menu ... 114
8.9.1 "Copyright" menu item ... 114
8.9.2 "Maxima" menu item ... 114
8.9.3 "Compare ep coefficients" menu item ... 114
8.9.4 "Help" menu item ... 114
8.9.5 "GGU on the web" menu item ... 114
8.9.6 "GGU support" menu item ... 114
8.9.7 "What's new?" menu item ... 114
8.9.8 "Language preferences" menu item ... 114

9 Tips and tricks 115
9.1 "?" buttons 115
9.2 Keyboard and mouse 116
9.3 Function keys 117
9.4 "Copy/print area" icon 118
10 Index 119

List of Figures:

Figure 1 Example system 1 11
Figure 2 Example system 2 23
Figure 3 Gabion body 27
Figure 4 Berm on the active side 29
Figure 5 Area load 30
Figure 6 At-rest earth pressure from area loads 31
Figure 7 Bounded surcharge (active side) 32
Figure 8 Two bounded surcharges 33
Figure 9 Double-bounded surcharge 33
Figure 10 Bounded surcharge (passive side) 34
Figure 11 Structural system of a wall without geogrid 35
Figure 12 Base inclination 36
Figure 13 Wall with geogrid 39
Figure 14 Two-part wedge failure mechanism 41
Figure 15 Equivalent system for weight calculation 43
Figure 16 Vertical section for earth pressure determination 44
Figure 17 Slip surface and earth pressure distribution 45
Figure 18 Logarithmic spiral 48
Figure 19 n-value and width for internal stability analysis 50
Figure 20 Body geometry 65
Figure 21 Passive earth pressure in front or superimposed 91
Figure 22 Birectangular earth pressure redistribution 92
Figure 23 Earth pressure redistribution in a trapezoidal 93
Figure 24 Earth pressure redistribution in a quadrilateral 93
Figure 25 User-defined earth pressure redistribution 94

1 Preface

The **GGU-GABION** program allows the analysis of retaining structures manufactured using concrete elements, stacked blocks and gabions. It is also possible to easily generate and analyse a noise abatement wall using gabions.

The governing basis for these analyses is the March 2003 ***FGSV*** (*Forschungsgesellschaft für Straßen und Verkehrswesen*) Fact Sheet 555. For analysis both the ***global safety factors*** to DIN 1054 (old) and the new ***partial safety factors*** to EC 7 may be taken into consideration.

The application is designed to allow simple data input or modification. The input is immediately shown on the screen, giving you optimum control over what you are doing. 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 "**?**" button (see also section 9.1).

Graphics output supports the true-type fonts supplied with WINDOWS, so that excellent layout is guaranteed. Colour output and graphics (e.g. files in formats BMP, JPG, PSP, TIF, etc.) are supported. PDF and DXF files can also be imported by means of the integrated **Mini-CAD** module (see the "**Mini-CAD**" manual).

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-GABION** program has the following special capabilities (and maximum values):

- Almost any kind of definition of gabion bodies may be defined.
- Base inclination of the gabion body
- Up to 50 soil layers
- Up to 40 berms on the active earth pressure side
- Up to 40 berms on the passive earth pressure side
- Analysis with active and increased active earth pressure and with at-rest earth pressure
- Coefficients of active earth pressure pursuant to DIN 4085
- Coefficients of passive earth pressure pursuant to DIN 4085, Streck, Caquot/Kerisel
- Analysis of settlements according to DIN 4019
- Verification of external stability
- Verification of inner stability to EC 2/DIN 1045-1 or DIN 4093
- Verification of sliding safety, overturning safety and maximum geogrid forces

- Convenient interface to the stability analysis application, **GGU-STABILITY**, for quick determination of safety factor against general failure
- Seismic effects via additional horizontal loads and/or altered earth pressure coefficients
- Analysis of static equilibrium in the EQU limit state
- Analysis of pull-out resistance of geogrids via gamma(B)
- Up to 50 additional earth pressure distributions
- Up to 20 area loads at any depth
- Structural analysis of the wall by means of a two-dimensional rod construction module based on finite-element methods
- Specification of up to 5 displacement boundary conditions (rotation, displacement in x or y directions) at any location
- Specification of up to 5 action boundary conditions (moment, shear force and normal force) at any location
- A variety of different earth pressure redistribution figures:
  - no redistribution
  - rectangle
  - 2 rectangles
  - triangle (maximum can be optionally placed at top, middle or bottom)
  - trapezoidal
  - quadrilateral with maximum at anchor locations or any other location
  - user-defined polygon
- Passive earth pressure can be superimposed (***global safety factor concept***) or located in front of the wall.
- Following computation of the system, earth pressure, water pressure, moment, horizontal force, vertical force and the relationship $V \cdot \mu / H$ are automatically displayed on the screen. The visualisation can be varied within wide limits.
- Display of the pressure line, the 1^{st} and 2^{nd} kernel width and the gravity axis
- Legends can be displayed on screen indicating soil characteristics, general computation data, data needed for settlement calculations and the main design data. Thus, virtually all the raw data necessary for the computation are shown on screen.
- **GGU-GABION**'s user interface is based on WYSIWYG (***What You See Is What You Get***), which means that what you see on the screen is virtually identical with what is printed. It also means that you can print out what you see on the screen at any point during the analysis.
- The use of true-type fonts guarantees excellent layout.
- Colour presentation of virtually all system geometries. Colours can be freely determined by the user. In particular, soil strata can also be coloured according to the German DIN 4022 conventions.
- Zoom function
- **Mini-CAD** system (additional text, lines, rectangles, circles, graphics etc.)
- By clicking the "**Copy/print area**" icon on the toolbar you can copy any part of the graphics to the clipboard, or save it as an EMF file (Enhanced Metafile Format). Using the "**Mini-CAD toolbar**" or "**Header toolbar**" modules, you can paste emf files into your graphics. Thus, the results of a slope failure analysis or of a grading analysis, for example, can easily be imported into the current graphics.

3 Licence protection

GGU software is provided with the WIBU-Systems CodeMeter software protection system. For this purpose the GGU-Software licences are linked to an USB dongle, the WIBU-Systems CmStick, or as CmActLicense to the respective PC hardware.

It is required for licence access that the CodeMeter runtime kit (CodeMeter software protection driver) is installed. Upon start-up and during running, the program **GGU-GABION** checks that a licence on a CmStick or as CmActLicense is available.

4 Language selection

GGU-GABION 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

By going to the "**File**" menu, a previously analysed system can be loaded by means of the "**Load**" menu item, or a new one created using "**New**".

The program simplifies system input by moving directly to the dialog box after clicking "**New**", which you can also open using the menu item "**File/New**" (see section 8.1.1). Once you have finished in this box you can generate the desired wall using the "**Common systems**".

The system will then be displayed on the screen and you will see nine menus at the top of the window:

- File
- Editor 1
- Editor 2
- Reinforced soil system
- System
- Evaluation
- Graphics preferences
- Page size + margins
- 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-GABION** 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 First steps

6.1 Worked example 1: Gabion wall

6.1.1 System description (Ex. 1)

As, from personal experience, the reading of user-manuals is a chore, there will now follow a short description of the main program functions. After studying this section you will be in a position to carry out analyses. You can take the details of the program from the following chapters.

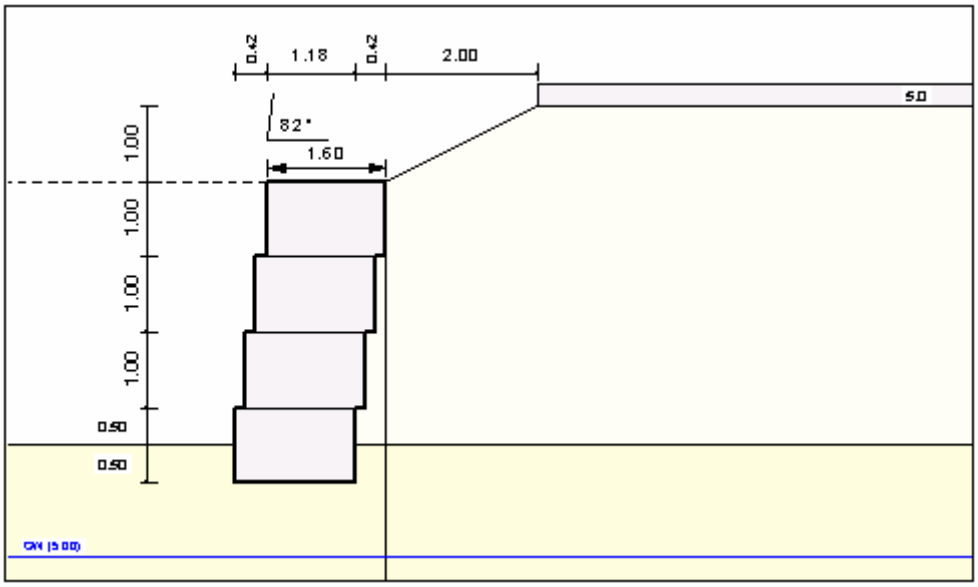


Figure 1 Example system 1

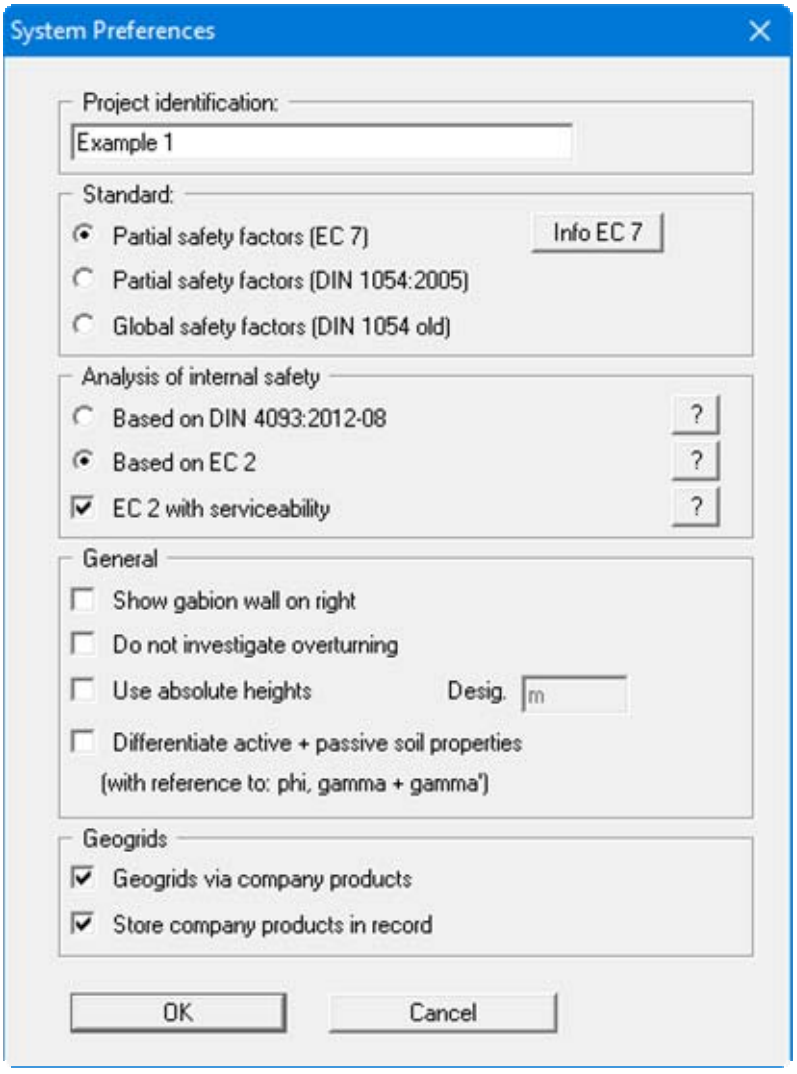
A 3 m high terrace with a 1 m high head berm, inclined at 1 : 2 is to be supported. A vertical load of 5 kN/m² acts on the berm. The angle of repose of the gabion elements is 82°. The design groundwater level is at 1.5 m below ground level (b GL). The backfill is a gravel-sand. Below this is a medium-compact sand. 4 gabion elements with a uniform height of 1.0 m each and a uniform width of 1.6 m are used. The following material properties are present:

- **Gabions**
  unit weight $\gamma = 18$ kN/m³
  friction coefficient in the joints $\mu = 0{,}6$
  design concrete compressive strength $f_{m,d}$ (EC 2) of the elements = 0,20 MN/m²
- **Gravel-sand**
  friction angle $\varphi = 35°$
  cohesion c = 0 kN/m²
  wet unit weight $\gamma = 18$ kN/m³
  wall friction angle (active) = 2/3 φ

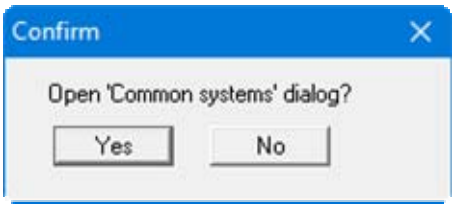
- **Sand**
  friction angle φ = 32,5°
  cohesion c = 0 kN/m²
  wet unit weight γ = 19 kN/m³
  buoyant unit weight γ ' = 10 kN/m³
  wall friction angle (active) = 2/3 φ
  no passive earth pressure used
  constrained modulus E_S = 60 MN/m²

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

After starting the program the logo is displayed. Select the menu item "**File**/**New**". The following dialog box will appear:



For this example, use the settings shown in the dialog box above and click "**OK**". First, another dialog box opens:

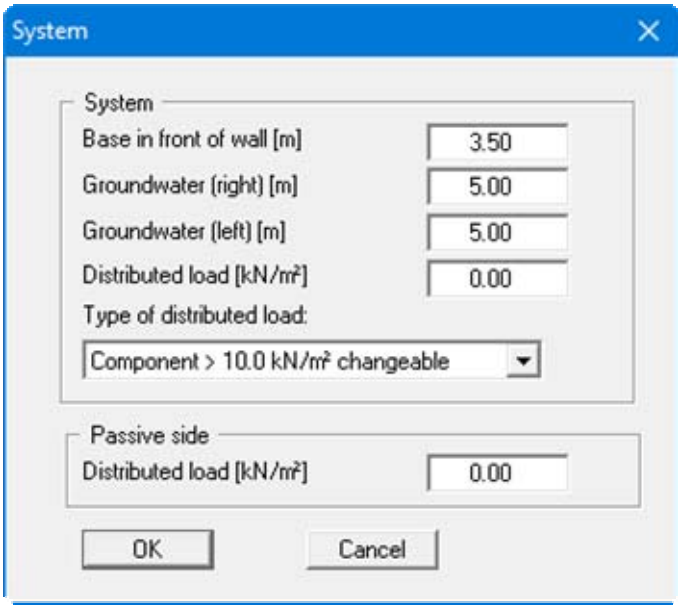


If you click "**Yes**", you can immediately generate the body's geometry (see section 6.1.5). However, for this example, click on "**No**" first.

A new system is then displayed on the screen and the complete menu bar is shown.

6.1.3 Step 2: Define the system (Ex. 1)

Select the menu item "**System**" from the "**Editor 1**" menu. The following dialog box will appear. Enter the figures shown below:

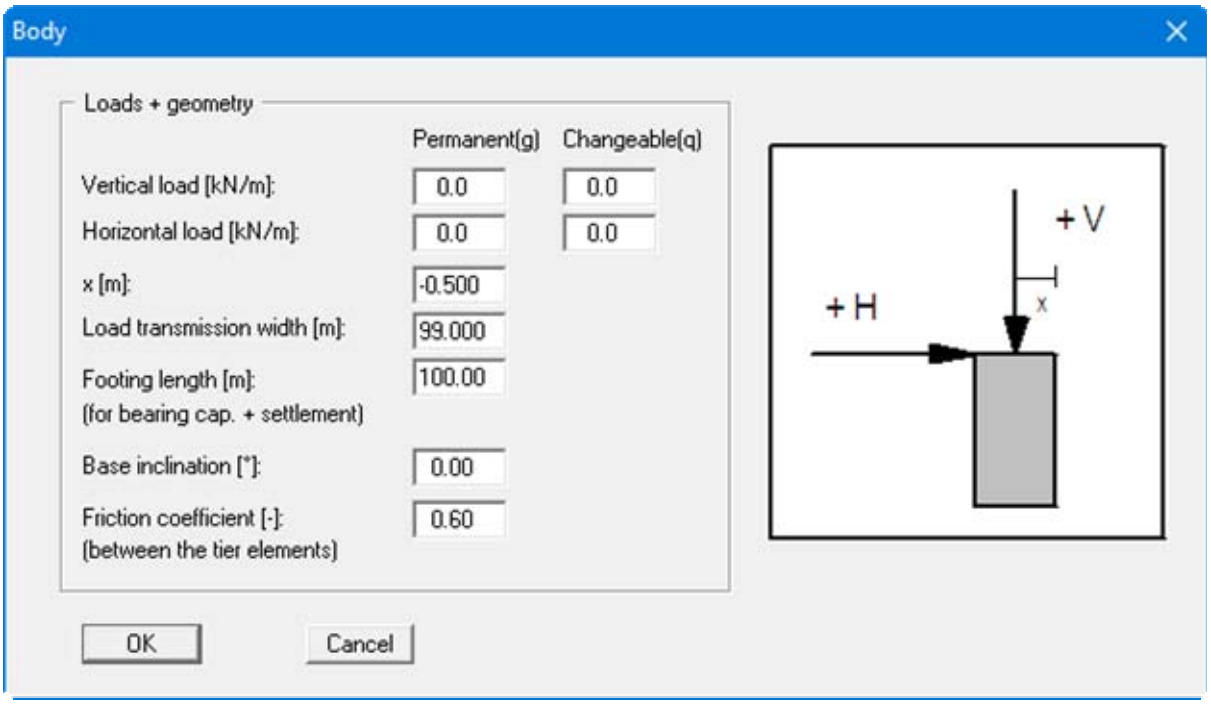


Because no absolute heights are used the top of the wall has a height of 0.0 m and all depths entered are positive downwards.

The 5 kN/m² live load for the example system is entered later using the menu item "**Editor 1/Berms (active side)**".

6.1.4 Step 3: Define body loads (Ex. 1)

Select the menu item "**Body (general)**" from the "**Editor 1**" menu.



Generally, gabions are not subject to a "**Vertical load**" or a "**Horizontal load**" at the top of the wall. In spite of this, values may be entered here if such loads do exist. Define the location of the vertical load from the top right edge of the gabion wall using "**x(load)**".

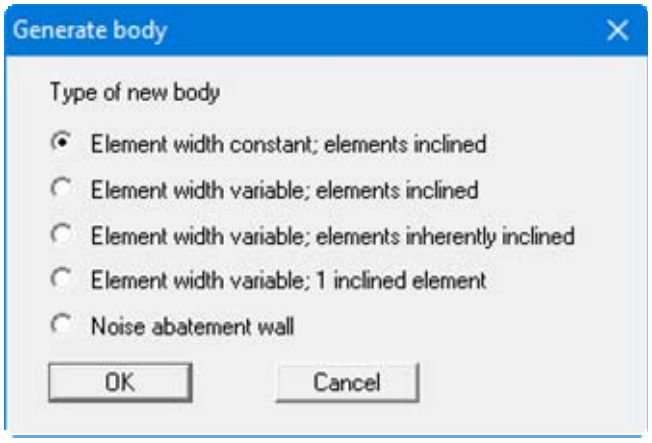
The "**Load transmission width**" is required for verification of the internal stability of the gabion wall to EC 2. If the load is not transmitted across the whole width, define the appropriate value. Otherwise, you may leave the default value of **99.0** unaltered.

The "**Footing length**" only influences the verification with regard to bearing capacity and settlements. You can also define any existing "**Base inclination**" in the above dialog box. The "**Friction coefficient**" between the tier elements (gabions) is required for verification of the internal sliding safety of the gabion wall.

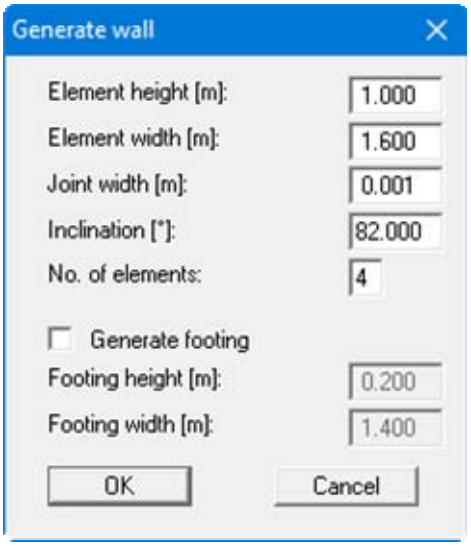
6.1.5 Step 4: Define body geometry (Ex. 1)

Select the menu item "**Editor 1/Body (geometry)**". Data input for body geometry is in horizontal sections. For this, you define the depth of the horizontal section and the two corresponding x values of the gabion wall in the dialog box (see section 8.2.4).

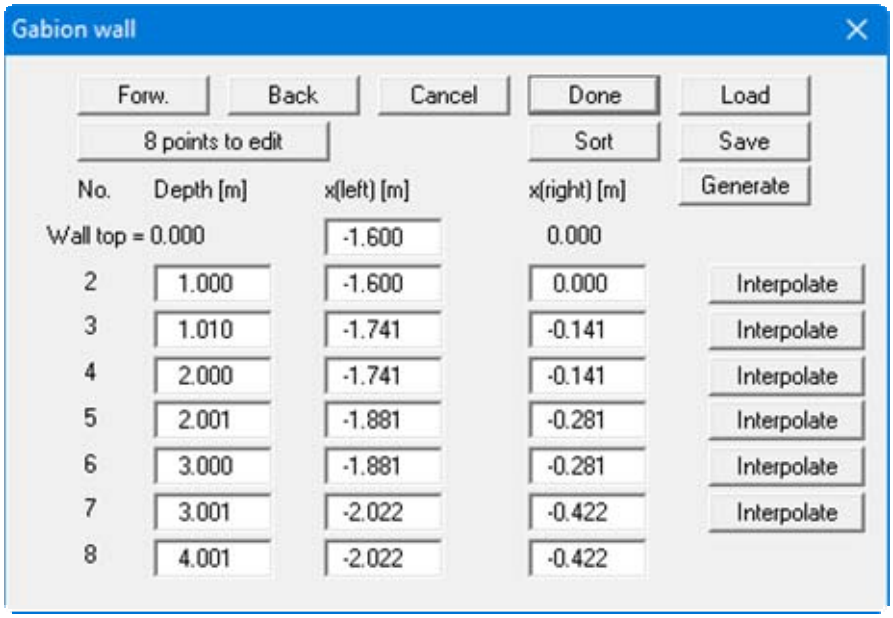
For most practical cases input can be automatically generated using the "**Generate**" button. When the "**Generate**" button is clicked, the following dialog box opens, allowing the type of elements to be specified.



For this example, confirm the default with "**OK**". A further dialog box opens.



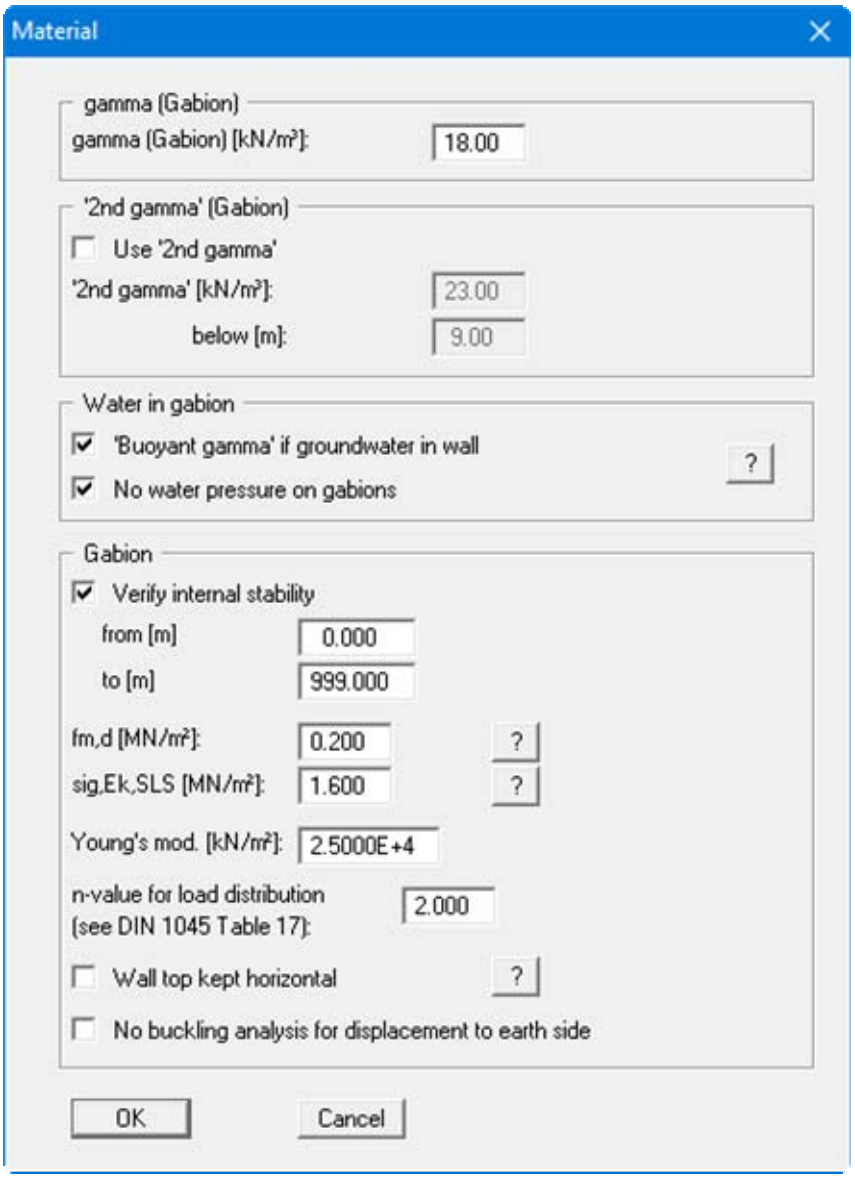
Once you have entered the data from the above dialog box and closed the box using "**OK**", the newly generated points are accepted.



The geometry of the gabion body has now been successfully entered. After closing the box using the "**Done**" button you will see the generated gabion elements represented on the screen.

6.1.6 Step 5: Define material of gabion (Ex. 1)

Select the menu item "**Editor 1/Material**" and enter the values shown in the dialog box.



The program allows you adopt user-defined unit weights to be used in the analysis, if the gabion wall rests on a footing, for example. It is also possible to consider a "**Buoyant gamma**" if the gabion wall is constructed within groundwater. "**Young's mod.**" designates the Young's modulus of the gabion material. This value is required for the structural analysis of the wall.

Using existing design practice the internal stability of a gabion wall is often incorrectly analysed. Failure cases show, however, that improvements are required in this area. The paper therefore contains the demand:

"The internal stability of the gabion elements of the wall is to be verified numerically, or via loading tests.

The inner stability of an element is influenced mainly by

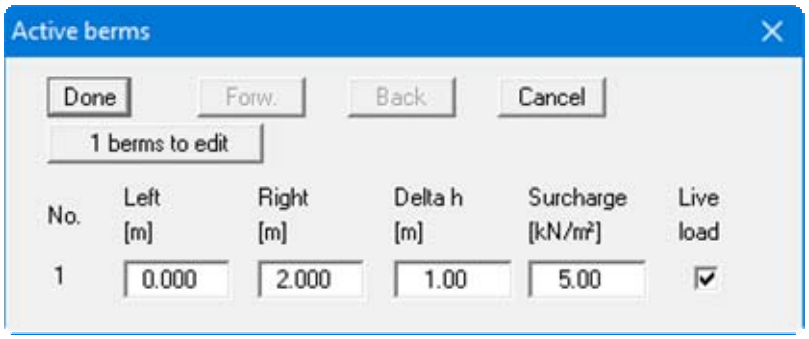
- *the fill material and*
- *the wire mesh.*

A reliable numerical determination of inner stability is barely possible, due to the complexity of interactions between the fill material and the wire mesh. If necessary, the verification concept of the DIN 1045 can be used, if an equivalent value for the design value of the concrete compressive strength of the gabion elements (fm,d) can be determined by modelling tests."

The program allows analysis of the internal stability to EC 2/DIN 1045-1 and DIN 4093 using this concrete compressive strength. Values for "**fm,d**" (design strength of the concrete) and "**n-value for load distribution**" are required here (see detailed explanation in section 7.25).

6.1.7 Step 6: Define berm (Ex. 1)

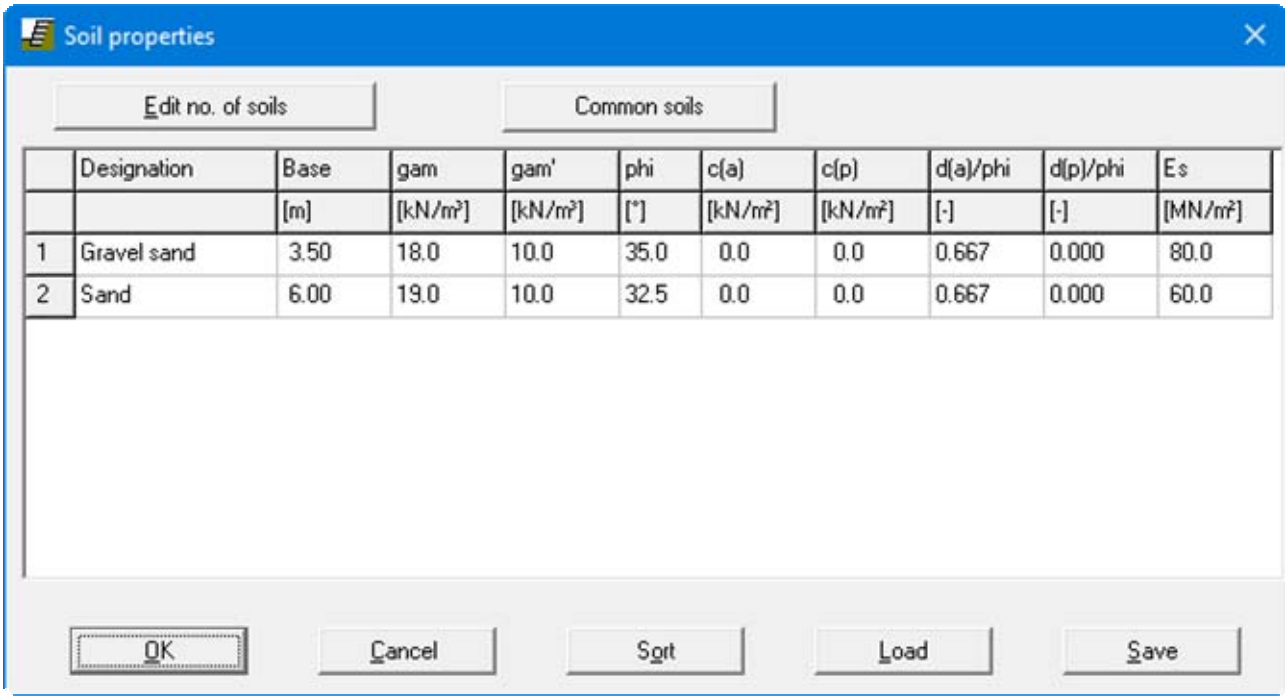
From the "**Editor 1**" menu select "**Berms (active side)**". Click "**0 berm(s) to edit**" and enter **1** as the new number of berms.



Enter the values from the above dialog box and confirm by clicking "**Done**".

6.1.8 Step 7: Define soils (Ex. 1)

Select the menu item "**Soils**" from the "**Editor 1**" menu. Edit the number of soils by selecting the "**Edit no. of soils**" button and then entering the number ***2***.

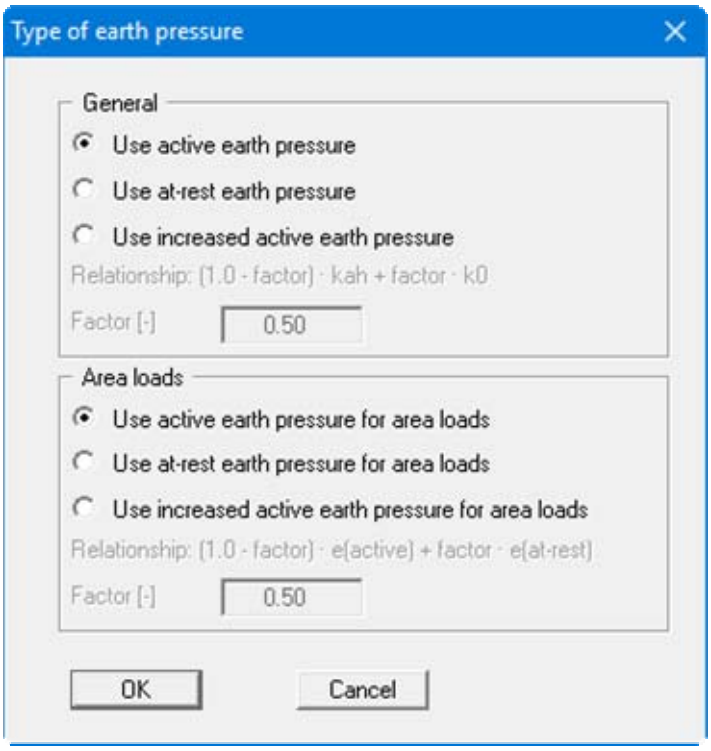


	Designation	Base	gam	gam'	phi	c(a)	c(p)	d(a)/phi	d(p)/phi	Es
		[m]	[kN/m³]	[kN/m³]	[°]	[kN/m²]	[kN/m²]	[-]	[-]	[MN/m²]
1	Gravel sand	3.50	18.0	10.0	35.0	0.0	0.0	0.667	0.000	80.0
2	Sand	6.00	19.0	10.0	32.5	0.0	0.0	0.667	0.000	60.0

Enter the values given in the above dialog box. You can enter differing cohesion and angles of friction for the active (a) and the passive sides (p).

6.1.9 Step 8: Define type of earth pressure (ex. 1)

Select the menu item "**Type of earth pressure**" from the "**Editor 1**" menu.

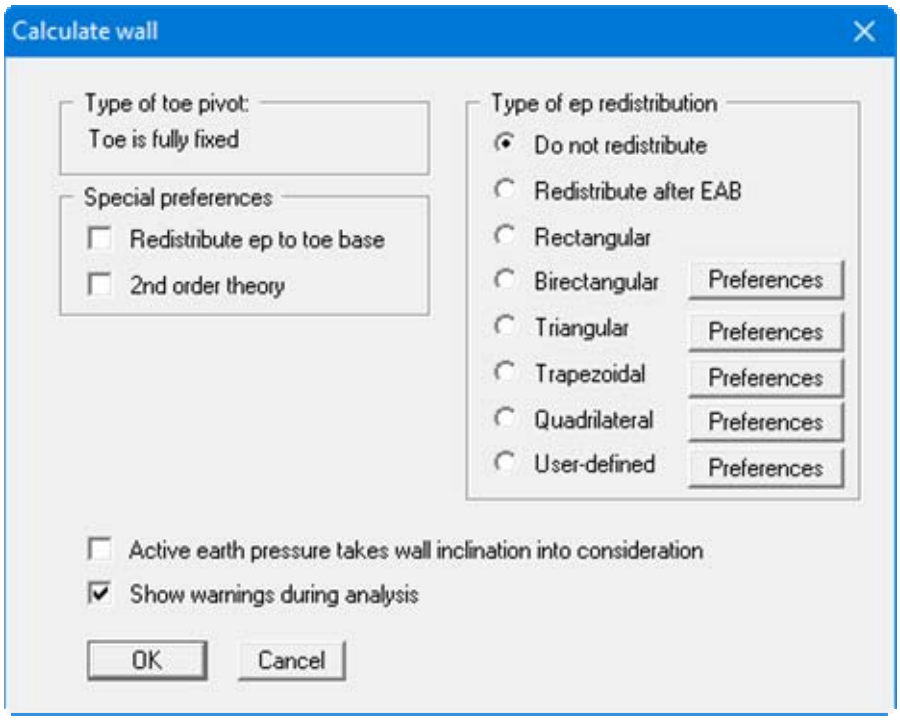


The necessary command buttons are already selected, so you need not change anything. The same applies to the remaining menu items in "**Editor 1**". However, you should click these items and take a look at them, in order to familiarise yourself with them.

This completes data input.

6.1.10 Step 9: Analyse system (Ex. 1)

Now select the "**Analyse**" menu item from the "**System**" menu.



After clicking the "**OK**" button the analysis begins. The program first performs an intensive plausibility check of the data input. Once the analysis is complete the system is displayed on the screen, together with the results. Besides the action effects the following are shown:

- Pressure line:
  Location of the resultant in each section
- 1st kernel width:
  If the pressure line is within the 1st kernel width the plan area is completely under pressure => compressive loading only.
- 2nd kernel width:
  If the pressure line is outside of the 1st kernel width but inside of the 2nd kernel width, a foundation gap occurs to a maximum of the position of the centroid.
- 'n-region':
  This designates the governing design area to EC 2/DIN 1045-1, which results from the n-value and the load transmission width.

6.1.11 Step 10: Evaluate and visualise the results (Ex. 1)

Amongst other things, four legends are displayed on the screen, containing the soil properties, the most important analysis data, analysis results and the results of the settlement analysis.

If you don't like the legend positions, they can be easily altered. Go to the "**Move objects**" menu item in the "**Graphics preferences**" menu and follow the instructions there. The size of the legends can be altered by changing the font sizes in the individual dialog boxes (see "**Graphics preferences**" menu).

Besides the system, the earth pressures, action effects, etc. are displayed for the whole height of the wall. These entries can also be edited. Go to the "**Graphics output preferences** " menu item in the "**System**" menu. Further evaluations can be displayed by going to the "**Evaluation**" menu.

The diagram can be printed on the selected printer ("**File/Print and export**" menu item). You can also print off a detailed protocol ("**File/Print output table**" menu item). The zoom function (see "**Graphics preferences/Zoom info**" menu item) allows you to magnify selected areas of the graphic. Double-clicking in the graphics at a particular point will cause a box to appear containing the corresponding state variables.

To further clarify the analysis results the graphics can be supplemented with text and graphic elements using the "**Mini-CAD**" module. You can save your work to a file. Go to the "**Save as**" menu item in the "**File**" menu.

6.2.1 System description (Ex. 2)

A noise abatement wall with an overall height of 4 metres on level ground will be analysed. Four gabion elements with a uniform height and width of 1.0 m x 1.0 m are used. The noise abatement wall is built on a 1.4 m wide footing, embedded 0.2 m in medium dense sand. The design ground-water table is at ground level. A wind load of 1.0 kN/m² is adopted.

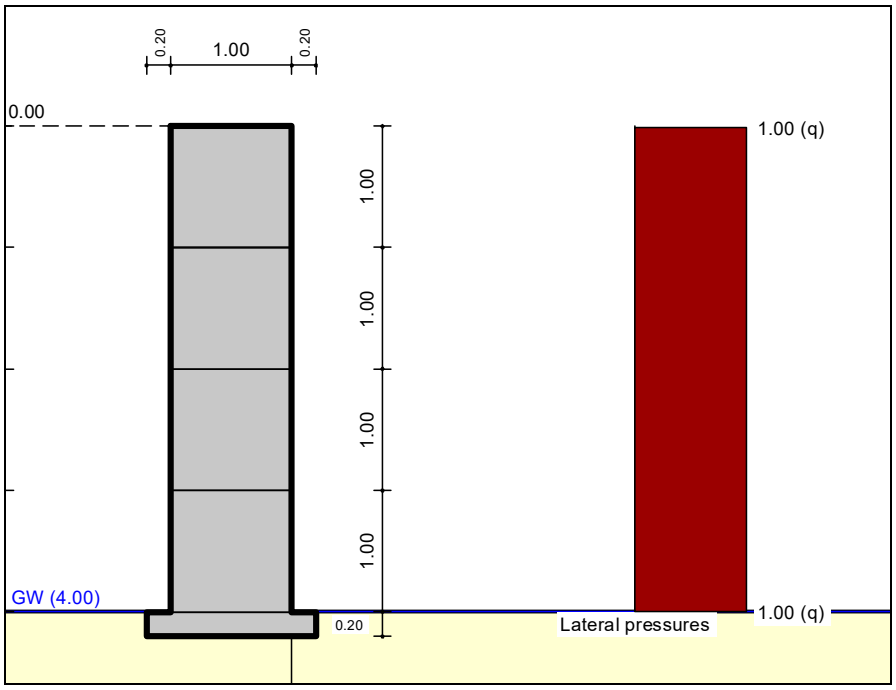


Figure 2 Example system 2

The following material properties are present:

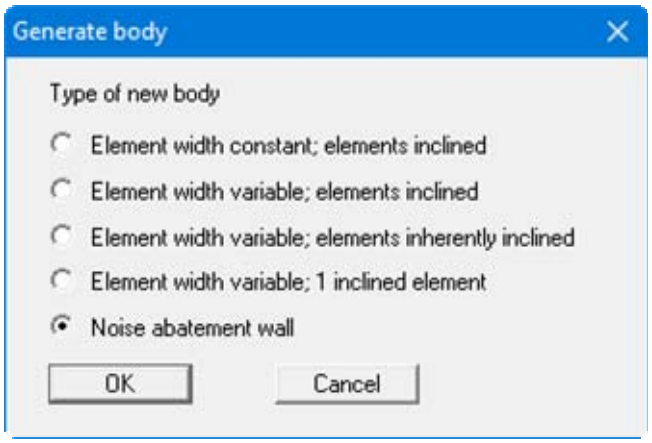
- **Gabions**
  unit weight γ = 16 kN/m³
  friction coefficient in the joints μ = 0,75
  design concrete compressive strength $f_{m,d}$ (EC 2) of the elements = 0,20 MN/m²
- **Sand**
  friction angle φ = 32,5°
  cohesion c = 0 kN/m²
  wet unit weight γ = 19 kN/m³
  buoyant unit weight γ' = 10 kN/m³
  no passive earth pressure used

Due to the wind loads the active wall friction angle must be set to zero to analyse noise abatement walls. Otherwise, the wind load would generate a favourably acting active earth pressure at the rear face of the wall.

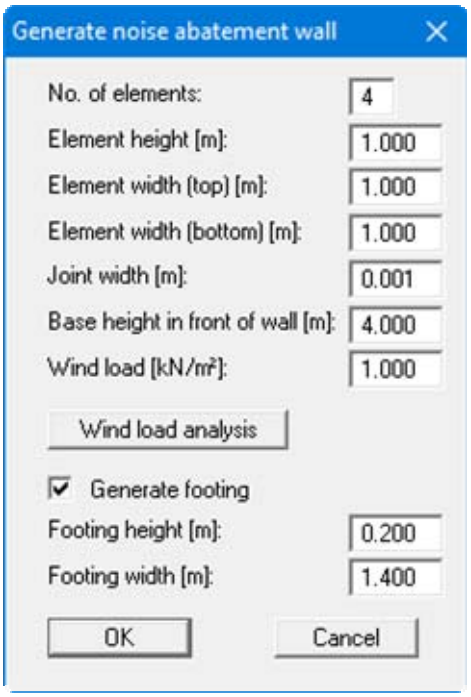
6.2.2 Step 1: Select analysis options and generate noise abatement wall (Ex. 2)

After starting the program you will see the **GGU-GABION** start-up screen. Select the "**File/New**" menu item and accept all presettings. If you prefer, you can enter a project identification, e.g. "**Example 2**".

After leaving the dialog box, you will again be queried on "**Common systems**". For this example, click the "**Yes**" button. Alternatively, you can go to the "**Editor 1/Body (geometry)**" menu item and click on the "**Generate**" button there. In both cases the following dialog box opens, in which you select "**Noise abatement wall**" as the type of new body.



After confirming by pressing "**OK**" you will see another dialog box, in which you use the following settings to automatically generate the noise abatement wall.



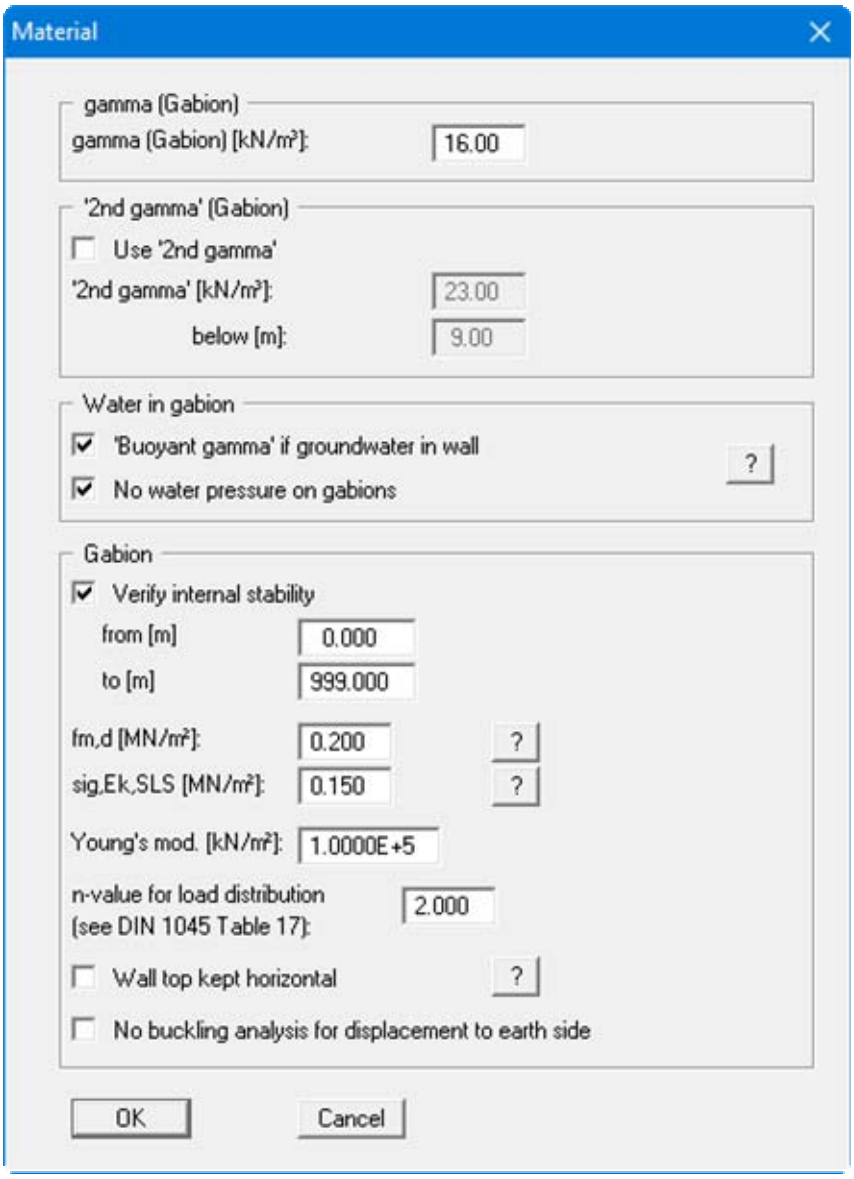
The "**Base height in front of wall**" is calculated from the top of the wall (= 0.0 m) and is positive downwards. An active berm of 4.0 m is generated in the noise abatement wall's active earth pressure zone in order to simulate the free-standing wall.

The wind load entered in the dialog box is taken into consideration by the system as a surcharge load. (see "**Editor 2 / Lateral pressures**" menu item, section 8.3.4) To prevent a favourably acting active earth pressure being generated at the rear face of the wall, the program automatically sets the active wall friction angle to zero.

After confirming your input by pressing "**OK**" you then move to the body geometry editor box and exit this box by pressing "**Done**". The entered system is now displayed on your screen. System visualisation can be optimised by pressing the [**F9**] function key.

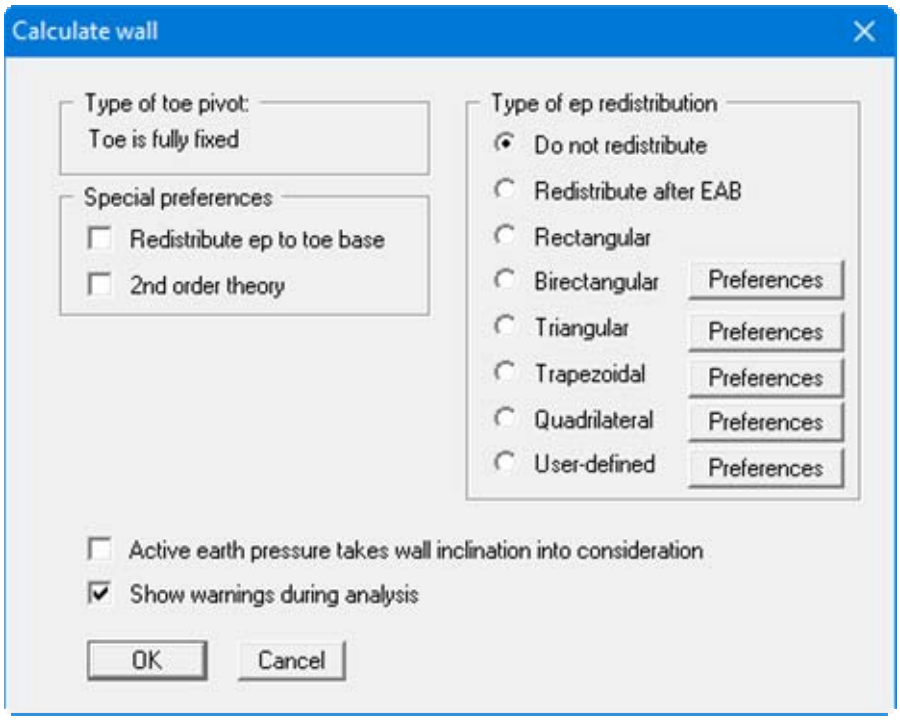
6.2.3 Step 2: Define material of gabion (Ex. 2)

Select the "**Editor 1/Material**" menu item and adapt the unit weight of the gabion elements.

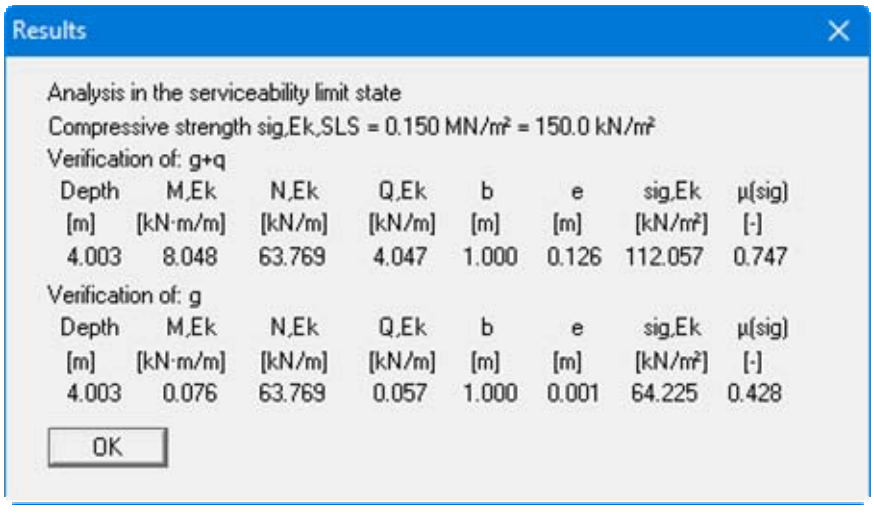


6.2.4 Step 3: Analyse system (Ex. 2)

Now select the "**Analyse**" menu item from the "**System**" menu.



After clicking the "**OK**" button the analysis begins. The program first performs an intensive plausibility check of the data input and returns the appropriate information. Once analysis is complete the results are shown in a message box.



After closing the message box the system is displayed on the screen, together with the results.

7 Theoretical principles

7.1 Body

The body is defined in horizontal sections of three values each:

- depth of cross-section below top of the wall;
- corresponding left and right wall boundary.

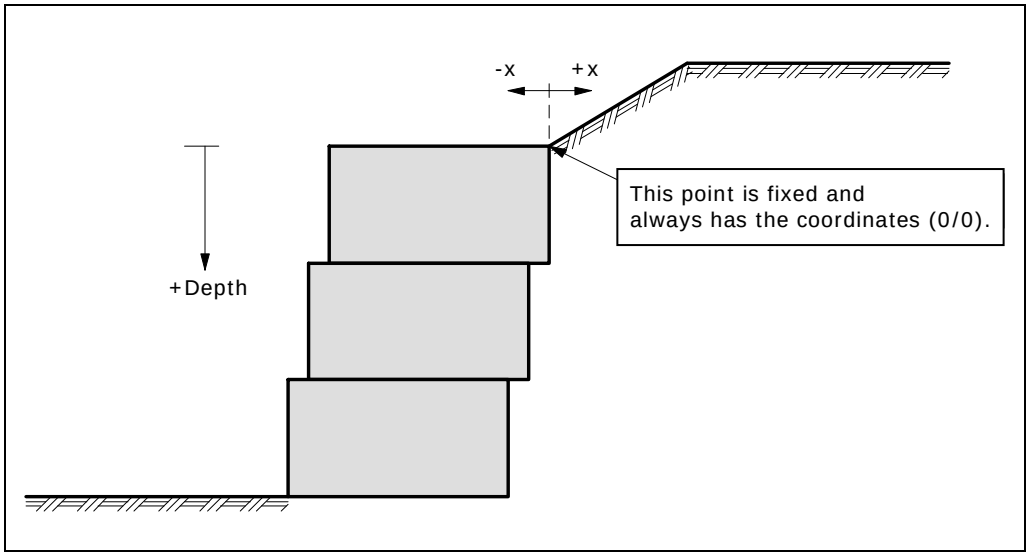


Figure 3 Gabion body

In order to define this body, you must enter the depths and the two corresponding x-values in six horizontal steps (also see section 8.2.4). All earth pressures (including from area loads) are calculated on the vertical line at x = 0.

7.2 Soil properties

Up to 50 soil layers can be taken into consideration. For each layer the following parameters must be given:

- Depth in metres below top of the wall, or absolute depth;
- Unit weight [kN/m³] of moist soil γ ;
- Unit weight [kN/m³] of buoyant soil γ' ;
- Friction angle [°];
- Cohesion (active and passive) [kN/m²];
- Active angle of wall friction as ratio δ_a/φ ;
- Passive angle of wall friction δ_p/φ ;
- Constrained modulus E_s [kN/m²] (for settlement analysis).

If you activate the "**Differentiate active + passive soil properties**" check box in the dialog box in "**File/New**" or "**Editor 1/Analysis options**", you can enter differing friction angles and unit weights for the active and the passive sides.

The program provides an expandable database with soil properties for common soils. The database can be opened via the "**Common soils**" button in the "**Editor 1/Soils**" editor box (see section 8.2.8).

7.3 Active earth pressure

Active earth pressure is analysed to DIN 4085. DIN 4085 provides two relationships for the coefficients of earth pressure k_{ah} (friction) and k_{ch} (cohesion). Alternatively, there is the option of determining the cohesion coefficient from $k_{ch} = k_{ah}^{-2}$, a method often found in older literature.

7.4 At-rest earth pressure

The at-rest earth pressure coefficient is calculated compliant with DIN 4085:2011.

7.5 Increased active earth pressure

The coefficient of increased active earth pressure, k_{eh}, is obtained from the coefficient of active earth pressure and at-rest earth pressure:

$$\mathbf{k_{eh} = (1.0 - f)\ k_{ah} + f\ k_0}$$

$$0.0 \le f \le 1.0$$

7.6 Passive earth pressure

The coefficient of passive earth pressure can be calculated using a number of methods:

- DIN 4085:2017;
- DIN 4085:1987;
- Streck,
- Caquot/Kerisel,
- DIN 4085:1987/Caquot/Kerisel.

7.7 Berms

GGU-GABION can handle berms on both the active and the passive sides of the wall. The berms may include surcharges. The effect on earth pressure is taken into consideration according to the Piling Handbook (Krupp Hoesch Stahl).

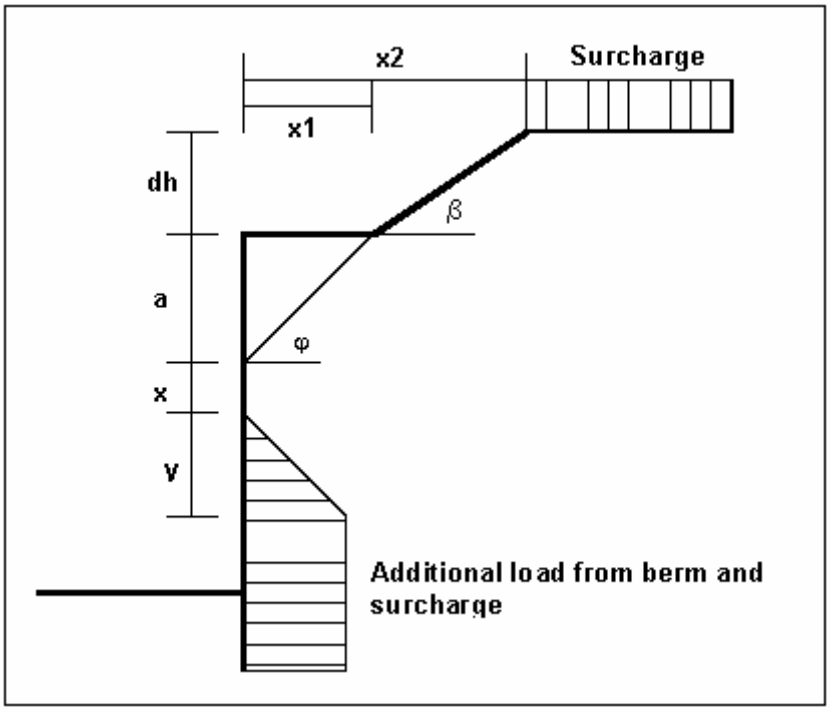


Figure 4 Berm on the active side

The following relationships apply for the parameters x and y:

$$x = k_{ah0} / (k_{ah\beta} - k_{ah0}) \cdot a$$

$$y = k_{ah0} / (k_{ah\beta} - k_{ah0}) \cdot x$$

$$\Delta e_{ahu} = \gamma \cdot dh + surcharge$$

γ = unit weight of soil in the berm area

If the angle β is greater than φ, it is assumed that $\beta = \varphi$ for analysis. Berms on the passive side are dealt with in exactly the same manner.

Up to 20 area loads can be positioned on the active side at any height.

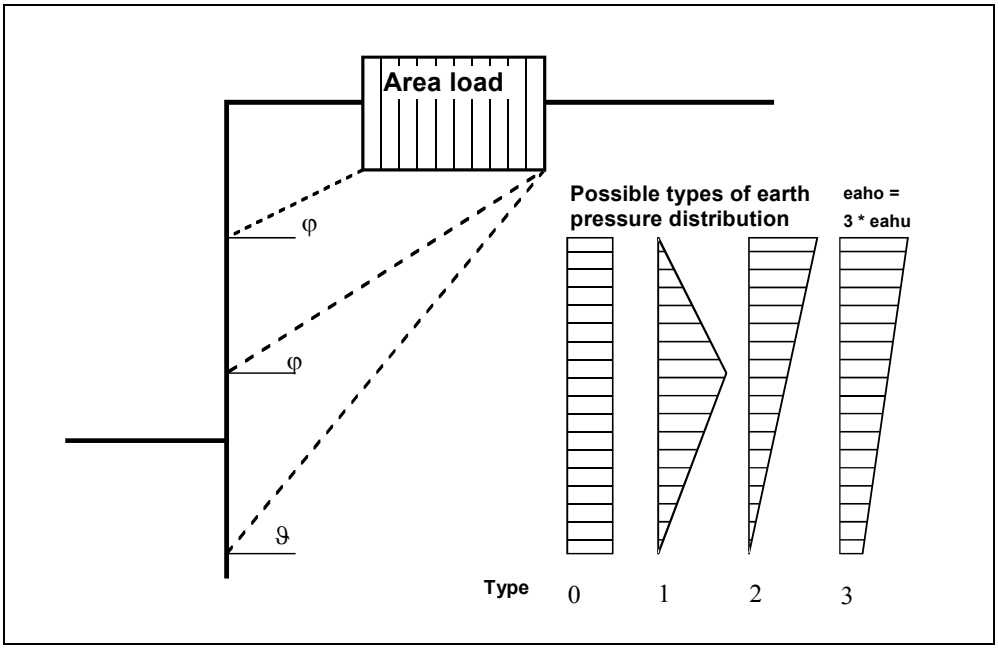


Figure 5 Area load

The slip surface angle for the active earth pressure resulting from the self-weight of the soil is adopted for analysis compliant to DIN 4085.

$$\vartheta_{ag} = \varphi + \arctan\left(\frac{\cos(\varphi - \alpha)}{\sin(\varphi - \alpha) + \sqrt{\frac{\sin(\varphi + \delta_a) \cdot \cos(\alpha - \beta)}{\sin(\varphi - \beta) \cdot \cos(\alpha + \delta_a)}}} \right)$$

When there are a number of soil layers, **GGU-GABION** moves from layer to layer applying the appropriate angles of friction. The type of the resulting earth pressure distribution can be specified in 4 different ways.

For at-rest earth pressure, the area loads are calculated by increasing by the factor k0/kah in compliance with DIN 4085:2011 section 6.4.3.

Alternatively, it is possible to determine the demands placed on a wall from area loads using the theory of elastic half-space. The two load concentration factors '**3**' and '**4**' can be taken into consideration (see also Figure 6).

- For over consolidated, cohesive soils the concentration factor '**3**' applies, where:
  $e_{op} = q/\pi\ (\beta_2 - \beta_1 + \cos\beta_1 \sin\beta_2 - \cos\beta_2 \sin\beta_2)$
- For non-cohesive soils, or for cohesive but not over consolidated soils, the concentration factor '**4**' applies, where:
  $e_{op} = q/4\ (\sin^3\beta_2 - \sin^3\beta_1)$

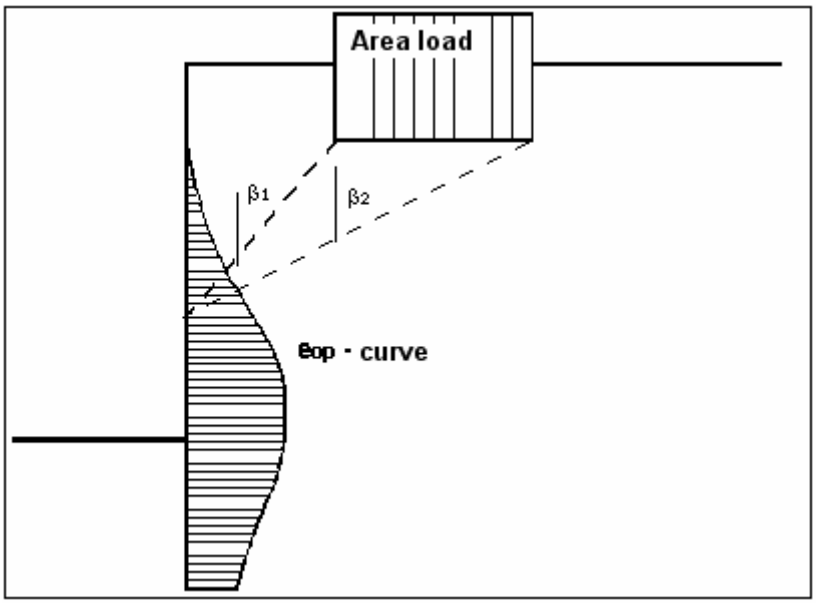


Figure 6 At-rest earth pressure from area loads

With regard to the kind of earth pressure, area loads can be defined independent of the global preferences (see menu item "**Editor 1/Type of earth pressure**", section 8.2.9).

7.9 Line loads

Line loads perpendicular to the wall axis (for example, from loads exerting a vertical force on the wall) are treated as shown in Fig. 4.20 on page 64 of the Piling Handbook 1977 (*Spundwand-Handbuch 1977*). Data is entered in the form of a number of discrete area loads (see section 8.3.1).

7.10 Bounded surcharges (active side)

Up to 40 bounded surcharges can be positioned at any height on the active side.

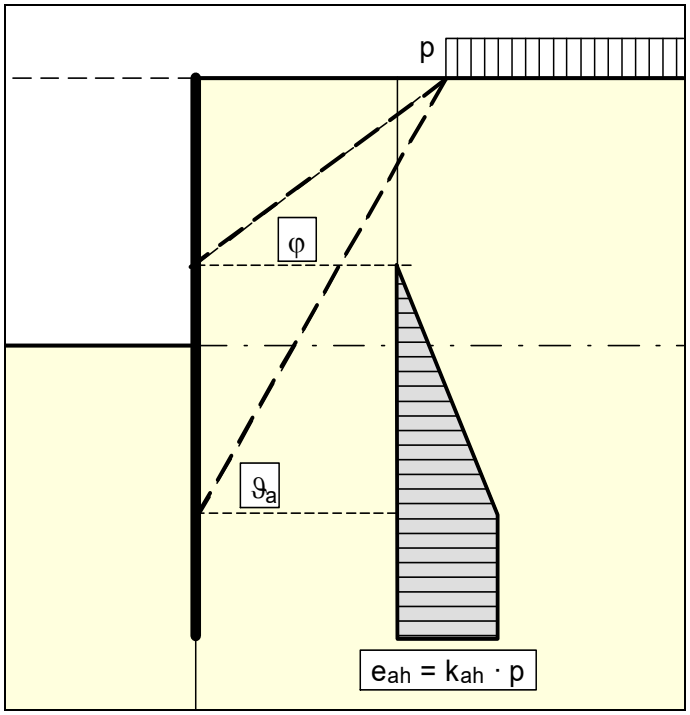


Figure 7 Bounded surcharge (active side)

The earth pressure coefficient k is acquired from k_{ah} for active earth pressure and from k_0 for at-rest earth pressure. If this option is activated, the resulting earth pressure is then redistributed.

If negative values are entered, e.g. in order to generate a double-bounded surcharge, the linear component between φ and ϑ may not be adopted.

7.11 Double-bounded surcharges (active side)

If two bounded surcharges are defined as follows:

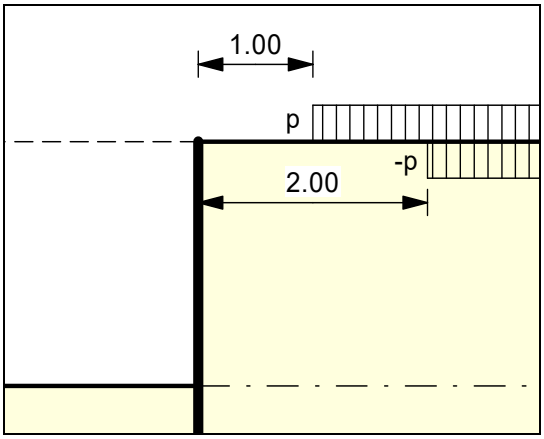


Figure 8 Two bounded surcharges

- Bounded surcharge 1:
  begins at x = 1,00 m and has the value p.
- Bounded surcharge 2:
  begins at x = 2,00 m and has the same value as surcharge 1 but with a negative sign
  (see the explanation on the previous page, section 7.10).

A double-bounded surcharge, which acts as a consequence of p at x = 1.00 to x = 2.00, is thus defined as the result of earth pressure analysis. However, the graphics are not really satisfactory and may be irritating for an examiner. Double-bounded surcharges were therefore introduced.

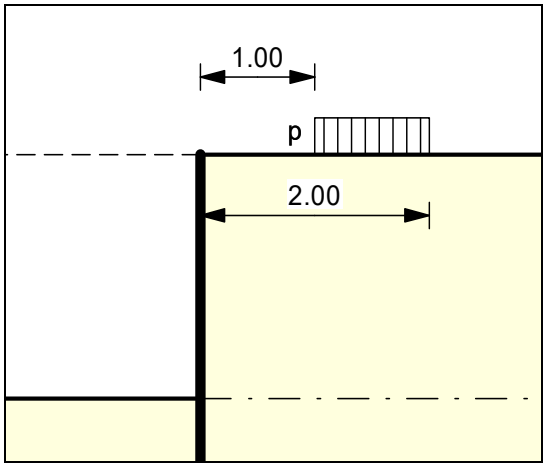


Figure 9 Double-bounded surcharge

The same result is achieved as for the definition provided by Figure 8 but with improved graphics.

The area loads described in section 7.8 are based on different assumptions for the resulting earth pressure, so the definition of a double-bounded surcharge does not provide the same result as an equivalent area load.

7.12 Bounded surcharges (passive side)

Up to 40 bounded surcharges may be adopted at any height on the passive side. The passive earth pressure is computed as follows:

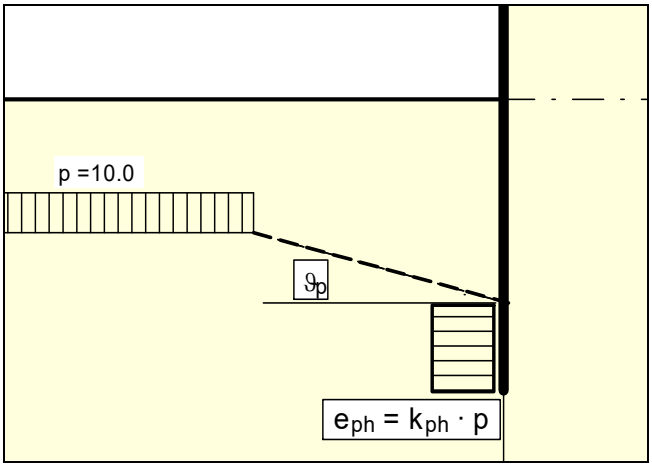


Figure 10 Bounded surcharge (passive side)

7.13 Double-bounded surcharges (passive side)

Double-bounded surcharges on the passive side can be modelled using two bounded surcharges (also see section 7.11). Double-bounded surcharges on the passive side were only implemented due to the better graphical representation.

7.14 Structural system

The structural analysis required to determine the state variables (displacement, moment, shear force and normal force) is by means of a rod construction module, which treats the wall and any geogrids as a single structural system. The wall is taken into consideration as a rod construction, running along the centroid of the wall.

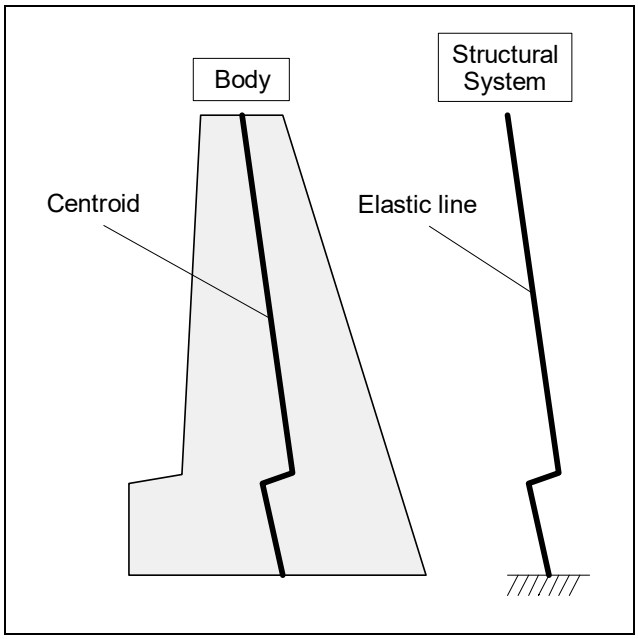


Figure 11 Structural system of a wall without geogrid

The stiffness EI of the wall is acquired from $E \cdot d^3/12$, where d is the horizontal thickness of the wall in the section under consideration. The stiffness of each element is assumed as constant, where the thickness is the thickness at the centre of the section.

If geogrids exist in the system, the point of acting is automatically placed in the centroid axis. The force components acting on the front and rear of the wall, such as soil weight, water pressure, loads, etc. produce a moment around the centroid axis, which **GGU-GABION** takes into consideration as a continuously distributed moment load on the rod axis.

The theoretical principles of the rod construction module can be traced back to an article by Duddeck/Ahrens (*Betonkalender* 1976 (Concrete Calendar), Volume 2). Basically, this is a finite element method based on the displacement method. The division of the wall and the struts into rod sections (finite elements) can be controlled by input (see "**System/Depth subdivisions**" menu item in section 8.5.3).

Using FEM produces equation systems in which the number of unknown is a function of the number of rods. The solution to the equation system is acquired in **GGU-GABION** using Cholesky's method, which is also employed in other GGU programs and is numerically highly stable. Nor have there been any numerical difficulties reported from other GGU applications.

Amongst the results of the rod construction analysis are the shear forces Q and the normal forces N with regard to the rod axis. In a post-processing calculation these forces are converted to the horizontal force H and the vertical force V.

7.15 Base inclination

Base inclination is generally carried out to increase the sliding safety to the required level.

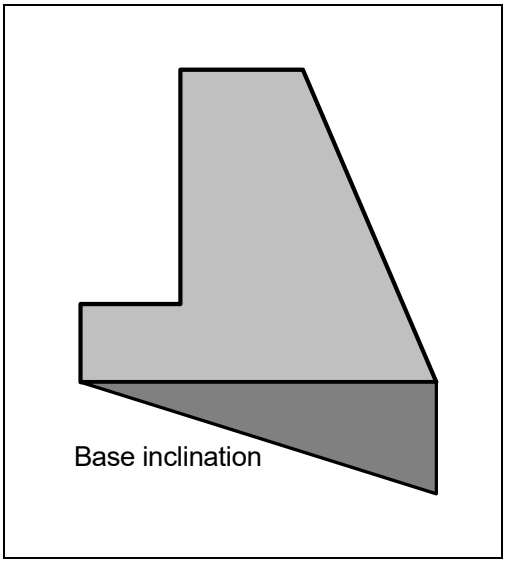


Figure 12 Base inclination

The idealisation of the body without base inclination to an equivalent structural system (elastic curve) presents no problems, because the stiffness of the individual partial calculations can be calculated from the horizontal thickness. However, the wedge shaped extension that normally occurs with base inclination is contradictory to the demands which arise when the wall is analysed as an elastic curve.

In order to disqualify contradictory results from the outset, the base inclination is **not** taken into consideration when determining the action effects. Only when verifying the ***safety against sliding*** are the active and passive earth pressures at the left and right of the wedge cumulated and this component allocated to the horizontal force when verifying the safety against sliding. When verifying the ***bearing capacity safety*** (see section 7.22.1), base inclination is taken into consideration. The additional horizontal force component is **not** incorporated in the analysis, leading to a more conservative result.

7.16 Bedding conditions of wall toe

For systems without geogrids the wall toe is fully fixed. In systems using geogrids, the wall toe can either be defined as fixed, freely pivoted or with an embedded base.

7.17 Action and displacement boundary conditions

GGU-GABION automatically sets the boundary conditions described in section 7.14 (Structural system). It is also possible to enter additional boundary conditions for any point on the wall. All six state variables can be defined as boundary conditions:

- displacement along x-axis;
- displacement along y-axis;
- rotation;
- horizontal force;
- vertical force;
- moment.

7.18 Earth pressure redistribution

According to ***EBGEO***, there is no requirement for redistribution of earth pressure. It is nevertheless possible to select from a large variety of redistribution figures. In addition, by defining a polygon, the user can create any redistribution figure. The figures available are given below:

- rectangle;
- 2 rectangles;
- triangle, maximum at top;
- triangle, maximum at centre;
- triangle, maximum at base;
- trapezoid;
- rectangle with maximum at anchor locations or at any point;
- user-defined redistribution figure by means of a polygon;
- EAB redistribution figures.

GGU-GABION gives you the option of including or excluding area loads in the redistribution.

7.19.1 Verifications

When you begin an analysis, the program requires the stiffnesses of the wall. The stiffness EI of the wall is acquired from $E \cdot d^3/12$, where d is the thickness of the wall in the section under consideration. The stiffness of each element is assumed as constant, where the thickness is the thickness at the centre of the section.

Once the structural analysis is complete the program performs the necessary verifications. If any verifications are not possible you will see an appropriate warning. The following verifications can be differentiated:

- verification of internal stability;
- verification of external stability;
- verification of serviceability limit state.

These are further differentiated:

- systems without reinforcement (geogrid) and
- systems with reinforcement (geogrid).

If, for whatever reason, you would sooner do without any of the verifications listed here, go to the "**Editor 1/Verifications/Safety factors**" menu item or the "**Editor 1/Verifications/Partial factors**" menu item and select the required verifications. The settlement analysis can be activated or deactivated by going to "**Editor 2/Settlements**".

7.19.2 Systems without geogrids

External stability

The following verifications must be performed:

- Sliding safety in the base plane (EC 7/DIN 1054):
  The verification is performed using the soil properties at the base plane
  (see section 7.20.1).
- Bearing capacity safety (DIN 4017):
  The verification is performed using the soil properties below the base plane
  (see section 7.22.1).
- General stability (DIN 4084):
  This verification can be performed using the **GGU-STABILITY** program after exporting the data (see "**File/Export**" menu item, section 8.1.6).

Internal stability

The internal stability analysis must verify that no material incompatibilities or overloading arises within the gabion wall. The following verifications must be performed:

- Sliding safety in tier joint planes (EC 7/DIN 1054, see section 7.20.2);
- Verification of gabions (EC 2/DIN 1045, see section 7.25.2, or DIN 4093, section 7.25.3).

Serviceability

The verifications of serviceability consist of:

- Verification of allowable eccentricity of the resultant in the base plane;
- Settlement considerations based on DIN 4019 (see section 7.23).

7.19.3 Systems with geogrids

Systems with geogrids are considered as reinforced soil systems with a retaining gabion wall.

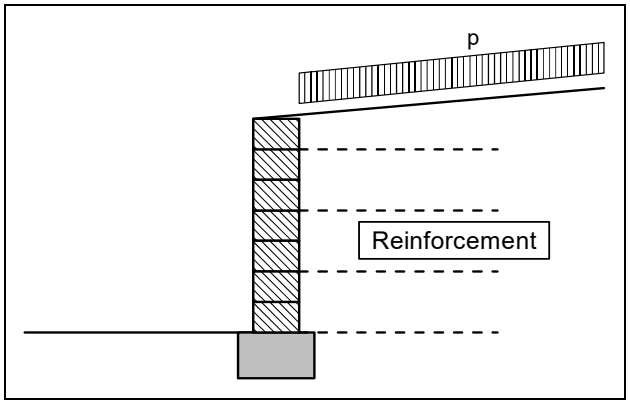


Figure 13 Wall with geogrid

Verification of the overall reinforced soil systems is based on section 6.7 of the Recommendations for Reinforcements using Geosynthetics (***Empfehlungen für Bewehrungen aus Geokunststoffen - EBGEO***). This requires the following verifications for the reinforced soil system:

Verification of the bearing capacity of reinforced soil systems

- Sliding safety verification (see also section 7.21.2);
- Bearing capacity safety verification (see also section 7.22.1);
- General stability verification (see also section 8.1.6);
- Verification of the bonding of the reinforcement with the gabion wall;
- Verification of internal stability of the reinforced soil system (see section 7.21.1).

Verification of the serviceability of reinforced soil systems

- Verification of allowable eccentricity of the resultant in the base plane of the reinforced soil system;
- Reinforced soil system settlement considerations based on DIN 4019 (see section 7.23).

Because the gabion wall can be of considerable thickness, in contrast to those systems dealt with in the ***EBGEO***, and because they may also possess a strip footing, bearing capacity and service-ability verifications are also required:

External stability (bearing capacity) of the gabion wall

- Sliding safety in the base plane (EC 7/DIN 1054, see section 7.20.1);
- Bearing capacity safety (DIN 4017, see section 7.22.1).

Internal stability (bearing capacity) of the gabion wall

- Sliding safety in tier joint planes (EC 7/DIN 1054, see section 7.20.2);
- Verification of gabions (if necessary to EC 2/DIN 1045-1, see section 7.25.2);
- Verification of the bonding of the reinforcement with the gabion wall.

Verification of the serviceability of the gabion wall

- Verification of allowable eccentricity of the resultant in the base plane;
- Settlement considerations based on DIN 4019 (see section 7.23).

7.20 Sliding safety to EC 7

7.20.1 Sliding safety in the base plane

The sliding safety is calculated using:

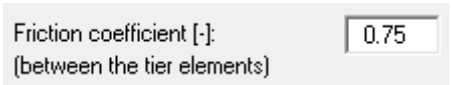
$$\eta = V_k \cdot \tan \varphi / H_d$$

V_k = characteristic vertical force
H_d = design value of horizontal force

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

7.20.2 Sliding safety in the tier joint planes

Enter a friction coefficient for verification of the sliding safety in the tier joint planes in the dialog box that opens in the "**Editor 1/Body (general)**" menu item :



The program calculates the sliding safety using:

$$\text{Friction coefficient} \cdot V_k / H_d$$

for all horizontal sections.

7.21.1 Internal stability

Verification of the internal stability of the reinforced soil system is regulated in ***EBGEO***. Verification uses the methods described in DIN 4084. The program assumes a failure system consisting of two slip planes (two-part wedge failure mechanism).

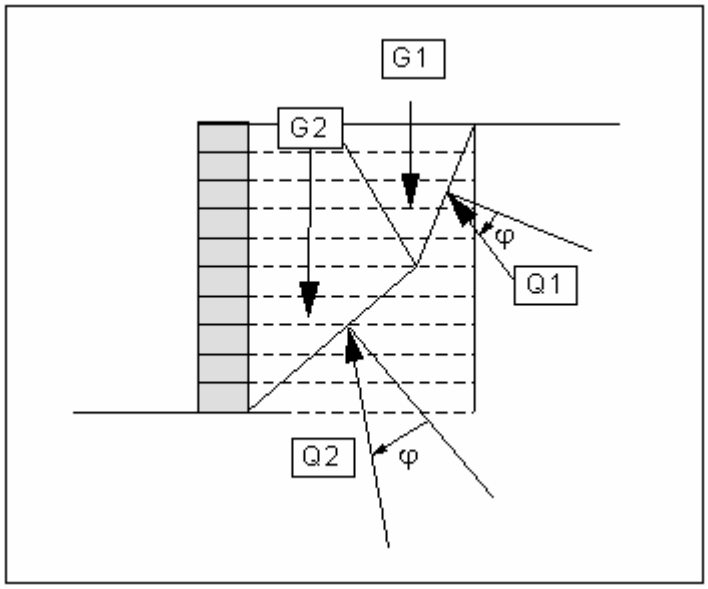


Figure 14 Two-part wedge failure mechanism

If one or more layers of reinforcement are intersected by this failure mechanism, the maximum force in a reinforcement layer must be determined according to ***EBGEO***, this involves examination of whether:

- the design resistance $R_{B,d}$ of the reinforcement layer is exceeded or;
- the pull-out to the "left" or "right" of the slip surface is governing ($R_{A,d}$).

The **design resistance $R_{B,d}$** of a geogrid according to ***EBGEO*** (section 6.1.3) is determined using:

$$\mathbf{R_{B,d} = R_{B,k0} / (A_1 \cdot A_2 \cdot A_3 \cdot A_4 \cdot A_5) / \gamma_M}$$

$R_{B,k0}$ = characteristic value of the short-term strength
γ_M = partial safety factor for the material resistance of the geogrid (in Load Case 1 to DIN 1054 = 1,40)
A_1 = creep strain or creep rupture reduction factor
A_2 = transport, installation and compaction reduction factor
A_3 = connections reduction factor
A_4 = environmental influence reduction factor
A_5 = dynamic actions reduction factor

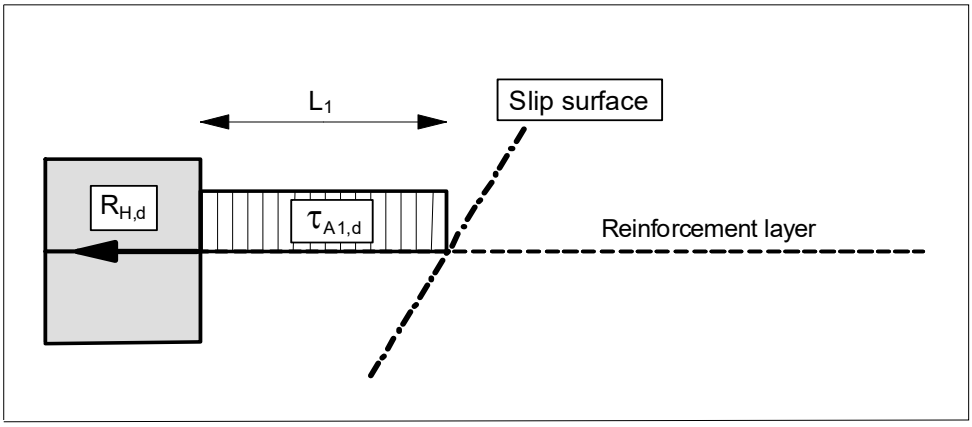
The following comparisons need to be considered when calculating the maximum **design value of the pull-out resistance $R_{A,d}$ $R_{A,d}$**:

- The maximum pull-out resistance ***within the slip body*** ("left") $R_{A1,d}$ is acquired from the force $R_{H,d}$ with which the geogrid is attached to the gabion wall and the bonding stress $\tau_{A1,d}$ that can be activated within the slip body.

  According to ***EBGEO*** a value of

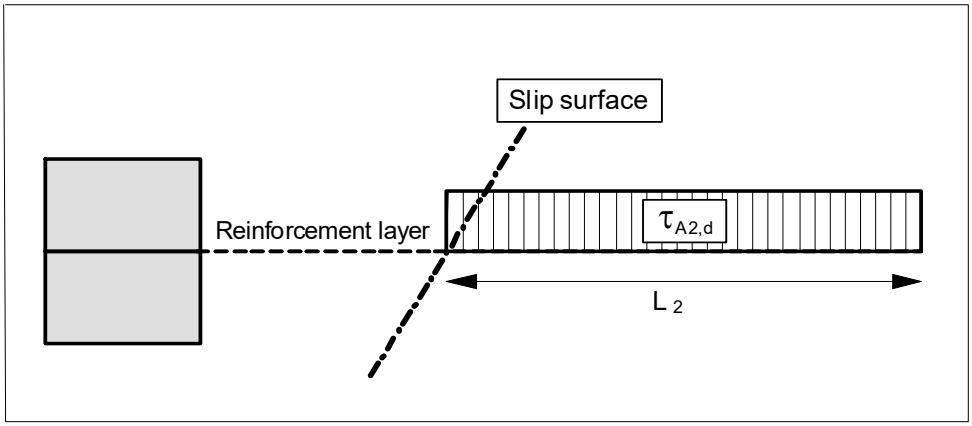
  $$R_{H,d} = 0{,}8 \cdot R_{B,d}$$

  can be adopted for $R_{H,d}$.



$$R_{A1,d} = R_{H,d} + 2 \cdot L_1 \cdot \tau_{A1,d}$$

- The maximum pull-out resistance $R_{A2,d}$ ***outside of the slip body*** ("right") is acquired from the bonding stress $\tau_{A2,d}$ behind the slip body.



$$R_{A2,d} = 2 \cdot L_2 \cdot \tau_{A2,d}$$

- In accordance with ***EBGEO*** the **bonding stress $\tau_{A,d}$** is acquired from:

$$\tau_{A,d} = f_{A,k} \cdot \sigma_{V,k} / \gamma_B = \lambda_k \cdot \tan \varphi_k \cdot \sigma_{V,k} / \gamma_B$$

$f_{A,k}$ = characteristic friction coefficient between geosynthetic and fill
λ_k = characteristic ratio of friction angle between geosynthetic and fill
φ_k = characteristic angle of internal friction of fill
$\sigma_{V,k}$ = characteristic normal stress due to reinforcement layer surcharge
γ_B = partial safety factor for the pull-out resistance of the reinforcement (in Load Case 1 to DIN 1054 = 1,4)

The minimum values of $R_{A1,d}$, $R_{A2,d}$ and $R_{B,d}$ are governing for stability analyses to DIN 4084.

The analysis of the structural system as a frame system results in support forces in a system using geogrids (tension forces in the geogrids). It is therefore also necessary to subsequently verify that the maximum acceptable force in the geogrids is greater than the calculated support force.

7.21.2 External stability

To verify the external stability of the reinforced soil system the weights and horizontal earth pressures, amongst other things, acting on the soil body must be calculated. This is done on equivalent systems. The data in Figure 15 can be used for the weight calculation.

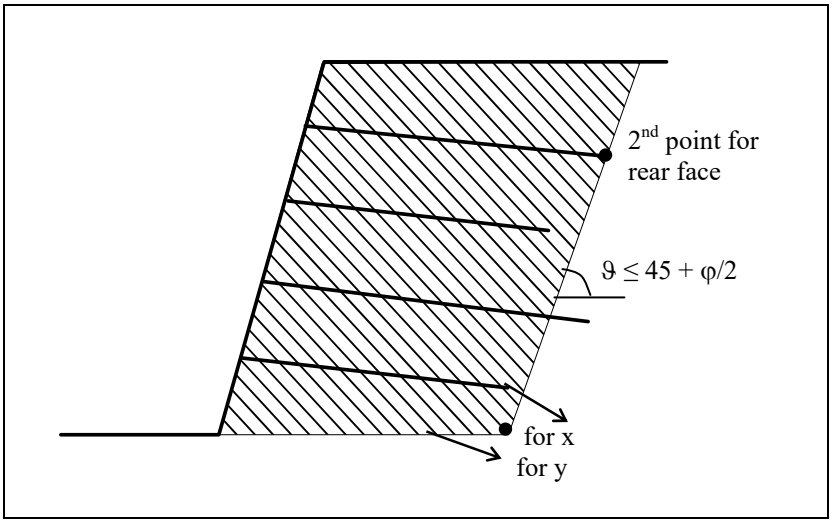


Figure 15 Equivalent system for weight calculation

The weight of the hatched area is determined. The lower right point of the system is given by the x-value of the lowest end of the geogrid and the y-value of the base of the gabion wall. The rear face corresponds to a line connecting this point to the end of the upper geogrid (irrespective of the length of any intervening geogrids). The inclination of the rear face of the wall can not become steeper than the inclination of the active earth pressure wedge (see further below).

The horizontal loading is given by the active earth pressure. The vertical section for which the earth pressure is determined is represented in Figure 16.

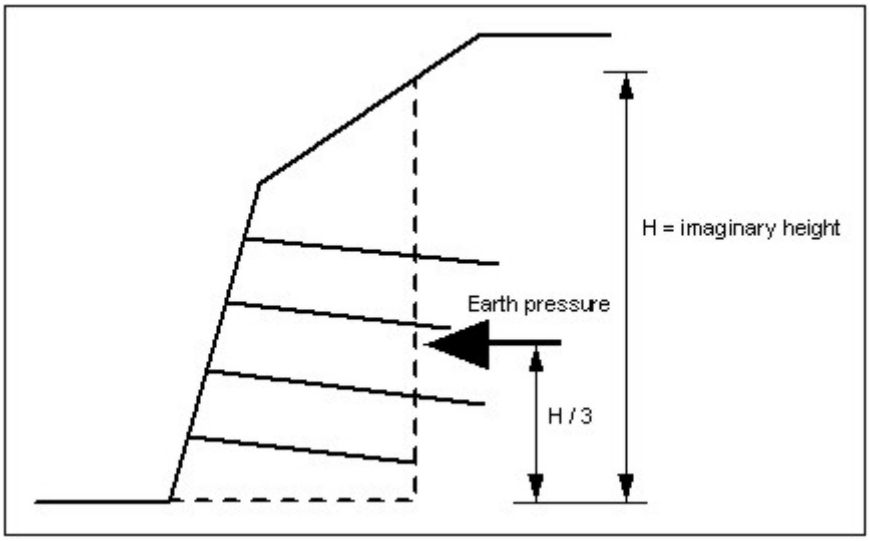


Figure 16 Vertical section for earth pressure determination

Calculation of earth pressure is by means of a variation of the rearward 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($\alpha = 0$)

Sliding safety η_G is calculated from:

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

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 reinforced soil system.

7.21.3 Overturning safety

Using the information provided in section 7.21.2 "**Verifications of reinforced soil system/External stability**" the moment and the vertical force in the base plane are calculated and the eccentricity e determined from the result. The moment resulting from the horizontal earth pressure is determined according to Figure 16 (Vertical section for earth pressure determination) in the section above. For example, the eccentricity in Load case 1 may not be greater than b/6.

7.21.4 Distribution of earth pressure on the gabion wall

Knowledge of the earth pressure distribution on the gabion wall is of critical importance for the verifications listed above. The factor f_q may be reduced on the basis of the ***EBGEO***.

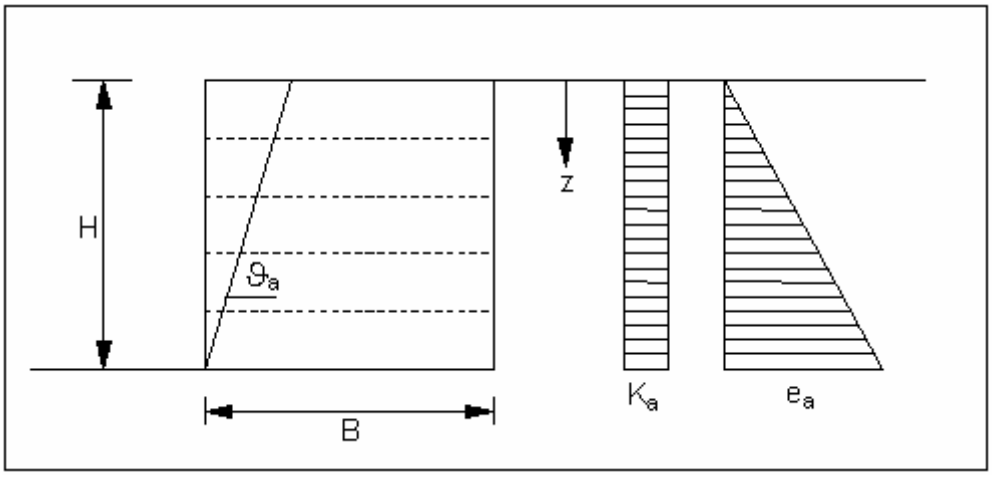


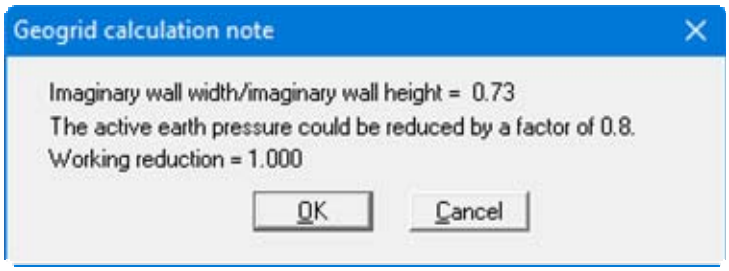
Figure 17 Slip surface and earth pressure distribution

The load q acting on the gabion wall q is calculated from:

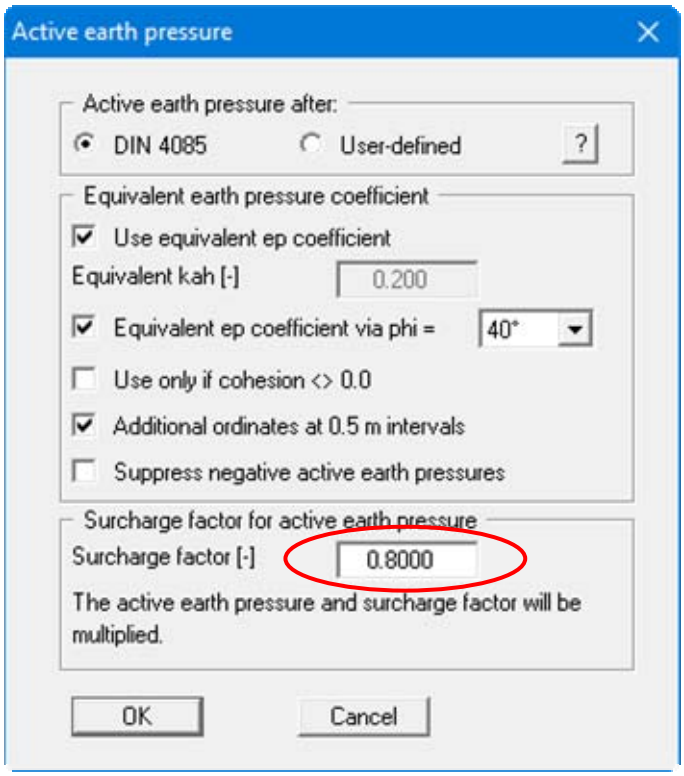
$$\mathbf{q = f_q \cdot k \cdot \gamma \cdot z}$$

f_q = 0.8 for B/H ≥ 0.7 and f_q = 1.0 for B/H ≤ 0.5
(linear interpolation between)
k = earth pressure coefficient according to Figure 17
γ = unit weight of fill
z = ordinate

The reduction factor f_q can be entered using the menu item "**Editor 1/Active earth pressure**". In a system using geogrids the program checks that the ratio H/B and, where necessary, informs either that a reduction in earth pressure is possible or that the earth pressure reduction used is too large.



If you close this dialog box using "**OK**", the program proceeds to calculate with the values provided. However, if you use the "**Cancel**" button, a query will appear asking if the allowable value determined in accordance with the above stipulations should be used. If you confirm this with "**Yes**" the program accepts this value for the reduction without you having to go to the menu item "**Editor 1/Active earth pressure**".



7.22.1 Analysis principles

The bearing capacity analysis will be carried out according to DIN 4017:2006. The following relationship applies:

$$\sigma_{0f,k} = c \cdot N_c + \gamma_1 \cdot d \cdot N_d + \gamma_2 \cdot b' \cdot N_b$$

Additionally, the depth coefficients T_c and T_d not documented in DIN 4107 may be taken into consideration.

- $\sigma_{0f,k}$ = characteristic bearing stress
- c = cohesion [kN/m²]
- N_c = bearing capacity coefficientequation cohesion
- γ_1 = unit weight of soil above footing base
- d = embedment depth of footing
- N_d = bearing capacity coefficient footing depth
- γ_2 = unit weight of soil below footing base
- b' = calculated depth of footing
- N_b = bearing capacity coefficient footing width

Bearing capacity coefficients N_c N_d and N_b

- $N_c = N_{c0} \cdot \nu_c \cdot i_c \cdot \lambda_c \cdot \xi_c \cdot T_c$
- $N_d = N_{d0} \cdot \nu_d \cdot i_d \cdot \lambda_d \cdot \xi_d \cdot T_d$
- $N_b = N_{b0} \cdot \nu_b \cdot i_b \cdot \lambda_b \cdot \xi_b$

The following values are adopted:

- N_{c0} N_{d0} N_{b0} according to DIN 4017
- ν_c ν_d ν_b according to DIN 4017 (shape coefficients)

> When entering the system data input of a length is principally of importance for the calculation of settlement. The program consistently uses the values for a and b, including for calculation of the shape coefficients ν_d and ν_b for strip footings, as this delivers somewhat more favourable values.
> $\nu_d = 1 + 00.2 \cdot b/a$ (instead of 1.0 for strip footings)
> $\nu_b = 1 + b/a \cdot \sin(\varphi)$ (instead of 1.0 for strip footings)

- i_c i_d i_b according to DIN 4017 (load inclination coefficients)
- λ_c λ_d λ_b according to DIN 4017 (ground inclination coefficient)
- ξ_c ξ_d ξ_b according to DIN 4017 (base inclination coefficients)
- T_c T_d = depth coefficients according to the Danish standard (not permitted to DIN 4017)
  $T_c = T_d = 1 + 0.35 \cdot GS / b \leq 2.0$
  GS = footing base
  b = footing width

7.22.2 Determining the mean governing soil properties

DIN 4017 instructs that the mean soil properties of the soil above the bearing capacity failure plane, which is composed of the two linear components of a logarithmic spiral, can be determined for stratified ground. The governing parameters are summarised in the figure:

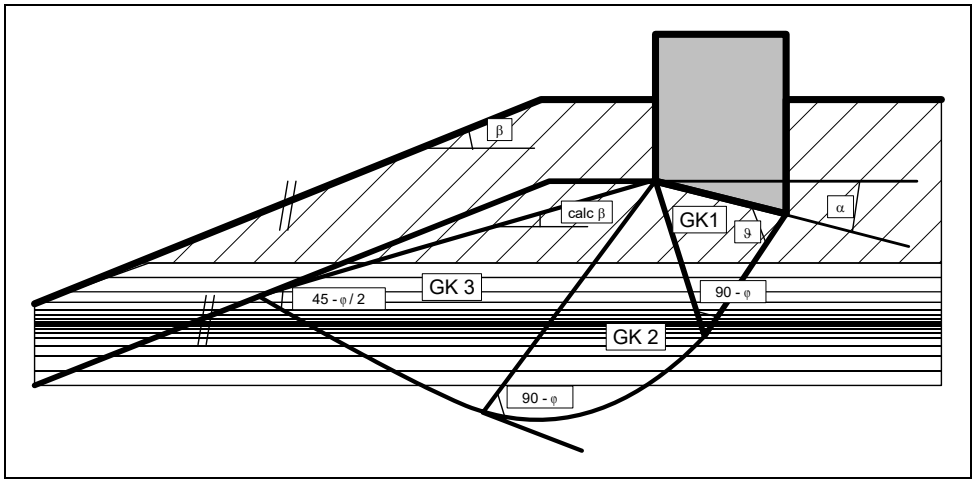


Figure 18 Logarithmic spiral

The mean governing soil properties are determined using the following relationships:

$$\mathbf{cal\ tan\ \varphi = \Sigma\ tan\ \varphi_i \cdot l_i' / \Sigma\ l_i}$$

$$\mathbf{cal\ c = \Sigma\ c_i \cdot l_i' / \Sigma\ l_i}$$

$$\mathbf{cal\ \gamma_2 = \Sigma\ \gamma_{2i} \cdot A_i' / \Sigma\ A_i}$$

l_i = length within individual layer
A_i = area of individual layer

The condition for the permissibility of the mean is that the mean friction angle demonstrates a maximum deviation of 5° to the true friction angles. This condition can be checked by the program. If it is not adhered to, the program reduces the largest friction angle in stages until the condition is met.

7.23 Settlements

Analysis of settlement is compliant with DIN 4019 using the relationships given in the Geotechnical Engineering Handbook (1990; Fourth Edition) (equations 8 and 14 from section 1.7 Stress analyses). The program determines the stresses at 0.05 m intervals or at layer boundaries and numerically integrates them.

The limiting depth can be defined in three different ways:

- with a fixed, user-defined value;
- as a multiple of the footing width;
- as the depth at which the total vertical stress exceeds the overburden stress by x% (generally 20%).

If the base of the lowest layer is exceeded during the settlement analysis, the analysis continues using the constrained modulus of this final layer.

For settlement analyses, any preconsolidation loading in kN/m² can be subtracted from the current soil pressure. Settlement analysis will then be performed with the reduced values. The overburden pressure is also reduced by this amount when calculating the limiting depth.

7.24 General stability

The general stability (see EAB, R 10, Para. 3) can be simply verified by exporting the data from **GGU-GABION** to **GGU-STABILITY** (GGU slope stability application).

7.25.1 Analysis to DIN 1045 (old)

The following data are required for internal stability analysis:

- **beta(r)**
  designates the nominal strength of the material (see also DIN 1045).
- **n-value**
  DIN 1045 states that a load distribution may be taken into consideration up to an inclination of 1:2 to the direction of the load. Depending on the soil pressure, other n-values can also be used (see DIN 1045; Table 17 for information on this). Together with the load transmission width, this allows calculation of the width to be adopted for the complete height when verifying internal stability.

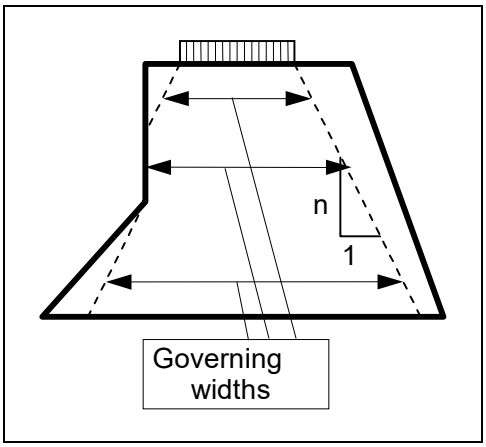


Figure 19 n-value and width for internal stability analysis

Internal stability analysis is performed using the following relationship to DIN 1045:

$$\textbf{allow. N} = \kappa \cdot 1 / 2.1 \cdot b \cdot \textbf{beta(r)} \cdot (1.0 - 2 \cdot e) \textbf{ [kN/m]}$$

b = governing width of concrete
e = eccentricity with regard to governing width
κ = reduction for slenderness of structural element

κ is a coefficient for considering the slenderness and unintentional eccentricity to DIN 1045 (Equation 20). κ is calculated from:

$$\kappa = 1 - \lambda/140 \cdot (1 - m/3)$$

m = e/k (e = M/V and k = b/6)
λ = slenderness (for rectangular sections = $s_k/b \cdot \sqrt{12}$)
s_k = buckling length

The buckling length s_k in systems without geogrids is given approximately by twice the separation between the top of the wall and half of the embedment depth. For walls that are horizontally restrained near the top by other engineering elements, the single separation may be adopted. The horizontal restraint can be specified using the "**Wall top kept horizontal**" check box. In systems which include geogrids, s_k is acquired from twice (once) the separation between the top of the walland the upper geogrid layer, or the separation between the individual geogrid layers. The slenderness λ is recalculated for every section using the respective governing gabion width. The program deals with the appropriate selections with the exception of the "**Wall top kept horizontal**" setting.

The allowable N calculated in this manner is compared to the existing normal force working N. The utilisation factor g is then acquired from the relationship:

g = working N/allowable N.

Internal stability analysis can also be performed using a partial section.

7.25.2 Analysis to EC 2/DIN 1045-1

The internal stability of the gabion structure can be analysed for unreinforced concrete in all governing horizontal sections using the following relationship based on EC 2/DIN 1045-1:

$$\mathbf{N_{R,d} = b \cdot f_{m,d} \cdot \varphi}$$

$$\varphi = 1{,}14 \cdot (1 - 2 \cdot e_{tot}/b) - 0.02\, s_k/b$$
$$(0 \le \varphi \le 1 - 2 \cdot e_{tot}/b)$$

$N_{R,d}$= design value of the acceptable longitudinal force
b = width of body in respective section
$f_{m,d}$ = equivalent design value of concrete compressive strength
e = eccentricity in respective section
e_{tot} = total eccentricity = $e_0 + e_a + e_\varphi$
e_0 = load eccentricity according to first order theory
e_a = additional unintentional eccentricity as a result of geometric imperfections
(If more precise data is not available, e_a may be assumed at = 0.5 l_0/200.)
e_φ = eccentricity resulting from creep (can be disregarded for gabions)
s_k = buckling length

The buckling length s_k is given approximately by twice the distance between the top of the wall and half of the embedment depth.

The equivalent design value of the concrete compressive strength $f_{m,d}$ is determined from load tests performed on a gabion basket.

The design value of the acceptable longitudinal compressive force $N_{R,d}$ and the design value of the acting longitudinal compressive force $N_{E,d}$ are determined in every governing section. The following condition must be adhered to:

$$\mathbf{N_{R,d} \ge N_{E,d}}$$

7.25.3 Analysis to DIN 4093

7.25.3.1 Ultimate limit state

The analysis of sufficient gabion strength is performed compliant with DIN 4093:2012-08 (Design of strengthened soil). The design value of the compressive strength of the gabion $f_{m,d}$ required for this purpose is derived from load tests.

DIN 4093:2012-08 stipulates that, in approximation, separate analyses may be performed during design for the acceptable compressive and shear stresses. Under design loads it must be demonstrated that the design values of the normal stresses $\sigma_{E,d}$ do not exceed $0.7 \cdot f_{m,d}$ and the design values of the shear stresses $\tau_{E,d}$ do not exceed $0.2 \cdot f_{m,d}$.

The normal stresses $\sigma_{E,d}$ are determined assuming a linear distribution.

$$\mathbf{\sigma_{E,d} = M_{E,d}/W + N_{E,d}/A \leq 0.7 \cdot f_{m,d}} \quad \textbf{(no foundation gap)}$$

$M_{E,d}$ = design value of the moment
W = section modulus
$N_{E,d}$ = design value of normal force
A = cross-sectional area

$$\mathbf{\sigma_{E,d} = 4 \cdot N_{E,d}/(3 \cdot b - 6 \cdot e) \leq 0.7 \cdot f_{m,d}} \quad \textbf{(foundation gap)}$$

$N_{E,d}$ = design value of normal force
b = width of cross-section
e = eccentricity

If analysis results in a foundation gap, the area of the gap may not be adopted in an analysis of the shear stresses.

The design value of the shear stress $\tau_{E,d}$ is given by:

$$\mathbf{\tau_{E,d} = 1.5 \cdot Q_{E,d}/A_v \leq 0.2 \cdot f_{m,d}}$$

$Q_{E,d}$ = design value of the shear force
A_v = cross-sectional area under shear loading

The characteristic value of the compressive strength of the gabion $f_{m,k}$ is determined from load tests. The corresponding design value of the compressive strength $f_{m,d}$ is given by:

$$\mathbf{f_{m,d} = \alpha \cdot f_{m,k}/\gamma_m}$$

α = coefficient for considering a long-term effect = 0.85
γ_m = partial safety factor for the compressive strength
(= 1.5 for DS-P and DS-T
= 1.3 for DS-A)

7.25.3.2 Serviceability limit state

In load tests on gabions, the failure stress is often only reached with large deformations. The stress $\sigma_{E,k,SLS}$ (SLS = serviceability limit state) at which the deformation of the gabion reaches the value allowable for the serviceability analysis is determined from the stress-deformation curve. The following analysis must then be performed:

$$\boldsymbol{\sigma_{E,k} = M_{E,k}/W + N_{E,k}/A \leq \sigma_{E,k,SLS}}\ \textbf{(no foundation gap)}$$

$M_{E,k}$ = characteristic value of the moment
W = section modulus
$N_{E,k}$ = characteristic value of normal force
A = cross-sectional area

$$\boldsymbol{\sigma_{E,k} = 4 \cdot N_{E,k}/(3 \cdot b - 6 \cdot e) \leq \sigma_{E,k,SLS}}\ \textbf{(foundation gap)}$$

$N_{E,k}$ = characteristic value of the normal force
b = width of cross-section
e = eccentricity

This serviceability analysis can also be selected for analysis compliant with ***EC 2***, which we recommend (see "**Editor 1/Analysis options**"/"**File/New**" dialog boxes, section 8.1.1).

7.25.3.3 Slenderness

DIN 4093:2012-08 stipulates that when analysing the ultimate limit state the slenderness $\lambda \leq 15$, if no buckling safety analysis is performed. This can not always be demonstrated. However, the normal force loading on gabion walls is generally insignificant, and the analysis concepts in EN 1993-1-1:2010-12 are helpful. EN 1993-1-1:2010-12 states that buckling safety analysis may be dispensed with if the following condition is adhered to:

$$\boldsymbol{N_{Ed}/N_{cr} \leq 0.04}$$

N_{Ed} = design value of the normal force
N_{cr} = ideal bifurcation load for the governing buckling case

$$\boldsymbol{N_{cr} = E \cdot I \cdot \pi^2/s_k^2}$$

E = Young's modulus of the gabion
I = moment of inertia
s_k = buckling length

According to expert opinion, the criterion to EN 1993-1-1:2010-12, Para. 6.3.1.2 (4), where $N_{Ed}/N_{cr} \leq 0.04$, is a ridiculously strict criterion and will be dispensed with in future standards. In future the tried and tested rule in Para. 5.2.1 applies, Eq. (5.1): $N_{Ed}/N_{cr} \leq 0.1$.

The investigation may therefore be performed using

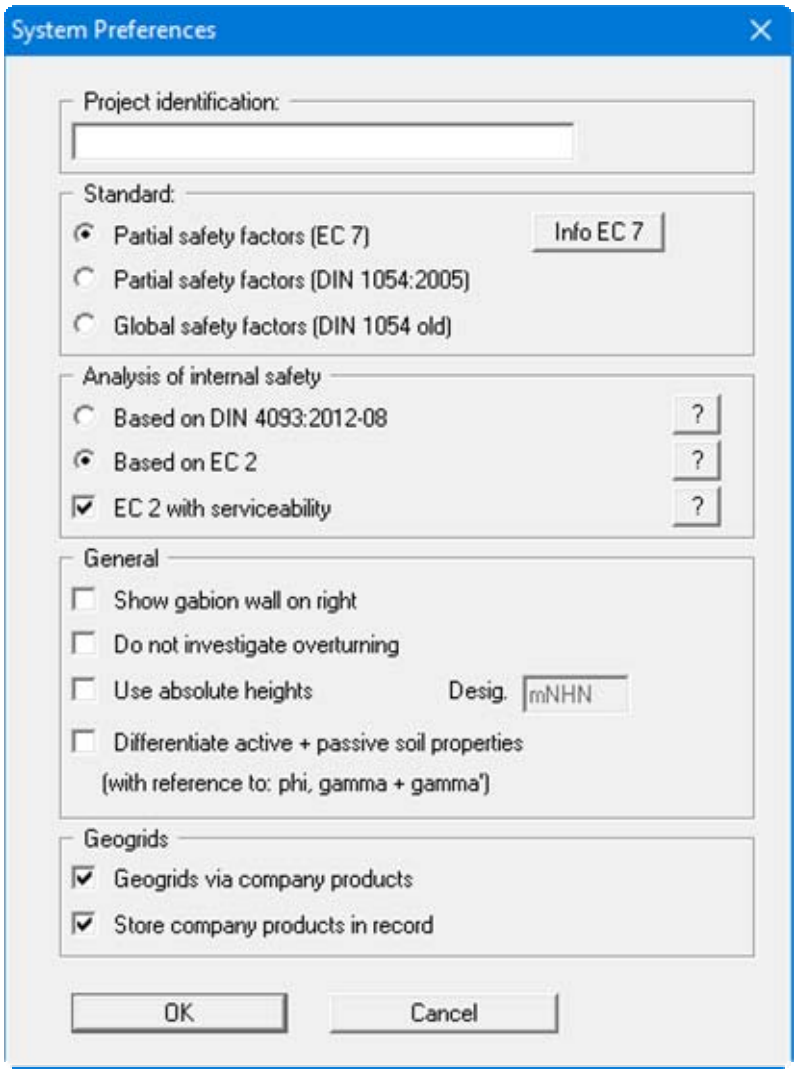
$$\boldsymbol{N_{Ed}/N_{cr} \leq 0.10.}$$

8 Description of menu items

8.1 File menu

8.1.1 "New" menu item

You can enter a new system using this menu item. You will see the following dialog box:



You can enter a dataset description ("**Project identification**"). of the problem going to process, which will then be used in the "**General legend**" (see section 8.7.8).

In the next group box the radio buttons are used to specify which safety concept to use for analysis and design.

By default, analysis of internal stability is based on EC 2 as an analysis of the unreinforced concrete (see section 7.25.2). We recommend analysing the serviceability limit state, including when using EC 2 (see section 7.25.3.2). Alternatively, you can analyse internal stability based on DIN 4093:2012-08 (see section 7.25.3). This then includes the serviceability analysis.

Additionally, gabion wall visualisation to the right can be activated. DIN 1054:2005 states that an overturning analysis may be dispensed with for Load Case 3 if sufficient bearing capacity has been demonstrated. This rule is no longer included in Eurocode 7; it is therefore always necessary to perform an overturning analysis. In justified exceptional cases, however, the analysis may be turned off using the "**Do not investigate overturning**" check box.

If you select the "**Use absolute heights**" check box, you can enter all depths and heights in **m AD** (heights are positive upwards). If you leave this box unselected, the top of the wall is assumed at **0.0** (height/depth) and all input of layer depths etc. is positive downwards. If, however, you want to work with absolute heights, enter the appropriate depths correspondingly altered. Thanks to ***WYSIWYG*** there is no danger of using incorrect data, since all input is immediately visible on the screen.

If your system uses differing soil properties on the active and the passive sides, activate the "**Differentiate active + passive soil properties**" check box in the above dialog box. You will then be presented with different input columns for entering the active and passive friction angle and unit weight soil properties in the "**Editor 1/Soils**" menu item (section 8.2.8). For better visualisation you can define the soil colours on the active and the passive sides differently using the ***Soil properties legend*** (see section 8.7.7).

If the "**Geogrids via company products**" check box is activated, it is possible to select the products of various geogrids manufacturers from a list stored in the program when entering geogrids (see menu item "**Reinforced soil system/Enter geogrids manually**" or "**Reinforced soil system/Generate geogrids**"). We also recommend activating the "**Store company products in record**" check box. 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 geogrids manufacturers can be selected in the menu item "**Editor 1/Geogrids table values**" (see section 8.4.10).

If the dialog box is changed from ***Global safety factor concept*** to ***Partial safety factor concept***, an additional dialog box for specifying the partial factors opens after your input has been confirmed. By pressing the '**Default values**' button the partial factors for the various load cases given in DIN 1054:2005 and EC 7 can be adopted. The partial factors entered can be edited at any time using the '**Editor 1/Verifications/Partial factors**' menu item (see section 8.2.15).

8.1.2 "Load" menu item

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

8.1.3 "Save" menu item

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

8.1.4 "Save as" menu item

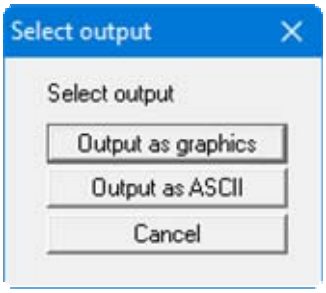
You can save data entered during program use to an existing file or to a new file, i.e. using a new file name. For reasons of clarity, it makes sense to use "**.gab**" as file suffix, as this is the suffix used in the file requester box for the menu item "**File/Load**". If you choose not to enter an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

8.1.5 "Print output table" menu item

8.1.5.1 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.

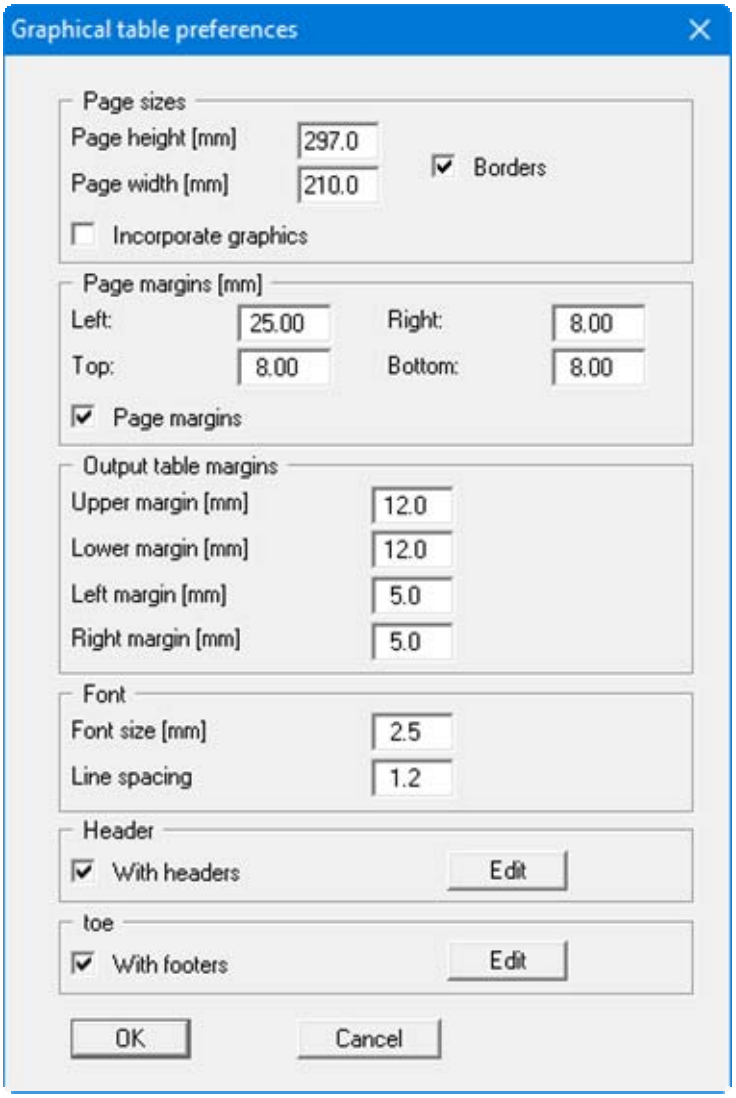
You have the option of designing and printing the output table as an annex to your report within the **GGU-GABION** application. To do this, select "**Output as graphics**" from the following options.



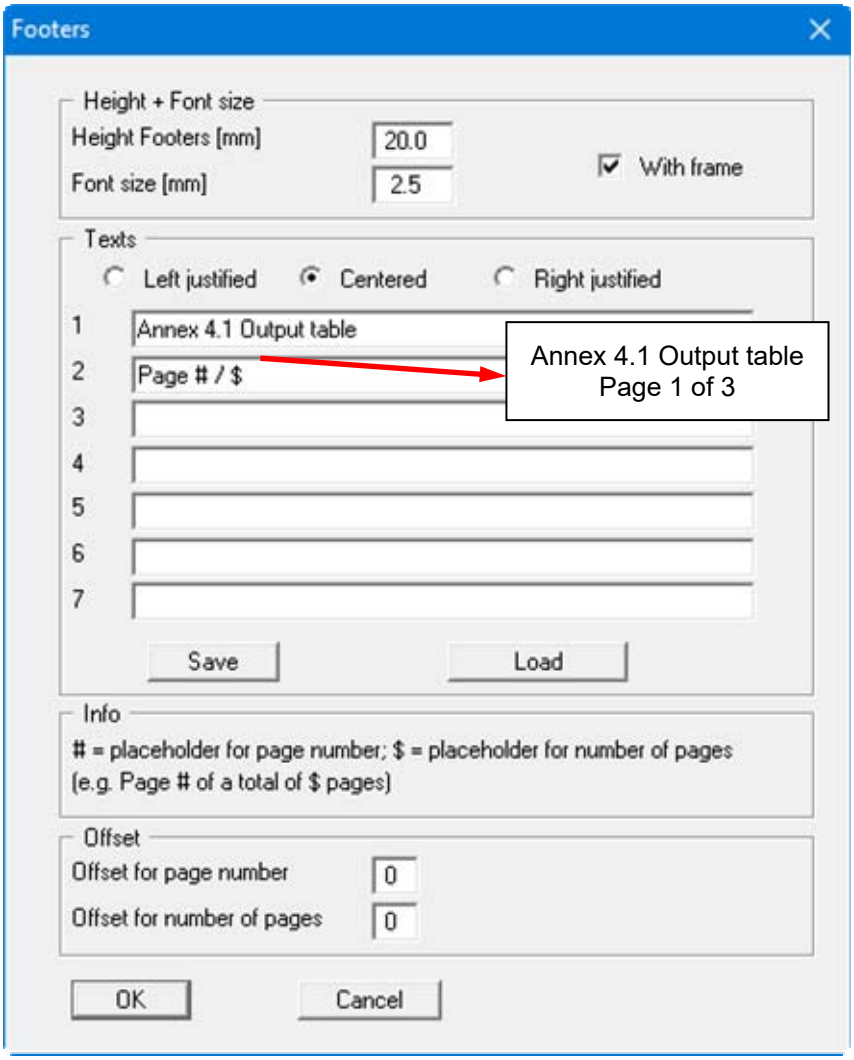
If you prefer to easily print or process the data in a different application, you have the possibility of sending them directly to the printer or to save them to a file using the "**Output as ASCII**" command button.

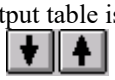
8.1.5.2 "Output as graphics" button

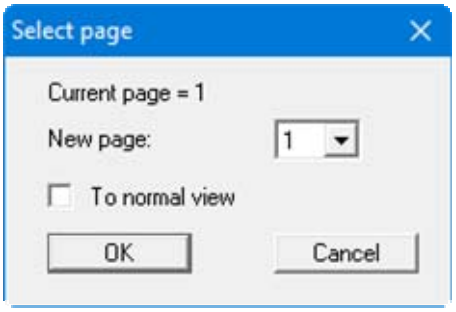
If you selected the "**Output as graphics**" button in the previous dialog box a further dialog box, in which you can define further preferences for result presentation.



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

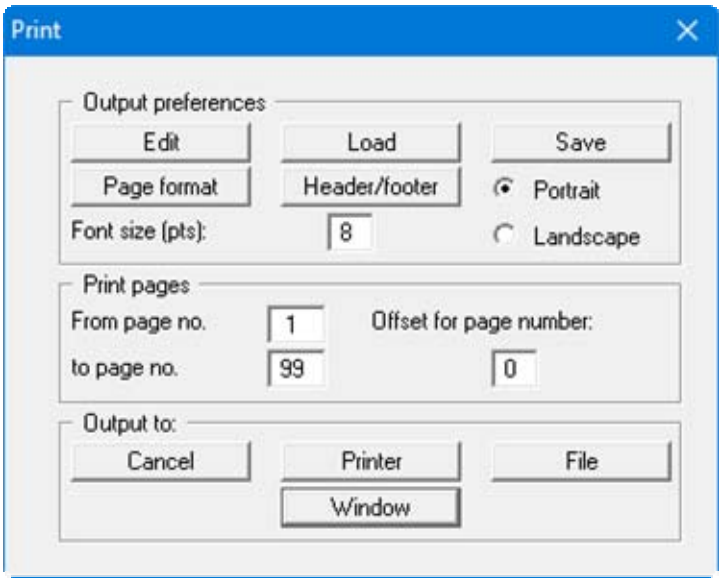


Automatic pagination can also be employed here if you work with the placeholders as described. After closing the dialog box using "**OK**" the output table is shown page by page on the screen. To navigate between the pages, use the arrow tools  in the toolbar. If you need to jump to a given page or back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



8.1.5.3 "Output as ASCII" button

You can have your analysis data sent to the printer, without further work on the layout, or save it to a file for further processing using a different program, e.g. a word processing application.



In the dialog box you can define output preferences.

- "**Output preferences**" group box
  Using the "**Edit**" button the current output preferences can be changed or a different printer selected. Using the "**Save**" button, all preferences from this dialog box can be saved to a file in order to have them available for a later session. If you select "**GGU-GABION.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.

8.1.6 "Export" menu item

The general stability can be simply verified by exporting the data from **GGU-GABION** to **GGU-STABILITY** (GGU slope stability application). After clicking this menu item an appropriate file ("**.boe**") can be generated with the required **GGU-STABILITY** version status.

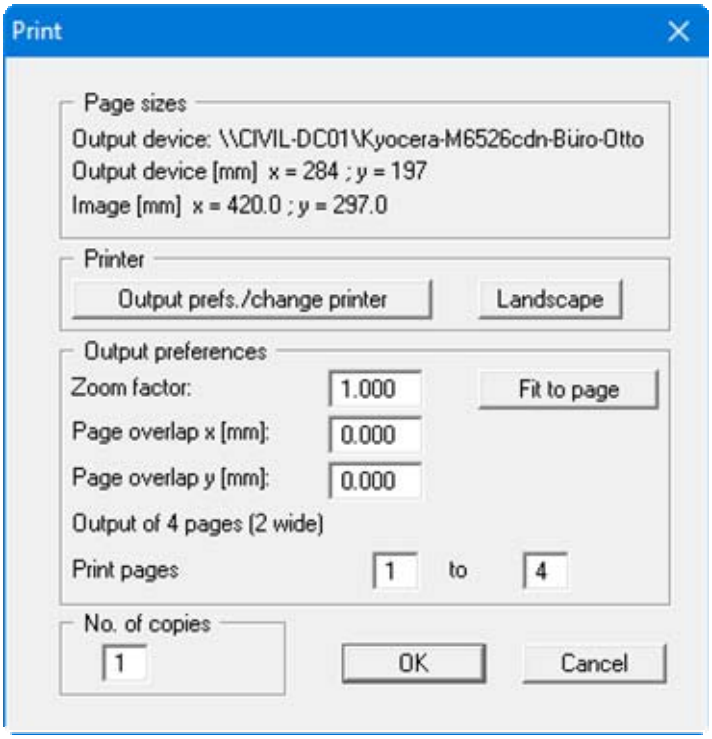
8.1.7 "Output preferences" menu item

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

8.1.8 "Print and export" menu item

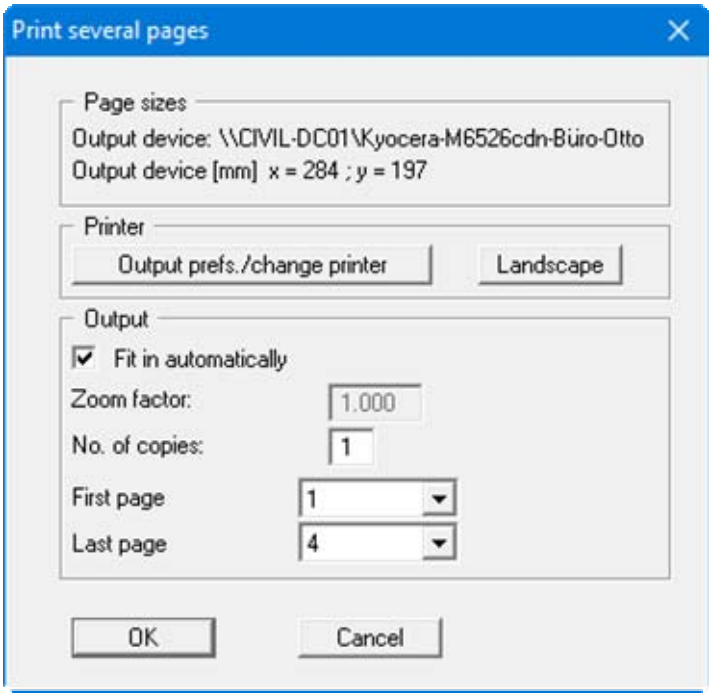
You can select your output format in a dialog box. You have the following options:

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



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

- "**DXF file**"
  allows output of the graphics to a DXF file. DXF is a common file format for transferring graphics between a variety of applications.
- "**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 in order to be further processed with third party software. Output is in the standardised EMF format (Enhanced Metafile format). Use of the Metafile format guarantees the best possible quality when transferring graphics.

If you select the "**Copy/print area**" tool from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see "**Tips and tricks**", section 9.4).

Using the "**Mini-CAD**" program module you can also import EMF files generated using other GGU applications into your graphics (see section 8.7.5).

- "**Mini-CAD**"
  allows export of the graphics to a file in order 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.

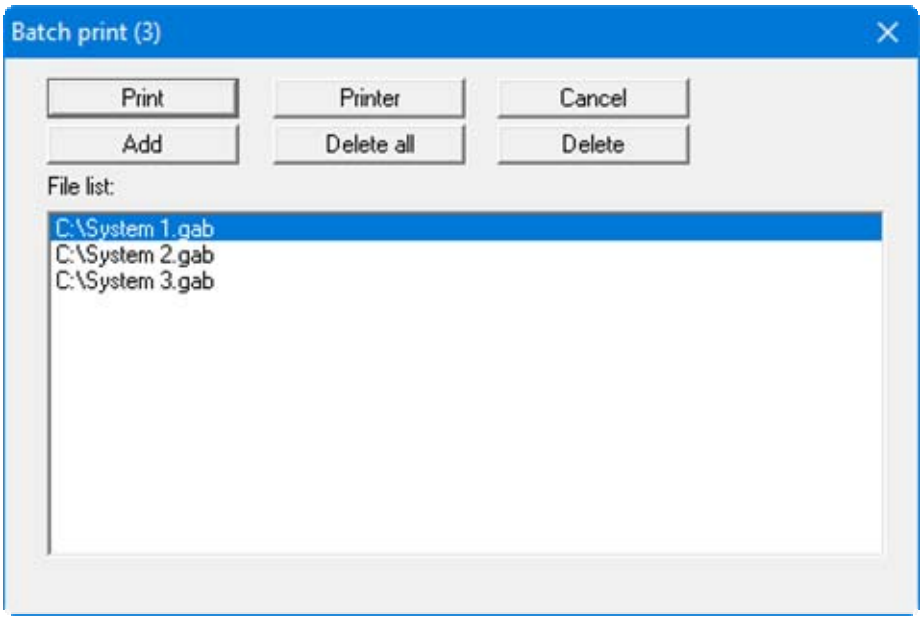
  By activating the "**Output global coordinates**" check box, the present graphics are saved in the system coordinates [m]. Otherwise they are saved in the page coordinates [mm]. If you import the **Mini-CAD** file saved using "**Global coordinates**" into a different **GGU** program, the coordinates 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.

8.1.9 "Batch print" menu item

If you would like to print several annexes 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.

8.1.10 "Exit" menu item

After a confirmation prompt, you can quit the program.

8.1.11 "1, 2, 3, 4" menu items

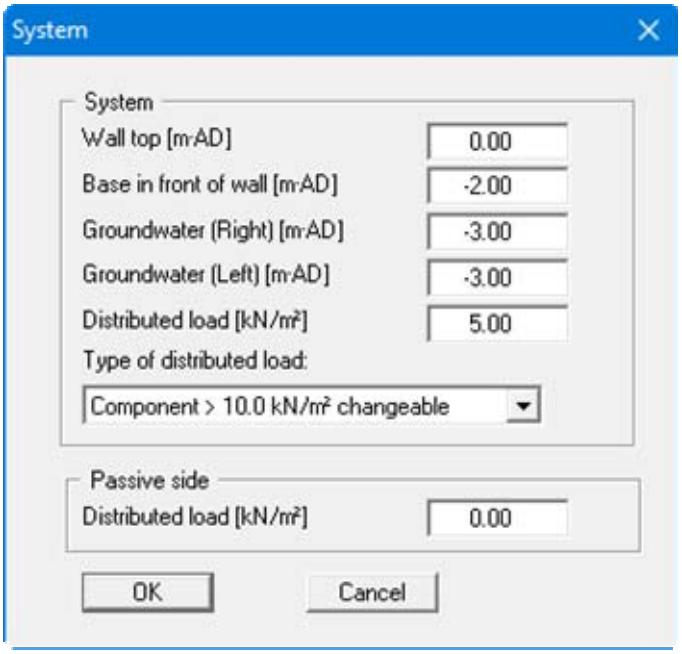
The "**1, 2, 3, 4**" menu items show the last four files worked on. By selecting one of these menu items the listed file will be loaded. If you have saved files in any other folder than the program folder, you can save yourself the occasionally onerous ***rummaging*** through various sub-folders.

8.2.1 "Analysis options" menu item

Using this menu item you can edit the default preferences of the current system. The dialog box corresponds to the box in the menu item "**File/New**" (see descriptions in section 8.1.1).

8.2.2 "System" menu item

A dialog box opens for defining the system:



In the upper group box, enter the depth of the base in front of wall and the groundwater levels. If you checked the "**Use absolute heights**" box when defining the system, an additional entry, "**Top of wall**", appears in the dialog box for specifying the absolute position. In this case, all heights are measured in m AD or m site zero, i.e. the y-axis is positive upwards. You can then enter a value, for example, of 86.42 [m AD] in the "**Top of wall**" field. All further input must then be with reference to this value.

If the height of a previously defined system is subsequently set to absolute heights, a query follows after leaving the dialog box above asking for confirmation of whether soil strata and defined elements such as anchors, for example, should be adapted to the new wall. Adaptation would mean that the depth of a soil layer entered as a positive value would be converted from, for example, 7.5 m to an absolute height of -7.5 m AD. If, then, you only convert your system to [m AD], do not select any elements in the query box and press the "**OK**" button.

Moreover, a distributed load can be defined. Using the ***global safety factors*** you will see here the "**Distributed load as live load**" check box. This is only of interest for verification of deep-seated stability. If the distributed load is defined as a live load, this load will only be adopted for verification of deep-seated stability if it acts excitingly.

If you are working with the ***partial safety factors***, decide whether the distributed load is "**Permanent**", "**Changeable**" or the "**Component above 10.0 kN/m² changeable**" (see the dialog box above). "**Component above 10.0 kN/m² changeable**" means, for example, that for an input of 13.5 kN/m², 10 kN/m² are adopted as permanent and 3.5 kN/m² as changeable in the analysis.

8.2.3 "Body (general)" menu item

Enter the loads acting on the gabions, the footing length, base inclination and friction coefficient. A more detailed description can be found in "**First steps**", section 6.1.4.

8.2.4 "Body (geometry)" menu item

Data input for the body geometry is carried out in horizontal sections. In the dialog box, define the respective depths of the horizontal sections and the two corresponding x-values of the gabion wall. For example, if you need to enter a gabion wall consisting of 2 elements (height = 0.5 m, width = 0.8 m) with a horizontal displacement of 0.1 m, you must define 4 horizontal sections.

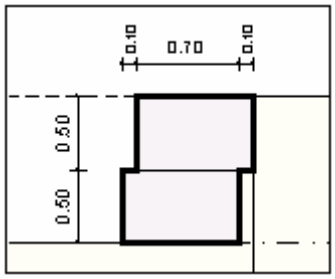
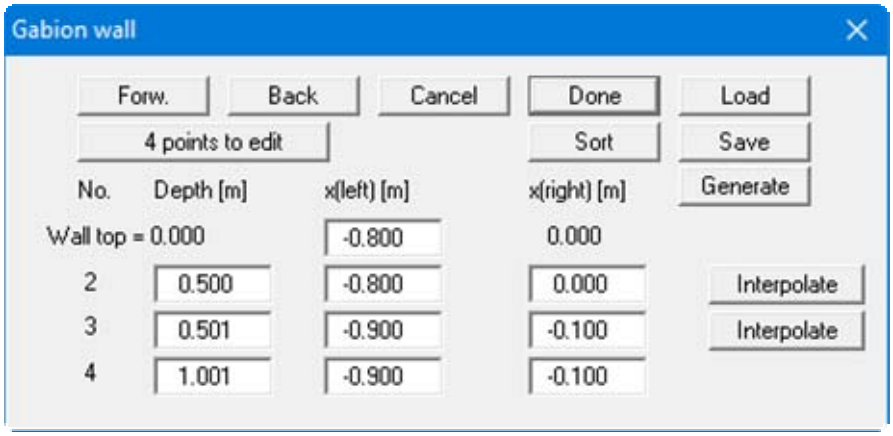


Figure 20 Body geometry

The corresponding data input is:

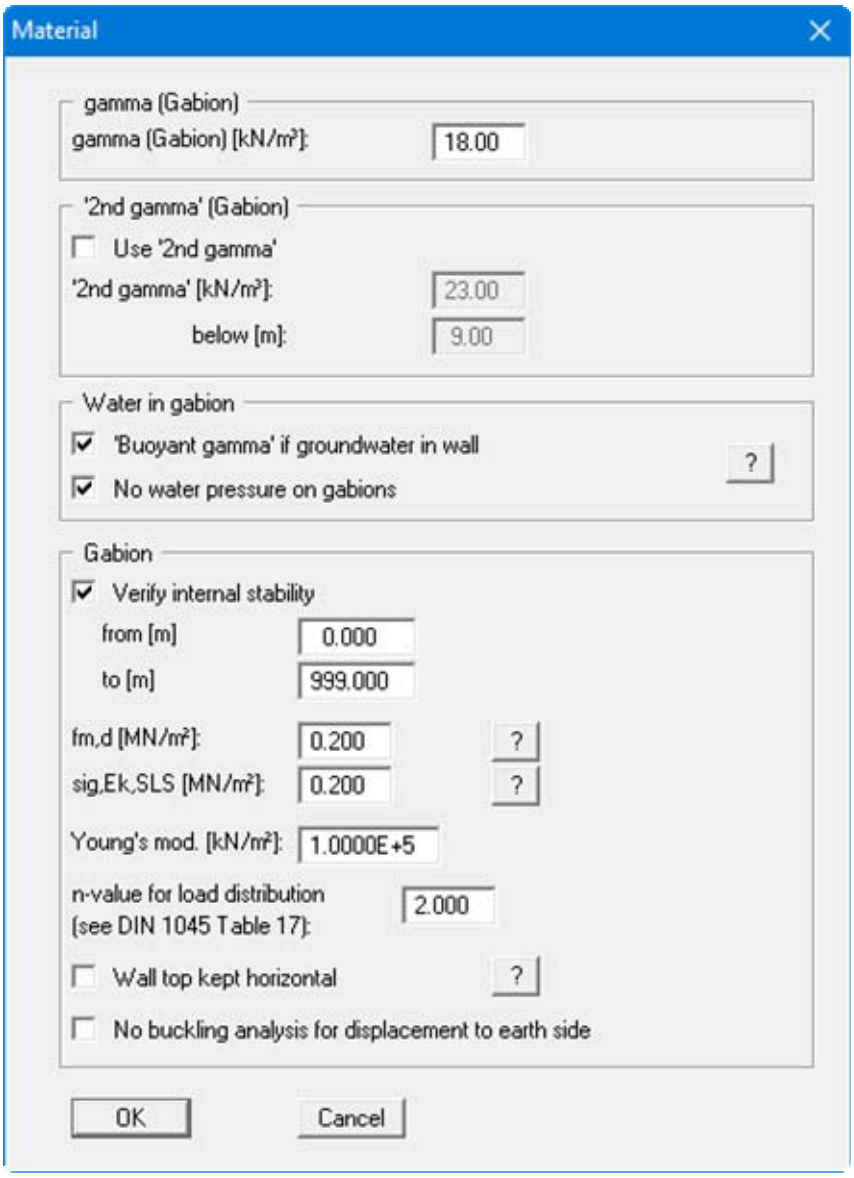


In the displacement area you define a small jump of, e.g., 0.001 m.

For most practical cases input can be automatically generated using the "**Generate**" button (see "**First steps: Worked example 1/Step 4**", section 6.1.5). In this way a gabion noise abatement wall can be very quickly generated, and the program automatically adapts the wind load and soil properties, among others, corresponding to your input (see "**First steps: Worked example 2 / Step 1**", section 6.2.2).

8.2.5 "Material" menu item

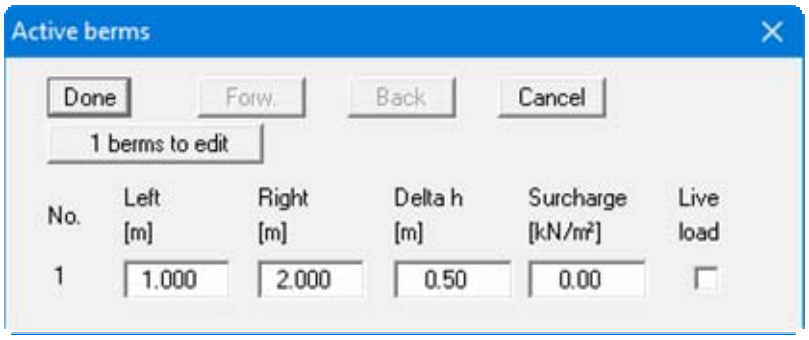
Enter the properties for the gabion material and for verification of the unreinforced concrete.



A more detailed description can be found in "**First steps**" (section 6.1.6) and in the "**Theoretical principles**" (section 7.25).

8.2.6 "Berms (active side)" menu item

You can define a maximum of 20 berms on the active side.



Enter the x-ordinates of the toe and head of the berm. With "**delta h**" you define the height of the berm, whereby negative values are also permitted. Finally, a "**Surcharge**" on the horizontal surface behind the head of the berm can be entered.

If more than one berm is present in the system, click "**x berms to edit**" and enter the number of berms.

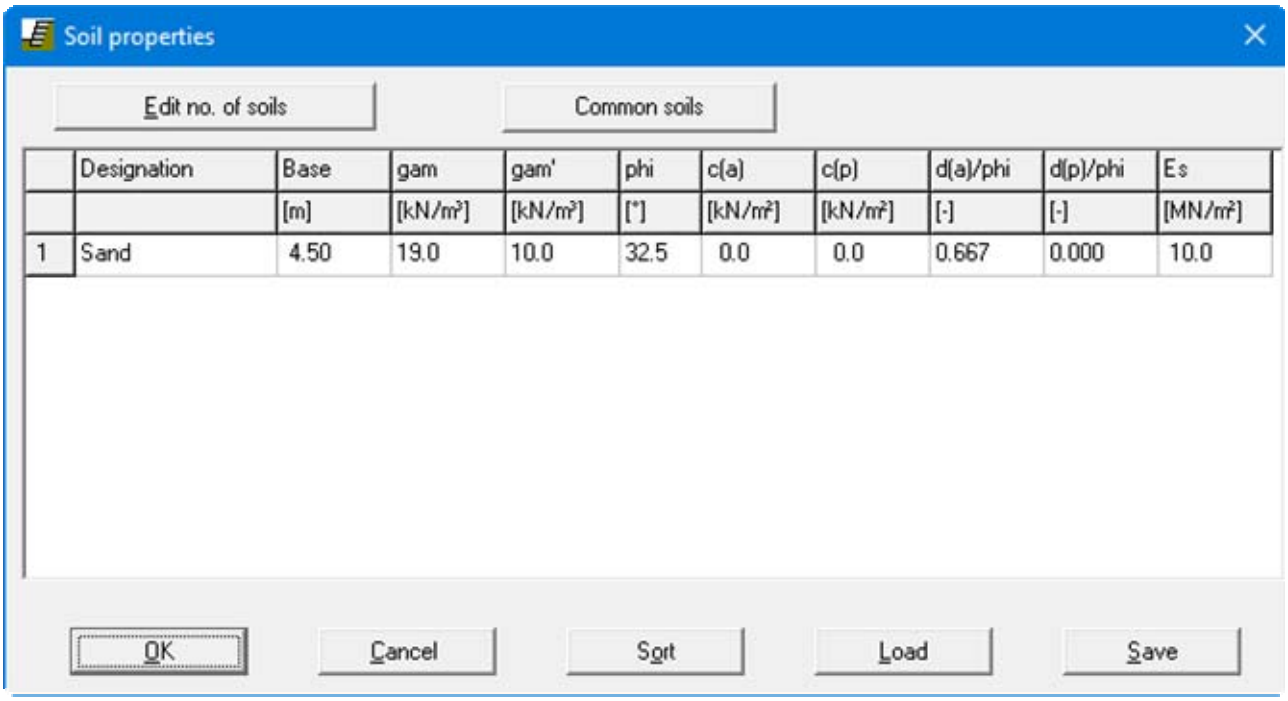
> Berms may not overlap. The program checks that this condition is adhered to and warns of any errors.

8.2.7 "Berms (passive side)" menu item

Berms on the passive side are defined in exactly the same manner as for the active side.

8.2.8 "Soils" menu item

You can define the soil properties in the following dialog box:



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.

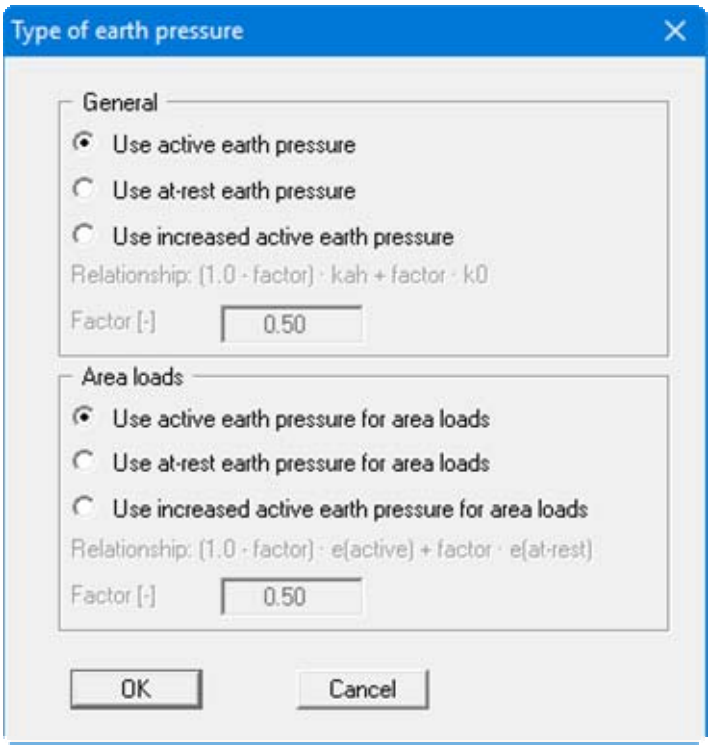
Layer depths are always with reference to the top of the wall, or are absolute heights (m AD), if this was selected in the initial dialog box of the "**File/New**" menu item.

According to EC 7, section 6.5.2.2 the passive earth pressure must be calculated with a wall friction angle = 0 when analysing the bearing capacity. If you have activated the "**Differentiate active + passive soil properties**" check box in the dialog box in "**File/New**" or "**Editor 1/Analysis options**", you can enter differing friction angles and unit weights for the active and the passive sides.

In stratified soils the number of layers must be entered under "**Edit no. of soils**". Clicking the "**Sort**" button sorts the soil layers according to depth; however, this is performed automatically when you click "**OK**" to leave the dialog box. This eliminates the possibility of input errors. You can also use this function to eliminate a soil from the table. Simply assign the soil to be eliminated a greater layer depth and then click the "**Sort**" button. The corresponding soil is now the last soil in the table and can be ***deleted*** by reducing the number of soils.

8.2.9 "Type of earth pressure" menu item

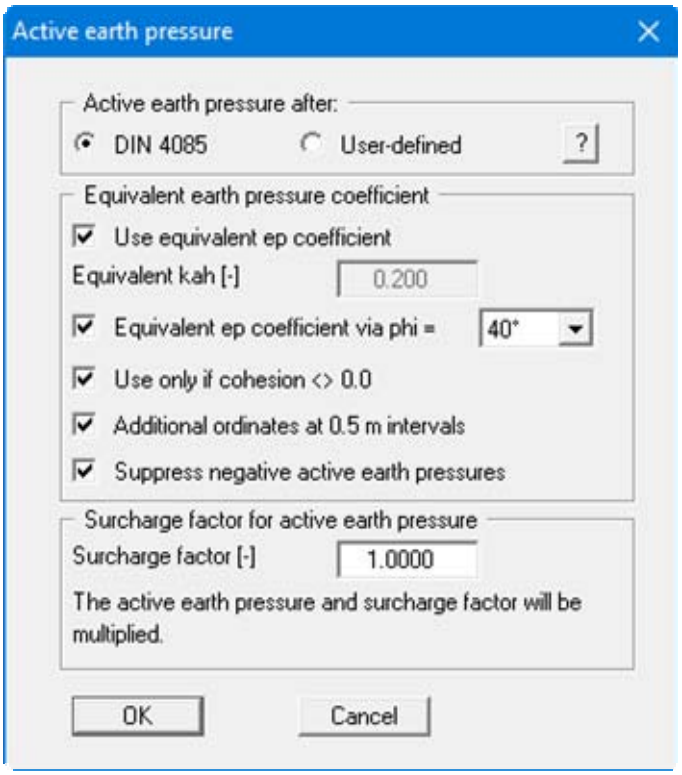
In this dialog box you define the type of earth pressure on which the analysis is to be based.



The options for ***area loads*** can be specified separately.

8.2.10 "Active earth pressure" menu item

You can specify active earth pressure preferences using this dialog box:



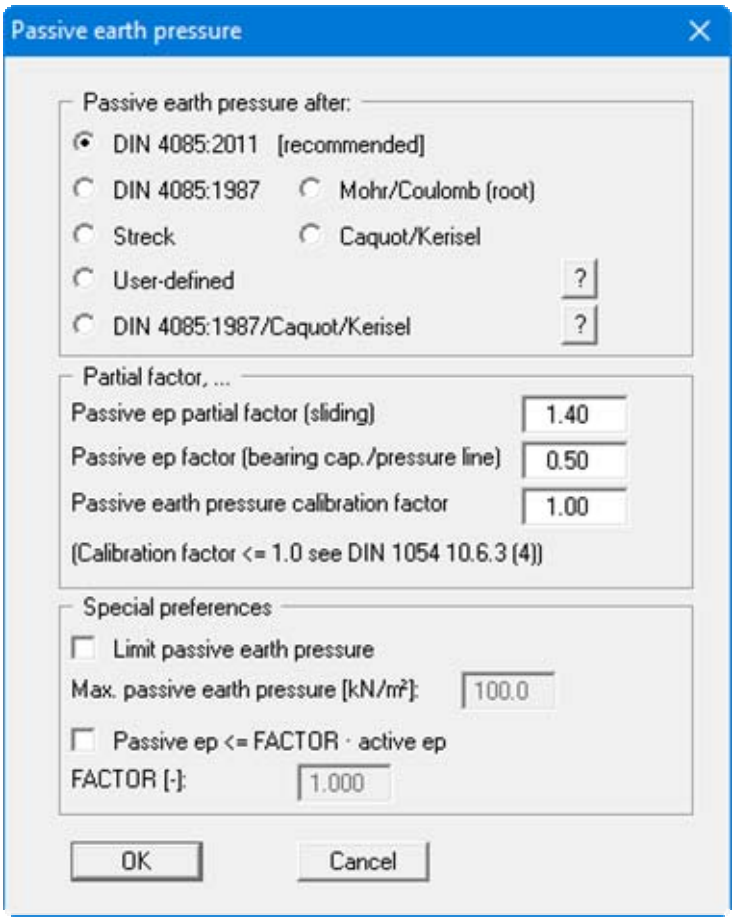
In the upper group box you specify the type of active earth pressure calculation.

The "**Use equivalent ep coefficient**" check box should only be deactivated in exceptional circumstances (see EAB R 4). The equivalent earth pressure coefficient can only be smaller than 0.2 in special circumstances (see EAB R 4). It only makes sense to deactivate this check box when re-examining existing analyses (for instance, all the examples used in the Piling Handbook). Alternatively, the equivalent earth pressure coefficient can be defined by means of a friction angle phi = 40°. This procedure also takes the defined wall friction angle into consideration.

A number of applications on the market also provide the option of a general increase in active earth pressure, apart from certain forms or earth pressure redistribution. In order to be able to check calculations performed with such an application, **GGU-GABION** also offers this possibility.

8.2.11 "Passive earth pressure" menu item

You can specify passive earth pressure preferences using this dialog box:



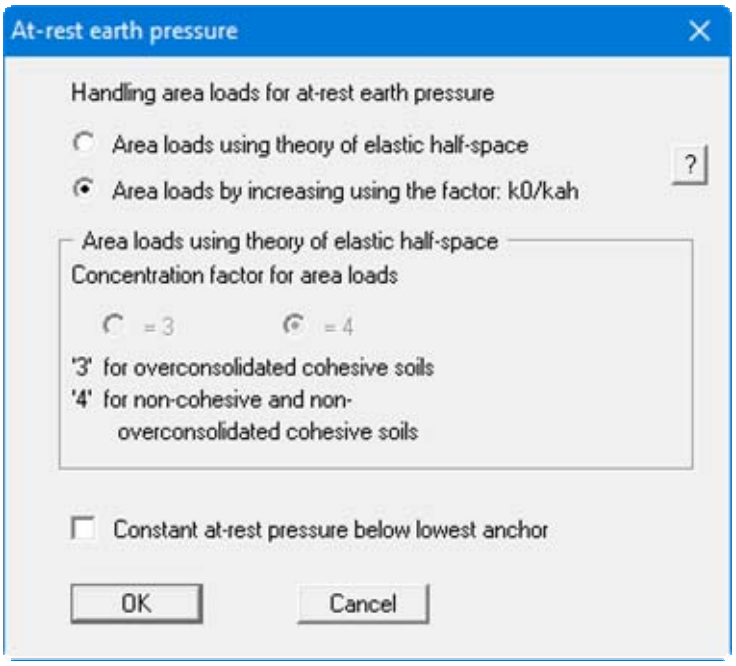
In the upper group box you specify the type of passive earth pressure calculation. The method of choice is "**DIN 4085:2011**". Only if you want to check examples from older literature or examine certain analysis results does it make sense to select a different method.

When you calculate intermediate construction phases large embedment depths may result, with large passive earth pressures for comparably small active earth pressures. This allows the location of the pressure line in the lower section of the wall to be displaced far to the earth side and even to leave the kernel width. In such situations you can limit the passive earth pressure to a user-defined value in the "**Special preferences**" group box and thus suppress the displacement of the pressure line.

8.2.12 "At-rest earth pressure" menu item

Using the at-rest earth pressure function, the area loads are calculated by increasing by the factor k0/kah in compliance with DIN 4085:2011 section 6.4.3.

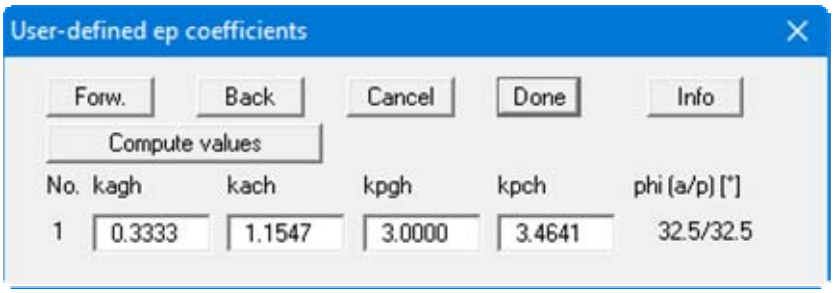
Alternatively, it is possible to determine the demands placed on the wall by area loads according to the theory of elastic half-space. The required concentration factor is specified in the following dialog box, if the upper check box is activated.



For gabion walls with at least two rows of geogrids, at-rest earth pressure can be kept constant from the lowest support (= anchor). This is effected by selecting the check box at the bottom of the dialog box.

8.2.13 "User-defined earth pressure coefficients" menu item

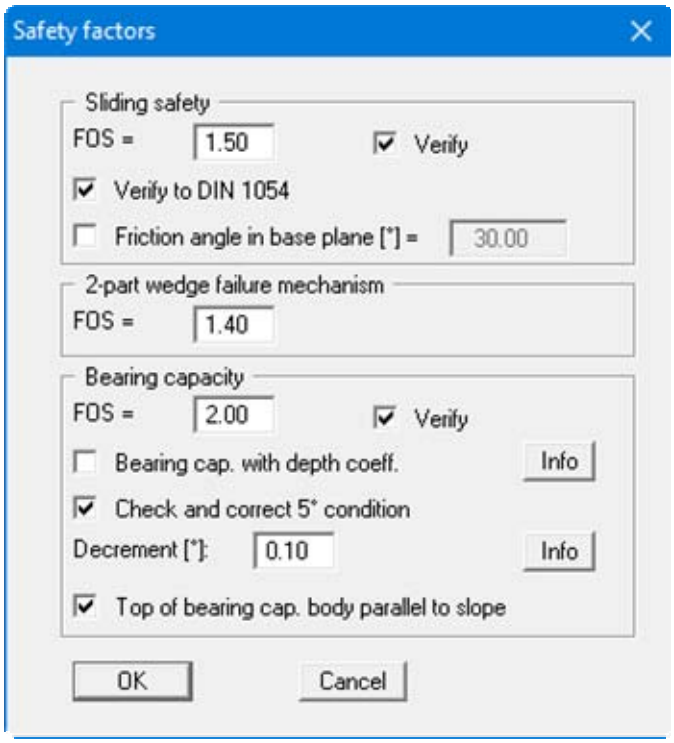
If you want to work with user-defined earth pressure coefficients instead of with those computed by the program you can enter them here. Enter the coefficients for horizontal ground. If necessary, **GGU-GABION** will convert them for sloping ground using equation values for k_{ah0} and $k_{ah\beta}$ (see "**Theoretical principles/Berms**", section 7.7).



If the "**Compute values**" button is pressed the earth pressure coefficients can be calculated by the program in accordance with the required standard and slope angle.

8.2.14 "Verifications/Safety factors" menu item

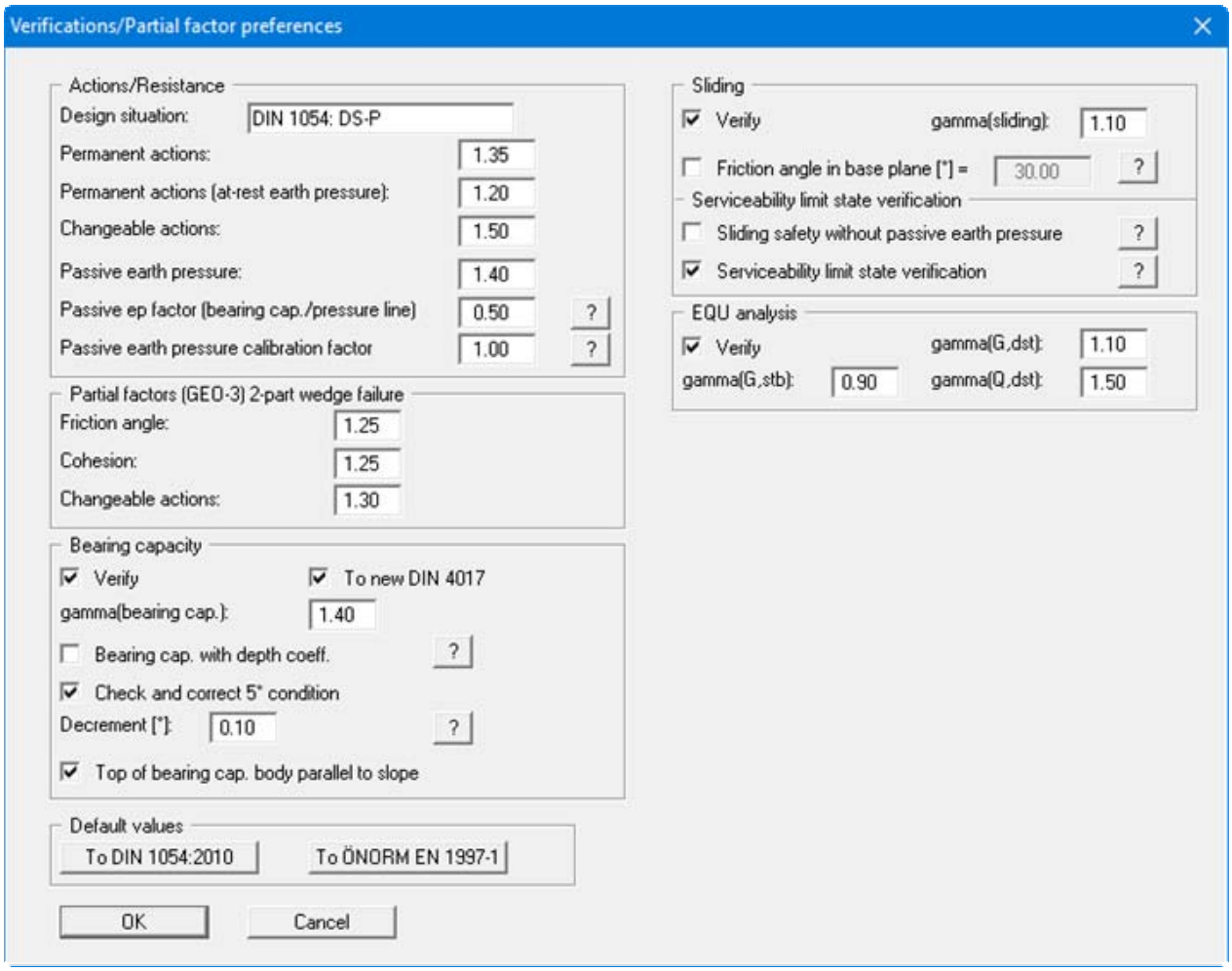
If you are analysing with ***global safety factors*** to DIN 1054 old, you can specify the verifications to be performed by the program by activating the corresponding check boxes in the dialog box shown below.



Then enter the safety factors for the individual verifications. The program's default values in the box are those required by the DIN 1054. If you do not want the program to perform the verifications, the corresponding check box must be deactivated. Pressing the two "**Info**" buttons, you will see further information.

8.2.15 "Verifications/Partial factors" menu item

If you are analysing with the ***partial safety factor concept*** , you will see the following dialog box for defining the partial factors.



In the "**Default values**" group box the partial factors for the various load cases and subsoil conditions given in the DIN 1054:2010 and in the EC 7 can be selected by means of the dialog box reached by clicking the "**To DIN 1054:2010**" button. The selected design situation is automatically entered at the top of the dialog box and shown later in the ***General legend***. However, you can also use your own designations. 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

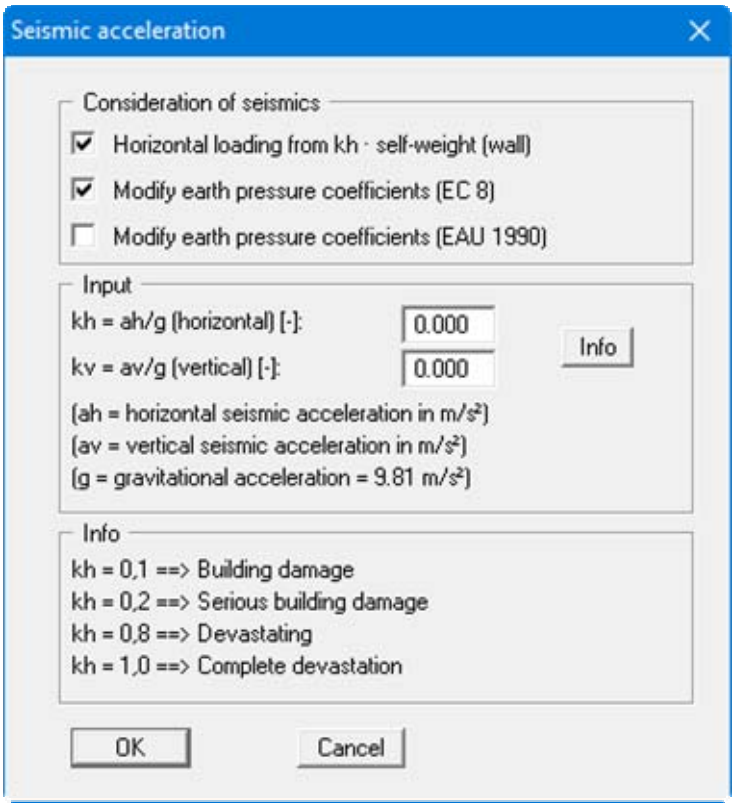
In addition, there is a seismic design situation (DS-E). In the DS-E design situation all partial factors = '**1,0**'. It is also possible to select the partial safety factors compliant with Austrian standards using the "**To ÖNORM EN 1997-1**" button.

The "**EQU analysis**" group box (limit state of loss of static equilibrium) is only available in EC 7.

8.2.16 "Seismic acceleration" menu item

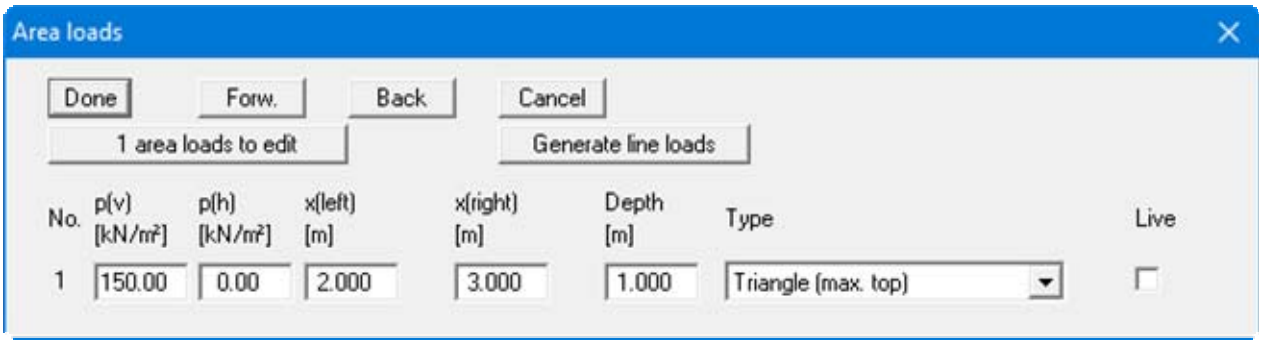
Seismic loads are given in multiples of gravitational acceleration. The following options, which can be selected individually or together, are available for taking seismic loads into consideration:

- The self-weight of the wall or equivalent wall is multiplied by the seismic acceleration factor and taken into consideration as additional horizontal loads in the analysis.
- Seismic loads are taken into consideration as described in EC 7 or EAU 1990, section 2.14, by increasing the active earth pressure coefficients and reducing the passive earth pressure coefficients.



8.3.1 "Area and line loads" menu item

Using this menu item you define area loads and line loads.

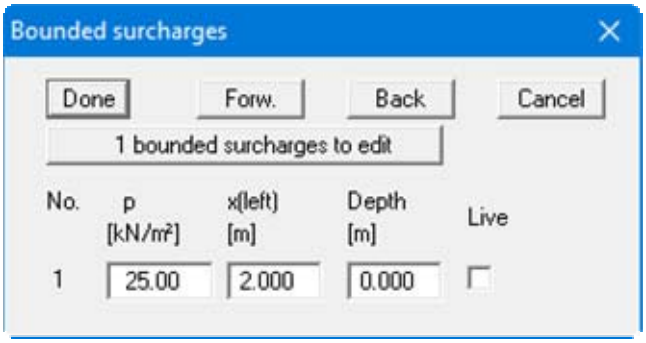


The "**x area loads to edit**" button allows you to determine the number of area loads to be considered. Subsequently you can enter size, ordinates and depth of the area loads. You must also enter the "**Type**" (shape) of the resultant horizontal forces on the wall (see also section 7.8).

Using the "**Generate line loads**" button, line loads acting vertically on the wall may be considered adopted as area loads (see section 7.9).

8.3.2 "Bounded surcharges" menu item

Bounded surcharges are defined in the following dialog box. You are first queried whether the surcharges are entered to be on the active or the passive side. The following dialog box opens for ***active side*** input:

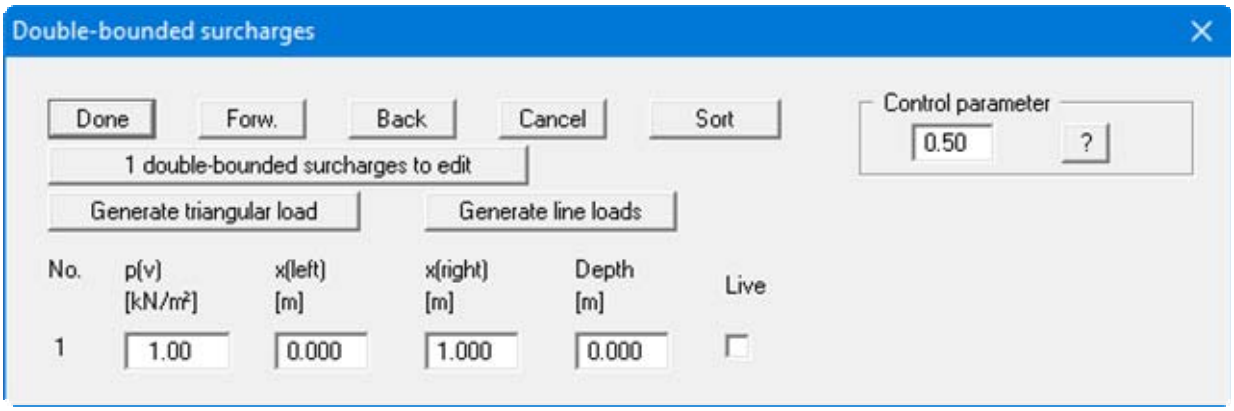


Using "**x bounded surcharges to edit**" you can determine the number of bounded surcharges. You can then enter the size of the surcharges, the ordinates and the depth in metres from the top of the wall or as absolute heights.

Input of bounded surcharges on the ***passive side*** is analogous, but without the "**Live**" check box.

8.3.3 "Double-bounded surcharges" menu item

Double-bounded surcharges are defined using this menu item. You are first queried whether the surcharges are entered to be on the active or the passive side. The following dialog box opens for ***active side*** input:



The number of surcharges can be edited using the "**x double-bounded surcharges to edit**" button. Then enter the size of the surcharges, the ordinates and the depth in metres from the top of the wall or as absolute heights.

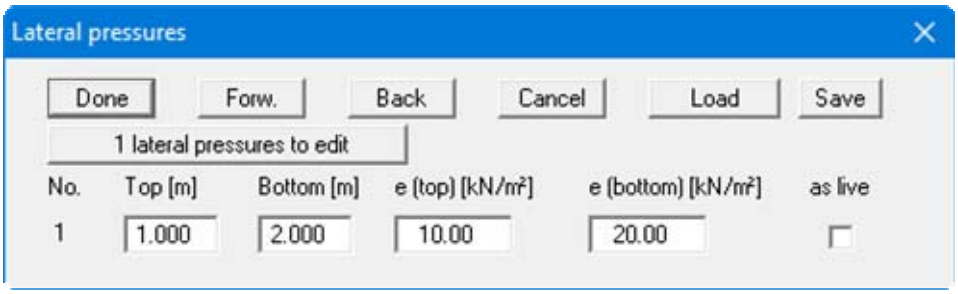
Double-bounded surcharges consist of 2 bounded surcharges with opposing signs. Whether or not the linear component of the resulting earth pressure (see Figure 7 in section 7.10) may be adopted at its full value for the bounded surcharge with the negative sign has not yet been conclusively clarified. This can be influenced using the control parameter:

- Control parameter = 0.0 ➔ no linear component
- Control parameter = 1.0 ➔ linear component at full value

Input of double-bounded surcharges on the ***passive side*** is analogous, but without the "**Live**" check box. There is no linear component for bounded surcharges on the passive side, meaning that a control parameter is not necessary.

8.3.4 "Lateral pressures" menu item

If, in addition to the diverse possibilities for determining earth pressure on the wall, you also need to take surcharges on the active side into account, this is where to enter them.

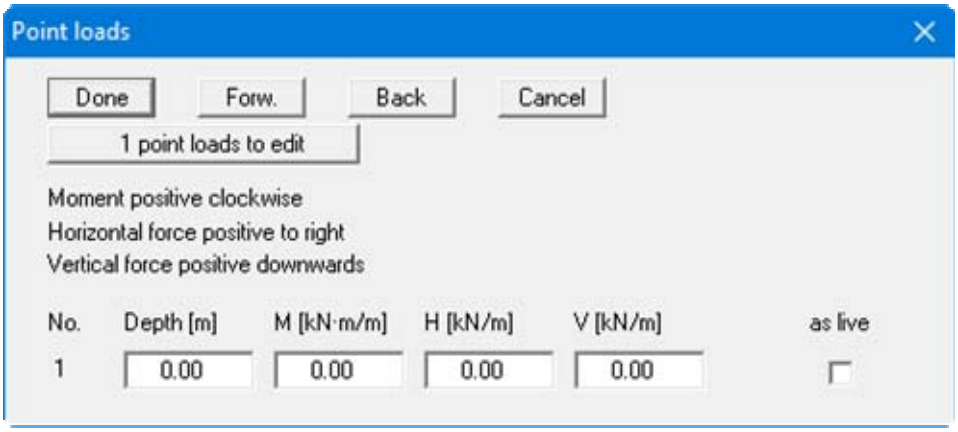


The number of lateral pressures can be modified using the "**x lateral pressures to edit**" button**.** Then enter the ordinates in metres from the top of the wall or as absolute heights, and the values for the lateral pressures. When analysing with ***partial safety factors*** you will also see the "**as live**" check box because of the differentiation between permanent and changeable lateral pressures.

If DIN 1054:2005 is adopted the static equilibrium for wind loads must be determined to DIN 1055. Wind loads are taken into consideration as surcharges. EQU analysis (limit state of loss of static equilibrium) is always performed if EC 7 is adopted.

8.3.5 "Action boundary conditions" menu item

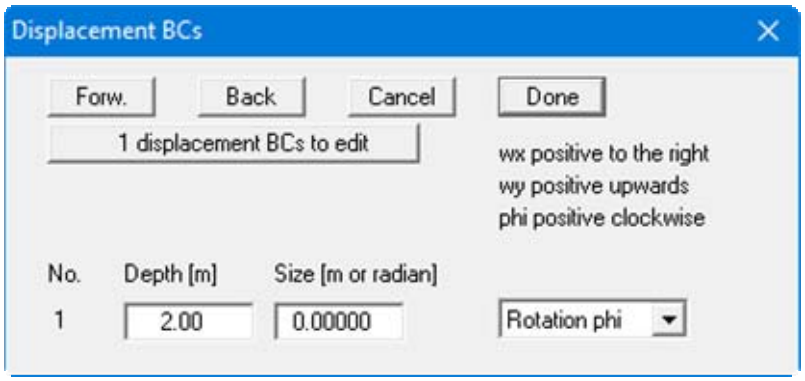
You can introduce additional action boundary conditions anywhere along the wall.



The direction of the forces is defined by means of the sign. In the example above a horizontal force of 15 kN/m has been entered at the top of the wall, acting towards the left. When analysing with the new ***partial safety factors*** you will also see the "**Live**" check box because of the differentiation between permanent and changeable loads.

8.3.6 "Displacement boundary conditions" menu item

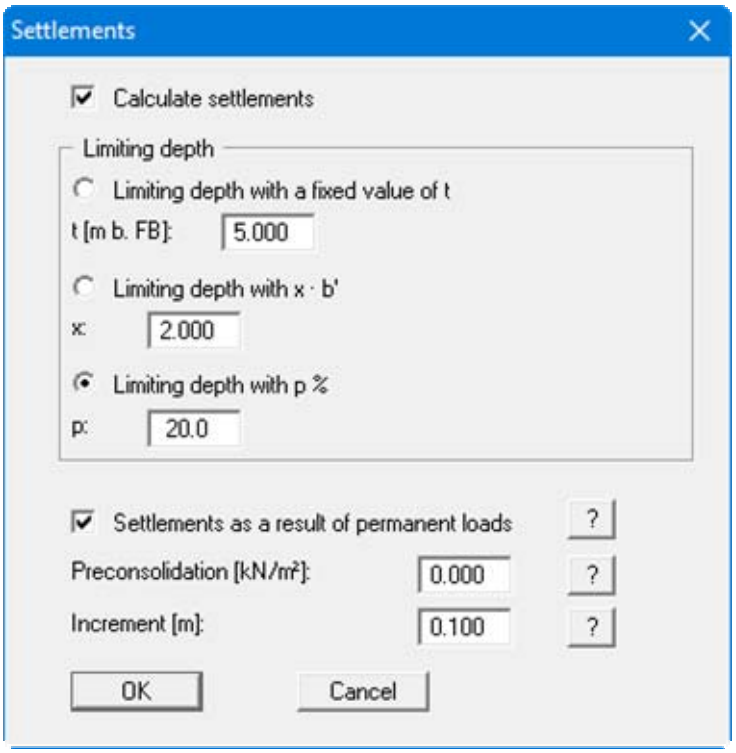
You can introduce additional displacement boundary conditions anywhere along the wall.



In the example above a "**Rotation phi**" of the wall of 0.0 has been entered 2.0 m below the top of the wall. The "**Displ. wx**" and "**Displ. wy**" switches stand for the horizontal and vertical displacements; the direction is defined by means of the sign.

8.3.7 "Settlements" menu item

Using this menu item, the type of limiting depth calculation can be defined in three different ways.



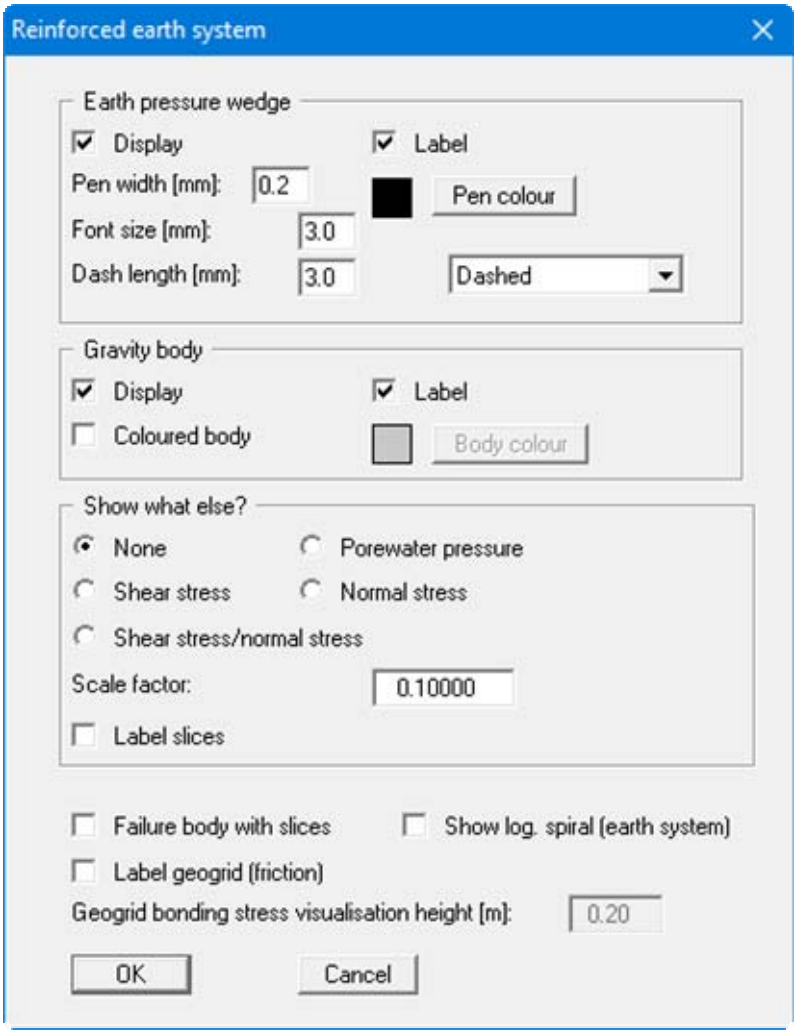
According to DIN 4019, settlement analysis can be terminated at a depth (limiting depth) in which the stress from soil self-weight and ground surcharges (overburden stress) is greater than 20% of the stresses caused by the body. The stress analysis for determining the limiting depth is performed for the mean soil pressure in the characteristic point. The lateral stresses on the active side of the wall are adopted for this limiting depth calculation. If the limiting depth lies below the depth of the lowest soil layer for this type of calculation, the values of this lowest layer are adopted for the settlement analysis.

In addition, a "**Preconsolidation**" can be defined. This preconsolidation in kN/m² is subtracted from the existing soil pressure. Settlement analysis will then be performed with the reduced values. The overburden stress is also reduced by this amount when calculating the limiting depth.

You can also specify whether a settlement analysis is to be performed at all. The "**Increments**" are only important for the graphical visualisation of the stress diagram.

8.4.1 "Graphic preferences" menu item

If you are analysing a gabion wall with geogrids you can define preferences for the visualisation here.



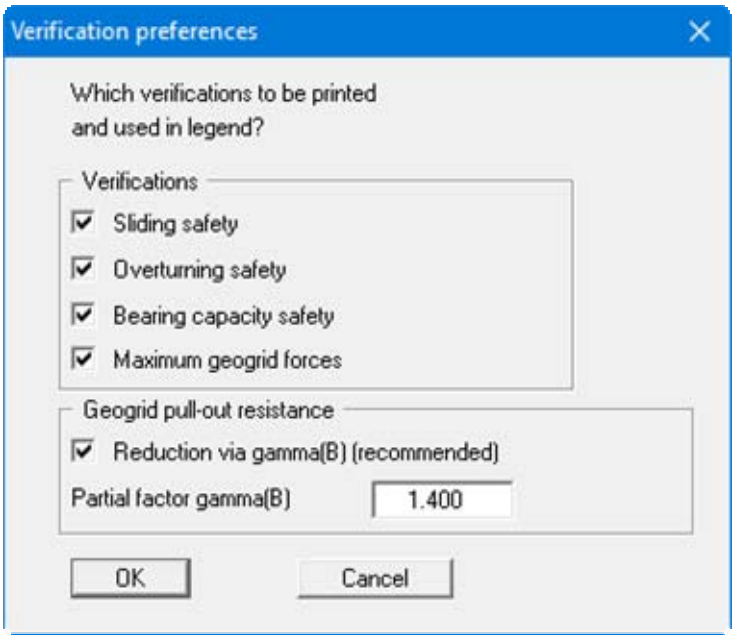
The earth pressure wedge and gravity body are explained in section 7.21.2. The visualisation of these elements can be influenced in the two upper group boxes of this dialog box.

In the "**Show what else?**" group box you define whether the visualisation of the two-part wedge failure mechanism with the lowest factor of safety should include additional information (e.g. the shear stress). The values for each slip body slice are shown perpendicular to the slip surface. The scale is given using the input field for the "**Scale factor**".

The failure body can be visualised together with slices. Moreover, the logarithmic spiral (bearing capacity) of the reinforced soil system can be shown. The geogrid bonding stresses are automatically calculated from the surcharges. These can also be visualised. The visualisation height can be entered in the input box at the bottom.

8.4.2 "Verifications" menu item

Using this menu item you can specify the entries made in the legend and the output table.

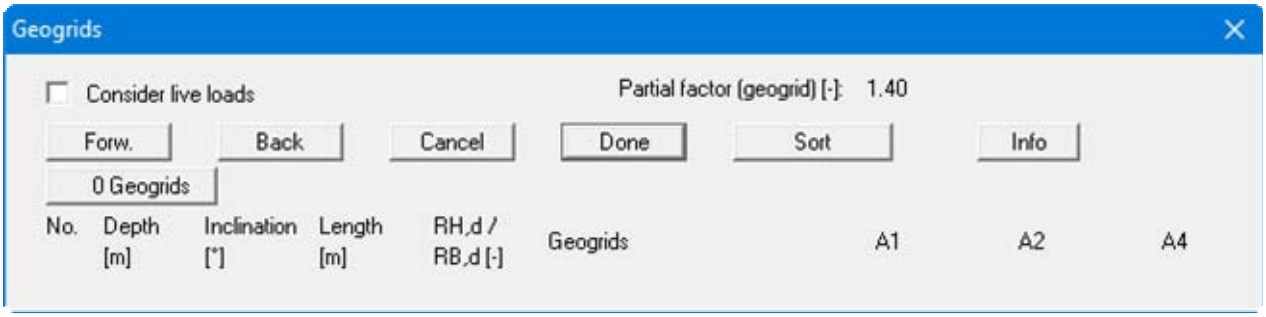


For example, if you do not want to calculate the bearing capacity safety, you can turn off the bearing capacity verification.

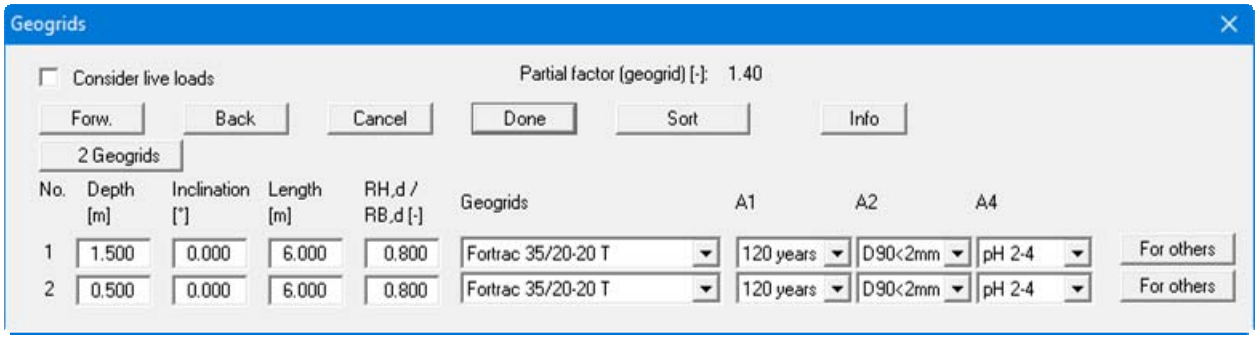
8.4.3 "Enter geogrids manually" menu item

Define the position and material properties of geogrids using this menu item. The defined geogrids are displayed after closing the dialog box.

When using geogrids it is possible to directly adopt certain products of various geogrid manufacturers. When the program starts the "**File/New**" or "**Editor 1/Analysis options**" dialog box therefore has the "**Geogrids via company products**" check box activated (see section 8.1.1). A dialog box will open, where the number of geogrids must be defined using the "**0 Geogrids**" button.



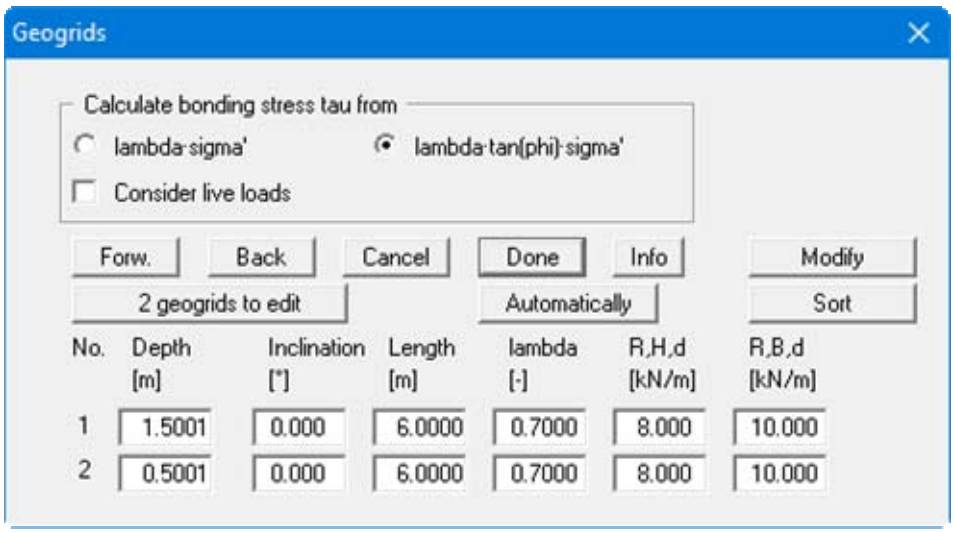
The required company product can then be selected for each of the geogrids. The coefficients defined for a geogrid, the length and the inclination can be adopted for the remaining geogrids by using the "**For others**" button.



A maximum of 8 geogrids are shown in the list. You can navigate through the list using the "**Forw.**" and "**Back**" buttons. Using the "**Sort**" button, you can sort the geogrids according to depth. For example, if you wish to delete the top geogrid from the table simply assign a very small depth (e.g. "-100") and press "**Sort**". The geogrid has now moved to the lowest position in the table. Now reduce the number of geogrids by **1**.

If geogrids by a different manufacturer are required, go to the menu item "**Editor 1/Geogrids table values**". Here, products by different manufacturers can be selected (see section 8.4.10).

If you do not work with company products, the following dialog box opens:



Where:

- $R_{B,d}$ = design resistance of geogrid;
- $R_{H,d}$ = design resistance of geogrid at junction (generally = 0.8 · $R_{B,d}$).

The ***bonding stress tau*** is automatically calculated by the program. Define the subsequent actions in the upper group box of the dialog box. Usual practice is to calculate the bonding stress tau from the ***effective stress*** (sigma') and the ***tangent of the friction angle*** phi and then to reduce this value by ***lambda***. The alternative to this is to calculate the bonding stress from a multiplication of the effective stress and a friction coefficient. In this case lambda no longer represents a reduction factor, but instead a friction coefficient.

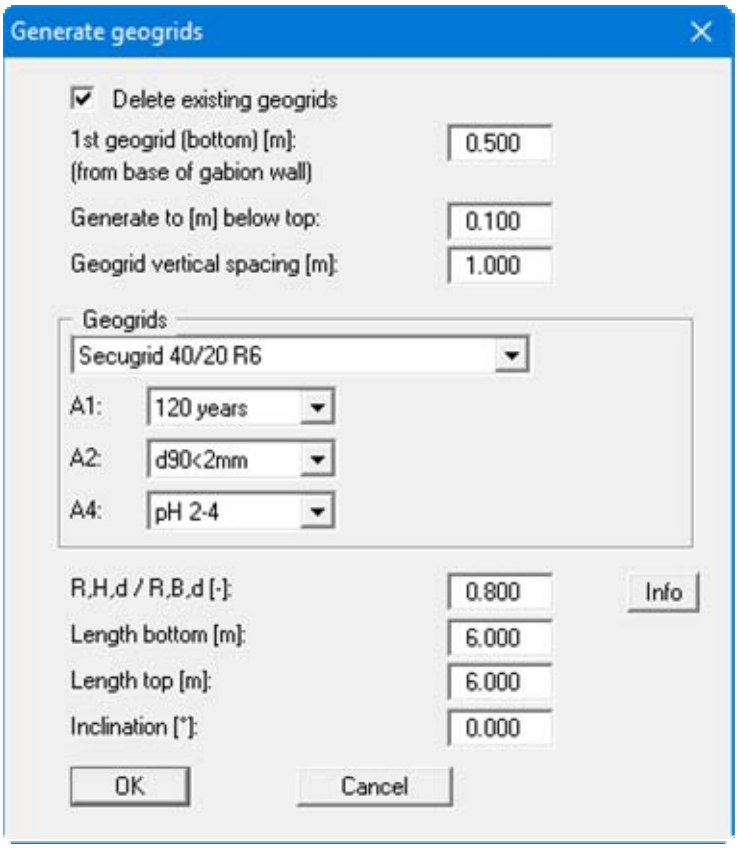
If you would rather define the geogrid yourself, click on the "**x geogrids to edit**" button and then enter the required number of geogrids. A maximum of 8 geogrids are shown in the list. You can navigate through the list using the "**Forw.**" and "**Back**" buttons.

Using the "**Modify**" button it is possible to simultaneously assign all geogrids certain values or to change the values entered. This function is available by going to the "**Reinforced soil system/Modify geogrids**" menu item (see section 8.4.5).

The "**Automatically**" button allows you to generate a completely new geogrid system. This function is also available by going to the "**Reinforced soil system/Generate geogrids**" menu item (see section 8.4.4).

8.4.4 "Generate geogrids" menu item

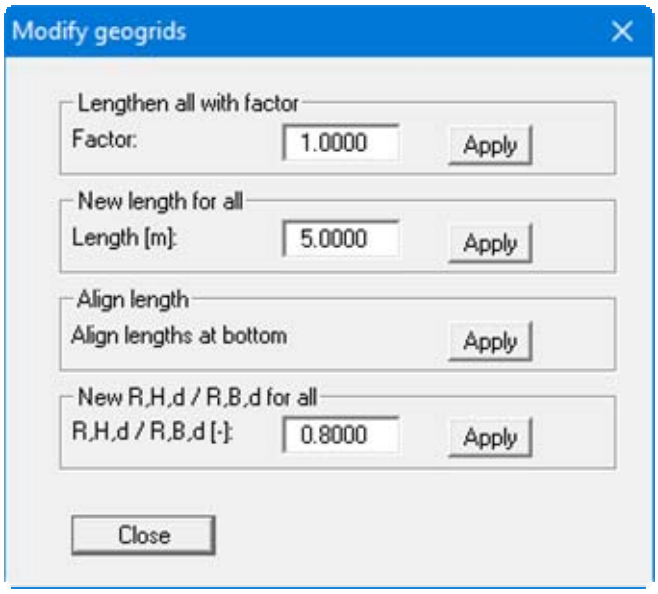
You can generate a completely new geogrid system using this menu item. Activate the "**Delete existing geogrids**" check box. If this check box is not activated you can add further geogrids to the previously existing ones.



If you do not work with company products, the dialog box for this menu item corresponds to the dialog box displayed when you press the "**Automatically**" button in the "**Reinforced soil system/Enter geogrids manually**" menu item.

8.4.5 "Modify geogrids" menu item

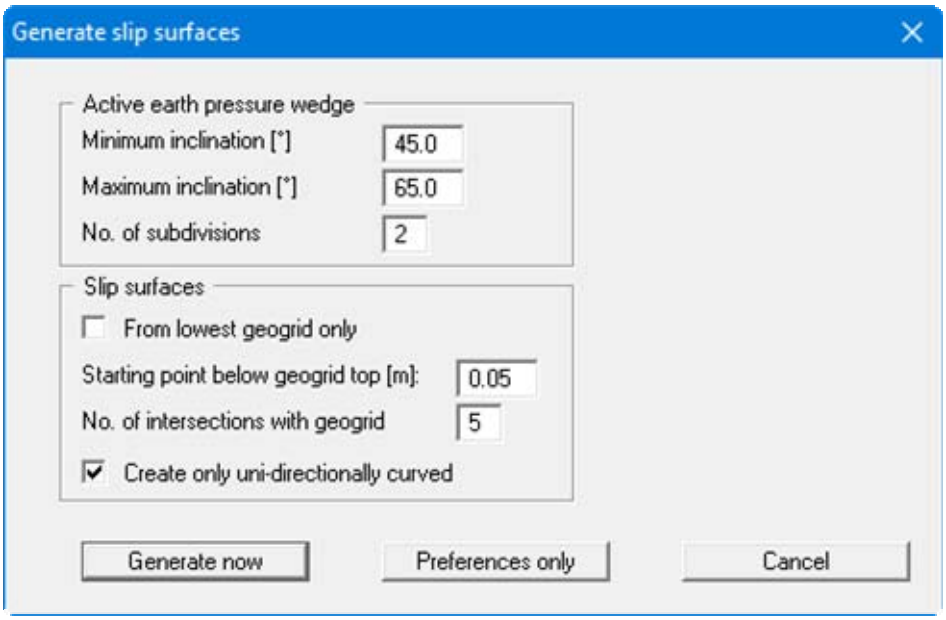
Using this menu item you can edit the data for all geogrids simultaneously in a single step. Enter a value and start the modification using the respective "**Apply**" button following the desired action. Only this action is subsequently performed.



If you do not work with company products, the dialog box for this menu item corresponds to the dialog box displayed when you press the "**Modify**" button in the "**Reinforced soil system/Enter geogrids manually**" menu item.

8.4.6 "Generate slip surfaces" menu item

By entering only a little data you can generate a number of slip surfaces using this menu item. Slip surfaces with 2 polygon sections are generated (two-part wedge failure mechanism).



The two-part wedge failure mechanism consists of a slip surface corresponding to the active earth pressure wedge. According to theory this earth pressure wedge has an inclination of 45° + φ/2 (see section 7.21.2). Values smaller than 45° are thus not possible. Values greater than 65° only result if the angle of friction of the soil is greater than 40°. The default values in the dialog box therefore cover the usual construction range. The input for "**No. of subdivisions**" indicates the number of ***intermediate angles*** to be generated between the two limiting angles.

In the lower group box it is possible to influence the left side of the two-part wedge failure mechanism. If you activate the "**From lowest geogrid only**" check box (not recommended), only failure mechanisms with ***bending points*** on the lowest geogrid are generated. The "**Starting point below geogrid top**" value defines the left-hand point of the failure mechanism. The input for "**No. of intersections with geogrid**" defines the number of central point of the failure mechanism.

The effects of the individual inputs are difficult to describe in words. So just try editing the input in the dialog box and then click "**Generate now**". You will see immediately the failure mechanisms generated by the program in the graphics. If you close the dialog box using the "**Preferences only**" button, the current analysis results will not be affected. However, when you start a new analysis that includes geogrids these settings will be used to generate the appropriate failure mechanisms.

8.4.7 "Geogrid forces" menu item

During analysis of the defined failure mechanisms the program calculates the maximum geogrid forces and divides these by the safety factor of the respective failure mechanism. Using this menu item you can view the maximum geogrid forces for each geogrid layer.

8.4.8 "Sliding, overturning, bearing capacity" menu item

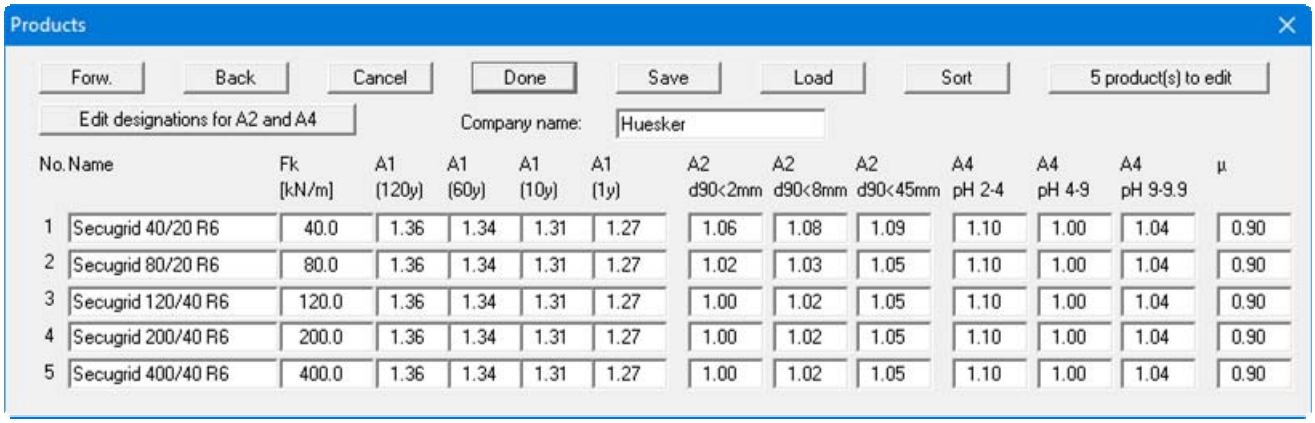
Verification is in accordance with the notes in the "**Theoretical principles**" section. You will see the results in dialog boxes.

8.4.9 "Calculate earth pressure + weight" menu item

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

8.4.10 "Geogrids table values" menu item

The geogrids of various manufacturers can be imported and supplemented using this menu item. Editing the data is not recommended! In general, in **GGU-GABION** importable files can be requested from the geogrid manufacturers. Data files from various manufacturers are installed in the program folder when the program is installed.



No.	Name	Fk [kN/m]	A1 (120y)	A1 (60y)	A1 (10y)	A1 (1y)	A2 d90<2mm	A2 d90<8mm	A2 d90<45mm	A4 pH 2-4	A4 pH 4-9	A4 pH 9-9.9	μ
1	Secugrid 40/20 R6	40.0	1.36	1.34	1.31	1.27	1.06	1.08	1.09	1.10	1.00	1.04	0.90
2	Secugrid 80/20 R6	80.0	1.36	1.34	1.31	1.27	1.02	1.03	1.05	1.10	1.00	1.04	0.90
3	Secugrid 120/40 R6	120.0	1.36	1.34	1.31	1.27	1.00	1.02	1.05	1.10	1.00	1.04	0.90
4	Secugrid 200/40 R6	200.0	1.36	1.34	1.31	1.27	1.00	1.02	1.05	1.10	1.00	1.04	0.90
5	Secugrid 400/40 R6	400.0	1.36	1.34	1.31	1.27	1.00	1.02	1.05	1.10	1.00	1.04	0.90

Other manufacturer's products can be imported using the "**Load**" button. When importing it is possible to specify whether to import the products of only one manufacturer or the products of several manufacturers together in one list. If you store your geogrids list in a "**GGU-GABION.ggu_geo**" file on the program level, the product list is automatically imported when the program is started.

In order to be able to select the geogrid products, the "**Geogrids via company products**" check box must be activated in the menu item "**File/New**" or "**Editor 1/Analysis options**" (see section 8.1.1). We also recommend activating the "**Store company products in record**" check box. 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 geogrid products can be selected in the menu item "**Reinforced soil system/Enter geogrids manually**" or "**Reinforced soil system/Generate geogrids**".

8.5.1 "Info" menu item

You will see information on the current system in a message box.

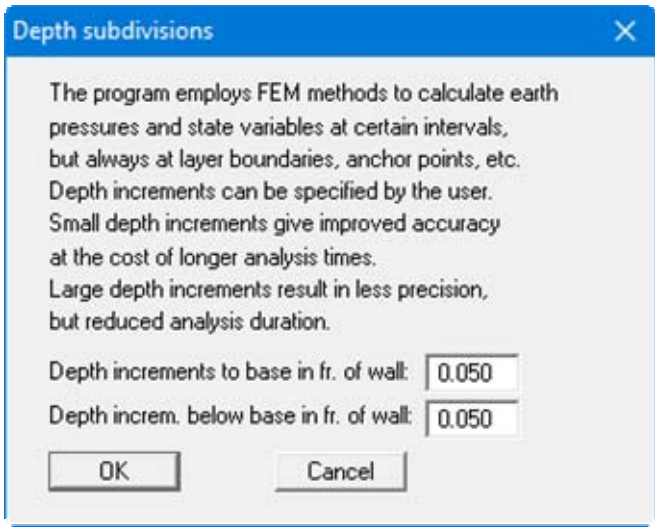
8.5.2 "Special preferences" menu item

The program performs a multitude of plausibility checks. After starting the analysis the preferences specified by the user are displayed in an info box; for problematical preferences separate information or warning are displayed. It is therefore recommended to leave the "**Show warnings in future**" check box activated.

If you do not want to see the automatic display when the analysis starts, deactivate the check box. You can subsequently view your special preferences using this menu item.

8.5.3 "Depth subdivisions" menu item

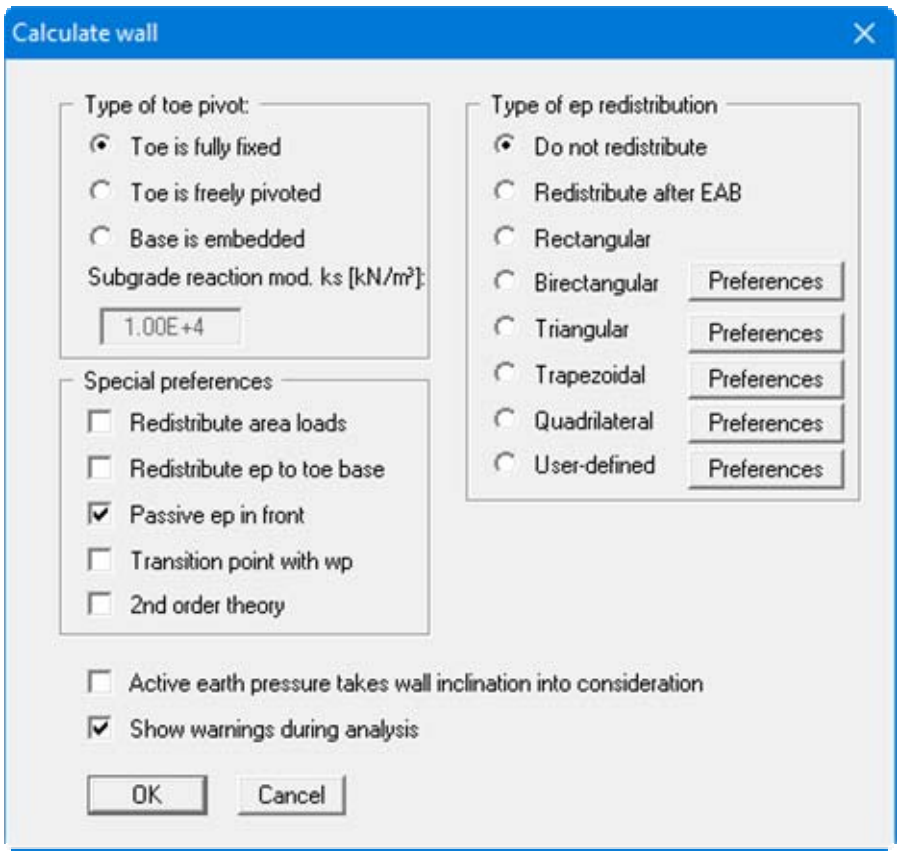
GGU-GABION uses the finite element method, which requires the system to be divided into a number of finite elements (rods) (see section 7.14). You can specify the size of these depth increments for the region above and below the base in front of wall.



8.5.4 "Analyse" menu item

8.5.4.1 Start dialog box

Once you have entered all data required to fully describe the system it can be analysed. After going to the "**System/Analyse**" menu item a ***start dialog box***, divided into three group boxes, opens (see below for descriptions). You can also initiate the analysis using the [**F5**] function key and see the same start dialog box.



When all preferences have been specified to your satisfaction press the "**OK**" button to start the analysis. The program first performs comprehensive plausibility checks on your input and notifies you of any inconsistencies. The actual analysis of the system then follows.

Once analysis is complete, the results are presented in message boxes and then visualised graphically on the screen.

8.5.4.2 "Type of toe pivot" group box

In the first group box of the start dialog box of the "**System/Analyse**" menu item you can specify the toe pivot. This is, of course, not possible in systems without geogrids. In such cases, the toe is always ***fully fixed***.

In systems with geogrids, a subgrade modulus for the base can be entered in place of "**Toe is fully fixed**" or "**Toe freely pivoted**". Activate the "**Base is embedded**" check box and enter a subgrade reaction.

8.5.4.3 "Special preferences" group box

In the "**Special preferences**" group box of the start dialog box of the "**System/Analyse**" menu item you can specify whether any ***area loads*** should be included in an earth pressure redistribution. You may "**Redistribute ep to toe base**" and not, as is usual, to the load transition point.

For an analysis employing the ***global safety factors*** used in DIN 1054 (old), the load transition point can be calculated with or without any prevalent water pressure ("**Transition point with wp**" check box). In addition, when analysing with the old safety factors you can specify whether the program should place the passive earth pressure in front of the wall. If the "**Passive ep in front**" check box is not activated, the passive and active earth pressures are added (superimposed). The new standard does not allow addition of the active earth pressure (action) and the passive earth pressure (resistance). The "**Passive ep in front**" check box therefore does not appear if the ***partial safety factor concept*** is selected, because the passive earth pressure is always in front.

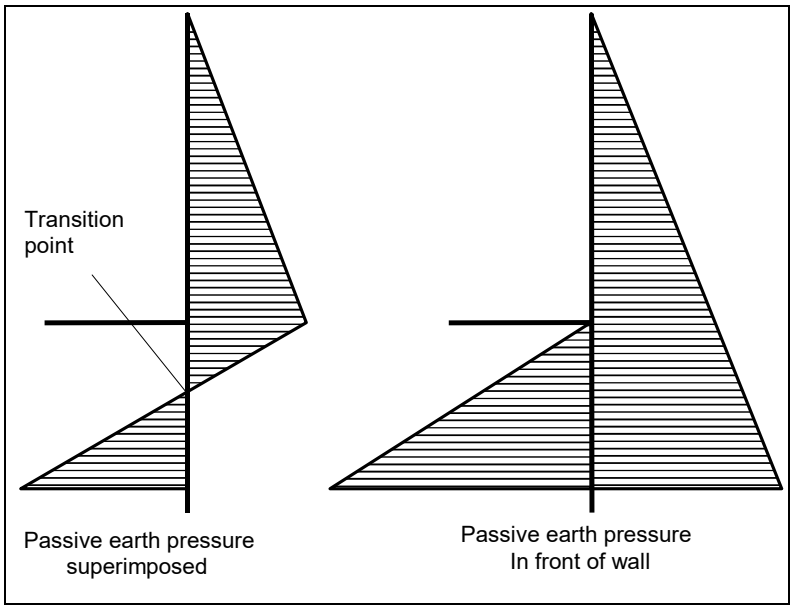


Figure 21 Passive earth pressure in front or superimposed

Finally, you also have the option of having the analysis performed according to "**2nd order theory**". This entails iteration, which increases computation time.

8.5.4.4 "Type of redistribution" group box

In the "**Type of redistribution**" group box of the start dialog box of the "**System/Analyse**" menu item the following options are available:

- **Do not redistribute**
  The analysis is performed using classical earth pressure redistribution.
- **Redistribute after EAB**
  For in-situ concrete walls, R 70 provide a total of 9 redistribution figures, depending on the location of anchors. **GGU-GABION** will automatically choose the appropriate figure. If no suitable figure is found, an error message will appear.
- "**Rectangular**"
  Earth pressure is redistributed in the form of a rectangle.
- "**Birectangular**"
  Earth pressure is redistributed in birectangular form. The relationship between the top and bottom earth pressure ordinates (eaho/eahu), as well as depth of the subdivision x, can be specified.

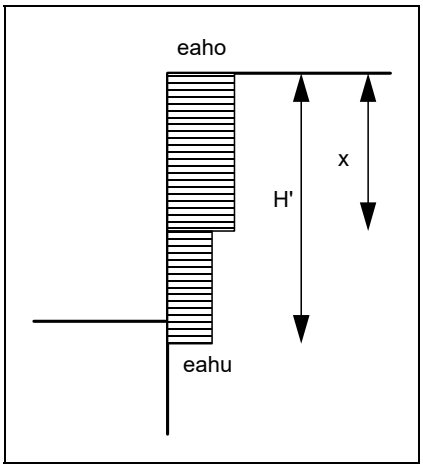


Figure 22 Birectangular earth pressure redistribution

- "**Triangular**"
  Earth pressure is redistributed in the form of a triangle. The associated "**Preferences**" button enables you to determine the position of the maximum (top, central, bottom).

- "**Trapezoidal**"
  Earth pressure is redistributed in the form of a trapezoidal. The associated "**Preferences**" button enables you to determine the eahu/eaho ratio.

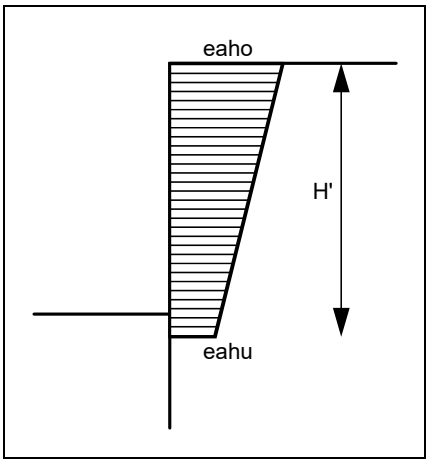


Figure 23 Earth pressure redistribution in a trapezoidal

- "**Quadrilateral**"
  The earth pressure is redistributed in a quadrilateral. After clicking the "**Preferences**" button you can select the ordinates at which the maximum should occur, either by entering the depth or, alternatively, the anchor positions. Activate the appropriate check boxes at the left of the dialog box. The ordinate at the load transition point is defined by the ratio eaho/eahu.

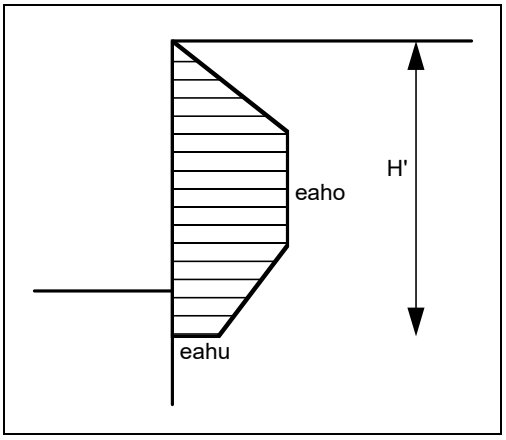
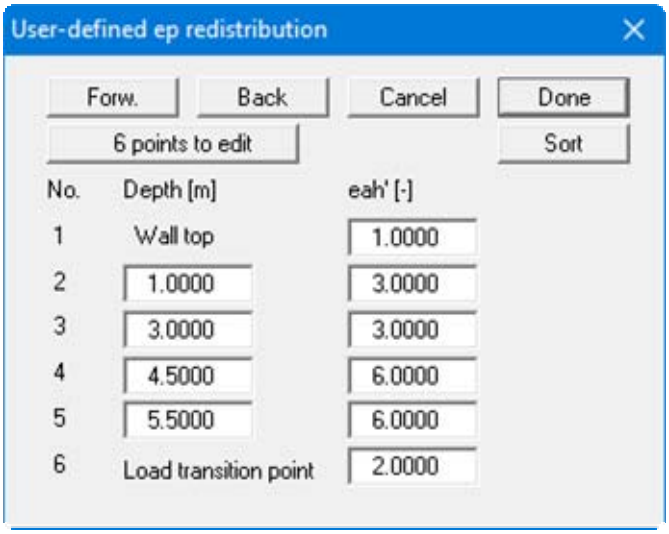


Figure 24 Earth pressure redistribution in a quadrilateral

- **User-defined"**
  If none of the offered redistribution figures meet your requirements, you have the option of creating your own by defining a polygon using the "**Preferences**" button.



You can define a number of depths between the top of the wall and the transition point, to each of which you can appoint appropriate earth pressure ordinates. In subsequent computations, earth pressure will be redistributed in exactly the area defined by the polygon you have created. Using the example in the above dialog box the following diagram is obtained:

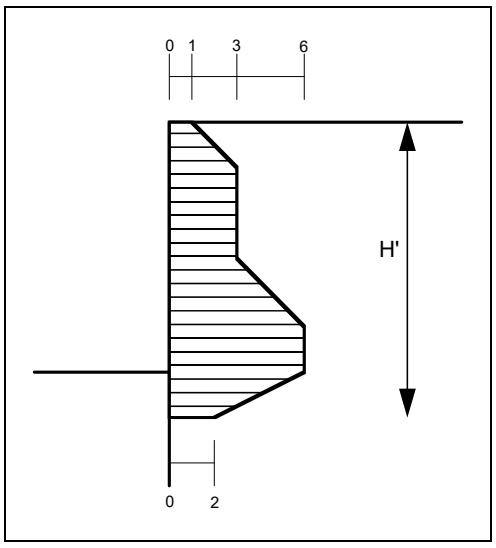
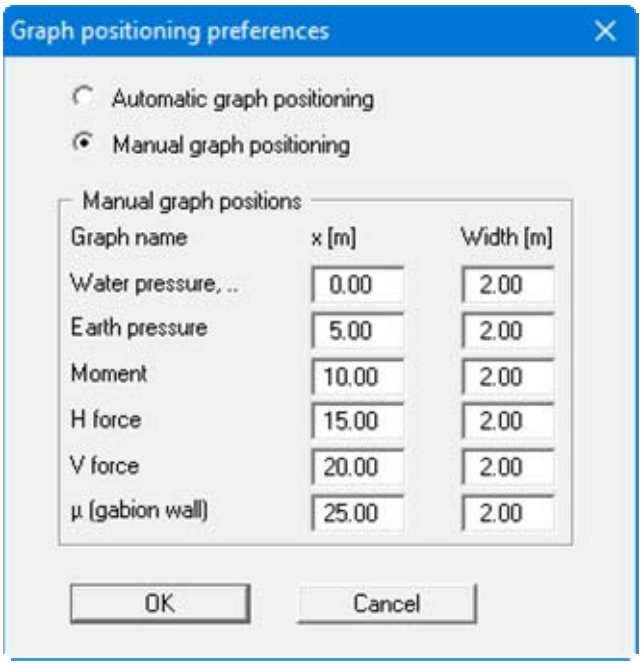


Figure 25 User-defined earth pressure redistribution

8.5.5 "Graph positioning preferences" menu item

If you are not happy with the automatic graph arrangement you can arrange them to suit your needs using this menu item. First, activate the "**Manual graph positioning**" radio button.

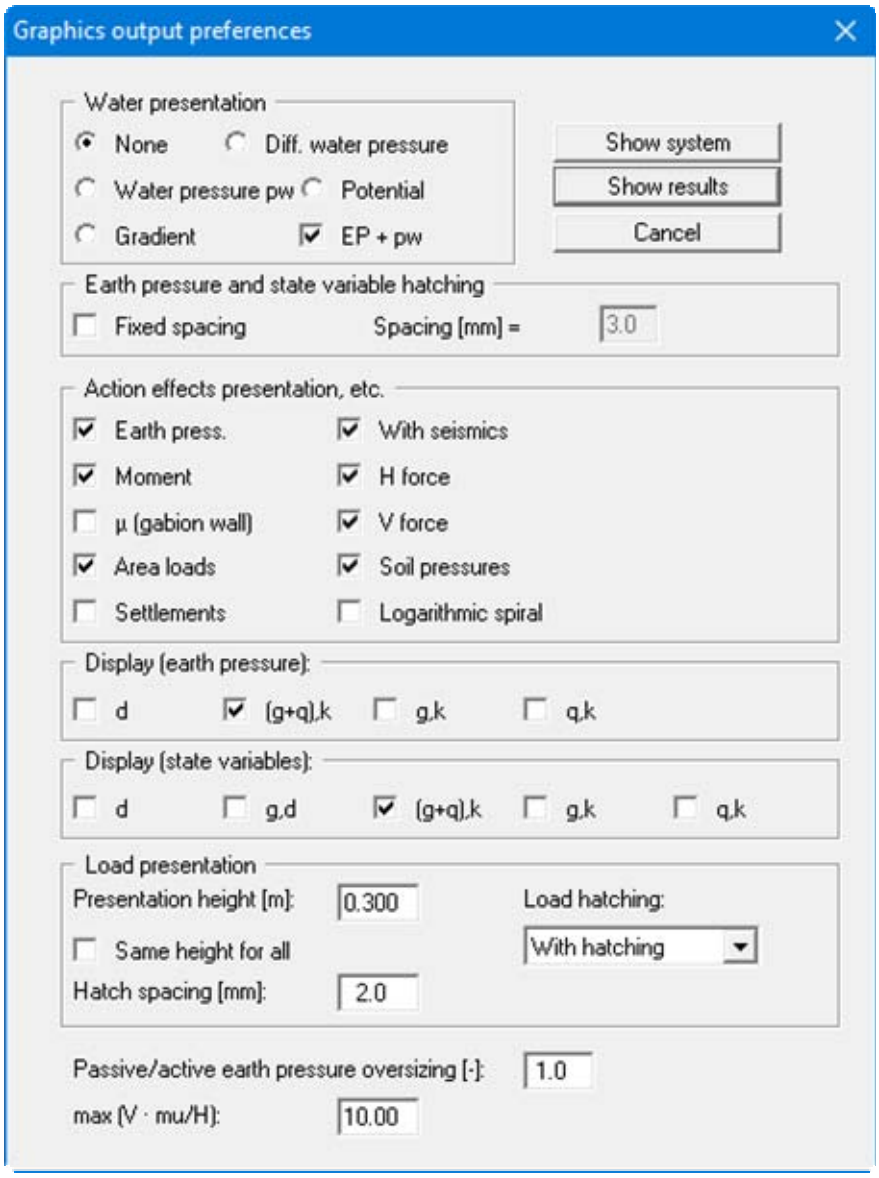


The graphs will then be shown at position "**x**" (central) with the specified "**Width**".

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

8.5.6 "Graphics output preferences" menu item

Among other things, the screen graphics consist of several diagrams, presenting depth-oriented results. This menu item opens a dialog box which allows you to visualise the selected state variables on the screen by activating the corresponding check boxes.



On the whole, the dialog box is self-explanatory. For example, if the "**Settlements**" check box is activated, the stresses below the footing for the characteristic point are visualised. When "**EP + pw**" is selected, the sum of earth pressure and water pressure (pw) is displayed in the diagram for earth pressure.

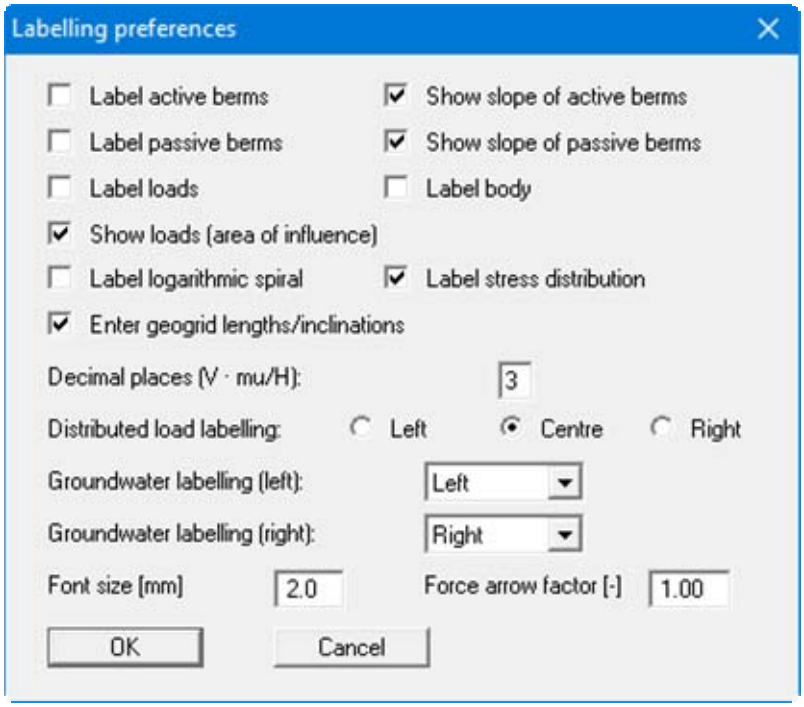
When analysing using ***partial safety factors*** you will also see the group boxes "**Display (earth pressure):**", "**Display (state variables):**" and "**Display (displacement):**", in which you can activate display of the permanent (g) and/or live loads (q). In addition, the design values (d) can be displayed.

You can also specify hatching and the presentation height of loads. If the "**Same height for all**" check box is not selected, load visualisation is based on load size, the height of the presentation indicating the maximum load.

Leave the dialog box by pressing "**Show system**". If the system has already been analysed you can leave the box by pressing "**Show results**" and then view the result graphics on the screen.

8.5.7 "Labelling preferences" menu item

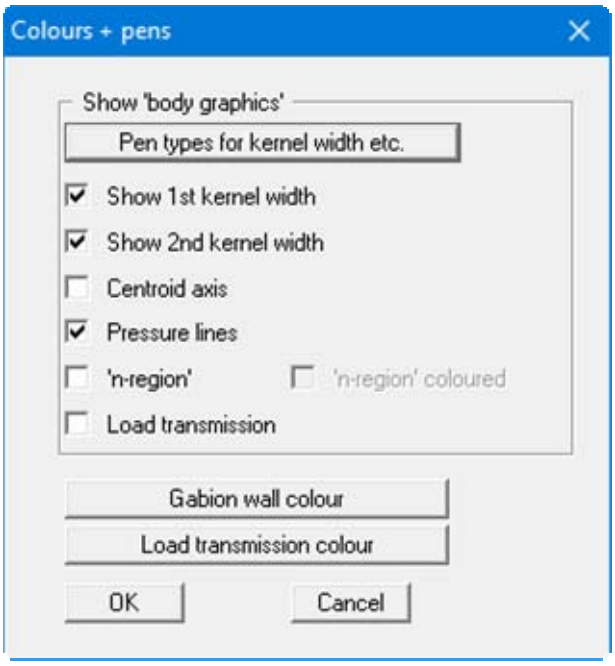
This menu item allows you to specify labelling preferences for the system visualisation and the result graphics.



In the dialog box, you activate the required check boxes and select the preferences for alignment or font sizes. In addition, groundwater and berms labelling can be edited.

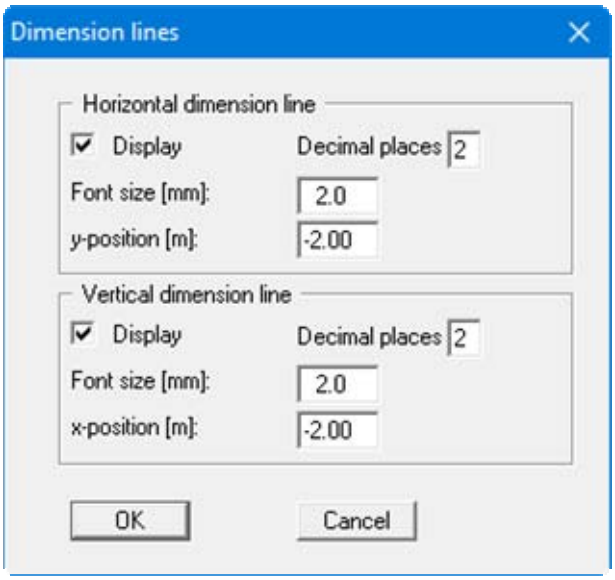
8.5.8 "Body pens + colours" menu item

The gabion body is visualised with a multitude of graphic elements (centroid axis, kernel width, etc.). Individual elements of the visualisation can be subdued, the pen types (dashed, dotted, etc.) and fill colours changed.



8.5.9 "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 gabion body is defined by means of the "**y-position**" for the horizontal dimension line and "**x-position**" for the vertical dimension line. Negative values define a position above or to the left of the gabion body. All values are in metres in the scale selected (see the menu item "**Page size + margins/Manual resize (editor)**" in section 8.8.3).

> 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.

8.5.10 "Display system" menu item

Once a system has been analysed, all the state variables are automatically shown on screen. So as not to overburden the screen, certain elements of the system (for example, surcharges) are no longer shown. If you want to view all the system data without state variables, clicking this menu item will enable you to do so.

8.5.11 "Display results" menu item

After a system has been analysed, all state variables are automatically presented on the screen. If you used the menu item "**System/Display system**" to return to the system visualisation, you can go to this menu item to return to the result presentation without renewed analysis. Of course, this only works if the system has already been analysed.

8.6.1 General note

All the following information regarding the analysis results can be sent to the printer or to a file by going to the menu items "**File/Print and export**" or "**File/Print output table**". However, it is always more desirable to be able to check over the analysis results without wasting a lot of paper. The following menu items were included in the program for just this reason.

8.6.2 "Earth pressure redistribution" menu item

Here you are provided with information relating to earth pressure redistribution. This can be particularly interesting if you have calculated according to the EAB, in which case it is probably a good idea to have a copy of the EAB at hand.

8.6.3 "Main output summary" menu item

A message box appears containing the main system analysis parameters. This function can also be initialized pressing the [**F6**] function key.

You are also informed that double-clicking the left mouse button (after you have closed the info box) on any part of the graphics will cause the corresponding state variables to be shown on screen.

8.6.4 "Bearing capacity" menu item

Here you will obtain information in a message box related to the bearing capacity analysis. This function can also be initialized pressing the [**F7**] function key.

8.6.5 "Settlements" menu item

Here you will obtain the results of the settlement analysis displayed in a message box. ***CP*** means that the settlements are calculated for the characteristic points.. This function can also be initialized pressing the [**F8**] function key.

8.6.6 "Sliding safety" menu item

The calculated sliding safety is displayed in a message box.

8.6.7 "Static equilibrium (DIN 1055)" menu item

Using the partial safety factors to DIN 1054:2005 the static equilibrium to DIN 1055 is displayed in a message box.

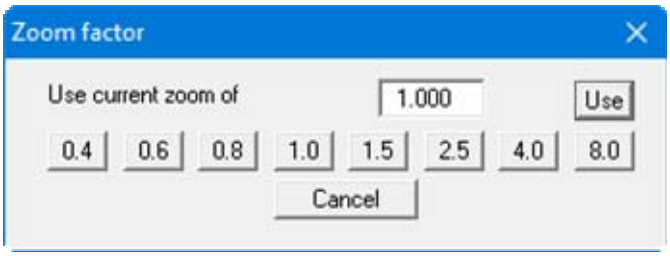
8.6.8 "EQU analysis" menu item

If the partial safety factor concept to EC 7 is adopted all available information on the EQU analysis (limit state of loss of static equilibrium) is displayed in a message box.

8.7.1 "Refresh and zoom" menu item

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

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



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

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

8.7.2 "Zoom info" menu item

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

8.7.3 "Legend font selection" menu item

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

8.7.4 "Pen colour and width" menu item

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

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

8.7.5 "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 you independent of the coordinate system and keeps you in the same position on the page. You should select the "**Header toolbar**" if you wish to place general information on the drawing (company logo, report numbers, plan numbers, stamp etc.). Once you have saved the header information to disk (see **Mini-CAD** user manual), you can load it into completely different systems (with different system coordinates). The saved header information will appear in exactly the same position on the page, which greatly simplifies the creation of general page information.

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

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

"**Next page**"/"**Previous page**"
Using this icon, you can navigate between the individual pages in the ***tabular representation***.

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

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

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

"**Move object**"
By clicking this icon you can move legends and graphical elements using the left mouse button (equivalent to menu item "**Graphics preferences/Move objects**").

"**Undo move object**"
By clicking this icon the last performed movement of graphical elements can be undone.

"**Copy/print area**"
Use this tool to copy only parts of the graphics in order to paste them, e.g. to a report. You will see information on this function and can then mark an area, which is copied to the clipboard or can be saved in a file. Alternatively you can send the marked area directly to your printer (see "**Tips and tricks**", section 9.4).

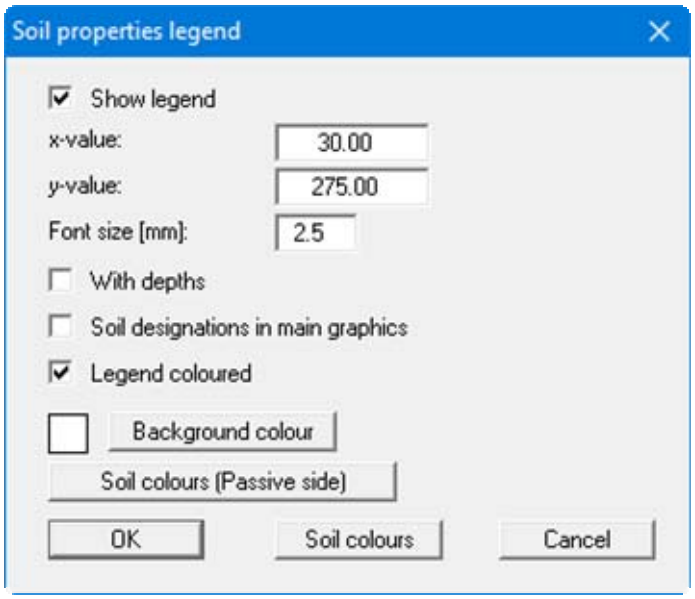
"**Colour on/off**"
If you need to remove the colour from the system presentation, to create a black and white print-out, for example, use this on/off switch.

"**Undo**"
By clicking this icon the last performed change (e.g. input of loads or movement of graphical elements made using the [**F11**] function key) can be undone.

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

8.7.7 "Soil properties legend" menu item

A legend with the soil properties of the individual layers will be displayed on your output sheet. If you have activated the "**Show legend**" check box, you can alter the type of visualisation using the dialog box of this menu item.



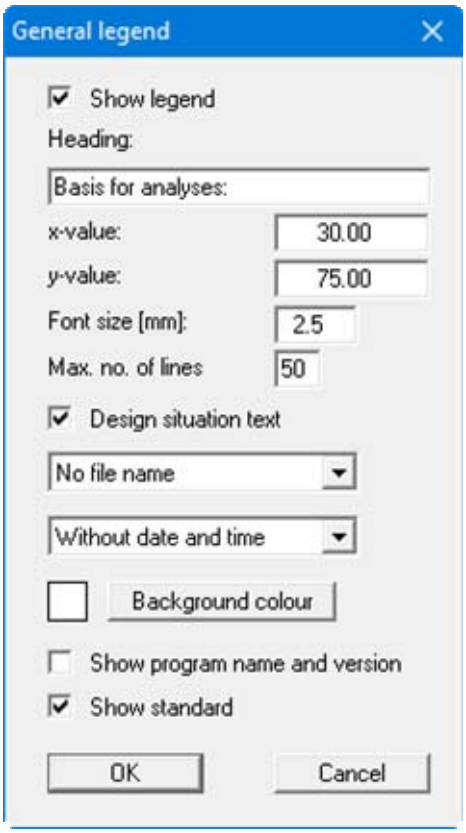
You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**". The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position with the left mouse button pressed.

- "**With depths**"
  The depths of the individual soil layers will be shown in the soil properties legend.
- "**Soil designations in main graphics**"
  The designations entered for the individual ground layers are displayed in the system graphics adjacent to the layer depths.
- "**Legend coloured**"
  The soil colours are displayed in the legend. Otherwise, they will be numbered.
- "**Background colour**"
  The legend can be assigned a background colour.
- "**Soil colours**"
  If, in the menu item "**Editor 1/Analysis options**", you have activated the "**Differentiate active + passive soil properties**" check box, the "**Soil colours (passive side)**" is also displayed.

  After clicking one of the buttons 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.

8.7.8 "General legend" menu item

A legend with general properties will be displayed on your output sheet. If you have activated the "**Show legend**" check box, you can alter the type of visualisation using the dialog box of this menu item.



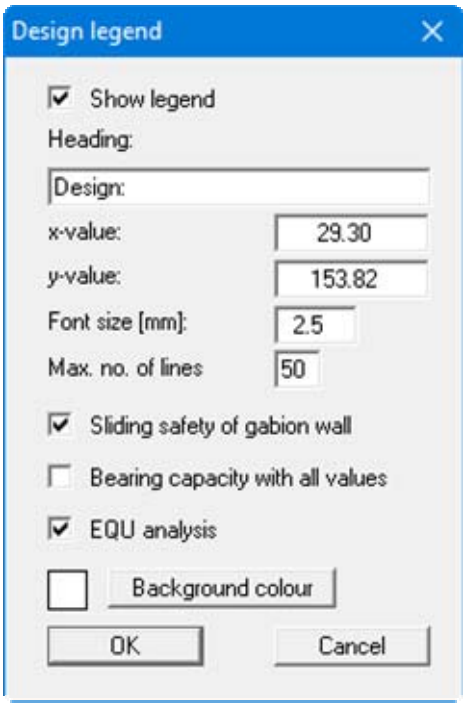
You can define and edit the position of the legend using the values "**x value**" and "**y value**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used.

> 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), on the adopted standard and on the current file (name, path, time info). Any project identification entered in the "**File/New**" or "**Editor 1/Analysis options**" dialog box will also be shown in the ***General legend*** . The legend can be assigned a background colour.

8.7.9 "Design legend" menu item

Following analysis and design a legend containing the principal system design results is displayed on the screen. Using this menu item you can alter the type of presentation if the "**Show legend**" check box is activated.



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.

By activating the "**Bearing capacity with all values**" check box the bearing capacity, shape and inclination coefficients will be displayed in the legend. Result display for the sliding safety (gabion wall) and the EQU analysis can also be activated. These values are automatically contained in the output table that can be opened using the "**File/Print output table**" menu item (see section 8.1.5). The legend can be assigned a background colour.

8.7.10 "Settlement legend" menu item

Following analysis and design a legend containing the principal results of the settlement analysis is displayed on the screen. Amongst other things, the share of the settlement for each individual layer can be taken from this legend. Using this menu item you can alter the type of presentation if the "**Show legend**" check box is activated.

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. The legend can be assigned a background colour.

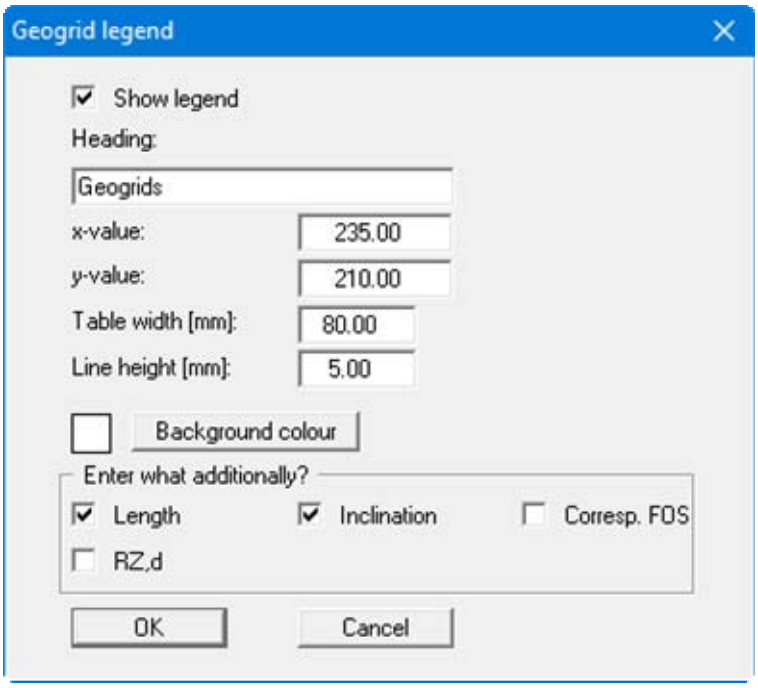
8.7.11 "Reinforced soil system legend" menu item

After you have analysed a system employing a reinforced soil body, a legend containing the basic assumptions and results of the analysis can be displayed. Using this menu item you can alter the type of presentation if the "**Show legend**" check box is activated.

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. The legend can be assigned a background colour.

8.7.12 "Geogrid legend" menu item

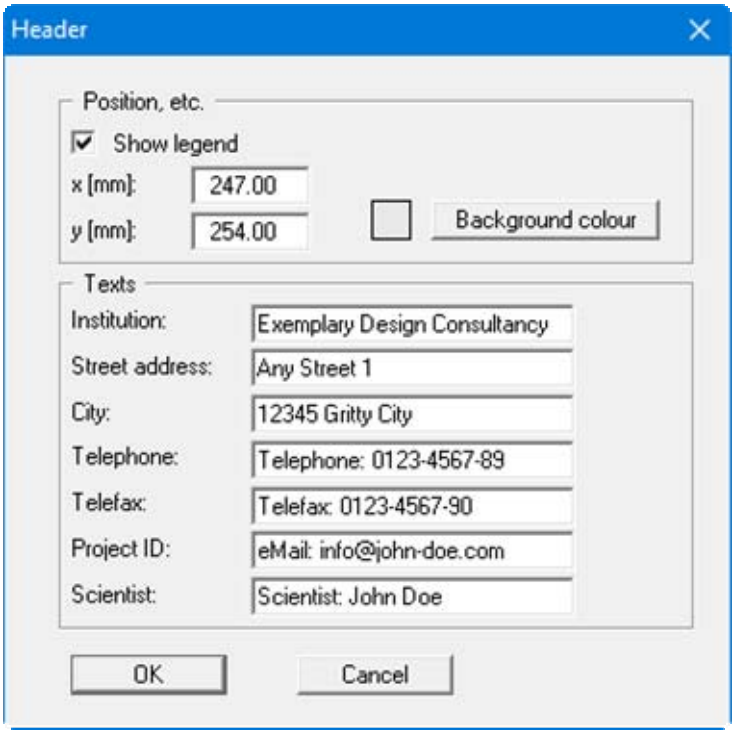
Where geogrids are employed in the system it is possible to display a corresponding legend by activating the "**Show legend**" check box in this dialog box.



The legend takes the form of a table. The font size can be controlled by means of "**Table width**" and "**Line height**". The table can be assigned a background colour. In the lower group box it is possible to select additional data to be included in the table.

8.7.13 "Quick header" menu item

Using this menu item it is possible to display a simple header containing company data. Activate the "**Show legend**" check box and enter your data.



You can define and edit the position of the header legend using the values "**x**" and "**y**". The legend can be assigned a background colour.

8.7.14 "Move objects" menu item

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

If "**Manual graph positioning**" has been selected in the "**System/Graph positioning preferences**" menu item (see section 8.5.5), the result graphs can also be repositioned using this function.

8.7.15 "Save graphics preferences" menu item

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

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

8.7.16 "Load graphics preferences" menu item

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

8.8.1 "Auto-resize" menu item

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

8.8.2 "Manual resize (mouse)" menu item

You can alter the image coordinates graphically with the mouse. In the dialog box you are offered two choices.

- 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.
- By activating the "**Redefine origin**" button you can change the image coordinates without altering the scale. In this way, you can quickly achieve the desired presentation with a suitably defined scale without annoying trial and error with the x/y coordinates. The previous scale preferences are not affected by this.

8.8.3 "Manual resize (editor)" menu item

You can alter the image coordinates by direct numerical input in a dialog box. This allows precise scale input. The coordinates refer to the ***drawing area***. This can be defined in the "**Page size + margins/Page size and margins**" menu item by means of the plot margins (see section 8.8.5).

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

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

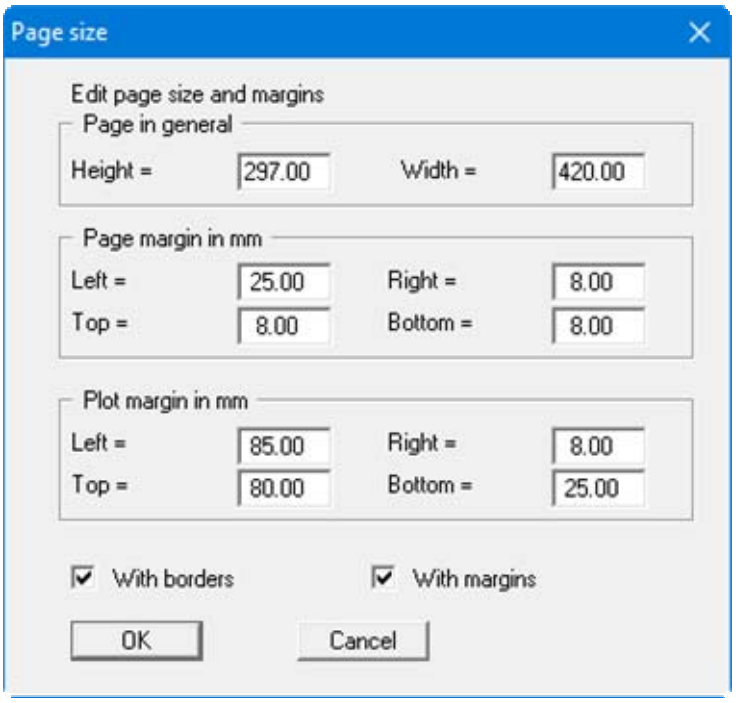
8.8.4 "Font size selection" menu item

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

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

8.8.5 "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.

8.8.6 "Undo" menu item

If you have carried out any changes to dialog boxes or moved objects to a different position on the screen after selecting the "**Graphics preferences/Move objects**" menu item or using the [**F11**] function key, this menu item will allow you to undo the movements. This function can also be reached by using the key combination [**Alt**] + [**Back**] or the appropriate tool in the toolbar (see section 8.7.6).

8.8.7 "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 "**Page size + margins/Undo**" will be restored. This function can also be reached by using the key combination [**Ctrl**] + [**Back**] or the appropriate tool in the toolbar (see section 8.7.6).

8.8.8 "Preferences" menu item

You can activate or deactivate the undo functions.

8.9.1 "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-GABION**.

8.9.2 "Maxima" menu item

Here you can check the defaults for maximum values.

8.9.3 "Compare ep coefficients" menu item

Here you can calculate the earth pressure coefficients for the given values of phi, delta and beta.

8.9.4 "Help" menu item

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

8.9.5 "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.

8.9.6 "GGU support" menu item

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

8.9.7 "What's new?" menu item

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

8.9.8 "Language preferences" menu item

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

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

9 Tips and tricks

9.1 "?" buttons

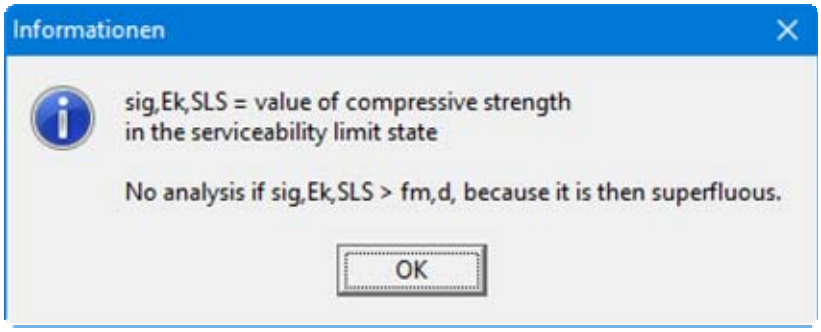
Reading of the manual can mostly be dispensed with, because

"**?**" 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 "**?**" button.

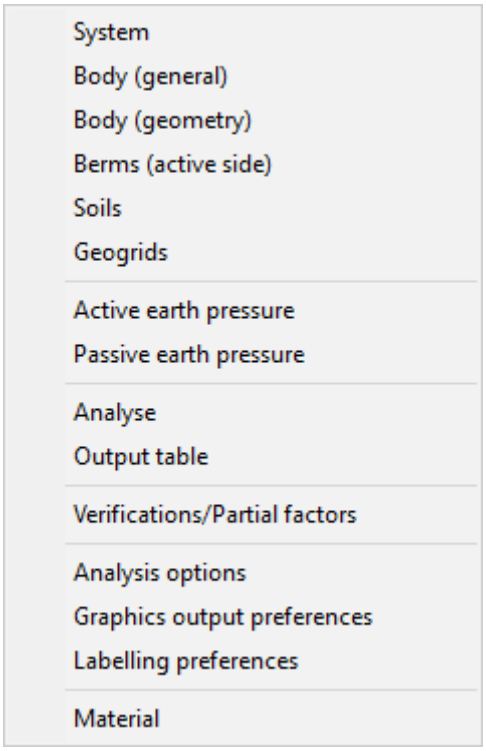
For example, the following check box can be found in the "**Editor 1/Material**" dialog box:

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



9.2 Keyboard and mouse

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



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

If you double-click in the system graphics with the left mouse button you will see an info box with the results at the selected depth. This gives you a quick overview of the results for any desired depth.

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

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

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

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

Change screen coordinates:

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

9.3 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.
- [**F5**] opens the menu item "**System/Analyse** ".
- [**F6**] opens the menu item "**Evaluation/Main output summary** ".
- [**F7**] opens the menu item "**Evaluation/Bearing capacity**".
- [**F8**] opens the menu item "**Evaluation/Settlements**".
- [**F9**] opens the menu item "**Page size + margins/Auto-resize**".
- [**F11**] activates the menu item "**Graphics preferences/Move objects**".

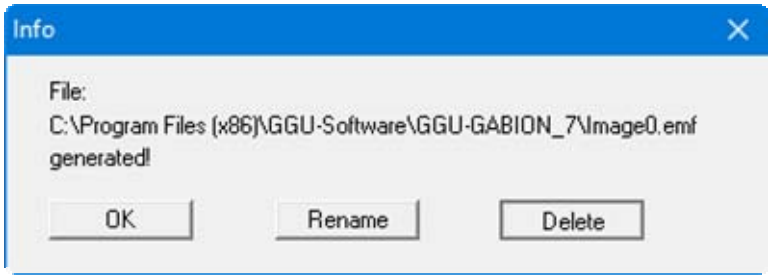
9.4 "Copy/print area" icon

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.

10 Index

2

2nd order theory, select for analysis 91

A

Absolute heights, activate use 55
Action boundary conditions, define..... 78
Active earth pressure,
  analysis principles..... 28
  edit analysis preferences 70
  select for analysis..... 69
Active wall friction angle, wind load approach 23
Angle of wall friction,
  enter/adopt from soil database..... 68
Area loads,
  consider in analysis 30
  define 76
  include into earth pressure redistribution 91
  select ep analysis method..... 72
  select type of ep analysis..... 69
  to DIN 4085 31
At-rest earth pressure,
  coefficient to DIN 4085 28
  keep constant..... 72
  select for analysis..... 69

B

Base in front of wall, define 64
Base inclination,
  coefficients in bearing capacity equation 47
  consider in analysis 36
  define 14
Bearing capacity failure plane, determine 48
Bearing capacity safety,
  analysis principles..... 47
  consider base inclination 36
  display analysis results for reinf. soil in a message box..... 88
  display analysis results in a message box... 100
  passive ep to EC 7..... 68
Bearing capacity,
  coefficients in bearing capacity equation 47
Berms,
  activate labelling 97
  define on active side..... 67
  define on passive side..... 67
Bonding stress,
  define visualisation height..... 81
  in accordance with EBGEO 43
  select calculation method 84
  within slip body..... 42
Bounded surcharges,
  consider in analysis 32, 34
  define 76
Buckling length, determination 51

C

Caquot/Kerisel,
  consider in passive ep analysis..... 29
  select for passive ep analysis..... 71
Centroid axis,
  activate/edit display..... 98
  consider acting forces..... 35
Characteristic point,
  display settlements 100
  limiting depth determination 80
Cholesky..... 35
Clipboard..... 61
Cohesion,
  coefficient determination methods 28
  enter/adopt from soil database..... 68
Colour,
  define for soils..... 104
  switch on/off 103
Colour/hatching, define for loads 97
Colour/pens,
  define for gabion body 98
  define for graphical elements 101
Company data,
  add via Mini-CAD..... 102
  add via quick header legend..... 109
Concrete,
  activate verification..... 18
  enter values for structural analysis 17
Constrained modulus,
  enter/adopt from soil database..... 68
  for settlement analysis..... 49
Context menu, open..... 116
Coordinates,
  alter via editor 111
  alter with mouse 111
  optimise/reset 111
  save/load 111
Copy/print area 61, 103, 118
Cutting borders, switch on/off..... 112

D

Danish standard 47
Database,
  for soil properties of common soils..... 28, 68
Dataset description,
  define..... 54
  display 105
Delete,
  geogrid 83
  soil layers 68
Depth coefficients,
  definition to Danish standard 47
Design compressive strength,
  define for concrete..... 18
Design resistance,
  according to EBGEO..... 41
  define for geogrid..... 84

Design situation,
activate display in legend ... 105
enter text for legend ... 74
Design, display results in legend ... 106
Dimension lines,
define ... 99
move with mouse ... 109
DIN 1045 ... 50
DIN 1055,
activate static equilibrium determination ... 78
DIN 4017 ... 47
DIN 4019 ... 49
DIN 4085,
consider in active ep analysis ... 28
consider in passive ep analysis ... 29
select for active ep analysis ... 70
select for passive ep analysis ... 71
DIN 4093,
concrete verification principles ... 52
select for analysis ... 54
Displacement boundary conditions, define ... 79
Displacement method ... 35
Displacement, define for gabion wall ... 79
Distributed load, define as live load ... 64
Double-bounded surcharges,
consider in analysis ... 33, 34
define ... 77
Drawing area, define ... 112
Duddeck/Ahrens ... 35
DXF file,
export ... 61
import via Mini-CAD ... 7

E

Earth pressure analysis,
display results in a message box ... 88
Earth pressure coefficients,
be calculated ... 114
calculate ... 72
user-defined ... 72
Earth pressure distribution, on gabion wall ... 45
Earth pressure redistribution,
display in a message box ... 100
to EAB ... 92
user-defined ... 94
Earth pressure reduction,
considering inclined wall ... 44
determine possible factor ... 45
Earth pressure wedge,
activate display ... 81
in earth pressure calculation ... 44
Earth pressure,
analysis via earth pressure wedge ... 44
edit preferences for active ... 70
edit preferences for passive ... 71
select for analysis ... 69
Earth quake, as design situation to EC 7 ... 74
EBGEO, earth pressure reduction to ... 45
EBGEO, Internal stability,
reinforced soil system ... 41
EC 7, load cases description ... 74
Eccentricity,
determine ... 44
unintentional ... 50
Editor window, output table ... 59
Elastic curve, for inclined body ... 36
Embedment depth, footing ... 47
EMF format ... 61
EQU analysis,
activate ... 74
display results in a message box ... 100
display results in legend ... 106
Equation solution to Cholesky ... 35
Equivalent earth pressure coefficient,
activate use ... 70
Equivalent system for weight calculation ... 43

F

Failure body, activate display ... 81
Failure system, internal stability verification ... 41
FGSV ... 7
File,
display name in legend ... 105
load/save ... 55
Finite element module, for structural system ... 35
Font selection ... 101
Font size,
for design legend ... 106
for dimension line ... 99
for general legend ... 105
for geogrid legend ... 108
for graphical elements ... 111
for reinforced soil system legend ... 107
for settlement legend ... 107
for soil properties legend ... 104
for system/result graphics ... 97
Footer, output table ... 57
Footing length, define ... 14
Foundation gap, explanation ... 21
Friction angle,
activate separate active/passive input ... 55
enter/adopt from soil database ... 68
Friction coefficient,
define ... 14
for sliding safety verification ... 40
Function keys ... 117

G

Gabion body,
define ... 27
define colour/pens ... 98
define geometry ... 65
define rotation/displacement ... 79
generate automatically ... 15
Gabion wall, change alignment ... 55
General page information,
add via Mini-CAD ... 102
General stability,
verification using file export ... 60
verification via file export ... 49
Geogrid forces, show maximum ... 87

Geogrids,
activate selection via company products 55
add automatically 85
display values in legend 108
enter/delete 83
enter/edit all values 86
enter/edit individual values 84
manage company products 88
select as company product 83
GGU-CAD file, export 61
GGUMiniCAD file, export 62
GGU-STABILITY 60
Global coordinates,
use for Mini-CAD output 62
Global safety factors/verifications define 73
Graph positions,
alter via numerical input 95
alter with mouse 109
Graphics, add via Mini-CAD 102
Gravity body, activate display 81
Ground inclination coefficients,
in bearing capacity equation 47
Groundwater,
activate labelling 97
define colour 101
enter level 64

H

Hatching, define for loads 97
Header CAD, application explanations 102
Header, output table 57
Horizontal force, in rod construction analysis .. 35
Horizontal loading, on soil body 43

I

Inclination,
active earth pressure wedge 43
rear face of wall 43
Increased active earth pressure,
coefficient 28
select for analysis 69
Internal stability,
select analysis method 54
to DIN 1045 old 50
to DIN 4093 52
to EC 2/DIN 1045-1 51

K

Kernel widths,
activate/edit display 98
explanation 21

L

Labelling, system/result graphics 97
Lambda, reduction factor/friction coefficient ... 84
Language preferences 114
Lateral pressures, define 78
Layout,
edit for output sheet 112
edit for output table 57
Legends, move with mouse 109
Licence protection 9
Limiting depth,
defining methods 49
select determination 80
Line loads,
define 76
perpendicular to wall axis 31
Live load, consider distributed load as 64
Load case,
adopt from DIN 1054-2010/EC 7 74
adopt from ÖNORM EN 1997-1 74
enter text for legend 74
Load concentration factor,
consider 31
define 72
Load distribution, for concrete verification 50
Load inclination coefficients,
in bearing capacity equation 47
Load transition point,
calculate with water pressure 91
using redistribution 93
Load transmission width,
activate/edit display 98
define 14
for concrete verification 50
Loads, edit graphical output preferences 97
Logarithmic spiral,
display below gabion wall 96
display below reinforced soil system 81
in bearing capacity failure plane 48

M

Manual, open as PDF file 114
Maximum values, display for system 114
Mean friction angle 48
Mean soil properties 48
Metafile, export 61
Mini-CAD,
application explanations 102
export file 62
output with global coordinates 62
Moment load, consider on rod axis 35
Moment, calculation in base plane 44
Mouse click functions 116
Mouse wheel functions 117

N

Navigation, output table 102
n-region,
activate/edit display 98
explanation 21
n-value, define for load distribution 18

O

Objects, move with mouse 109
ÖNORM EN 1997-1,
select partial safety factors 74
Output preferences 59, 60

Output table,
edit output as ASCII ... 59
edit output as graphics ... 57
navigation ... 58
select output format ... 56
switch to system graphics ... 58, 102
Overburden stress,
consider in settlement analysis ... 80
Overturning safety,
analysis ... 44
deactivate analysis ... 55
display analysis results for reinforced soil system in a message box ... 88

P

Page,
copy/print section ... 103, 118
define format ... 112
define/switch margins on/off ... 112
Pagination, automatic ... 58, 59
Partial safety factors,
define ... 74
select from ÖNORM EN 1997-1 ... 74
Passive earth pressure,
analysis methods for coefficients ... 29
edit analysis preferences ... 71
for bearing capacity safety ... 68
in front/superimposed ... 91
limit ... 71
PDF file, import via Mini-CAD ... 7, 102
Pen preferences,
edit for gabion body ... 98
edit for graphical elements ... 101
Piling Handbook,
consideration of line loads ... 31
effect on earth pressure ... 29
Plausibility checks,
show during analysis ... 90
view results ... 89
Plot margins, define ... 112
Preconsolidation,
consider in settlement analysis ... 49, 80
Presentation height, define for loads ... 97
Pressure line,
activate/edit display ... 98
explanation ... 21
Print,
graphics ... 60
output table ... 61
section ... 61, 103, 118
several files ... 63
Program,
display name in legend ... 105
save/load preferences ... 110
show improvements ... 114
show information ... 114
Project data, add via Mini-CAD ... 102
Project identification,
define ... 54
display ... 105
Pull-out resistance, maximum design value ... 42

R

Redistribution figures,
available ... 37
select ... 92
Reinforced soil system,
display values in legend ... 107
Reinforcement force, maximum ... 41
Result graphics,
display ... 99
edit preferences ... 96
label ... 97
position ... 95
position with mouse ... 109
Results, display for selected depth ... 100, 116
Rod construction module ... 35
Rods,
define for system ... 89
effect on analysis ... 35
Rotation, define for gabion wall ... 79

S

Safety concept, select ... 54
Safety factors, define ... 73
Scale,
alter with mouse ... 111
define via editor ... 111
determine automatically ... 111
Scroll the screen ... 116
Seismic acceleration,
as design situation to EC 7 ... 74
consider loads ... 75
Serviceability,
systems with geogrids ... 39
systems without geogrids ... 39
to DIN 4093 ... 53
Settlement analysis,
display results in a message box ... 100
display results in legend ... 107
edit preferences ... 80
limiting depth definition ... 49
subtraction preconsolidation loading ... 49
Settlements, display share in legend ... 107
Shape coefficients, for strip footings ... 47
Short-term strength ... 41
Slenderness, structural element ... 50
Sliding safety,
consider base inclination ... 36
display analysis results for reinforced soil system in a message box ... 88
display analysis results in a message box ... 100
in base plane ... 40
in tier joint planes ... 40
Slip surfaces,
generate/edit ... 87
internal stability verification ... 41
Slope stability application ... 60
Smart icons, for menu items ... 102
Soil colours,
activate display ... 104
define separate active/passive soils ... 104

Soil designations,
activate presentation in system graphics 104
define 68
Soil layers,
adapt to absolute heights 64
define/delete 68
maximum number 28
Soil numbers, activate display 104
Soil properties,
activate separate active/passive input..... 55
display in legend 104
enter/adopt from soil database..... 68
Stability,
systems with geogrids 39
systems without geogrids 38
Standard, display in legend..... 105
Static equilibrium to DIN 1055,
activate determination 78
display analysis results in a message box... 100
Status bar main program, activate 102
Stiffness, wall 35, 38
Streck,
consider in passive ep analysis..... 29
select for passive ep analysis..... 71
Stress analysis, for limiting depth..... 80
Subgrade reaction modulus, for toe pivot..... 90
Support forces, reinforced soil system..... 43
System,
analyse 90
display..... 99
display properties in legend 105
division into rods..... 89
edit graphics output preferences..... 96
show information 89, 114
structural 35

T

Tension forces, in the geogrids..... 43
Theory of elastic half-space..... 31
Toe pivot, define..... 90
Toolbar, edit for menu items 102
Transition point,
calculate with water pressure 91
using redistribution..... 93
Translation, activate 114
True-type font..... 101
Two-part wedge failure mechanism,
internal stability verification 41
slip surface generation..... 87

U

Unit weight,
activate separate active/passive input..... 55
enter/adopt from soil database..... 68
User-defined ep coefficients, enter 72
Utilisation factor, unreinforced concrete 51

V

Verifications,
activate for reinforced soil system..... 82
activate to DIN 1054 old 73
activate to DIN 1054-2010/EC7..... 74
Version number,
display in a message box..... 114
display in legend 105
Vertical force,
calculation in base plane 44
in rod construction analysis..... 35
Vertical section for ep determination..... 43

W

Wall friction angle,
enter/adopt from soil database..... 68
wind loads approach..... 23
Wall toe, select bedding conditions 90
Wall top, kept horizontal,
consider in analysis 51
Weight analysis,
display analysis results in a message box..... 88
What you see is what you get 101
Wind loads,
active wall friction angle approach 23
consider as lateral pressures 78
consider for noise abatement wall..... 25

Y

Young's modulus, define for gabion material ... 17

Z

Zoom factor, define for full-screen display 101
Zoom function, activate..... 101, 102, 116