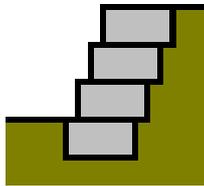


Analysis of retaining structures manufactured using  
concrete elements, stacked blocks and gabions

---

# GGU-GABION

VERSION 6



Last revision: February 2016  
Copyright: Prof. Dr. Johann Buß  
Technical implementation and sales: Civilserve GmbH, Steinfeld

---

**Contents:**

- 1 Preface ..... 7**
- 2 Capabilities ..... 7**
- 3 Licence protection and installation ..... 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 excavation (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 (Ex. 2) ..... 24
    - 6.2.3 Step 2: Generate noise abatement wall (Ex. 2) ..... 24
    - 6.2.4 Step 3: Define material of gabion (Ex. 2) ..... 25
    - 6.2.5 Step 4: Analyse system (Ex. 2) ..... 26
- 7 Theoretical principles ..... 27**
  - 7.1 Body ..... 27
  - 7.2 Soil properties ..... 27
  - 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 ..... 28
  - 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 earth 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 outer skin.....	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
<b>8</b>	<b>Description of menu items.....</b>	<b>54</b>
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	"Printer preferences" menu item.....	60
8.1.8	"Print and export" menu item .....	60
8.1.9	"Batch print" menu item .....	62
8.1.10	"Exit" menu item.....	62
8.1.11	"1, 2, 3, 4" menu items.....	62
8.2	Editor 1 menu.....	63
8.2.1	"Analysis options" menu item.....	63
8.2.2	"Excavation" menu item .....	63
8.2.3	"Body (general)" menu item .....	64
8.2.4	"Body (geometry)" menu item .....	64
8.2.5	"Material" menu item.....	65

8.2.6	"Berms (active side)" menu item .....	66
8.2.7	"Berms (passive side)" menu item .....	66
8.2.8	"Soils" menu item .....	67
8.2.9	"Type of earth pressure" menu item .....	68
8.2.10	"Active earth pressure" menu item .....	69
8.2.11	"Passive earth pressure" menu item .....	70
8.2.12	"At-rest earth pressure" menu item .....	71
8.2.13	"User-defined earth pressure coefficients" menu item.....	71
8.2.14	"Verifications/Safety factors" menu item .....	72
8.2.15	"Verifications/Partial factors" menu item .....	73
8.2.16	"Seismic acceleration" menu item .....	74
8.3	Editor 2 menu .....	75
8.3.1	"Area and line loads" menu item .....	75
8.3.2	"Bounded surcharges" menu item.....	75
8.3.3	"Double-bounded surcharges" menu item .....	76
8.3.4	"Lateral pressures" menu item .....	76
8.3.5	"Action boundary conditions" menu item.....	77
8.3.6	"Displacement boundary conditions" menu item.....	77
8.3.7	"Settlements" menu item.....	78
8.4	Reinforced earth system menu .....	79
8.4.1	"Graphic preferences" menu item .....	79
8.4.2	"Verifications" menu item .....	80
8.4.3	"Enter geogrids manually" menu item .....	81
8.4.4	"Generate geogrids" menu item .....	83
8.4.5	"Modify geogrids " menu item .....	84
8.4.6	"Generate slip surfaces" menu item.....	85
8.4.7	"Geogrid forces" menu item .....	85
8.4.8	"Sliding, overturning, bearing capacity" menu item.....	86
8.4.9	"Calculate earth pressure + weight" menu item.....	86
8.4.10	"Geogrids table values" menu item.....	86
8.5	System menu .....	87
8.5.1	"Info" menu item .....	87
8.5.2	"Special preferences" menu item.....	87
8.5.3	"Depth subdivisions" menu item .....	87
8.5.4	"Analyse" menu item .....	88
8.5.4.1	Start dialog box .....	88
8.5.4.2	"Type of toe pivot" group box.....	88
8.5.4.3	"Special preferences" group box .....	89
8.5.4.4	"Type of redistribution" group box .....	90
8.5.5	"Graph positioning preferences" menu item.....	93
8.5.6	"Graphics output preferences" menu item .....	94
	"Labelling preferences" menu item.....	95
8.5.7	"Body pens + colours" menu item .....	96
8.5.8	"Dimension lines" menu item .....	97
8.5.9	"Display system" menu item.....	97
8.5.10	"Display results" menu item .....	97

8.6	Evaluation menu.....	98
8.6.1	General note.....	98
8.6.2	"Earth pressure redistribution" menu item.....	98
8.6.3	"Main output summary" menu item.....	98
8.6.4	"Bearing capacity" menu item.....	98
8.6.5	"Settlements" menu item.....	98
8.6.6	"Sliding safety" menu item.....	98
8.6.7	"Static equilibrium (DIN 1055)" menu item.....	98
8.6.8	"EQU analysis" menu item.....	98
8.7	Graphics preferences menu.....	99
8.7.1	"Refresh and zoom" menu item.....	99
8.7.2	"Zoom info" menu item.....	99
8.7.3	"Legend font selection" menu item.....	99
8.7.4	"Pen colour and width" menu item.....	99
8.7.5	"Mini-CAD toolbar" and "Header toolbar" menu items.....	100
8.7.6	"Toolbar preferences" menu item.....	100
8.7.7	"Soil properties legend" menu item.....	102
8.7.8	"General legend" menu item.....	103
8.7.9	"Design legend" menu item.....	104
8.7.10	"Settlement legend" menu item.....	104
8.7.11	"Legend reinforced earth system" menu item.....	105
8.7.12	"Geogrid legend" menu item.....	105
8.7.13	"Quick header" menu item.....	106
8.7.14	"Move objects" menu item.....	106
8.7.15	"Save graphics preferences" menu item.....	107
8.7.16	"Load graphics preferences" menu item.....	107
8.8	Page size + margins menu.....	108
8.8.1	"Auto-resize" menu item.....	108
8.8.2	"Manual resize (mouse)" menu item.....	108
8.8.3	"Manual resize (editor)" menu item.....	108
8.8.4	"Font size selection" menu item.....	108
8.8.5	"Page size and margins" menu item.....	109
8.8.6	"Undo" menu item.....	110
8.8.7	"Restore" menu item.....	110
8.8.8	"Preferences" menu item.....	110
8.9	Info menu.....	111
8.9.1	"Copyright" menu item.....	111
8.9.2	"Maxima" menu item.....	111
8.9.3	"Compare ep coefficients" menu item.....	111
8.9.4	"Help" menu item.....	111
8.9.5	"GGU on the web" menu item.....	111
8.9.6	"GGU support" menu item.....	111
8.9.7	"What's new?" menu item.....	111
8.9.8	"Language preferences" menu item.....	111

<b>9 Tips and tricks.....</b>	<b>112</b>
9.1 "?" buttons.....	112
9.2 Keyboard and mouse.....	112
9.3 Function keys.....	114
9.4 "Copy/print area" icon.....	115
<b>10 Index.....</b>	<b>116</b>

**List of Figures:**

<i>Figure 1 Example system 1.....</i>	<i>11</i>
<i>Figure 2 Example system 2.....</i>	<i>23</i>
<i>Figure 3 Gabion body.....</i>	<i>27</i>
<i>Figure 4 Berm on the active side.....</i>	<i>29</i>
<i>Figure 5 Area load.....</i>	<i>30</i>
<i>Figure 6 At-rest earth pressure from area loads.....</i>	<i>31</i>
<i>Figure 7 Bounded surcharge (active side).....</i>	<i>32</i>
<i>Figure 8 Two bounded surcharges.....</i>	<i>33</i>
<i>Figure 9 Double-bounded surcharge.....</i>	<i>33</i>
<i>Figure 10 Bounded surcharge (passive side).....</i>	<i>34</i>
<i>Figure 11 Structural system of a wall without geogrid.....</i>	<i>35</i>
<i>Figure 12 Base inclination.....</i>	<i>36</i>
<i>Figure 13 Wall with geogrid.....</i>	<i>39</i>
<i>Figure 14 Two-part wedge failure mechanism.....</i>	<i>41</i>
<i>Figure 15 Equivalent system for weight calculation.....</i>	<i>43</i>
<i>Figure 16 Vertical section for earth pressure determination.....</i>	<i>44</i>
<i>Figure 17 Slip surface and earth pressure distribution.....</i>	<i>45</i>
<i>Figure 18 Logarithmic spiral.....</i>	<i>48</i>
<i>Figure 19 n-value and width for internal stability analysis.....</i>	<i>50</i>
<i>Figure 20 Body geometry.....</i>	<i>64</i>
<i>Figure 21 Passive earth pressure in front or superimposed.....</i>	<i>89</i>
<i>Figure 22 Birectangular earth pressure redistribution.....</i>	<i>90</i>
<i>Figure 23 Earth pressure redistribution in a trapezoidal.....</i>	<i>91</i>
<i>Figure 24 Earth pressure redistribution in a quadrilateral.....</i>	<i>91</i>
<i>Figure 25 User-defined earth pressure redistribution.....</i>	<i>92</i>

---

## 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. 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 20 berms on the active earth pressure side
- Up to 20 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 or user-defined values
- Coefficients of passive earth pressure pursuant to DIN 4085, Streck, Caquot/Kerisel or user-defined values
- 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<sup>st</sup> and 2<sup>nd</sup> 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 and installation

---

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

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

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

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

---

### 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
- ?

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**". After clicking the "**New**" menu item a dialog box opens for specifying general preferences for your new system (see Section 8.1.1). After leaving the box you then see nine menus in the menu bar:

- File
- Editor 1
- Editor 2
- Reinforced earth 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.

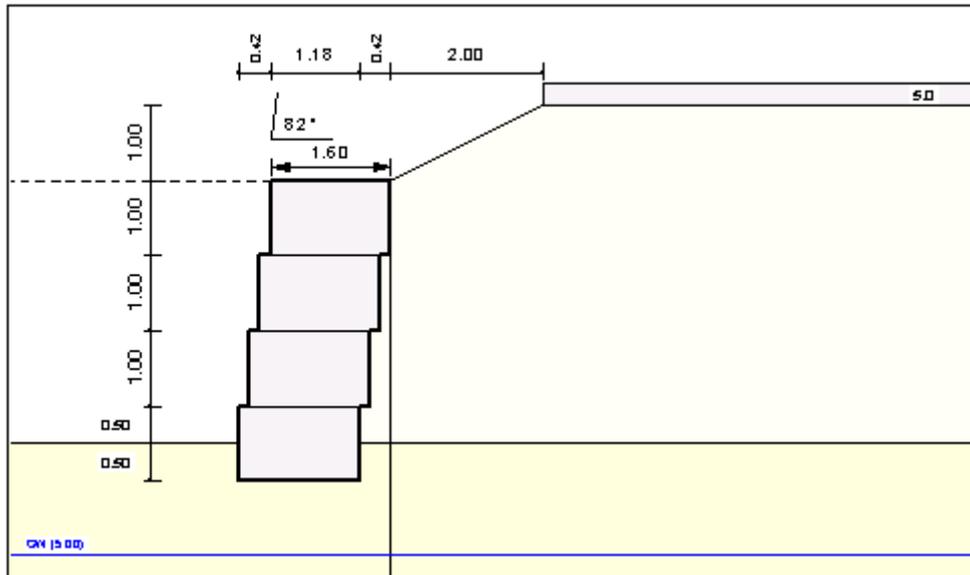


Figure 1 Example system 1

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

- **Sand**  
friction angle  $\varphi = 32,5^\circ$   
cohesion  $c = 0 \text{ kN/m}^2$   
wet unit weight  $\gamma = 19 \text{ kN/m}^3$   
buoyant unit weight  $\gamma' = 10 \text{ kN/m}^3$   
wall friction angle (active) =  $2/3 \varphi$   
no passive earth pressure used  
constrained modulus  $E_S = 60 \text{ MN/m}^2$

### 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:

If you select the "**Use absolute heights**" check box you can enter all depths or 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. In this example the check box "**Use absolute heights**" is not selected.

For this example, use the settings shown in the dialog box above and click "OK". A new system is then displayed on the screen and you will see the complete menu bar.

### 6.1.3 Step 2: Define the excavation (Ex. 1)

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

Excavation	
Excavation base [m]	3.50
Groundwater (right) [m]	5.00
Groundwater (left) [m]	5.00
Distributed load [kN/m <sup>2</sup> ]	0.00
Type of distributed load:	Component > 10.0 kN/m <sup>2</sup> changeable

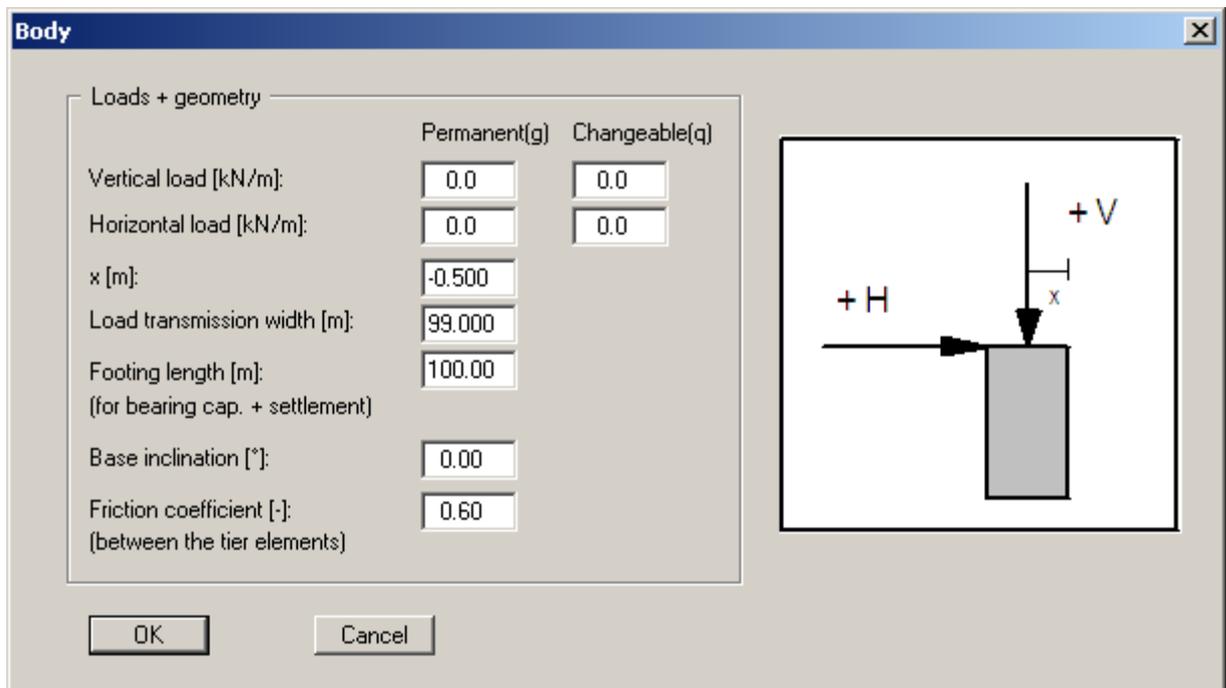
Passive side	
Distributed load [kN/m <sup>2</sup> ]	0.00

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

The 5 kN/m<sup>2</sup> 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)**".

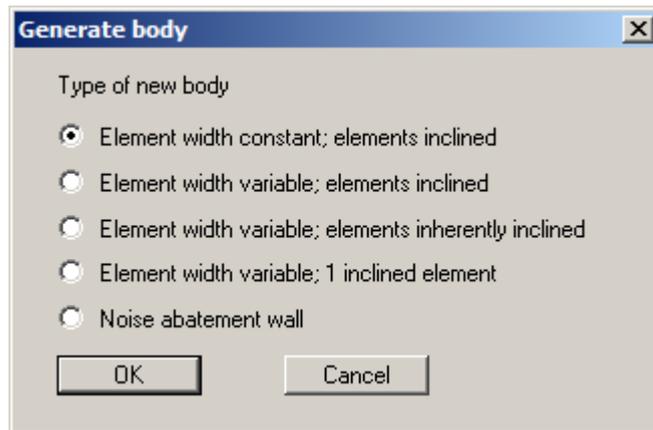
The "**Load transmission width**" is required for verification of the internal stability of the gabion wall to EC 2. If the load is not transmitted across the whole width, define the appropriate value. Otherwise, you may leave the default value of **99.0** unaltered.

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.

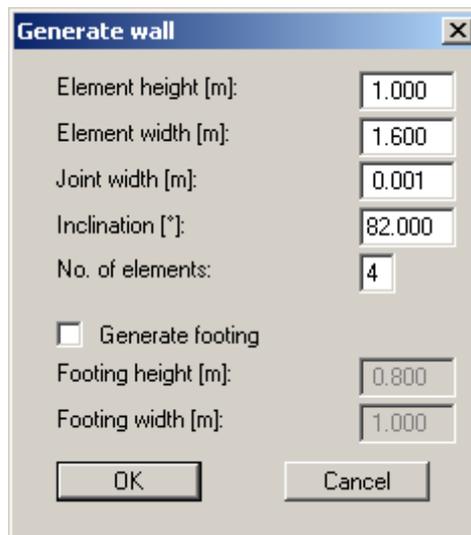
### 6.1.5 Step 4: Define body geometry (Ex. 1)

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.

**Outer skin** [X]

Forw. Back Cancel Done Load

8 points to edit Sort Save

No. Depth [m] x(left) [m] x(right) [m] Generate

Wall top = 0.000 -1.600 0.000

2	1.000	-1.600	0.000	Interpolate
3	1.000	-1.741	-0.141	Interpolate
4	2.000	-1.741	-0.141	Interpolate
5	2.001	-1.881	-0.281	Interpolate
6	3.000	-1.881	-0.281	Interpolate
7	3.001	-2.022	-0.422	Interpolate
8	4.001	-2.022	-0.422	

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 dialog box 'Material' contains the following fields and values:

- gamma (Gabion): 18.00
- '2nd gamma' (Gabion):
  - Use '2nd gamma'
  - '2nd gamma' [kN/m<sup>3</sup>]: 23.00
  - below [m]: 9.00
- Water in gabion:
  - 'Buoyant gamma' if groundwater in wall
  - No water pressure on gabions
- Gabion:
  - Verify internal stability
    - from [m]: 0.000
    - to [m]: 999.000
    - fm,d [MN/m<sup>2</sup>]: 0.200
    - sig.Ek,SLS [MN/m<sup>2</sup>]: 1.600
    - Young's mod. [kN/m<sup>2</sup>]: 2.5000E+4
    - n-value for load distribution (see DIN 1045 Table 17): 2.000
  - Wall top kept horizontal
  - No buckling analysis for displacement to earth side

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 ( $f_{m,d}$ ) 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.

No.	Left [m]	Right [m]	Delta h [m]	Surcharge [kN/m <sup>2</sup> ]	Live load
1	0.000	2.000	1.00	5.00	<input checked="" type="checkbox"/>

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.

The dialog box titled "Soil properties" contains a button labeled "Edit no. of soils" at the top. Below it is a table with the following data:

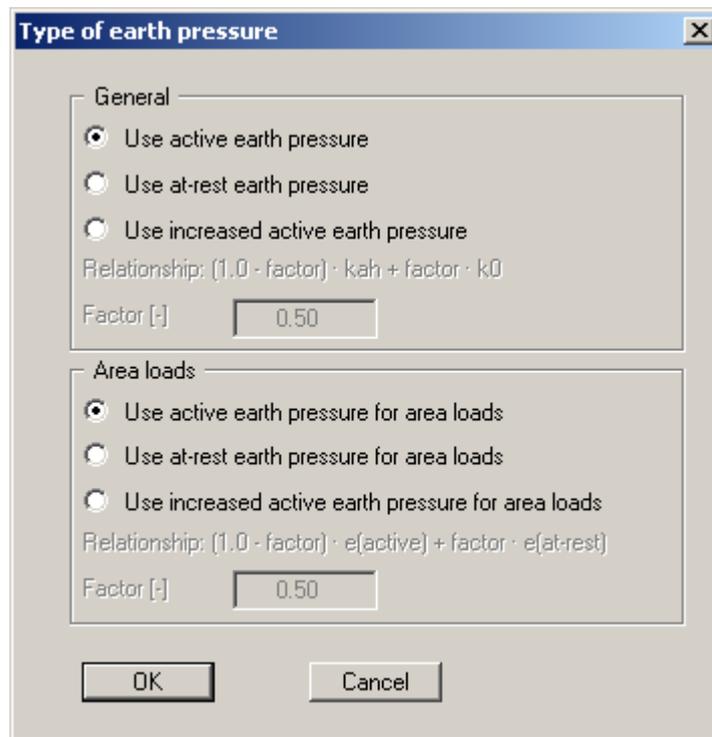
	Designation	Base	gam	gam'	phi	c(a)	c(p)	d(a)/phi	d(p)/phi	Es
		[m]	[kN/m <sup>2</sup> ]	[kN/m <sup>2</sup> ]	[°]	[kN/m <sup>2</sup> ]	[kN/m <sup>2</sup> ]	[-]	[-]	[MN/m <sup>2</sup> ]
1	Gravel sand	3.50	18.0	10.0	35.0	0.0	0.0	0.667	0.000	80.0
2	Sand	6.00	19.0	10.0	32.5	0.0	0.0	0.667	0.000	60.0

At the bottom of the dialog box, there are five buttons: "OK", "Cancel", "Sort", "Load", and "Save".

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.

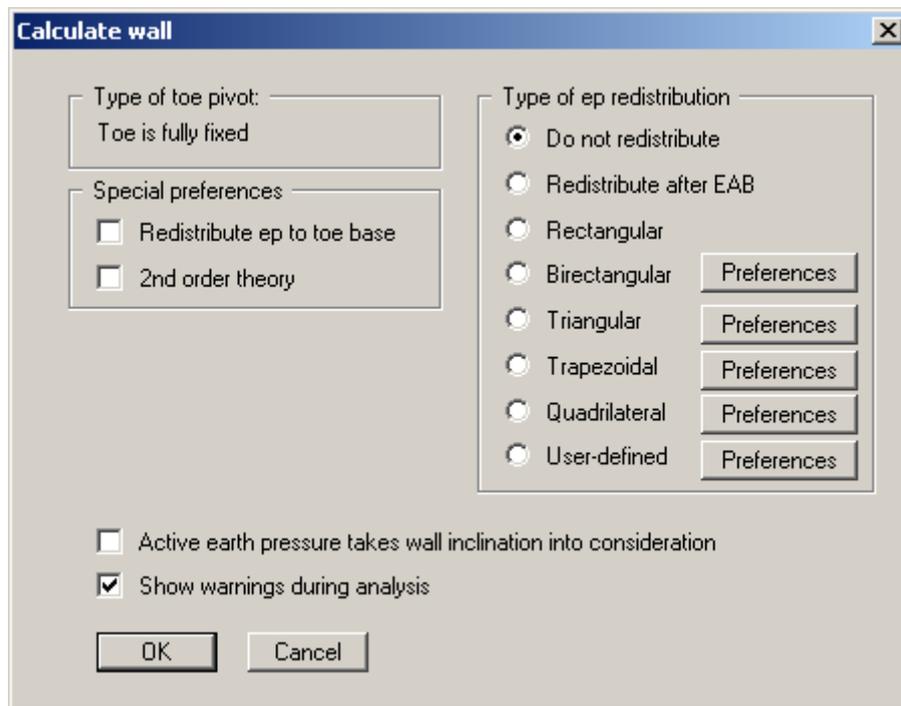


The necessary command buttons are already selected, so you need not change anything. The same applies to the remaining menu items in "Editor 1". However, you should click these items and take a look at them, in order to familiarise yourself with them.

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
- 1<sup>st</sup> kernel width:  
If the pressure line is within the 1<sup>st</sup> kernel width the plan area is completely under pressure => compressive loading only.
- 2<sup>nd</sup> kernel width:  
If the pressure line is outside of the 1<sup>st</sup> kernel width but inside of the 2<sup>nd</sup> 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 Worked example 2: Noise abatement wall

### 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/m<sup>2</sup> is adopted.

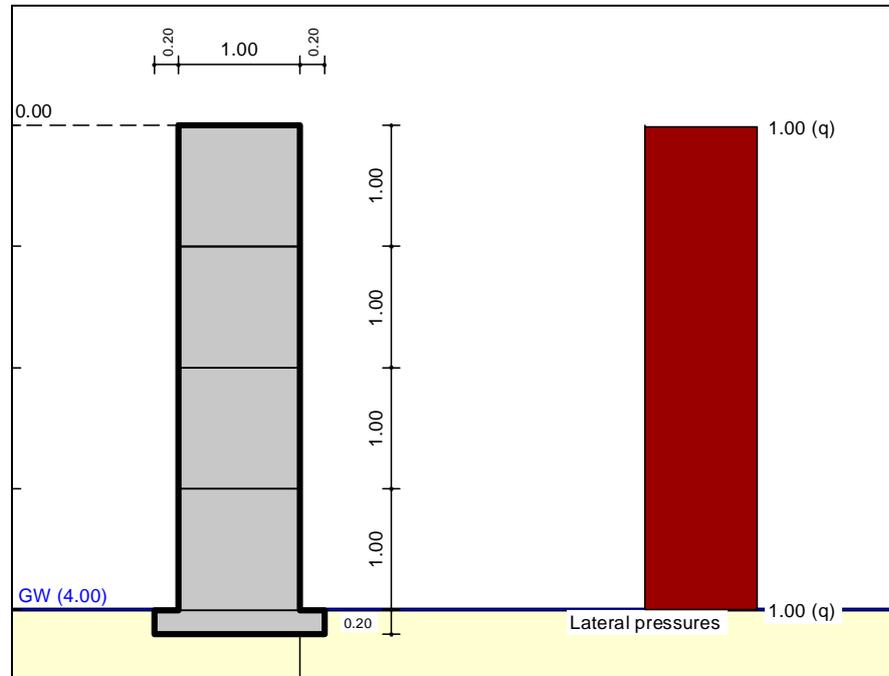


Figure 2 Example system 2

The following material properties are present:

- **Gabions**
  - unit weight  $\gamma = 16 \text{ kN/m}^3$
  - friction coefficient in the joints  $\mu = 0,75$
  - design concrete compressive strength  $f_{m,d}$  (EC 2) of the elements =  $0,20 \text{ MN/m}^2$
- **Sand**
  - friction angle  $\varphi = 32,5^\circ$
  - cohesion  $c = 0 \text{ kN/m}^2$
  - wet unit weight  $\gamma = 19 \text{ kN/m}^3$
  - buoyant unit weight  $\gamma' = 10 \text{ kN/m}^3$
  - no passive earth pressure used

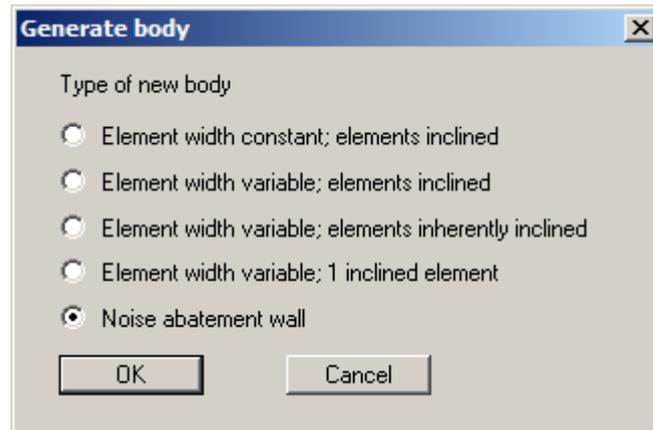
Due to the wind loads the active wall friction angle must be set to zero to analyse noise abatement walls. Otherwise, the wind load would generate a favourably acting active earth pressure at the rear face of the wall.

### 6.2.2 Step 1: Select analysis options (Ex. 2)

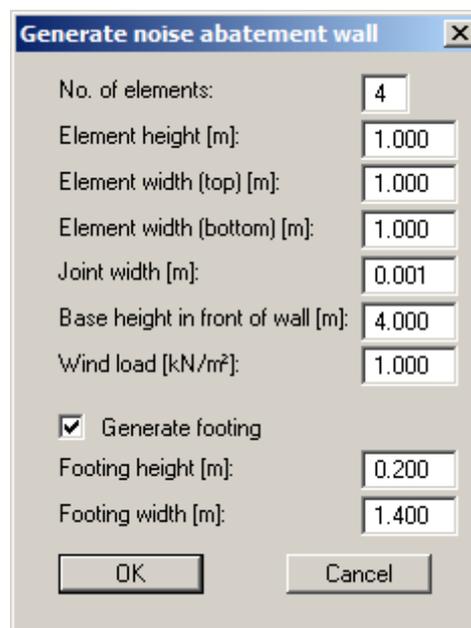
After starting the program you will see the **GGU-GABION** start-up screen. Select the **"File/New"** menu item and accept all presettings. If you prefer, you can enter a project identification, e.g. **"Example 2"**. A new system is then displayed on the screen.

### 6.2.3 Step 2: Generate noise abatement wall (Ex. 2)

Select the **"Editor 1/Body (geometry)"** menu item. To easily generate the gabion elements, click the **"Generate"** button. In the subsequent dialog box, select the **"Noise abatement wall"** option button 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.

### 6.2.4 Step 3: Define material of gabion (Ex. 2)

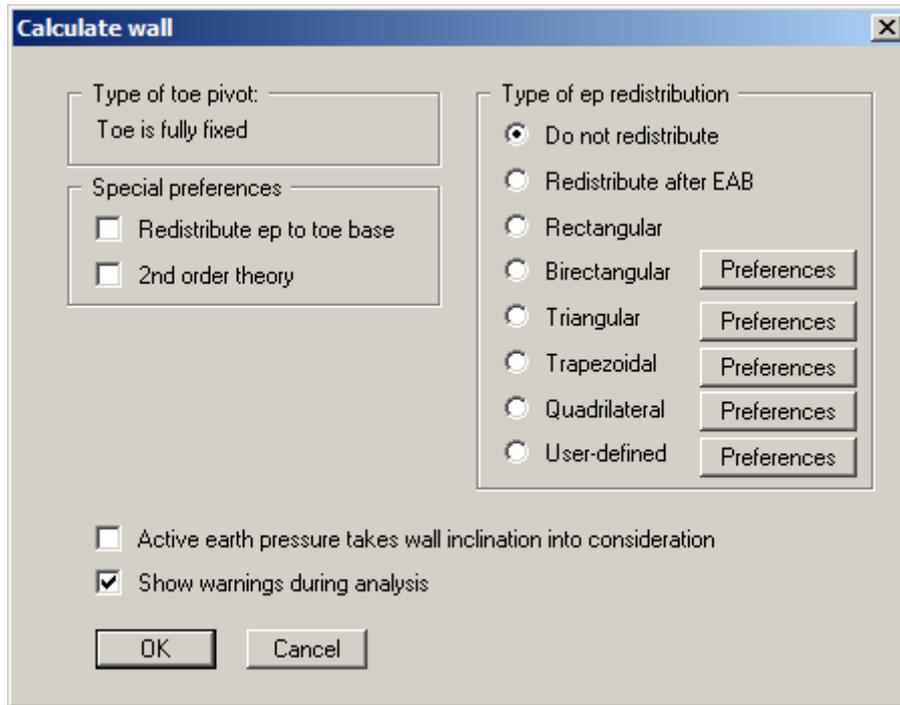
Select the **"Editor 1/Material"** menu item and adapt the unit weight of the gabion elements.

The screenshot shows the 'Material' dialog box with the following settings:

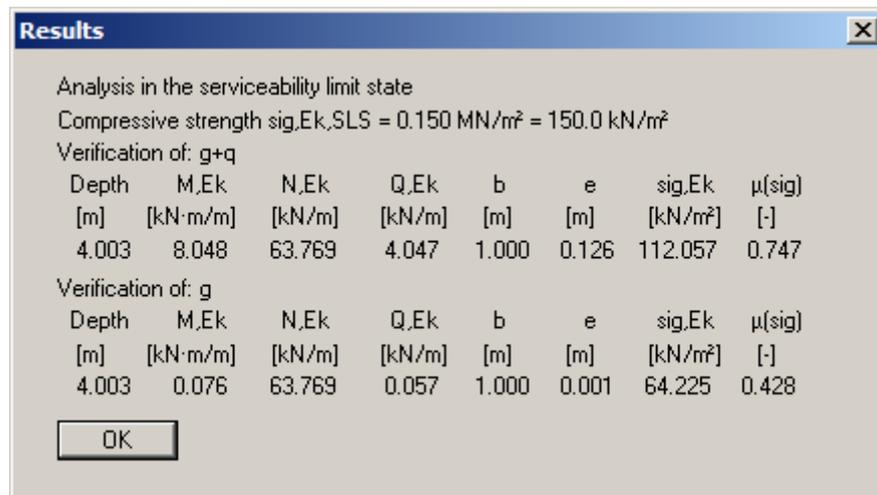
- gamma (Gabion)**: 16.00 [kN/m³]
- '2nd gamma' (Gabion)**:
  - Use '2nd gamma'
  - '2nd gamma' [kN/m³]: 23.00
  - below [m]: 9.00
- Water in gabion**:
  - 'Buoyant gamma' if groundwater in wall
  - No water pressure on gabions
  - Help button: ?
- Gabion**:
  - Verify internal stability
  - from [m]: 0.000
  - to [m]: 999.000
  - fm,d [MN/m²]: 0.200 (Help: ?)
  - sig.Ek,SLS [MN/m²]: 0.150 (Help: ?)
  - Young's mod. [kN/m²]: 1.0000E+5
  - n-value for load distribution (see DIN 1045 Table 17): 2.000
  - Wall top kept horizontal (Help: ?)
  - No buckling analysis for displacement to earth side

### 6.2.5 Step 4: 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.

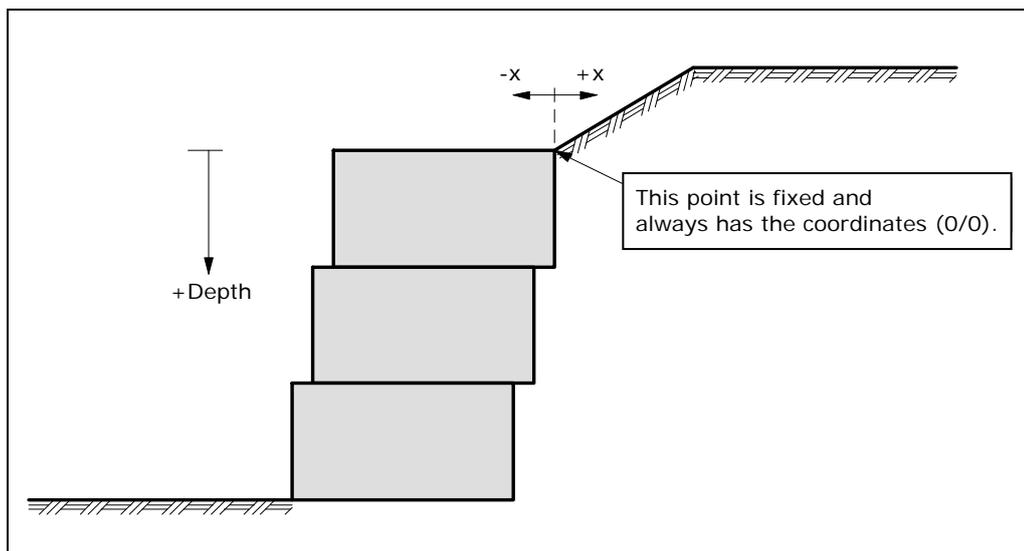


Figure 3 Gabion body

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/m<sup>3</sup>] of moist soil  $\gamma$  ;
- Unit weight [kN/m<sup>3</sup>] of buoyant soil  $\gamma'$  ;
- Friction angle [°];
- Cohesion (active and passive) [kN/m<sup>2</sup>];
- Active angle of wall friction as ratio  $\delta_a/\varphi$  ;
- Passive angle of wall friction  $\delta_p/\varphi$  ;
- Constrained modulus  $E_s$  [kN/m<sup>2</sup>] (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.

### **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. **GGU-GABION** also provides the option of applying user-defined earth pressure coefficients, which can be entered in tabulated form for each type of soil.

### **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:

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

$$0.0 \leq f \leq 1.0$$

### **7.6 Passive earth pressure**

---

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

- DIN 4085:2011;
- DIN 4085:1987;
- Streck,
- Caquot/Kerisel;
- DIN 4085:1987/Caquot/Kerisel.

Besides calculated values, you can also define your own.

## 7.7 Berms

GGU-GABION can handle 20 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).

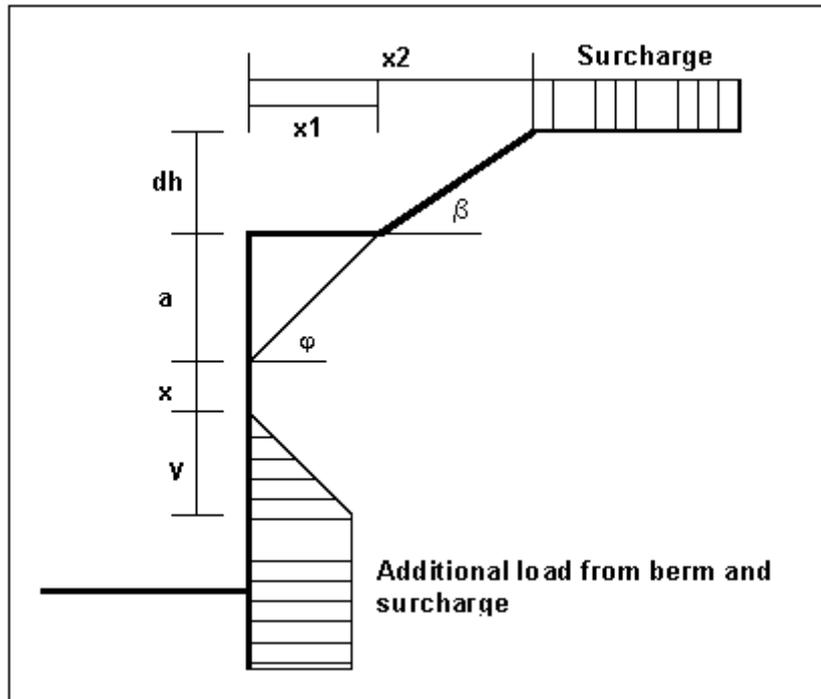


Figure 4 Berm on the active side

The following relationships apply for the parameters  $x$  and  $y$ :

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

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

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

$\gamma$  = unit weight of soil in the berm area

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

## 7.8 Area loads

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

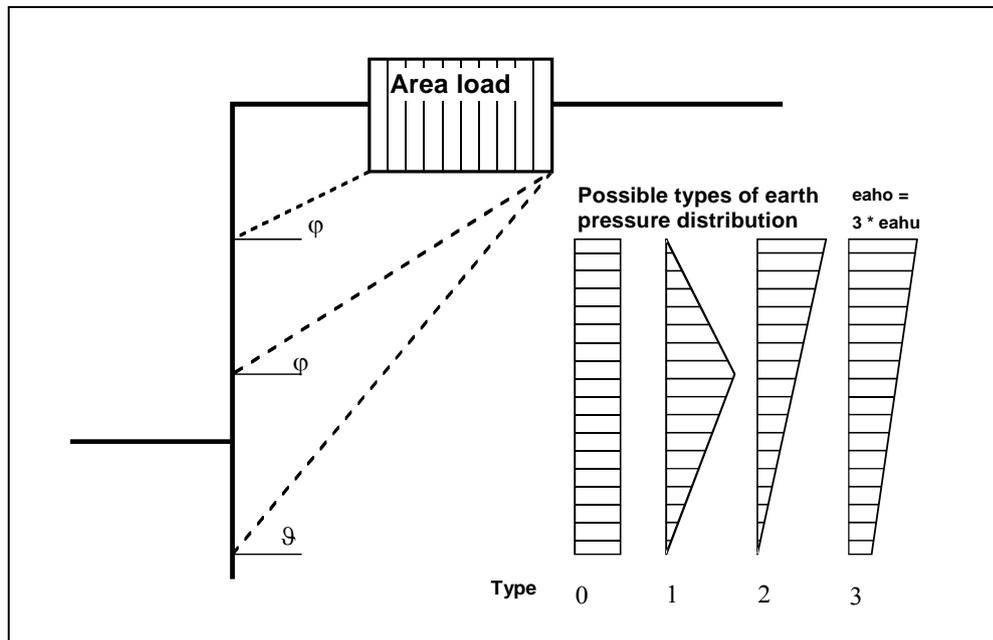


Figure 5 Area load

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  $k_0/k_{ah}$  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:  

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

$$e_{op} = q/4 (\sin^3\beta_2 - \sin^3\beta_1)$$

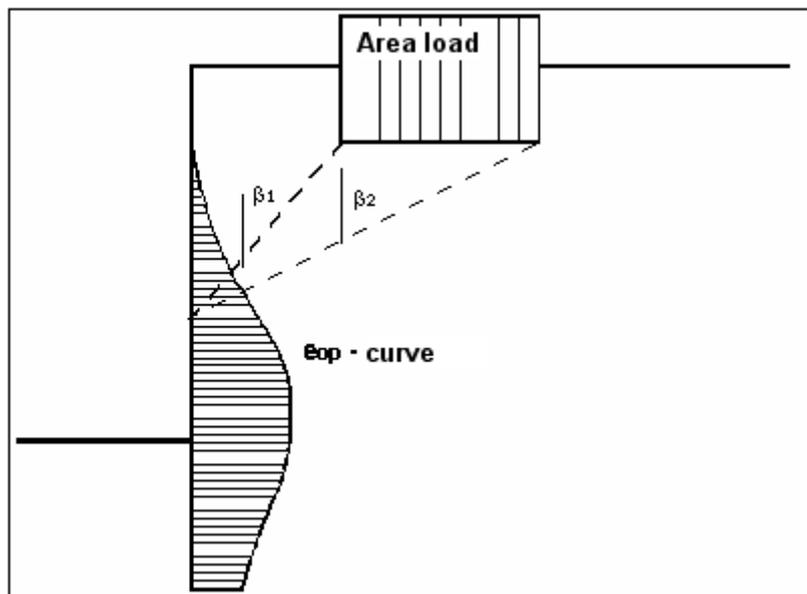


Figure 6 At-rest earth pressure from area loads

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 10 bounded surcharges can be positioned at any height on the active side.

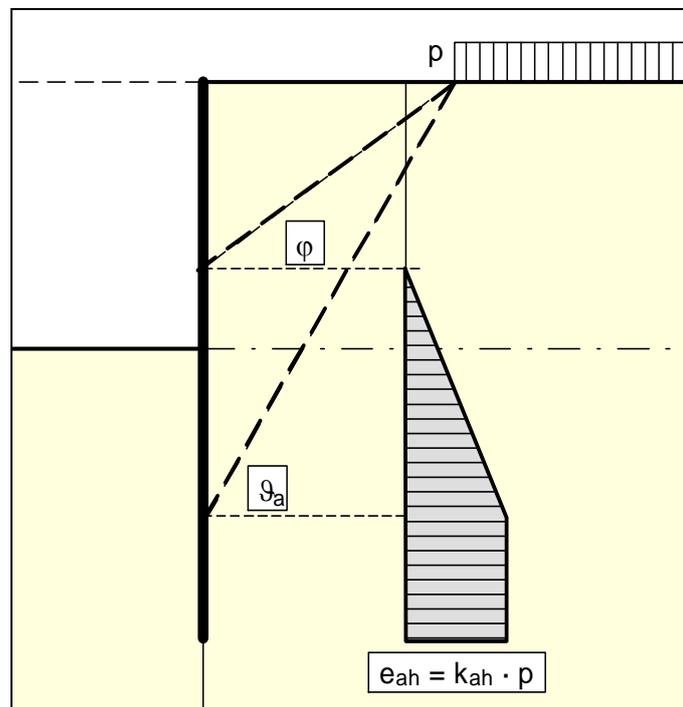


Figure 7 Bounded surcharge (active side)

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  $\varphi$  and  $\vartheta$  may not be adopted.

### 7.11 Double-bounded surcharges (active side)

If two bounded surcharges are defined as follows:

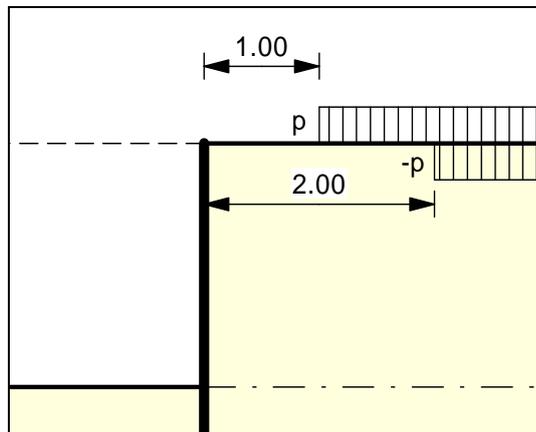


Figure 8 Two bounded surcharges

- 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. The analysis result is correct in terms of the adopted approach. However, the graphics are not really satisfactory and may be irritating for an examiner. Double-bounded surcharges were therefore introduced.

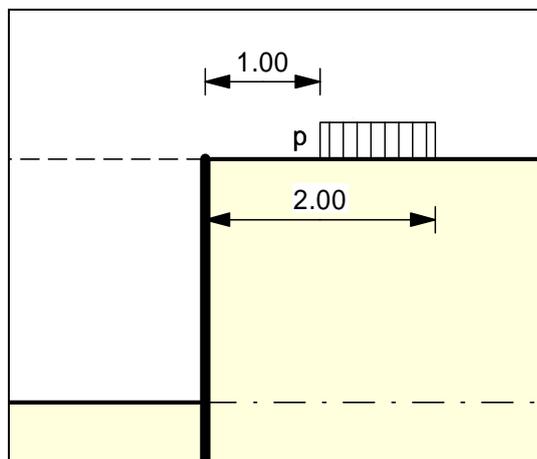


Figure 9 Double-bounded surcharge

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. Generally speaking, the earth pressure from double-bounded surcharges is less than that from area loads.

### 7.12 Bounded surcharges (passive side)

---

Up to 10 bounded surcharges may be adopted at any height on the passive side. The passive earth pressure is computed as follows:

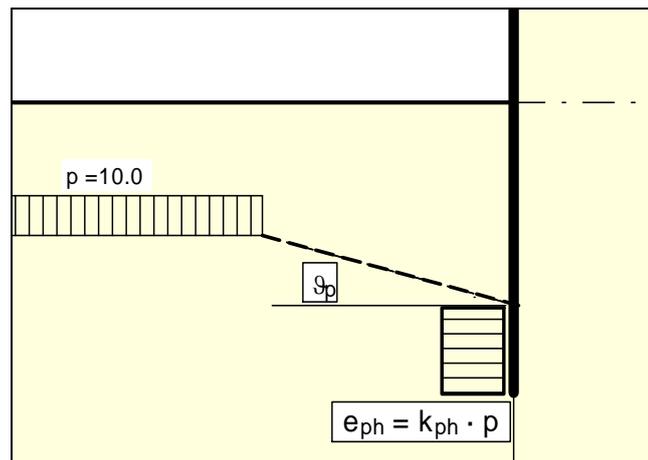


Figure 10 Bounded surcharge (passive side)

### 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.

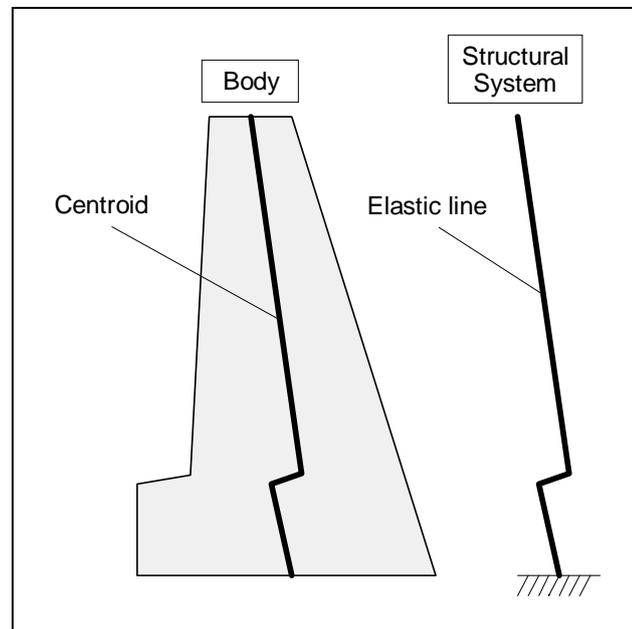


Figure 11 Structural system of a wall without geogrid

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.

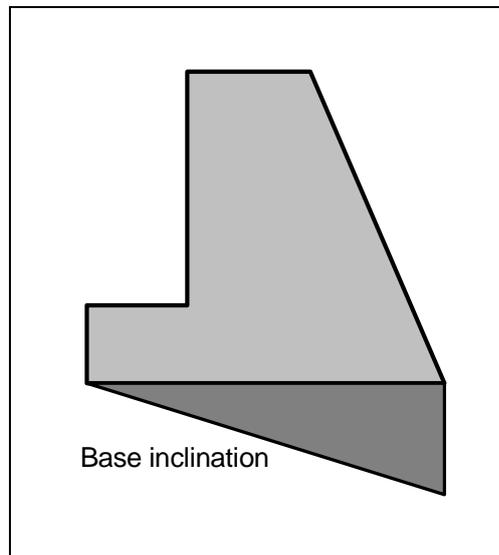


Figure 12 Base inclination

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 Verification concept

---

### 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 earth walls with an outer skin of gabions.

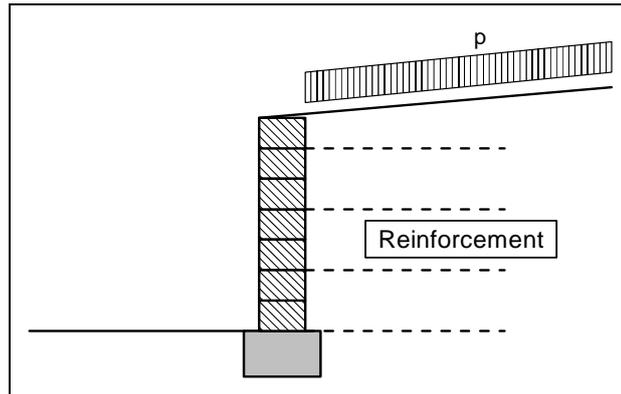


Figure 13 Wall with geogrid

Verification of the overall reinforced earth 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 earth system:

#### Verification of the bearing capacity of reinforced earth 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 outer skin;
- Verification of internal stability of the reinforced earth system (see Section 7.21.1).

#### Verification of the serviceability of reinforced earth systems

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

Because the outer skin 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 serviceability verifications are also required:

**External stability (bearing capacity) of the "outer skin manufactured from gabions"**

- 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 "outer skin manufactured from gabions"**

- 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 outer skin (gabions).

**Verification of the serviceability of the "outer skin manufactured from gabions"**

- 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:

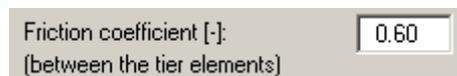
$$\eta = V_k \cdot \tan \varphi / H_d$$

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

The friction angle  $\varphi$  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 :



Friction coefficient [-]:   
(between the tier elements)

The program calculates the sliding safety using:

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

for all horizontal sections.

## 7.21 Verifications reinforced earth system

### 7.21.1 Internal stability

Verification of the internal stability of the reinforced earth 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).

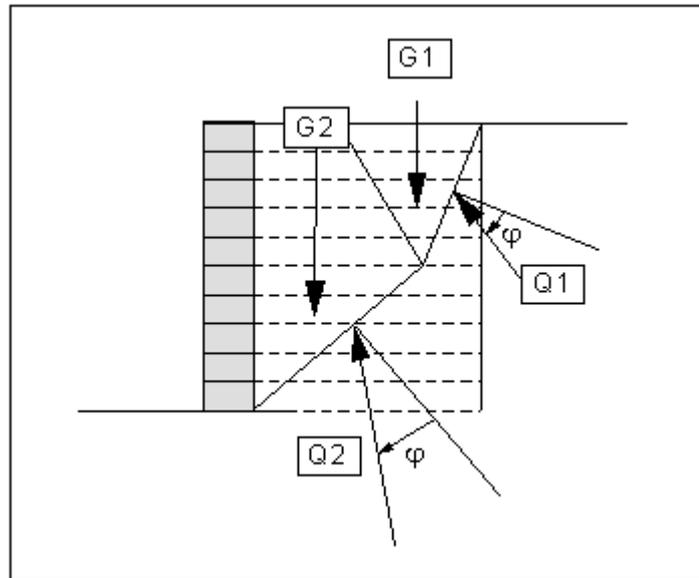


Figure 14 Two-part wedge failure mechanism

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:

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

- $R_{B,k0}$  = characteristic value of the short-term strength  
 $\gamma_M$  = 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

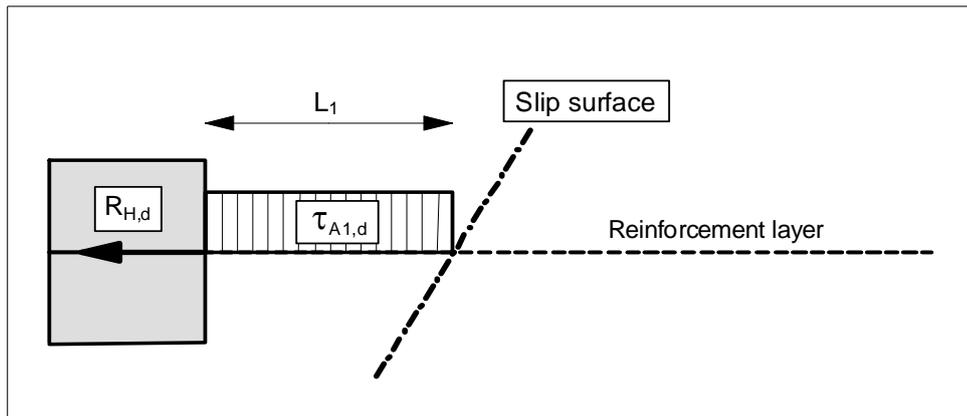
The following comparisons need to be considered when calculating the maximum **design value of the pull-out resistance**  $R_{A,d}$   $R_{A,d}$ :

- 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 outer skin and the bonding stress  $\tau_{A1,d}$  that can be activated within the slip body.

According to **EBGEO** a value of

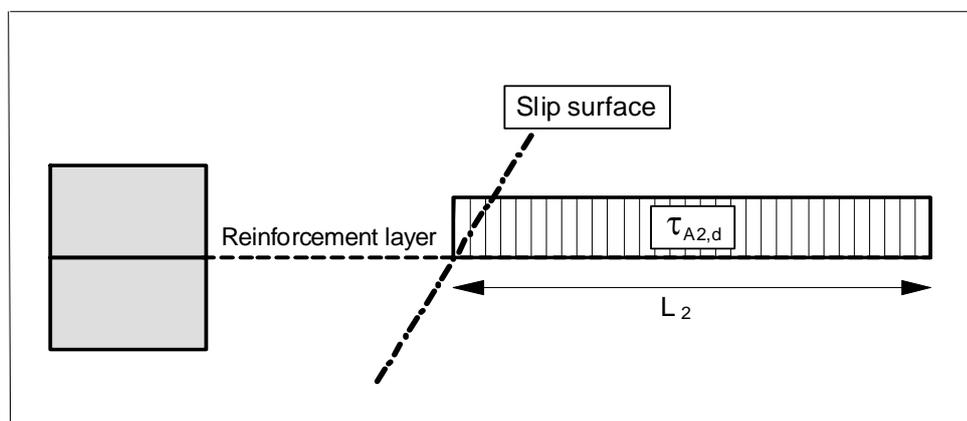
$$R_{H,d} = 0,8 \cdot R_{B,d}$$

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
- $\lambda_k$  = characteristic ratio of friction angle between geosynthetic and fill
- $\varphi_k$  = characteristic angle of internal friction of fill
- $\sigma_{V,k}$  = characteristic normal stress due to reinforcement layer surcharge
- $\gamma_B$  = 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 earth system the weights and horizontal earth pressures, amongst other things, acting on the earth body must be calculated. This is done on equivalent systems. The data in Figure 15 can be used for the weight calculation.

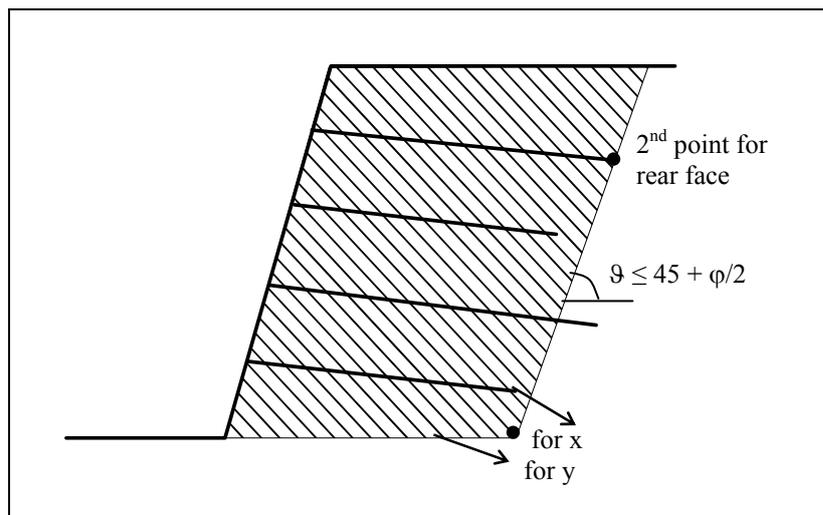


Figure 15 Equivalent system for weight calculation

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 outer skin of gabions. 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.

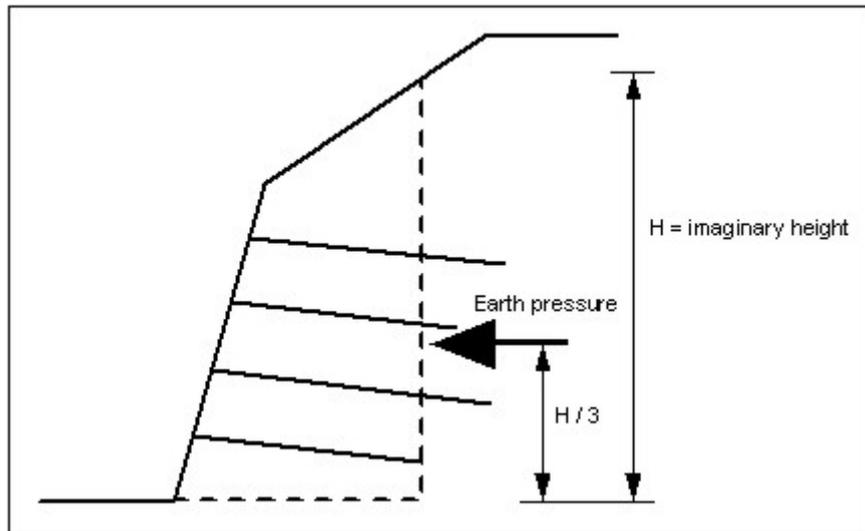


Figure 16 Vertical section for earth pressure determination

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  $k_{ah}$  for both a vertical rear face ( $\alpha = 0$ ) and an inclined rear face ( $\alpha \neq 0$ ), after determining the earth pressure:

$$\text{Reduction factor (earth pressure)} = k_{ah}(\alpha)/k_{ah}(\alpha = 0)$$

Sliding safety  $\eta_G$  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  $\varphi$  is obtained from the mean of the soils in the base of the reinforced earth system.

### 7.21.3 Overturning safety

Using the information provided in Section 7.21.2 "**Verifications of reinforced earth 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 outer skin

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

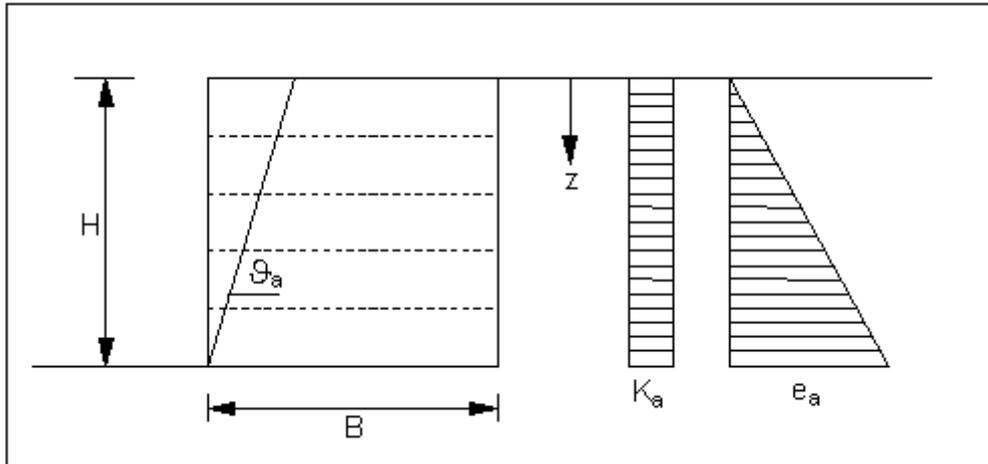


Figure 17 Slip surface and earth pressure distribution

The load  $q$  acting on the outer skin  $q$  is calculated from:

$$q = f_q \cdot k \cdot \gamma \cdot z$$

Vertical walls with solid outer skin:

$$f_q = 0.8 \text{ for } B/H \geq 0.7 \text{ and } f_q = 1.0 \text{ for } B/H \leq 0.5$$

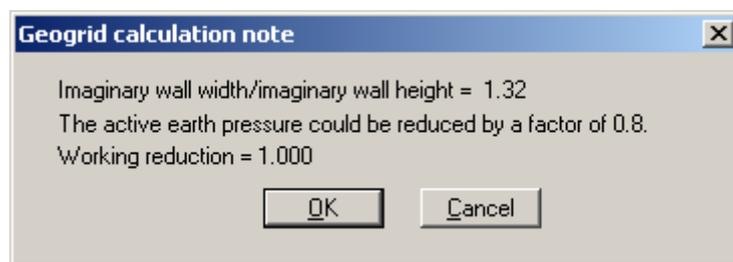
(linear interpolation between)

$k$  = earth pressure coefficient according to Figure 17

$\gamma$  = 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".

Active earth pressure

Active earth pressure after:

DIN 4085     User-defined    ?

Equivalent earth pressure coefficient

Use equivalent ep coefficient  
Equivalent kah [-]    0.200

Equivalent ep coefficient via  $\phi = 40^\circ$

Additional ordinates at 0.5 m intervals

Surcharge factor for active earth pressure

Surcharge factor [-]    0.8000

The active earth pressure and surcharge factor will be multiplied.

OK    Cancel

## 7.22 Bearing capacity safety

### 7.22.1 Analysis principles

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

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

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

- $\sigma_{of,k}$  = characteristic bearing stress
- $c$  = cohesion [kN/m<sup>2</sup>]
- $N_c$  = bearing capacity coefficient cohesion
- $\gamma_1$  = unit weight of soil above footing base
- $d$  = embedment depth of footing
- $N_d$  = bearing capacity coefficient footing depth
- $\gamma_2$  = 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 $N_b$

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

The following values are adopted:

- $N_{c0}$ ,  $N_{d0}$ ,  $N_{b0}$  according to DIN 4017
- $v_c$ ,  $v_d$ ,  $v_b$  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  $v_d$  and  $v_b$  for strip footings, as this delivers somewhat more favourable values.

$$v_d = 1 + 0.02 \cdot b/a \quad (\text{instead of } 1.0 \text{ for strip footings})$$

$$v_b = 1 + b/a \cdot \sin(\varphi) \quad (\text{instead of } 1.0 \text{ for strip footings})$$

- $i_c$ ,  $i_d$ ,  $i_b$  according to DIN 4017 (load inclination coefficients)
- $\lambda_c$ ,  $\lambda_d$ ,  $\lambda_b$  according to DIN 4017 (ground inclination coefficient)
- $\xi_c$ ,  $\xi_d$ ,  $\xi_b$  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:

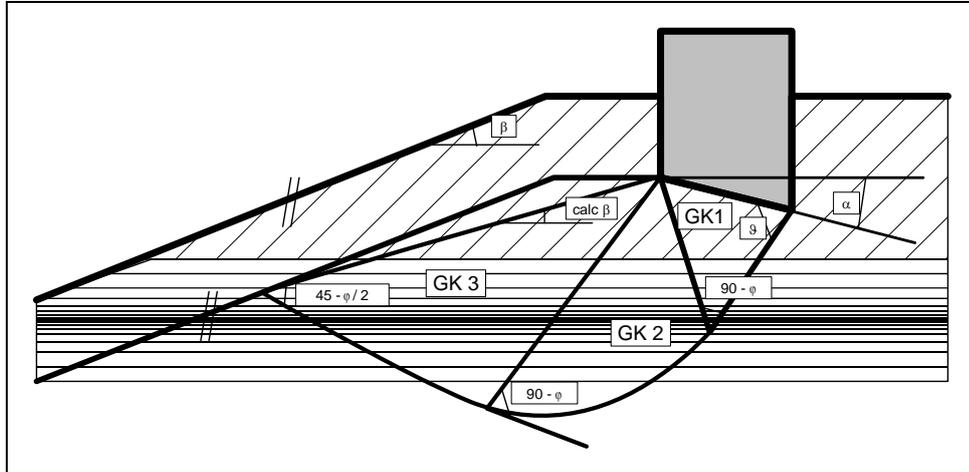


Figure 18 Logarithmic spiral

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

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

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

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

$l_i$  = 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  $5^\circ$  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/m<sup>2</sup> 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 Analysis of internal stability

### 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.

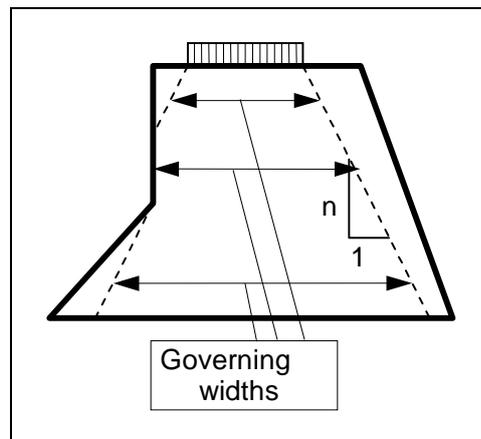


Figure 19 n-value and width for internal stability analysis

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

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

b = governing width of concrete

e = eccentricity with regard to governing width

$\kappa$  = reduction for slenderness of structural element

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

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

m = e/k (e = M/V and k = b/6)

$\lambda$  = 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 wall and the upper geogrid layer, or the separation between the individual geogrid layers. The slenderness  $\lambda$  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:

$$g = \text{working N} / \text{allowable N.}$$

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:

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

$$\varphi = 1,14 \cdot (1 - 2 \cdot e_{tot}/b) - 0,02 s_k/b$$

$$(0 \leq \varphi \leq 1 - 2 \cdot e_{tot}/b)$$

$N_{R,d}$  = 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 =  $0.5 l_0/200$ .)

$e_\varphi$  = 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:

$$N_{R,d} \geq N_{E,d}$$

### 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.

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

$M_{E,d}$  = design value of the moment

$W$  = section modulus

$N_{E,d}$  = design value of normal force

$A$  = cross-sectional area

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

$N_{E,d}$  = 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:

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

$Q_{E,d}$  = 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:

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

$\alpha$  = coefficient for considering a long-term effect = 0.85

$\gamma_m$  = 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:

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

$M_{E,k}$  = characteristic value of the moment

$W$  = section modulus

$N_{E,k}$  = characteristic value of normal force

$A$  = cross-sectional area

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

$N_{E,k}$  = 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:

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

$N_{Ed}$  = design value of the normal force

$N_{cr}$  = ideal bifurcation load for the governing buckling case

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

$E$  = 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

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

---

## 8 Description of menu items

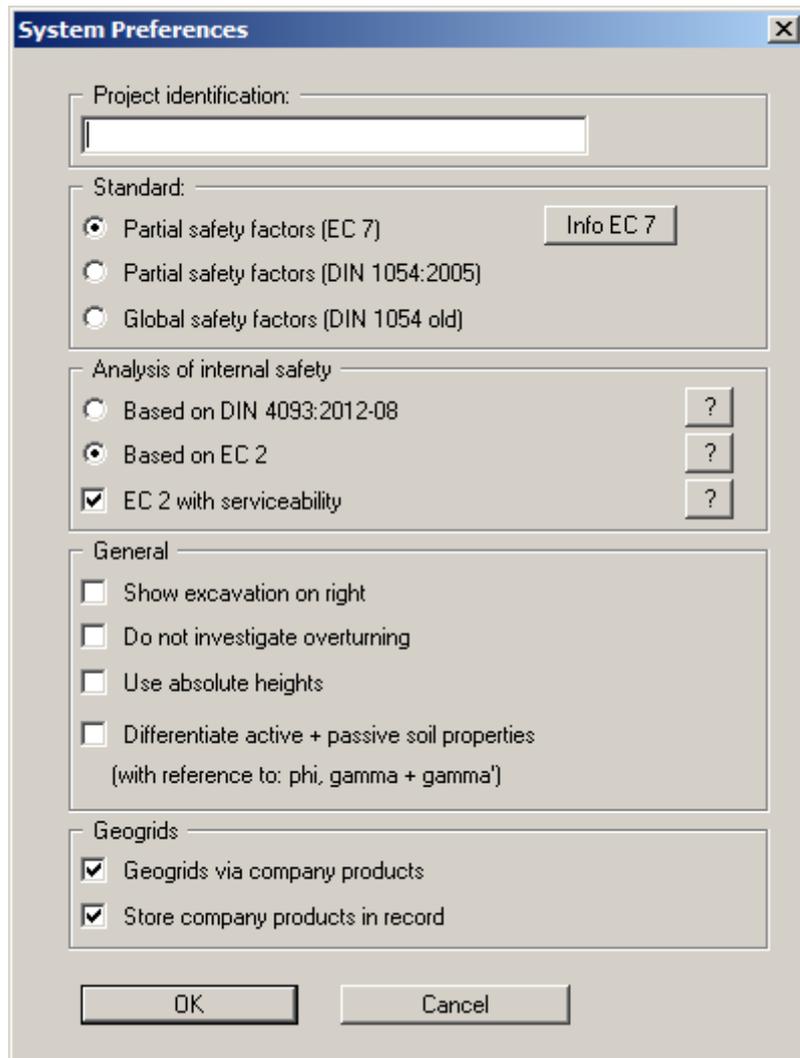
---

### 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, excavation 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 this check box is not selected, the top of the wall is assumed to have a height of **0.0** and all further entries, depth of soil layers, for example, are positive downwards.

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).

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 earth system/Enter geogrids manually**" or "**Reinforced earth 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

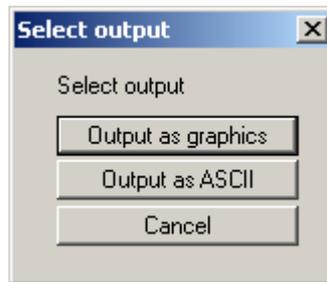
You can save data entered during program use to an existing file or to a new file, i.e. using a new file name. For reasons of clarity, it makes sense to use ".gab" as file suffix, as this is the suffix used in the file requester box for the menu item "**File/Load**". If you choose not to enter an extension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

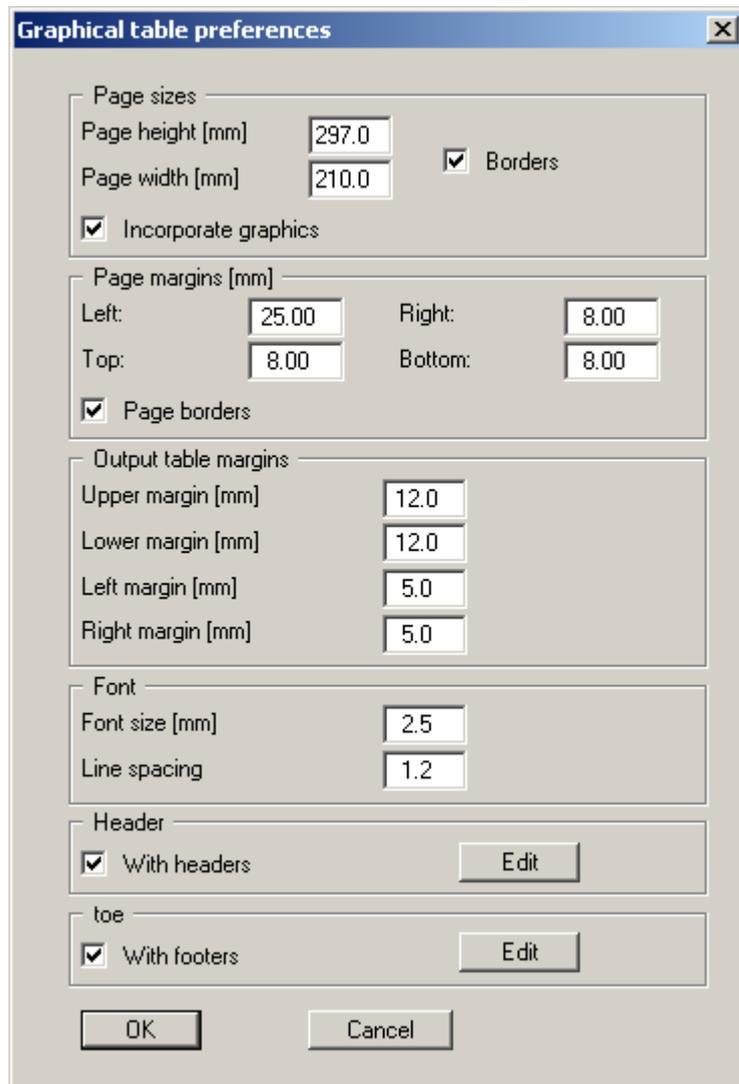
You have the option of designing and printing the output table as an annex to your report within the **GGU-GABION** application. To do this, select "**Output as graphics**" from the following options.



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.

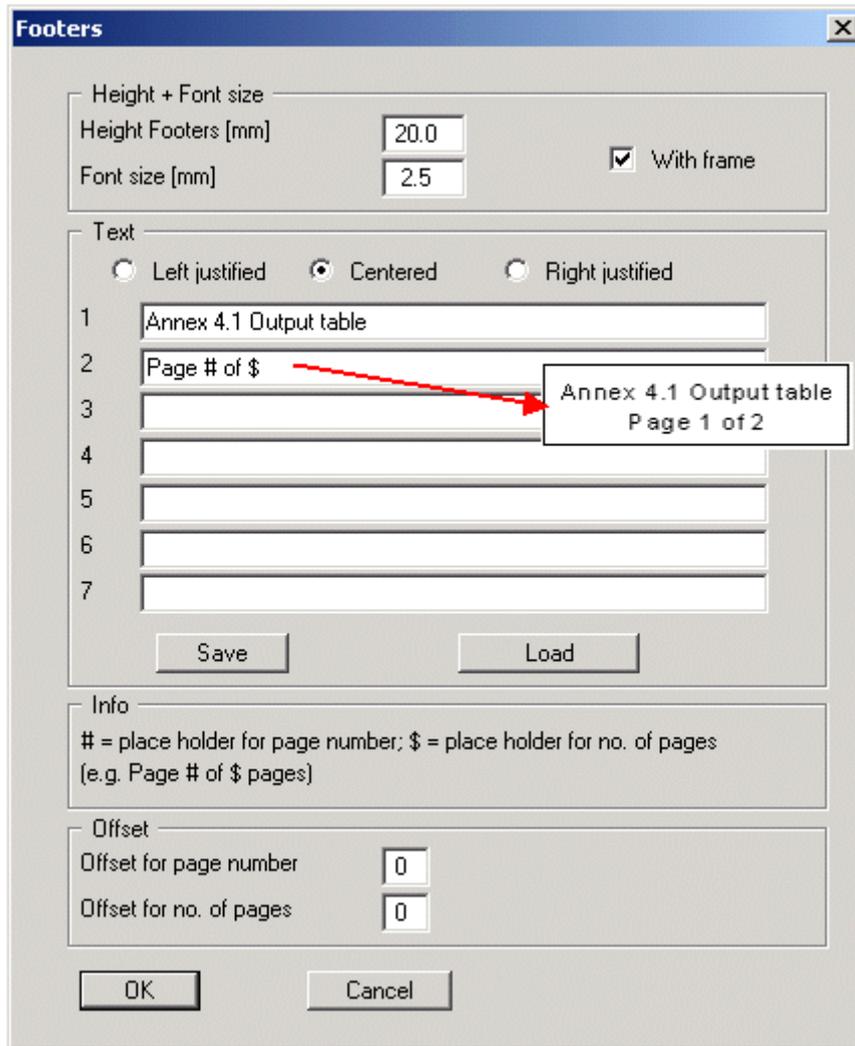


The dialog box titled "Graphical table preferences" contains several sections for configuring output settings:

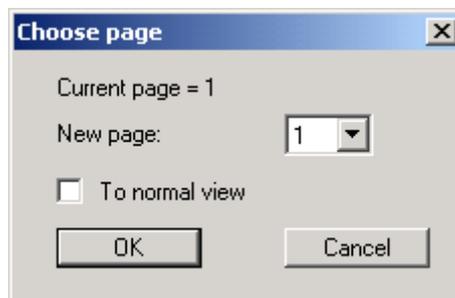
- Page sizes:** Page height [mm] is 297.0, Page width [mm] is 210.0. There is a checked checkbox for "Borders".
- Page margins [mm]:** Left: 25.00, Right: 8.00, Top: 8.00, Bottom: 8.00. There is a checked checkbox for "Page borders".
- Output table margins:** Upper margin [mm] is 12.0, Lower margin [mm] is 12.0, Left margin [mm] is 5.0, Right margin [mm] is 5.0.
- Font:** Font size [mm] is 2.5, Line spacing is 1.2.
- Header:** A checked checkbox for "With headers" and an "Edit" button.
- Footer:** A checked checkbox for "With footers" and an "Edit" button.

At the bottom of the dialog are "OK" and "Cancel" buttons.

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.

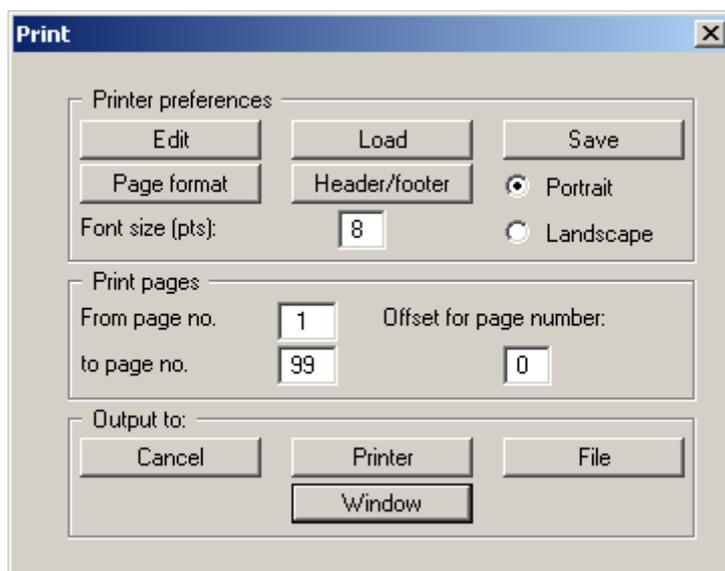


Automatic pagination can also be employed here if you work with the placeholders as described. After closing the dialog box using "OK" the output table is shown page by page on the screen. To navigate between the pages, use the arrow tools   in the toolbar. If you need to jump to a given page or back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

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

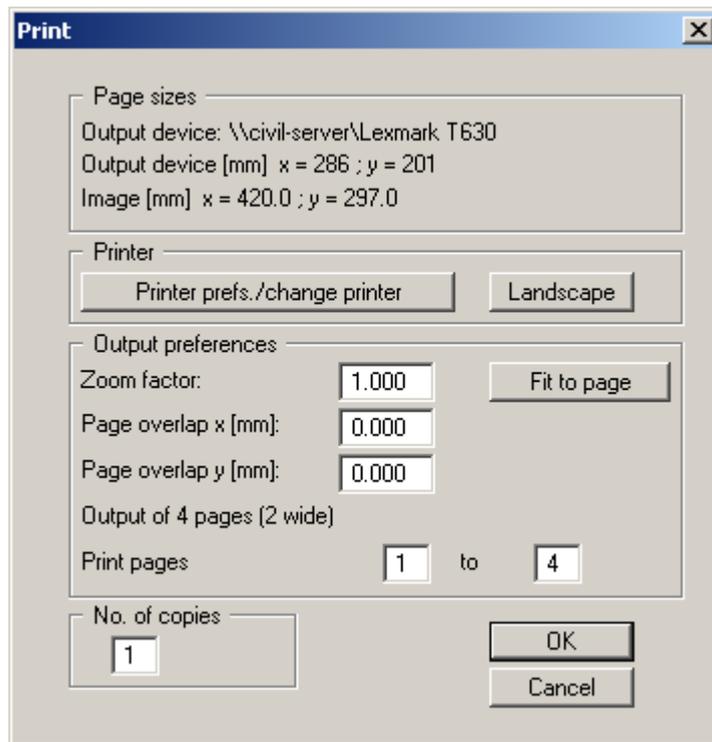
### 8.1.7 "Printer preferences" menu item

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

### 8.1.8 "Print and export" menu item

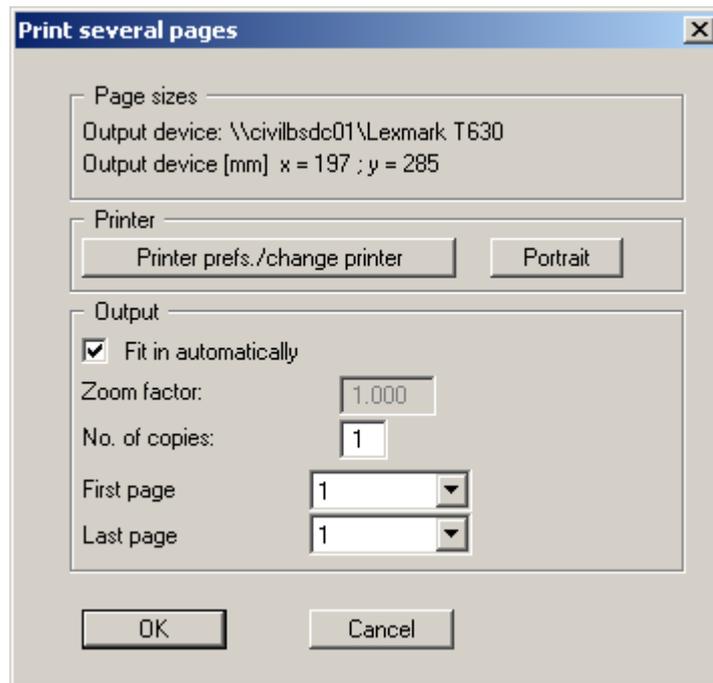
You can select your output format in a dialog box. You have the following options:

- **"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/Printer preferences"**. But you may also select a different printer in the following dialog box by pressing the **"Printer prefs./change printer"** button.



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

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



- Here, you can select the table pages to be printed. In order to achieve output with a zoom factor of 1 (button "Fit in automatically" is deactivated), you must adjust the page format to suit the size format of the output device. To do this, use the dialog box in "File/Print output table" button "Output as graphics".
- **"DXF file"**  
allows output of the graphics to a DXF file. DXF is a common file format for transferring graphics between a variety of applications.
- **"GGUCAD file"**  
allows output of the graphics to a file, in order to enable further processing with the **GGUCAD** program. Compared to output as a DXF file this has the advantage that no loss of colour quality occurs during export.
- **"Clipboard"**  
The graphics are copied to the WINDOWS clipboard. From there, they can be imported into other WINDOWS programs for further processing, e.g. into a word processor. In order to import into any other WINDOWS program you must generally use the "Edit/Paste" function of the respective application.
- **"Metafile"**  
allows output of the graphics to a file in order to be further processed with third party software. Output is in the standardised EMF format (Enhanced Metafile format). Use of the Metafile format guarantees the best possible quality when transferring graphics.

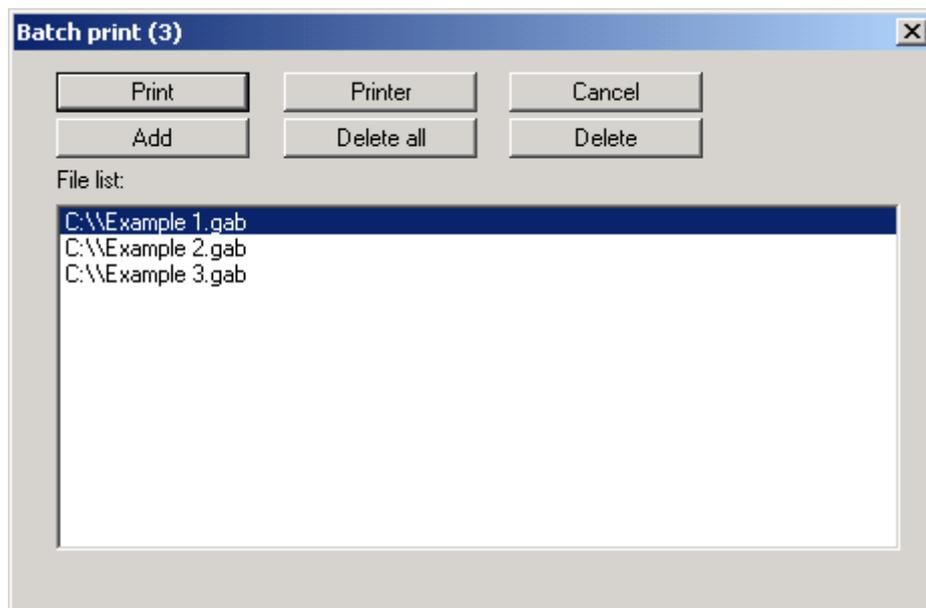
If you select the "Copy/print area" tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see "Tips and tricks", Section 9.4).

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

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

#### 8.1.9 "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 printer preferences is achieved by pressing the **"Printer"** button.

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

#### 8.1.10 "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 Editor 1 menu

---

### 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 "Excavation" menu item

A dialog box opens for defining the excavation:

Excavation	
Wall top [m]	0.00
Excavation base [m]	-2.00
Groundwater (right) [m]	-2.30
Groundwater (left) [m]	-2.30
Distributed load [kN/m <sup>2</sup> ]	10.00
Type of distributed load:	Component > 10.0 kN/m <sup>2</sup> changeable

Passive side	
Distributed load [kN/m <sup>2</sup> ]	0.00

In the upper group box, enter the depth of the excavation level 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/m<sup>2</sup> changeable**" (see the dialog box above). "**Component above 10.0 kN/m<sup>2</sup> changeable**" means, for example, that for an input of 13.5 kN/m<sup>2</sup>, 10 kN/m<sup>2</sup> are adopted as permanent and 3.5 kN/m<sup>2</sup> 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.

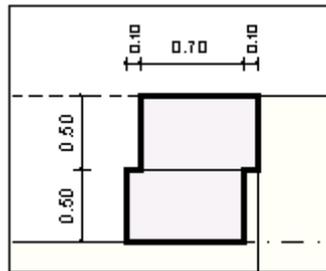


Figure 20 Body geometry

The corresponding data input is:

The screenshot shows the "Outer skin" dialog box with the following data input:

No.	Depth [m]	x(left) [m]	x(right) [m]
Wall top = 0.000		-0.800	0.000
2	0.500	-0.800	0.000
3	0.501	-0.900	-0.100
4	1.001	-0.900	-0.100

Buttons: Forward, Back, Cancel, Done, Load, 4 points to edit, Sort, Save, Generate, Interpolate.

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 2**", Section 6.2.3).

### 8.2.5 "Material" menu item

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

The screenshot shows the "Material" dialog box with the following settings:

- gamma (Gabion)**: 18.00 [kN/m<sup>3</sup>]
- '2nd gamma' (Gabion)**:
  - Use '2nd gamma'
  - '2nd gamma' [kN/m<sup>3</sup>]: 23.00
  - below [m]: 9.00
- Water in gabion**:
  - 'Buoyant gamma' if groundwater in wall
  - No water pressure on gabions
- Gabion**:
  - Verify internal stability
    - from [m]: 0.000
    - to [m]: 999.000
  - fm,d [MN/m<sup>2</sup>]: 0.200
  - sig.Ek,SLS [MN/m<sup>2</sup>]: 0.150
  - Young's mod. [kN/m<sup>2</sup>]: 1.0000E+5
  - n-value for load distribution (see DIN 1045 Table 17): 2.000
  - Wall top kept horizontal
  - No buckling analysis for displacement to earth side

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.

No.	Left [m]	Right [m]	Delta h [m]	Surcharge [kN/m <sup>2</sup> ]	Live load
1	1.000	2.000	0.50	0.00	<input type="checkbox"/>

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:

	Designation	Base	gam	gam'	phi	c(a)	c(p)	d(a)/phi	d(p)/phi	Es
		[m]	[kN/m <sup>2</sup> ]	[kN/m <sup>2</sup> ]	[°]	[kN/m <sup>2</sup> ]	[kN/m <sup>2</sup> ]	[-]	[-]	[MN/m <sup>2</sup> ]
1	Sand	-4.50	19.0	10.0	32.5	0.0	0.0	0.667	0.000	10.0

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.

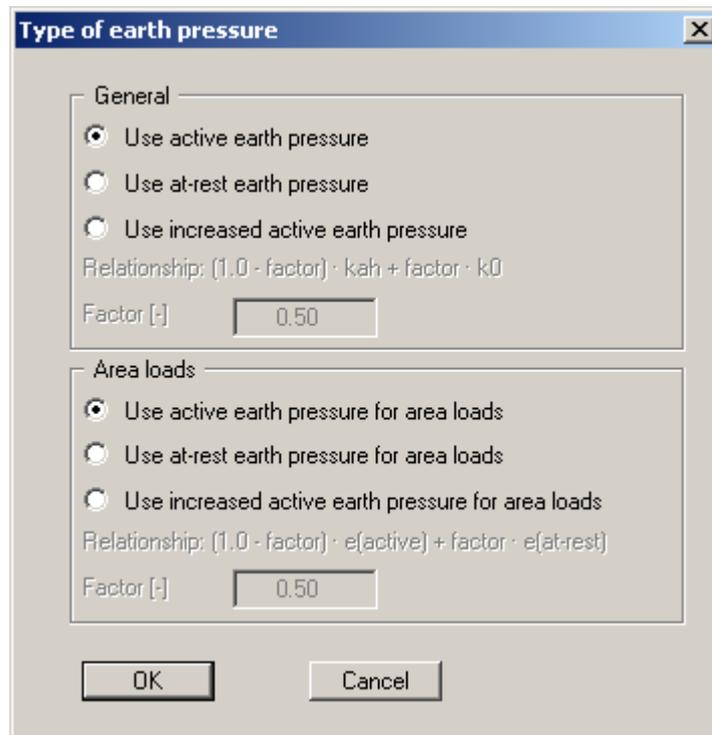
In stratified soils the number of layers must be entered under "**Edit no. of soils**". Clicking the "**Sort**" button sorts the soil layers according to depth; however, this is performed automatically when you click "**OK**" to leave the dialog box. This eliminates the possibility of input errors.

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:



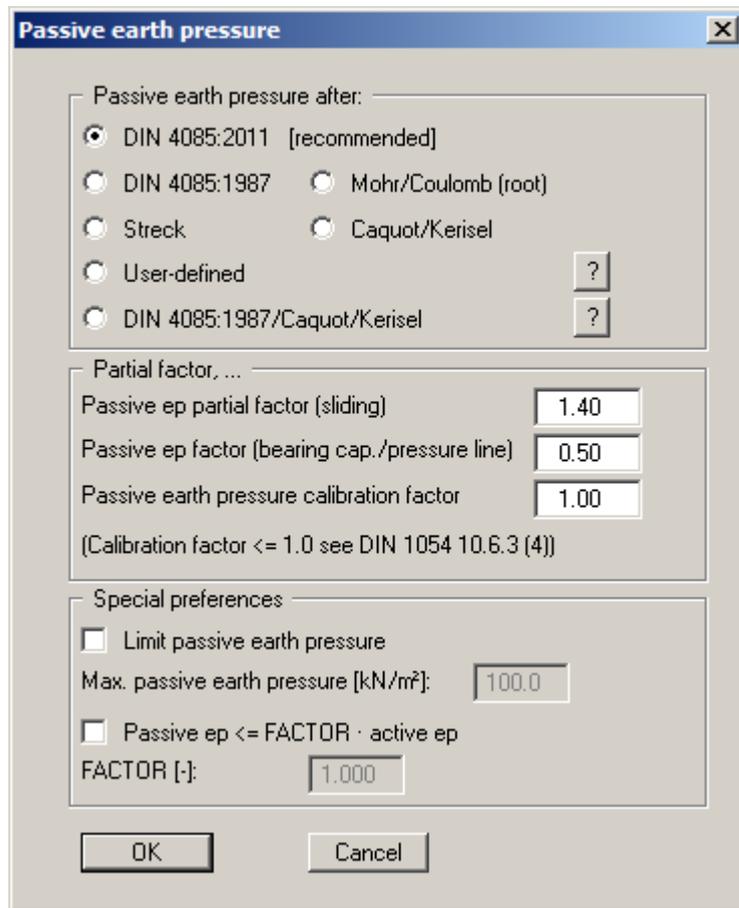
In the upper group box you specify the type of active earth pressure calculation. The method of choice is "**DIN 4085**". The other two methods are only of interest if you wish to analyse an example from older literature sources or check certain results. User-defined earth pressure coefficients can be entered using the menu item "**Editor 1/User-defined earth pressure coefficients**" (see Section 8.2.13).

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  $\phi = 40^\circ$ . 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:



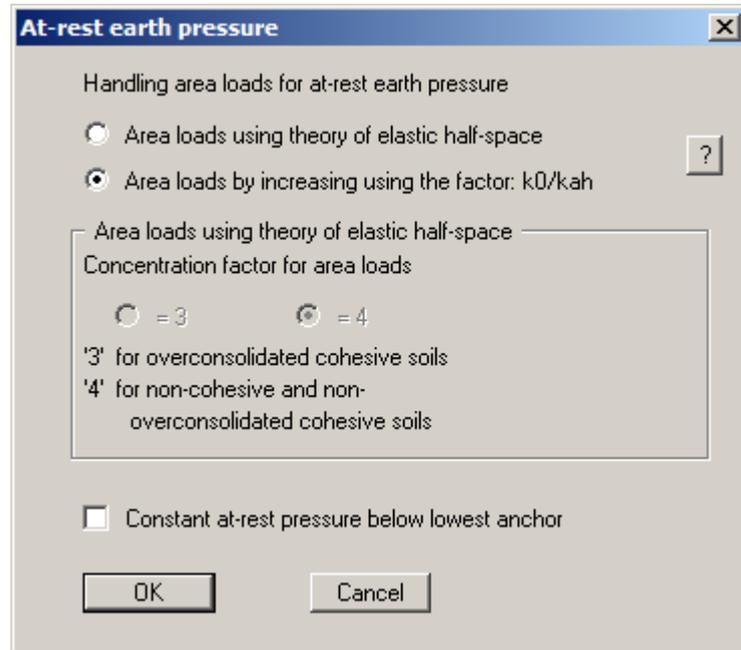
In the upper group box you specify the type of passive earth pressure calculation. The method of choice is "**DIN 4085:2011**". Only if you want to check examples from older literature or examine certain analysis results does it make sense to select a different method. User-defined earth pressure coefficients can be entered using the menu item "**Editor 1/User-defined earth pressure coefficients**" (see Section 8.2.13).

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  $k_0/k_{ah}$  in compliance with DIN 4085:2011 Section 6.4.3.

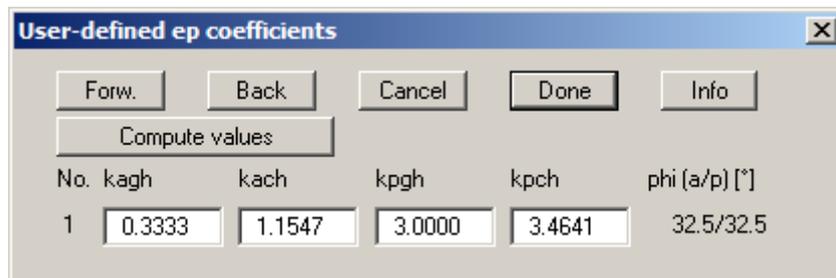
Alternatively, it is possible to determine the demands placed on the wall by area loads according to the theory of elastic half-space. The required concentration factor is specified in the following dialog box, if the upper check box is activated.



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.

The screenshot shows a dialog box titled "Safety factors" with the following settings:

- Sliding safety:** FOS = 1.50,  Verify,  Verify to DIN 1054, Friction angle in base plane [°] = 30.00
- 2-part wedge failure mechanism:** FOS = 1.40
- Bearing capacity:** FOS = 2.00,  Verify,  Bearing cap. with depth coeff. (Info),  Check and correct 5° condition, Decrement [°]: 0.10 (Info),  Top of bearing cap. body parallel to slope

Buttons: OK, Cancel

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.

The dialog box "Verifications/Partial factor preferences" contains the following settings:

- Actions/Resistance:**
  - Permanent actions: 1.35
  - Permanent actions (at-rest earth pressure): 1.20
  - Changeable actions: 1.50
  - Passive earth pressure: 1.40
  - Passive ep factor (bearing cap./pressure line): 0.50
  - Passive earth pressure calibration factor: 1.00 (Calibration factor  $\leq 1.0$ )
- Partial factors (GEO-3) 2-part wedge failure:**
  - Friction angle: 1.25
  - Cohesion: 1.25
  - Changeable actions: 1.30
- Bearing capacity:**
  - Verify  To new DIN 4017
  - gamma(bearing cap.): 1.40
  - Bearing cap. with depth coeff. [Info]
  - Check and correct 5° condition
  - Decrement [°]: 0.10 [Info]
  - Top of bearing cap. body parallel to slope
- Sliding:**
  - Verify gamma(sliding): 1.10
  - Friction angle in base plane [°] = 30.00
- Serviceability limit state verification:**
  - Sliding safety without passive earth pressure [Info]
  - Serviceability limit state verification [Info]
- EQU analysis:**
  - gamma(G.dst): 1.10 gamma(G.stb): 0.90
  - gamma(Q.dst): 1.50
- Default values:**
  - To DIN 1054:2010
  - To ÖNORM EN 1997-1

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

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 load case designations were altered for the EC 7 partial safety factor concept:

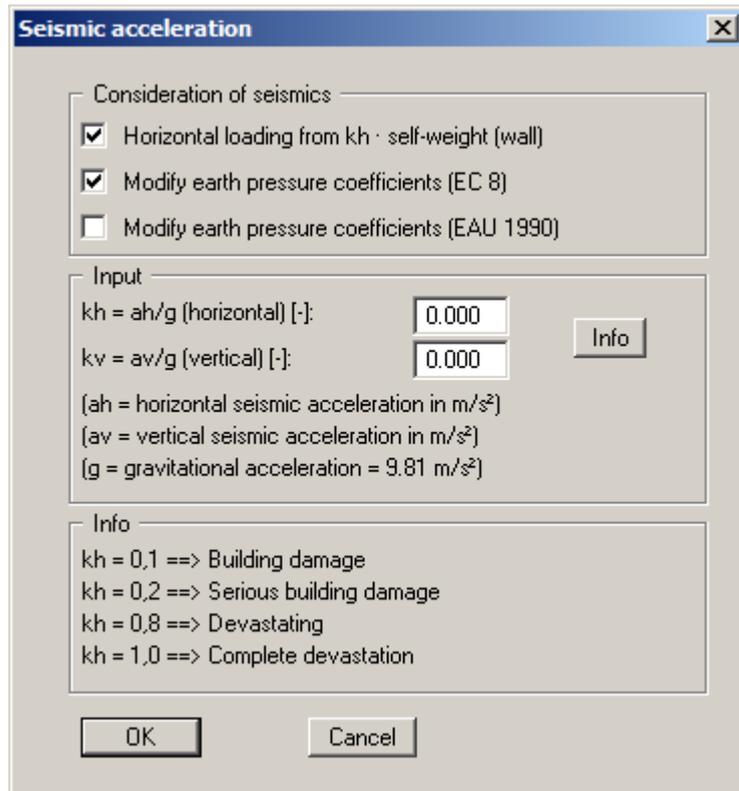
- Load Case 1 is now DS-P: Persistent Design Situation
- Load Case 2 is now DS-T: Transient Design Situation
- Load Case 3 is now DS-A: Accidental Design Situation

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.

### 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 Editor 2 menu

#### 8.3.1 "Area and line loads" menu item

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

No.	p(v) [kN/m <sup>2</sup> ]	p(h) [kN/m <sup>2</sup> ]	x(left) [m]	x(right) [m]	Depth [m]	Type	Live
1	150.00	0.00	2.000	3.000	1.000	Triangle (max. top)	<input type="checkbox"/>

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:

No.	p [kN/m <sup>2</sup> ]	x(left) [m]	Depth [m]	Live
1	25.00	2.000	0.000	<input type="checkbox"/>

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:

No.	p(v) [kN/m <sup>2</sup> ]	x(left) [m]	x(right) [m]	Depth [m]	Live
1	1.00	0.000	1.000	0.000	<input type="checkbox"/>

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.

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

### 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.

No.	Top [m]	Bottom [m]	e (top) [kN/m <sup>2</sup> ]	e (bottom) [kN/m <sup>2</sup> ]	as live
1	1.000	2.000	10.00	20.00	<input type="checkbox"/>

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.

No.	Depth [m]	M [kN*m/m]	H [kN/m]	V [kN/m]	as live
1	0.00	0.00	0.00	0.00	<input type="checkbox"/>

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

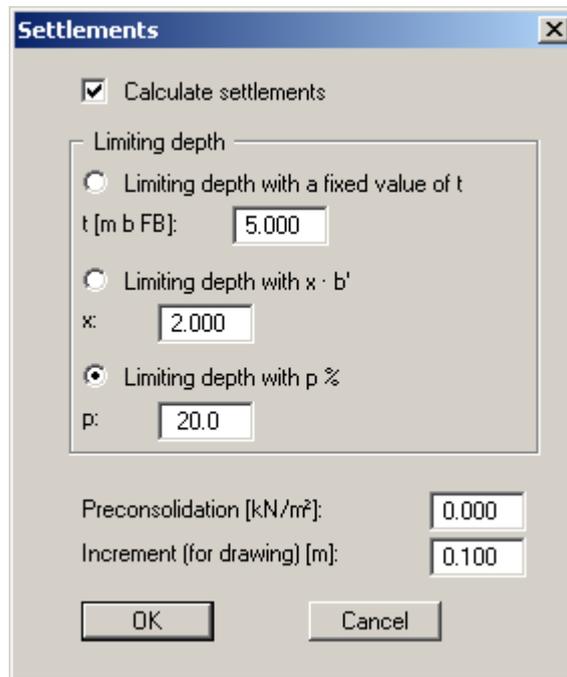
You can introduce additional displacement boundary conditions anywhere along the wall.

No.	Depth [m]	Size [m or radian]
1	2.00	0.00000

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/m<sup>2</sup> 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 Reinforced earth system menu

### 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 screenshot shows a dialog box titled "Reinforced earth system" with the following settings:

- Earth pressure wedge:**
  - Show
  - Label
  - Line strength [mm]: 0.2
  - Font size [mm]: 3.0
  - Dash length [mm]: 3.0
  - Pen colour: [Black swatch]
  - Dashed: [Dashed dropdown]
- Gravity body:**
  - Display
  - Label
  - Coloured body
  - Body colour: [Grey swatch]
- Show what else?:**
  - None
  - Shear stress
  - Shear stress/normal stress
  - Porewater pressure
  - Normal stress
  - Scale factor: 0.10000
  - Label slices
- Failure body with slices
- Show log. spiral (earth system)
- Label geogrid (friction)
- Geogrid bonding stress visualisation height [m]: 0.20

Buttons: OK, Cancel

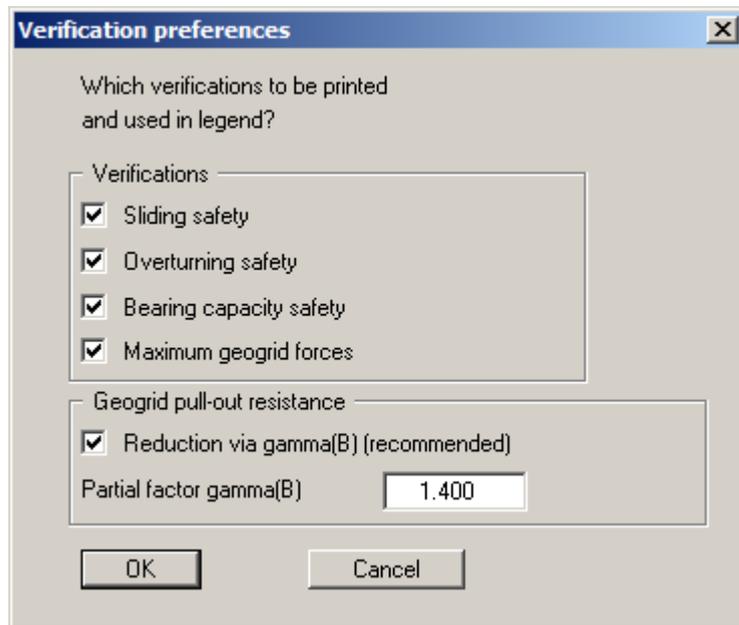
The earth pressure wedge and gravity body are explained in Section 7.21.2. The visualisation of these elements can be influenced in the two upper group boxes of this dialog box.

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 earth 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.

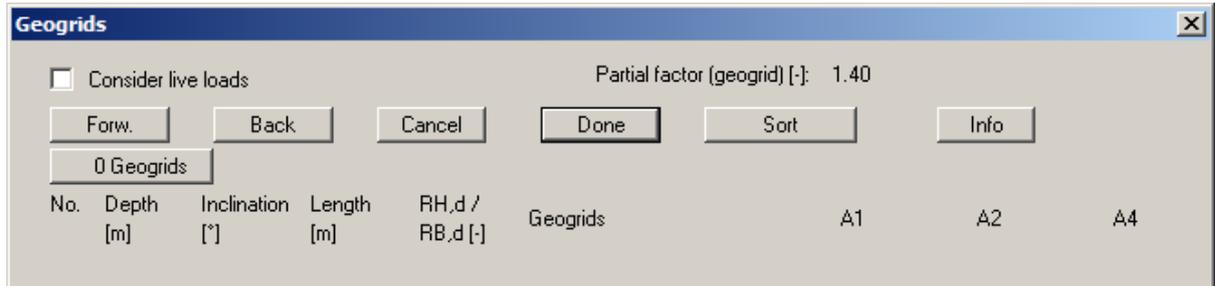


For example, if you do not want to calculate the bearing capacity safety, you can turn off the bearing capacity verification.

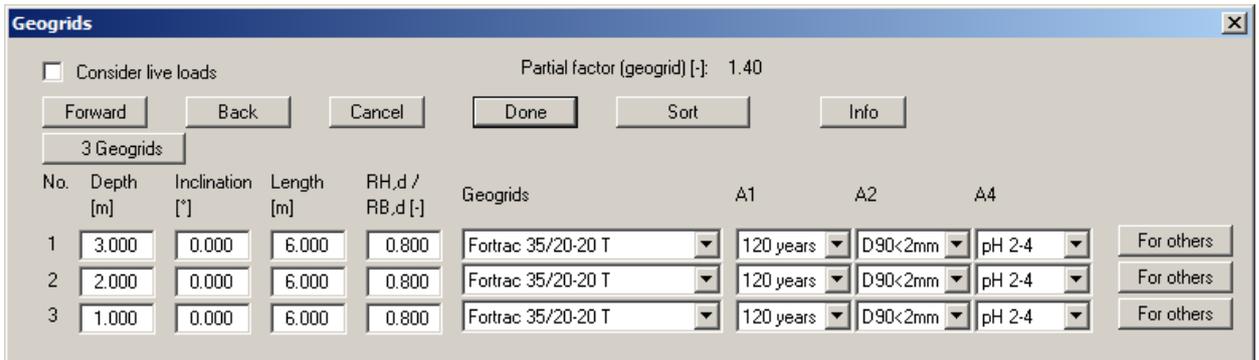
### 8.4.3 "Enter geogrids manually" menu item

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:

No.	Depth [m]	Inclination [°]	Length [m]	lambda [-]	R,H,d [kN/m]	R,B,d [kN/m]
1	1.5001	0.000	6.0000	0.7000	8.000	10.000
2	0.5001	0.000	6.0000	0.7000	8.000	10.000

Where:

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

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 earth 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 earth 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.

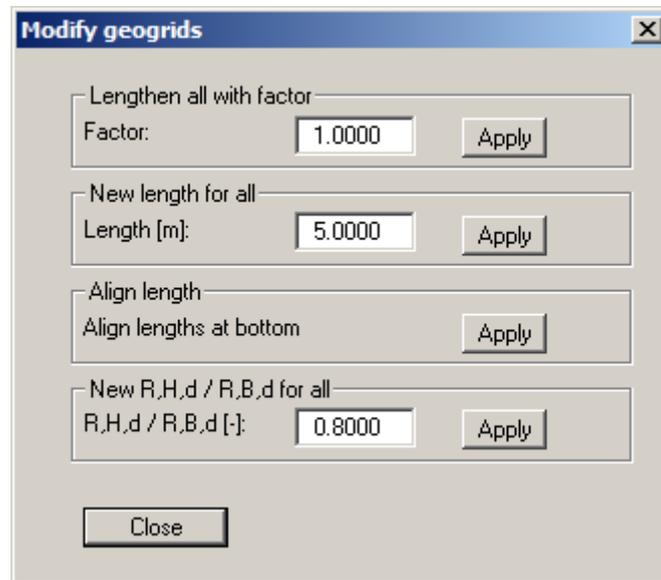
The dialog box "Generate geogrids" contains the following elements:

- Delete existing geogrids
- 1st geogrid (bottom) [m]: 0.500 (from base of outer skin)
- Generate to [m] below top: 0.100
- Geogrid vertical spacing [m]: 1.000
- Geogrids section:
  - Fortrac 35/20-20 T (dropdown)
  - A1: 120 years (dropdown)
  - A2: D90<2mm (dropdown)
  - A4: pH 2-4 (dropdown)
- R,H,d / R,B,d [-]: 0.800 (with Info button)
- Length bottom [m]: 6.000
- Length top [m]: 6.000
- Inclination [°]: 0.000
- OK and Cancel buttons

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 earth system/Enter geogrids manually**" menu item.

#### 8.4.5 "Modify geogrids " menu item

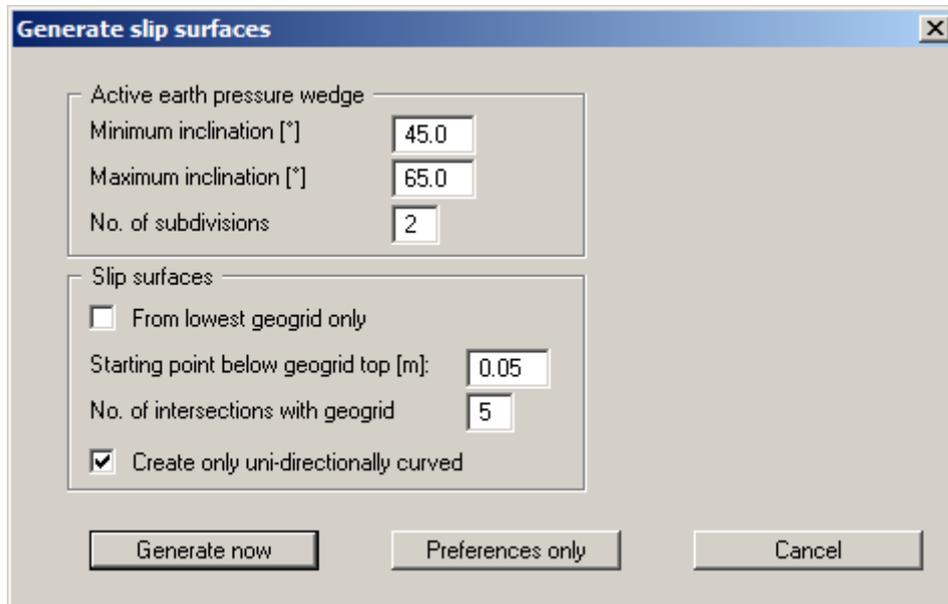
Using this menu item you can edit the data for all geogrids simultaneously in a single step. Enter a value and start the modification using the respective "Apply" button following the desired action. Only this action is subsequently performed.



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 earth 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  $45^\circ + \phi/2$  (see Section 7.21.2). Values smaller than  $45^\circ$  are thus not possible. Values greater than  $65^\circ$  only result if the angle of friction of the soil is greater than  $40^\circ$ . 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

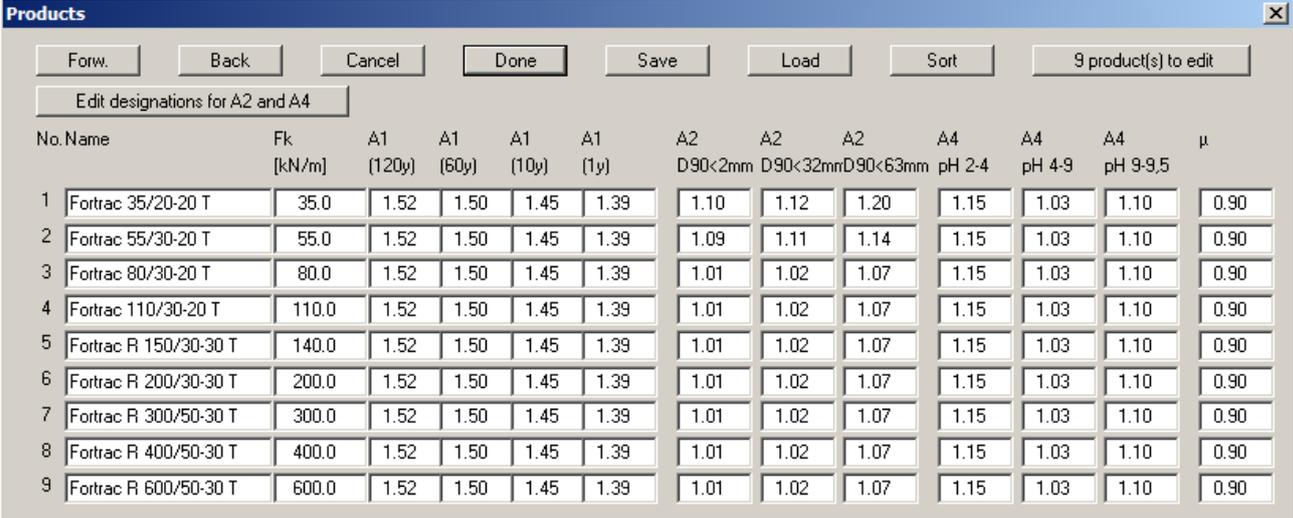
Verification is in accordance with the notes in the "Theoretical principles" section. You will see the results in dialog boxes.

#### 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<32mm	A2 D90<63mm	A4 pH 2-4	A4 pH 4-9	A4 pH 9-9,5	μ
1	Fortrac 35/20-20 T	35.0	1.52	1.50	1.45	1.39	1.10	1.12	1.20	1.15	1.03	1.10	0.90
2	Fortrac 55/30-20 T	55.0	1.52	1.50	1.45	1.39	1.09	1.11	1.14	1.15	1.03	1.10	0.90
3	Fortrac 80/30-20 T	80.0	1.52	1.50	1.45	1.39	1.01	1.02	1.07	1.15	1.03	1.10	0.90
4	Fortrac 110/30-20 T	110.0	1.52	1.50	1.45	1.39	1.01	1.02	1.07	1.15	1.03	1.10	0.90
5	Fortrac R 150/30-30 T	140.0	1.52	1.50	1.45	1.39	1.01	1.02	1.07	1.15	1.03	1.10	0.90
6	Fortrac R 200/30-30 T	200.0	1.52	1.50	1.45	1.39	1.01	1.02	1.07	1.15	1.03	1.10	0.90
7	Fortrac R 300/50-30 T	300.0	1.52	1.50	1.45	1.39	1.01	1.02	1.07	1.15	1.03	1.10	0.90
8	Fortrac R 400/50-30 T	400.0	1.52	1.50	1.45	1.39	1.01	1.02	1.07	1.15	1.03	1.10	0.90
9	Fortrac R 600/50-30 T	600.0	1.52	1.50	1.45	1.39	1.01	1.02	1.07	1.15	1.03	1.10	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 earth system/Enter geogrids manually" or "Reinforced earth system/Generate geogrids".

## 8.5 System menu

---

### 8.5.1 "Info" menu item

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

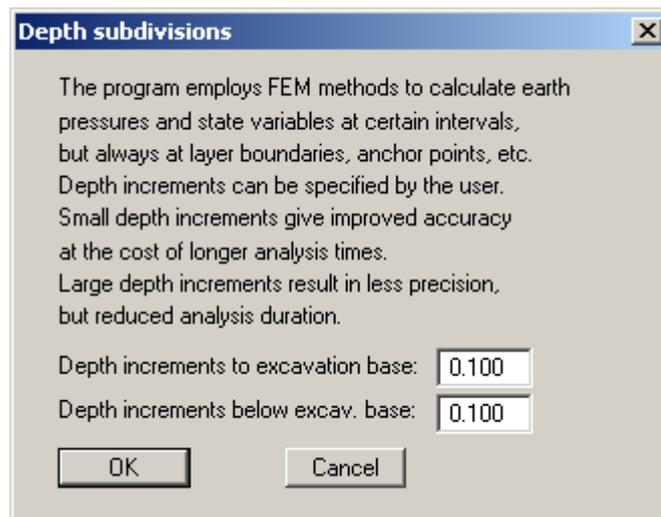
### 8.5.2 "Special preferences" menu item

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 excavation base.



## 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.

The screenshot shows a dialog box titled "Calculate wall" with a close button (X) in the top right corner. The dialog is organized into several sections:

- Type of toe pivot:** Three radio buttons: "Toe is fully fixed" (selected), "Toe freely pivoted", and "Base is embedded". Below this is a text box for "Subgrade reaction mod. ks [kN/m²]" containing the value "1.00E+4".
- Special preferences:** A group box containing five checkboxes: "Redistribute area loads" (unchecked), "Redistribute ep to toe base" (unchecked), "Passive ep in front" (checked), "Transition point with wp" (unchecked), and "2nd order theory" (unchecked).
- Type of ep redistribution:** A group box containing seven radio buttons: "Do not redistribute" (selected), "Redistribute after EAB", "Rectangular", "Birectangular", "Triangular", "Trapezoidal", and "User-defined". Each of the last five options has a "Preferences" button next to it.
- Bottom section:** Two checkboxes: "Active earth pressure takes wall inclination into consideration" (unchecked) and "Show warnings during analysis" (checked). At the very bottom are "OK" and "Cancel" buttons.

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.

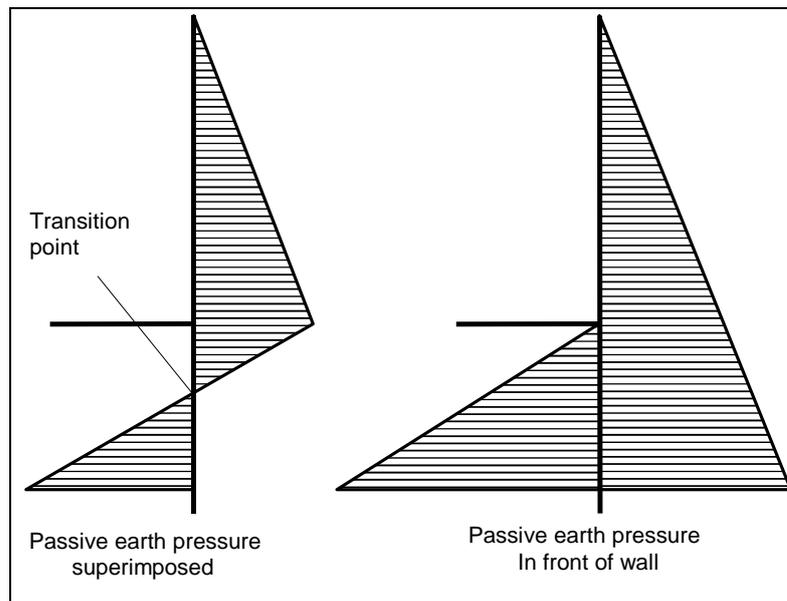


Figure 21 Passive earth pressure in front or superimposed

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.

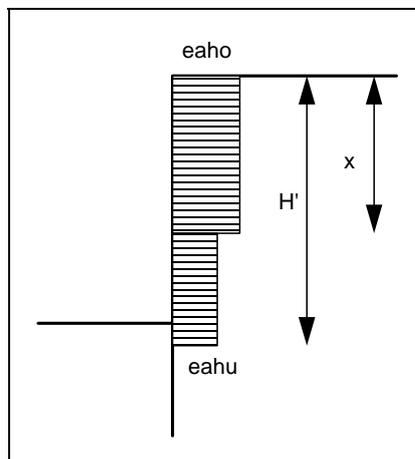
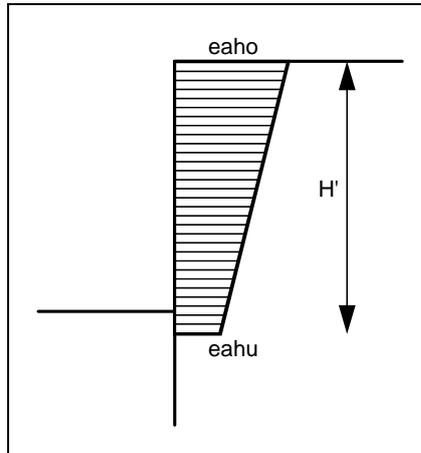


Figure 22 Birectangular earth pressure redistribution

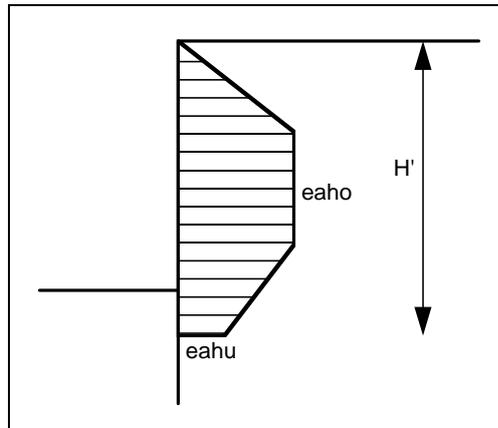
- **"Triangular"**  
Earth pressure is redistributed in the form of a triangle. The associated "Preferences" button enables you to determine the position of the maximum (top, central, bottom).

- **"Trapezoidal"**  
Earth pressure is redistributed in the form of a trapezoidal. The associated **"Preferences"** button enables you to determine the  $e_{ahu}/e_{aho}$  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  $e_{aho}/e_{ahu}$ .



*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.

No.	Depth [m]	eah' [-]
1	Wall top	1.0000
2	1.0000	3.0000
3	3.0000	3.0000
4	4.5000	6.0000
5	5.5000	6.0000
6	Load transition point	2.0000

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:

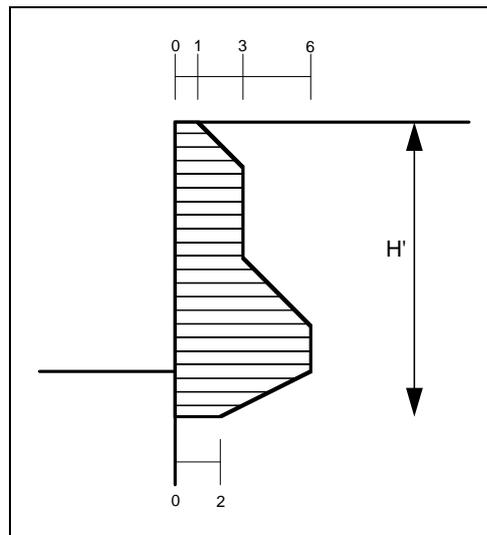
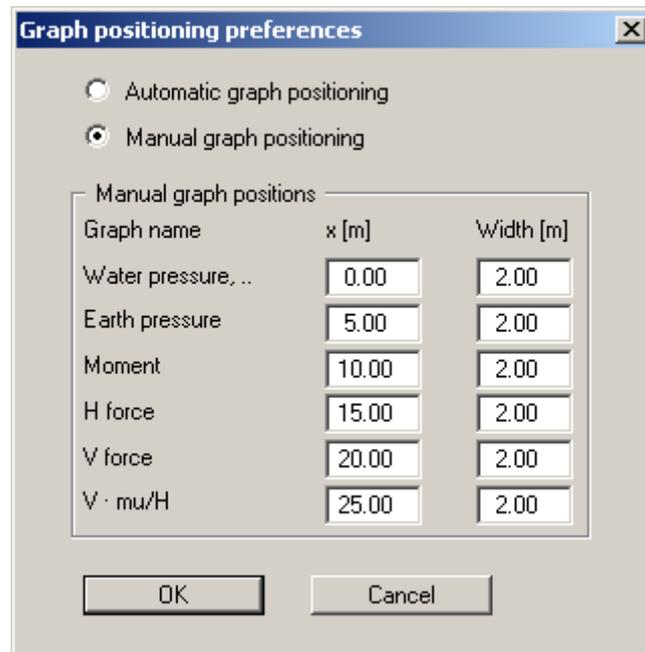


Figure 25 User-defined earth pressure redistribution

### 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.

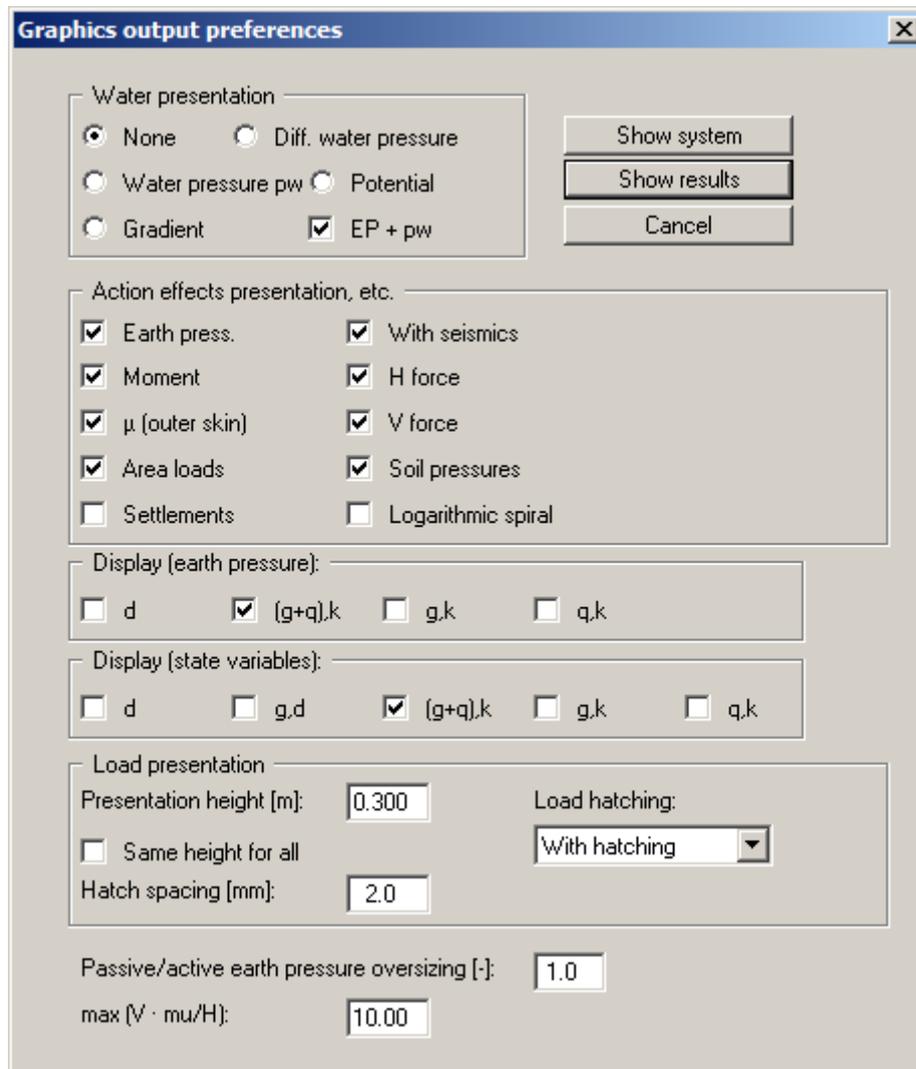


The graphs will then be shown at position "x" (central) with the specified "**Width**".

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.

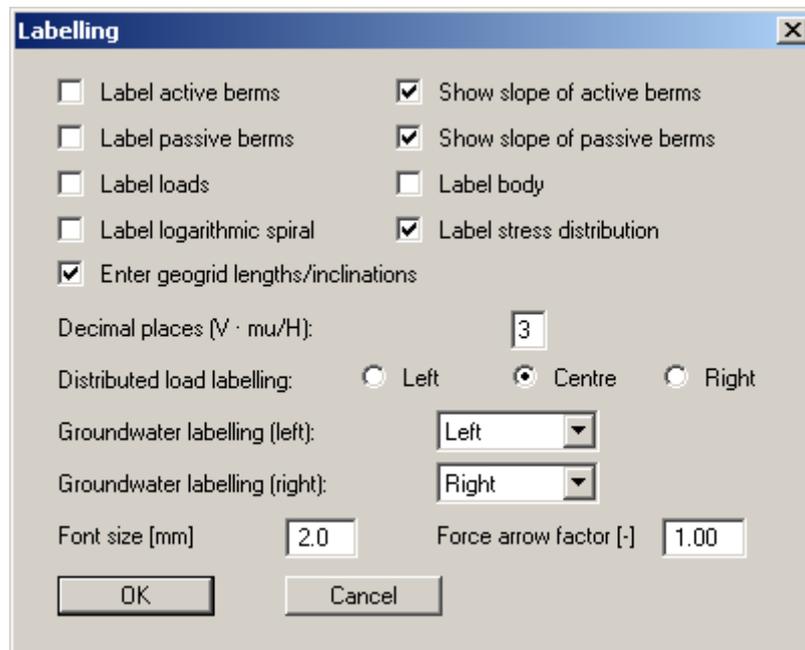
When analysing using *partial safety factors* you will also see the group boxes "Display (earth pressure):", "Display (state variables):" and "Display (displacement):", in which you can activate display of the permanent (g) and/or live loads (q). In addition, the design values (d) can be displayed.

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.

### "Labelling preferences" menu item

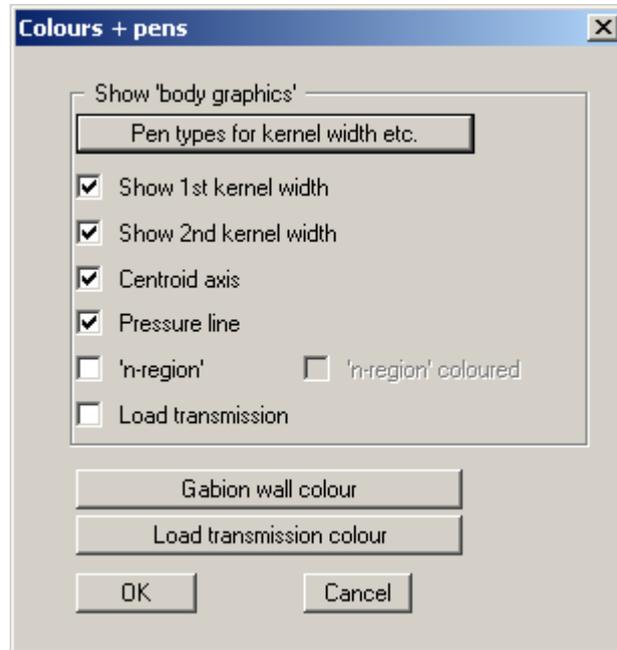
This menu item allows you to specify labelling preferences for the system visualisation and the result graphics.



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.

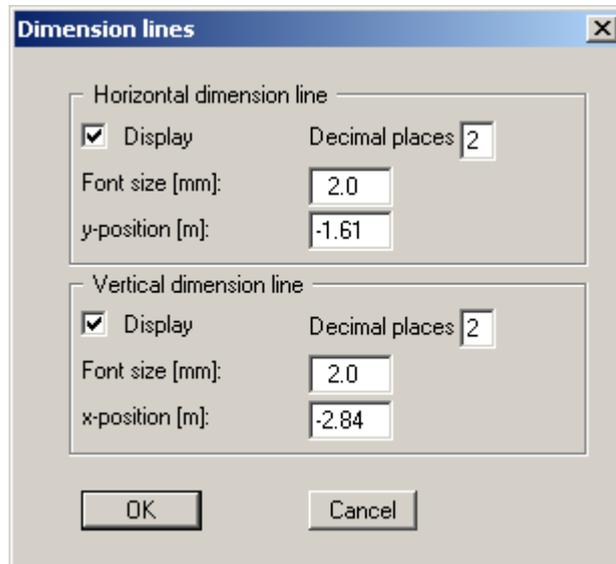
### 8.5.7 "Body pens + colours" menu item

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.



### 8.5.8 "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.

### 8.5.9 "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.

### 8.5.10 "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 Evaluation menu

---

### 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.

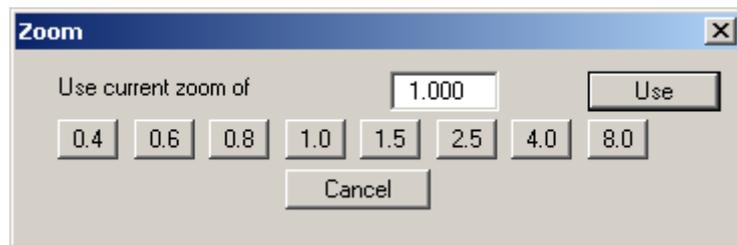
## 8.7 Graphics preferences menu

---

### 8.7.1 "Refresh and zoom" menu item

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

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



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

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

### 8.7.2 "Zoom info" menu item

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

### 8.7.3 "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 to the graphics and add lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.). The same pop-up menu opens for both menu items, the icons and functions used are described in more detail in the **Mini-CAD** manual provided. The differences between the Mini-CAD and Header CAD are as follows:

- Objects created with **Mini-CAD** are based on the coordinate system (generally in metres), in which the drawing is produced, and are shown accordingly. You should use the "**Mini-CAD toolbar**" when you wish to add information to the system (for example, labelling of slope inclinations or the location of any 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.

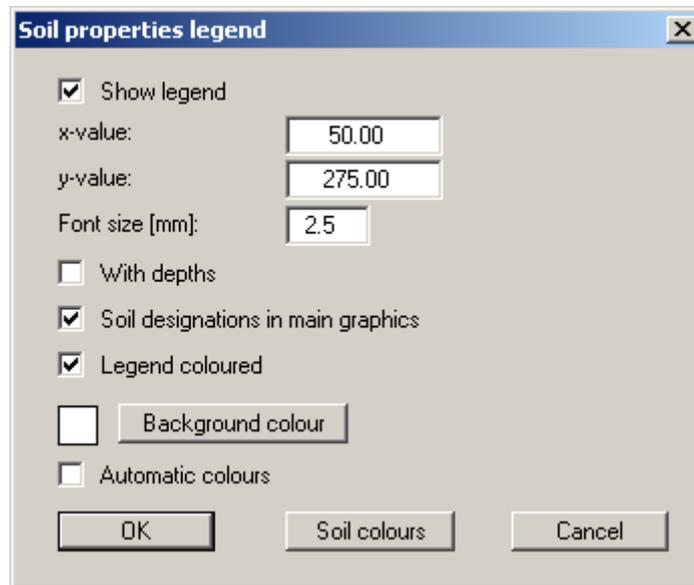


### "Restore "

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.

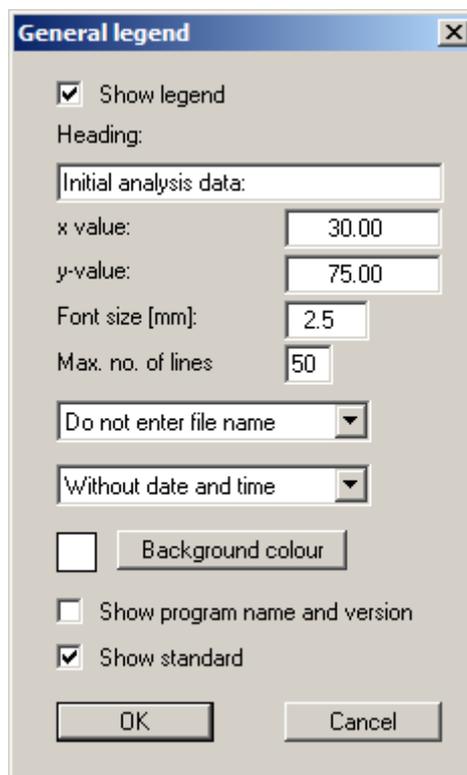
- "**With depths**"  
The depths of the individual soil layers will be shown in the soil properties legend.
- "**Soil designations in main graphics**"  
The designations entered for the individual ground layers are displayed in the system graphics adjacent to the layer depths.
- "**Legend coloured**"  
The soil colours are displayed in the legend. Otherwise, they will be numbered.
- "**Background colour**"  
The legend can be assigned a background colour.
- "**Automatic colours**"  
The soils are assigned soils colours automatically by the program. If the check box is not selected, the soil colours individually defined using the "**Soil colours**" button will be adopted.

- **"Soil colours"**

You will see a dialog box, in which you can define your preferences. After clicking the button with the desired number you can assign each soil layer a new number or reorganise using the **"Soil colours/Reorganise"** command button. You can save your colour preferences to a file with **"Soil colours/Save"** and use them for different systems by means of the **"Soil colours/Load"** command button. In the lower group box you can also transfer the colour preferences to the Windows colour management dialog box, or vice versa, as user-defined colour preferences for example. You can read a further description by pressing the **"Info"** button.

### 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.



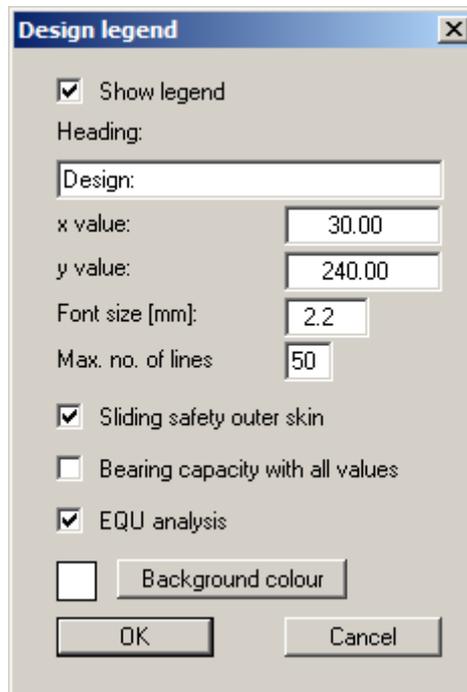
You can define and edit the position of the legend using the values **"x value"** and **"y value"**. You control the size of the legend using **"Font size"** and **"Max. no. of lines"**; where necessary, several columns are used.

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 (outer skin) 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 "Legend reinforced earth system" menu item

After you have analysed a system employing a reinforced earth system, 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 dialog box titled "Geogrid legend" contains the following elements:

- Show legend
- Heading: Geogrids
- x-value: 235.00
- y-value: 210.00
- Table width [mm]: 80.00
- Line height [mm]: 5.00
- Background colour
- Enter what additionally?
  - Length
  - Inclination
  - Corresp. FOS
  - RZ,d
- OK
- Cancel

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.

The screenshot shows a dialog box titled "Header". It contains two main sections: "Position, etc." and "Texts".

**Position, etc. section:**

- A checked checkbox labeled "Show legend".
- Input fields for "x [mm]:" (value: 247.00) and "y [mm]:" (value: 254.00).
- A "Background colour" button.

**Texts section:**

- Institution: Exemplary Design Consultancy
- Street address: Any Street 1
- City: 12345 Gritty City
- Telephone: Telephone: 0123-4567-89
- Telefax: Telefax: 0123-4567-90
- Project ID: eMail: info@john-doe.com
- Scientist: Scientist: John Doe

At the bottom of the dialog are "OK" and "Cancel" buttons.

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 Page size + margins menu

---

### 8.8.1 "Auto-resize" menu item

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

### 8.8.2 "Manual resize (mouse)" menu item

You can 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.

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

- Page in general:** Height = 297.00, Width = 420.00
- Page margin in mm:** Left = 25.00, Right = 8.00, Top = 8.00, Bottom = 8.00
- Plot margin in mm:** Left = 25.00, Right = 8.00, Top = 50.00, Bottom = 25.00

At the bottom, the "With borders" and "With margins" checkboxes are checked. The "OK" and "Cancel" buttons are also present.

- **"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 Info menu

---

### 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](http://www.ggu-software.com). Keep in touch with new program versions and the regular *download* offers.

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

### 8.9.6 "GGU support" menu item

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

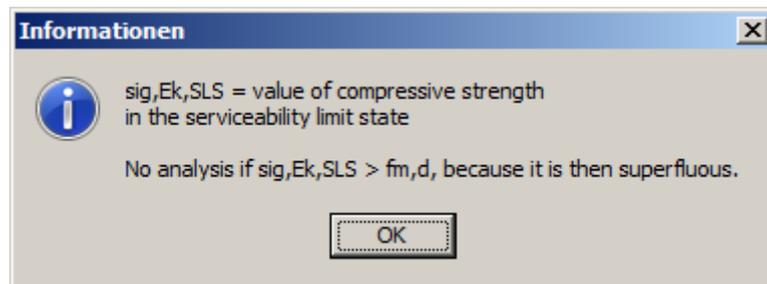
"?" buttons 

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:

sig,Ek,SLS [MN/m<sup>2</sup>):  

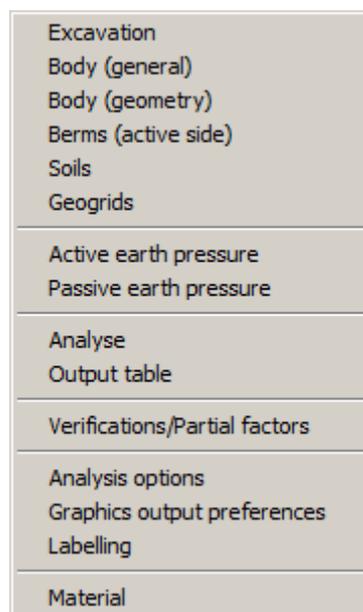
If you click on the question mark, you will see the following message box:



### 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

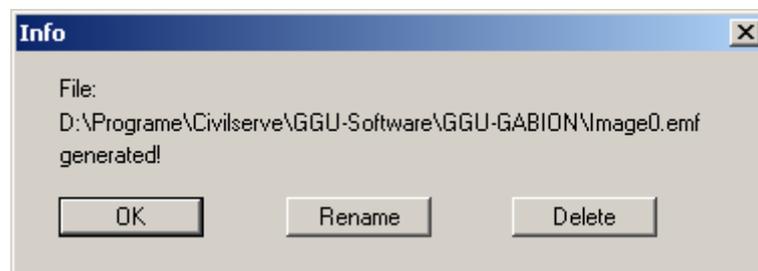
---

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

2<sup>nd</sup> order theory, select for analysis ..... 89

### A

Absolute heights, use ..... 55  
Action boundary conditions, define ..... 77  
Active earth pressure, analysis principles ..... 28  
Active earth pressure, edit preferences ..... 69  
Active earth pressure, select for analysis ..... 68  
Active wall friction angle, wind load  
  approach ..... 23  
Angle of wall friction, enter ..... 67  
Area loads, consider in analysis ..... 30  
Area loads, define ..... 75  
Area loads, include into earth pressure  
  redistribution ..... 89  
Area loads, to DIN 4085 ..... 31, 71  
At-rest earth pressure, coefficient to  
  DIN 4085 ..... 28  
At-rest earth pressure, keep constant ..... 71  
At-rest earth pressure, select for analysis ..... 68

### B

Base inclination coefficients ..... 47  
Base inclination, consider in analysis ..... 36  
Base inclination, define ..... 14  
Bearing capacity coefficients,  
  in bearing capacity failure analysis ..... 47  
Bearing capacity failure plane, determine ..... 48  
Bearing capacity safety, analysis principles ..... 47  
Bearing capacity safety, consider  
  base inclination ..... 36  
Bearing capacity safety, display analysis  
  results for reinf. earth in a message box ..... 86  
Bearing capacity safety, display analysis  
  results in a message box ..... 98  
Bearing capacity safety, passive ep to EC 7 ..... 67  
Berms, activate labelling ..... 95  
Berms, define on active side ..... 66  
Berms, define on passive side ..... 66  
Body, define ..... 27  
Bonding stress, define visualisation height ..... 79  
Bonding stress, in accordance with EBGeo ..... 43  
Bonding stress, select calculation method ..... 82  
Bonding stress, within slip body ..... 42  
Bounded surcharges, consider in analysis .. 32, 34  
Bounded surcharges, define ..... 75  
Buckling length, determination ..... 51

### C

Caquot/Kerisel, select for analysis ..... 70  
Centroid axis, activate/edit display ..... 96  
Centroid axis, consider acting forces ..... 35  
Characteristic point, display settlements ..... 98  
Characteristic point, limiting depth  
  determination ..... 78

Cholesky ..... 35  
Clipboard ..... 61  
CodeMeter stick ..... 9  
Cohesion coefficient, determination methods... 28  
Cohesion, enter ..... 67  
Colour, define for soils ..... 103  
Colour, switch on/off ..... 101  
Colour/hatching, define for loads ..... 94  
Colour/pens, define for gabion body ..... 96  
Colour/pens, define for graphical elements ..... 99  
Company data, add via Mini-CAD ..... 100  
Company data, add via quick header legend .. 106  
Concrete, activate verification ..... 18  
Concrete, enter values for structural analysis ... 17  
Constrained modulus, for settlement analysis .. 49  
Context menu, open ..... 112  
Coordinates, alter via editor ..... 108  
Coordinates, alter with mouse ..... 108  
Coordinates, optimise/reset ..... 108  
Coordinates, save/load ..... 108  
Copy/print area ..... 61, 101, 115  
Cutting borders, switch on/off ..... 109

### D

Danish standard ..... 47  
Dataset description, define ..... 54  
Dataset description, display ..... 103  
Delete, geogrid ..... 81  
Delete, soil layers ..... 67  
Depth coefficients, definition ..... 47  
Design compressive strength, define for  
  concrete ..... 18  
Design resistance of geogrid, define ..... 82  
Design resistance, according to EBGeo ..... 41  
Design, display results in legend ..... 104  
Dimension lines, define ..... 97  
Dimension lines, move with mouse ..... 106  
DIN 1045 ..... 50  
DIN 1055, activate static equilibrium  
  determination ..... 76  
DIN 4017 ..... 47  
DIN 4019 ..... 49  
DIN 4084 ..... 38  
DIN 4085, consider in analysis ..... 28  
DIN 4085, select for analysis ..... 69, 70  
DIN 4093 ..... 52  
DIN 4093, select for analysis ..... 54  
Displacement boundary conditions, define ..... 77  
Displacement method ..... 35  
Displacement, define for gabion wall ..... 77  
Distributed load, define as live load ..... 63  
Double-bounded surcharges, consider  
  in analysis ..... 33, 34  
Double-bounded surcharges, define ..... 76  
Drawing area, define ..... 109  
Duddeck/Ahrens ..... 35  
DXF file, export ..... 61  
DXF file, import ..... 7

## E

Earth pressure analysis, display results in a message box .....	86
Earth pressure coefficients, be calculated .....	111
Earth pressure coefficients, calculate .....	71
Earth pressure coefficients, user-defined .....	71
Earth pressure distribution, on outer skin .....	45
Earth pressure redistribution, display in a message box .....	98
Earth pressure redistribution, to EAB .....	90
Earth pressure redistribution, user-defined .....	92
Earth pressure reduction, considering inclined wall .....	44
Earth pressure reduction, determine possible factor .....	45
Earth pressure wedge, activate display .....	79
Earth pressure wedge, in earth pressure calculation .....	44
Earth pressure, calculation via earth pressure wedge .....	44
Earth pressure, edit preferences for active .....	69
Earth pressure, edit preferences for passive .....	70
Earth pressure, select for analysis .....	68
Earth quake, as design situation to EC 7 .....	73
EBGEO, earth pressure reduction to .....	45
EBGEO, Internal stability, reinforced earth system .....	41
EC 7, load cases description .....	73
Eccentricity, determine .....	44
Eccentricity, unintentional .....	50
Editor window, output table .....	59
Elastic curve, for inclined body .....	36
Embedment depth, footing .....	47
EMF format .....	61
EQU analysis, display results in a message box .....	98
EQU analysis, display results in legend .....	104
Equation solution to Cholesky .....	35
Equivalent earth pressure coefficient, activate use .....	69
Equivalent system for weight calculation .....	43
Excavation, change alignment .....	55
Excavation, define base .....	63

## F

Failure body, activate display .....	79
Failure system, internal stability verification ...	41
FGSV .....	7
File, display name in legend .....	103
File, load/save .....	55
Finite element module, for structural system ...	35
Font selection .....	99
Font size, design legend .....	104
Font size, dimension line .....	97
Font size, edit for graphical elements .....	108
Font size, general legend .....	103
Font size, geogrid legend .....	105
Font size, reinforced earth system legend .....	105
Font size, settlement legend .....	105
Font size, soil properties legend .....	102
Font size, system/result graphics .....	95
Footer, output table .....	57

Footing length, define .....	14
Foundation gap, explanation .....	21
Friction angle, enter .....	67
Friction coefficient, define .....	14
Friction coefficient, for bonding stress determination .....	43
Friction coefficient, for sliding safety verification .....	40
Function keys .....	114

## G

Gabion body, define colour/pens .....	96
Gabion body, define geometry .....	64
Gabion body, define rotation/displacement .....	77
Gabion body, generate automatically .....	15
General page informations, add via Mini-CAD .....	100
General stability, verification using file export .....	60
Geogrid forces, show maximum .....	85
Geogrids, activate selection via company products .....	55
Geogrids, add automatically .....	83
Geogrids, display values in legend .....	105
Geogrids, enter/delete .....	81
Geogrids, enter/edit all values .....	84
Geogrids, enter/edit individual values .....	82
Geogrids, manage company products .....	86
Geogrids, select as company product .....	81
GGUCAD file, export .....	61
GGUMiniCAD file, export .....	62
GGU-STABILITY .....	60
Global safety factors/verifications define .....	72
Graph positions, alter via numerical input .....	93
Graph positions, alter with mouse .....	106
Graphics, add via Mini-CAD .....	100
Gravity body, activate display .....	79
Ground inclination coefficients .....	47
Groundwater level, enter .....	63
Groundwater, activate labelling .....	95
Groundwater, define colour .....	99

## H

Hatching, define for loads .....	94
Header CAD, use .....	100
Header, output table .....	57
Horizontal force, in rod construction analysis ..	35
Horizontal loading, on earth body .....	43

## I

Inclination, active earth pressure wedge .....	43
Inclination, rear face of wall .....	43
Increased active earth pressure, coefficient .....	28
Increased active earth pressure, select for analysis .....	68
Installation .....	9
Internal stability, to DIN 1045 old .....	50
Internal stability, to DIN 4093 .....	52
Internal stability, to EC 2/DIN 1045-1 .....	51

## K

Kernel widths, activate/edit display.....	96
Kernel widths, explanation.....	21
Knowledge Base, access.....	111

## L

Labelling, system/result graphics.....	95
Lambda, reduction factor/friction coefficient...	82
Language preferences.....	111
Lateral pressures, define.....	76
Layout, output table.....	57
Legends, move with mouse.....	106
Licence protection.....	9
Limiting depth, defining methods.....	49
Limiting depth, select determination.....	78
Line loads, define.....	75
Line loads, perpendicular to wall axis.....	31
Live load, consider distributed load as.....	63
Load cases, adopt from ÖNORM EN 1997-1...	73
Load cases, adopt in accordance with DIN 1054-2010/EC 7.....	73
Load concentration factor, consider.....	31
Load concentration factor, define.....	71
Load distribution, for concrete verification.....	50
Load inclination coefficients.....	47
Load transition point, calculate with water pressure.....	89
Load transition point, using redistribution.....	91
Load transmission width, define.....	14
Load transmission, activate/edit display.....	96
Loads, edit graphical output preferences.....	94
Logarithmic spiral, activate display below reinforced earth system.....	79
Logarithmic spiral, display below gabion wall.....	94
Logarithmic spiral, in bearing capacity failure plane.....	48

## M

Manual, open as PDF file.....	111
Maximum values, display for system.....	111
Mean friction angle.....	48
Mean soil properties.....	48
Metafile, export.....	61
Mini-CAD file, export.....	62
Mini-CAD, use.....	100
Moment load, consider on rod axis.....	35
Moment, calculation in base plane.....	44
Mouse click functions.....	113
Mouse wheel functions.....	113

## N

Navigation, output table.....	100
Noise abatement wall, generate automatically..	24
n-region, activate/edit display.....	96
n-region, explanation.....	21
n-value, define for load distribution.....	18

## O

Objects, move with mouse.....	106
ÖNORM EN 1997-1, select partial factors.....	73
Output table, navigation.....	58
Output table, output as ASCII.....	59
Output table, output as graphics.....	57
Output table, select output format.....	56
Output table, switch to system graphics.....	100
Overburden stress, consider in settlement analysis.....	78
Overturning analysis, deactivate.....	55
Overturning safety, calculation.....	44
Overturning safety, display analysis results for reinforced earth system in a message box....	86

## P

Page format, define.....	109
Page margins, define/switch on/off.....	109
Page section, copy/print.....	101, 115
Pagination, automatic.....	58, 59
Partial factors, define.....	73
Partial safety factors, select from ÖNORM EN 1997-1.....	73
Passive earth pressure, calculation methods for coefficients.....	28
Passive earth pressure, for bearing capacity safety.....	67
Passive earth pressure, in front/superimposed..	89
Passive earth pressure, specify calculation.....	70
Pen preferences, edit for gabion body.....	96
Pen preferences, edit for graphical elements....	99
Piling Handbook, consideration of line loads...	31
Piling Handbook, effect on earth pressure.....	29
Plausibility check, show during analysis.....	88
Plausibility checks, view results.....	87
Plot margins, define.....	109
Preconsolidation, consider in settlement analysis.....	78
Presentation height, define for loads.....	94
Pressure line, activate/edit display.....	96
Pressure line, explanation.....	21
Print, graphics.....	60
Print, output table.....	61
Print, section.....	61, 101, 115
Print, several files.....	62
Printer preferences.....	59, 60
Program, display name in legend.....	103
Program, preferences.....	107
Program, save/load preferences.....	107
Program, show improvements.....	111
Program, show information.....	111
Project data, add via Mini-CAD.....	100
Project identification, define.....	54
Project identification, display.....	103
Pull-out resistance, maximum design value.....	42

## R

Redistribution figures, available.....	37
Redistribution figures, select.....	90
Reinforced earth system, display values in legend.....	105

Reinforcement force, maximum .....	41
Result graphics, display .....	97
Result graphics, edit preferences .....	94
Result graphics, label .....	95
Result graphics, position .....	93, 106
Results, display for selected depth .....	98, 113
Rod construction module .....	35
Rods, define for system .....	87
Rods, effect on analysis .....	35
Rotation, define for gabion wall .....	77

## S

Safety concept, select .....	54
Safety factors, define .....	72
Scale, alter with mouse .....	108
Scale, define via editor .....	108
Scale, determine automatically .....	108
Scroll the screen .....	113
Seismic loads, consider .....	74
Serviceability, systems with geogrids .....	39
Serviceability, systems without geogrids .....	39
Settlement analysis, display results in legend .....	104
Settlement analysis, edit preferences .....	78
Settlement analysis, limiting depth definition ..	49
Settlement analysis, subtraction preconsolidation loading .....	49
Settlements, display share in legend .....	104
Shape coefficients, for strip footings .....	47
Short-term strength .....	41
Slenderness, structural element .....	50
Sliding safety, consider base inclination .....	36
Sliding safety, display analysis results for reinforced earth system in a message box ...	86
Sliding safety, display analysis results in a message box .....	98
Sliding safety, in base plane .....	40
Sliding safety, in tier joint planes .....	40
Slip surfaces, generate/edit .....	85
Slip surfaces, internal stability verification .....	41
Slope stability application .....	60
Smarticons, for menu items .....	100
Soil colours/numbers, activate display .....	102
Soil designations, activate presentation in system graphics .....	102
Soil designations, define .....	67
Soil layers, adapt to absolute heights .....	63
Soil layers, define/delete .....	67
Soil layers, maximum number .....	27
Soil properties, activate separate input active/passive values .....	55
Soil properties, define .....	67
Soil properties, display in legend .....	102
Stability, systems with geogrids .....	39
Stability, systems without geogrids .....	38
Standard, display in legend .....	103
Static equilibrium to DIN 1055, activate determination .....	76
Static equilibrium to DIN 1055, display analysis results in a message box .....	98
Status bar main program, activate .....	100
Stiffness, wall .....	35, 38

Streck, select for analysis .....	70
Stress analysis, for limiting depth .....	78
Subgrade reaction modulus, for toe pivot .....	88
Support forces, reinforced earth system .....	43
System properties, display in legend .....	103
System, analyse .....	88
System, display .....	97
System, division into rods .....	87
System, edit graphics output preferences .....	94
System, show information .....	87, 111
System, structural .....	35

## T

Tension forces, in the geogrids .....	43
Theory of elastic half-space .....	31
Toe pivot, define .....	88
Toolbar, edit for menu items .....	100
Transition point, calculate with water pressure .....	89
Transition point, using redistribution .....	91
Translation, activate .....	111
True-type font .....	99
Two-part wedge failure mechanism, internal stability verification .....	41
Two-part wedge failure mechanism, slip surface generation .....	85

## U

Unit weight, enter .....	67
Utilisation factor, unreinforced concrete .....	51

## V

Verifications, activate for reinforced earth system .....	80
Verifications, activate to DIN 1054 old .....	72
Verifications, activate to DIN 1054-2010/ EC7 .....	73
Version number, display in a message box .....	111
Version number, display in legend .....	103
Vertical force, calculation in base plane .....	44
Vertical force, in rod construction analysis .....	35
Vertical section for ep determination .....	43

## W

Wall friction angle, wind loads approach .....	23
Wall toe, bedding conditions .....	36
Wall toe, select bedding conditions .....	88
Wall top, kept horizontal, consider in analysis .....	51
Weight analysis, display analysis results in a message box .....	86
What you see is what you get .....	99
Wind loads, active wall friction angle approach .....	23
Wind loads, consider as lateral pressures .....	76
Wind loads, consider for noise abatement wall .....	25

## **Y**

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

## **Z**

Zoom factor, define for full-screen display ..... 99  
Zoom function, activate..... 99, 100, 113