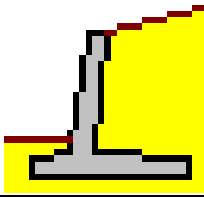


Analysis of cantilever walls

GGU-CANTILEVER

VERSION 3



Last revision: September 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	10
4 Language selection.....	10
5 Starting the program	11
6 First steps using a worked example.....	12
6.1 Example system.....	12
6.2 Step 1: Select analysis options	13
6.3 Step 2: Define the excavation.....	14
6.4 Step 3: Define load on cantilever wall	15
6.5 Step 4: Define geometry of cantilever wall.....	16
6.6 Step 5: Define soils	17
6.7 Step 6: Define type of earth pressure	17
6.8 Step 7: Analyse and design the system	18
6.9 Step 8: Evaluate and visualise the results.....	19
7 Theoretical principles	20
7.1 Cantilever wall	20
7.1.1 Definition.....	20
7.1.2 Determination of wall loads for reinforced concrete design.....	21
7.1.3 Analysis of stability of overall system.....	22
7.2 Soil properties	25
7.3 Active earth pressure.....	25
7.4 At-rest earth pressure	25
7.5 Increased active earth pressure.....	26
7.6 Passive earth pressure	26
7.7 Water pressure.....	27
7.7.1 Classical water pressure approach	27
7.7.2 Water pressure approach using flow conduits	28
7.7.3 Recommendation for water pressure approach.....	29
7.7.4 What is a potential?	29
7.8 Berms	31
7.9 Area loads.....	32
7.10 Line loads	33
7.11 Bounded surcharges (active side).....	34
7.12 Double-bounded surcharges (active side)	35
7.13 Bounded surcharges (passive side)	36
7.14 Double-bounded surcharges (passive side).....	36
7.15 Structural system.....	37
7.16 Design	38
7.17 Action and displacement boundary conditions	38
7.18 Prestressing	39
7.19 Earth pressure redistribution	39
7.20 Overturning stability	39

7.21 Bearing capacity safety	40
7.22 Sliding safety.....	41
7.23 General stability	42
7.24 Settlements	42
7.25 Hydraulic heave	42
7.25.1 Hydraulic heave safety using global safety factors.....	42
7.25.2 Utilisation factor (hydraulic heave) using partial safety factors	43
7.26 Buoyancy.....	43
7.26.1 Buoyancy safety using global safety factors.....	43
7.26.2 Utilisation factor (buoyancy) using partial safety factors.....	44
7.27 Verification of deep-seated stability.....	45
7.28 Heave of anchor soil.....	46
7.29 Reinforced concrete design	46
8 Description of menu items.....	47
8.1 File menu.....	47
8.1.1 "New" menu item.....	47
8.1.2 "Load" menu item.....	48
8.1.3 "Save" menu item	48
8.1.4 "Save as" menu item	48
8.1.5 "Print output table" menu item.....	49
8.1.5.1 Selecting the output format	49
8.1.5.2 Button "Output as graphics"	50
8.1.5.3 Button "Output as ASCII"	52
8.1.6 "Export" menu item	53
8.1.7 "Printer preferences" menu item.....	53
8.1.8 "Print and export" menu item	53
8.1.9 "Batch print" menu item	55
8.1.10 "Exit" menu item.....	55
8.1.11 "1, 2, 3, 4" menu items.....	55
8.2 Editor 1 menu.....	56
8.2.1 "Analysis options" menu item.....	56
8.2.2 "Excavation" menu item	56
8.2.3 "Wall (general)" menu item	57
8.2.4 "Wall (geometry)" menu item.....	57
8.2.5 "Berms (active side)" menu item	57
8.2.6 "Berms (passive side)" menu item	57
8.2.7 "Soils" menu item	58
8.2.8 "Type of earth pressure" menu item	59
8.2.9 "Active earth pressure" menu item	60
8.2.10 "Passive earth pressure" menu item.....	61
8.2.11 "At-rest earth pressure" menu item.....	62
8.2.12 "User-defined earth pressure coefficients" menu item.....	62
8.2.13 "Groundwater" menu item	63
8.2.14 "Verifications/Safety factors" menu item	64
8.2.15 "Verifications/Partial factors" menu item	65
8.2.16 "Buoyancy + Hydraulic heave" menu item.....	66

8.2.17	"Seismic acceleration" menu item	67
8.2.18	"Deep-seated stability/Heave of anchor soil" menu item.....	68
8.3	Editor 2 menu	69
8.3.1	"Lateral pressures" menu item	69
8.3.2	"Area and line loads" menu item	69
8.3.3	"Compaction earth pressure" menu item.....	70
8.3.4	"Bounded surcharges" menu item.....	71
8.3.5	"Double-bounded surcharges" menu item	72
8.3.6	"Action boundary conditions" menu item.....	73
8.3.7	"Displacement boundary conditions" menu item.....	73
8.3.8	"Anchors" menu item.....	74
8.3.9	"Struts" menu item.....	75
8.3.10	"Prestressing" menu item.....	76
8.3.11	"Groundwater potentials" menu item.....	76
8.3.12	"Settlements" menu item.....	77
8.3.13	"Predeformation info" menu item	78
8.3.14	"Predeformation preferences" menu item	78
8.3.15	"Anchor steel design" menu item.....	80
8.4	System menu	81
8.4.1	"Info" menu item	81
8.4.2	"Special preferences" menu item	81
8.4.3	"Depth subdivisions" menu item	81
8.4.4	"Analyse" menu item	82
8.4.4.1	Start dialog box	82
8.4.4.2	"Type of toe pivot" group box.....	83
8.4.4.3	"Type of redistribution" group box	83
8.4.4.4	"Special preferences" group box	85
8.4.5	"Optimise" menu item.....	86
8.4.6	"Design defaults" menu item	87
8.4.7	"Graph positioning preferences" menu item.....	88
8.4.8	"Graphics output preferences" menu item	89
8.4.9	"Labelling preferences" menu item.....	90
8.4.10	"Wall pens + colours" menu item	91
8.4.11	"Dimension lines" menu item	91
8.4.12	"Display system" menu item.....	92
8.4.13	"Display results" menu item	92
8.5	Evaluation menu.....	93
8.5.1	General notes	93
8.5.2	"Earth pressure redistribution" menu item.....	93
8.5.3	"Main output summary" menu item.....	93
8.5.4	"Interim results" menu item	93
8.5.5	"Bearing capacity" menu item	93
8.5.6	"Sliding safety" menu item	93
8.5.7	"Settlements" menu item.....	93
8.5.8	"Deep-seated stability summary" menu item.....	94
8.5.9	"Hydraulic heave FOS summary" menu item.....	95

8.5.10	"Buoyancy FOS summary" menu item	95
8.5.11	"Anchor and strut summary" menu item	95
8.5.12	"Heave of anchor soil" menu item	95
8.5.13	"EQU analysis" menu item	95
8.6	Graphics preferences menu	96
8.6.1	"Refresh and zoom" menu item	96
8.6.2	"Zoom info" menu item	96
8.6.3	"Legend font selection" menu item	96
8.6.4	"Pen colour and width" menu item	97
8.6.5	"Mini-CAD toolbar" and "Header toolbar" menu items	97
8.6.6	"Toolbar preferences" menu item	98
8.6.7	"Soil properties legend" menu item	99
8.6.8	"General legend" menu item	100
8.6.9	"Design legend" menu item	101
8.6.10	"Settlement legend" menu item	102
8.6.11	"Move objects" menu item	102
8.6.12	"Save graphics preferences" menu item	102
8.6.13	"Load graphics preferences" menu item	102
8.7	Page size + margins menu	103
8.7.1	"Auto-resize" menu item	103
8.7.2	"Manual resize (mouse)" menu item	103
8.7.3	"Manual resize (editor)" menu item	103
8.7.4	"Zoom" menu item	103
8.7.5	"Font size selection" menu item	103
8.7.6	"Page size and margins" menu item	104
8.7.7	"Undo" menu item	105
8.7.8	"Restore" menu item	105
8.7.9	"Preferences" menu item	105
8.8	Info menu	106
8.8.1	"Copyright" menu item	106
8.8.2	"Maxima" menu item	106
8.8.3	"Compare earth pressure coefficients" menu item	106
8.8.4	"Help" menu item	106
8.8.5	"GGU on the web" menu item	106
8.8.6	"GGU support" menu item	106
8.8.7	"What's new?" menu item	106
8.8.8	"Language preferences" menu item	106
9	Tips and tricks	107
9.1	"?" buttons	107
9.2	Keyboard and mouse	108
9.3	Function keys	109
9.4	"Copy/print area" icon	110
10	Index	111

List of figures:

<i>Figure 1 Illustration of the worked example</i>	12
<i>Figure 2 Base of wall (stem) Example 1</i>	21
<i>Figure 3 Base of wall (stem) Example 2</i>	21
<i>Figure 4 Equivalent wall inclination</i>	22
<i>Figure 5 Determination of angle β</i>	23
<i>Figure 6 Classical water pressure approach</i>	27
<i>Figure 7 Resultant water pressure</i>	27
<i>Figure 8 Water pressure approach using flow conduits</i>	28
<i>Figure 9 Resultant water pressure with flow conduits</i>	28
<i>Figure 10 Definition of potential</i>	30
<i>Figure 11 Berm on the active side</i>	31
<i>Figure 12 Area load</i>	32
<i>Figure 13 At-rest earth pressure from area loads</i>	33
<i>Figure 14 Bounded surcharge (active side)</i>	34
<i>Figure 15 Two bounded surcharges</i>	35
<i>Figure 16 Double-bounded surcharge</i>	35
<i>Figure 17 Bounded surcharge (passive side)</i>	36
<i>Figure 18 Logarithmic spiral</i>	40
<i>Figure 19 Horizontal equivalent plane</i>	41
<i>Figure 20 Compound "deep slip planes"</i>	45
<i>Figure 21 Compound "deep slip plane", which is not investigated</i>	45
<i>Figure 22 Birectangular earth pressure redistribution</i>	83
<i>Figure 23 Earth pressure redistribution in a trapezoidal</i>	84
<i>Figure 24 Earth pressure redistribution in a quadrilateral</i>	84
<i>Figure 25 User defined earth pressure redistribution</i>	85

1 Preface

The **GGU-CANTILEVER** program system allows the analysis of cantilever walls.

For analysis and design both the *global safety factors* to DIN 1054 (old) and the *partial safety factors* to EC 7 may be taken into consideration. Details of special features can be found in Section 2 of this manual.

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

GGU-CANTILEVER has the following characteristics and maximum capabilities:

- Almost any user-defined cantilever wall possible
- Base inclination of cantilever wall
- Up to 50 soil layers
- Up to 10 berms on the active earth pressure side
- Up to 10 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 to DIN 4085
- Coefficients of passive earth pressure to DIN 4085, Streck, Caquot/Kerisel
- Classical water pressure approach and, alternatively, by means of a flow conduit parallel to the wall in accordance with potential theory (analysed using finite-element methods). Inconsistencies in subsurface hydraulics apparent when using the classical approach are avoided when using flow conduits, and varying permeabilities are correctly considered as far as the approach allows. User-defined potentials can be applied at any point along the flow conduit. This allows correct, problem-free consideration of systems with several groundwater storeys and/or aquifuges, or with artesian conditions.

- Optional consideration of hydraulic gradients on the active and the passive earth pressure sides
- Consideration of prestressing of anchors and struts possible
- Analysis of safety factors against deep-seated failure with optimisation of anchor lengths
- Analysis of stability against overturning
- Analysis of bearing capacity safety factor
- Analysis of settlements
- Analysis of sliding safety factor
- Analysis of safety against hydraulic heave
- Analysis of safety against buoyancy
- Reinforced concrete design
- Convenient interface to the stability analysis application, **GGU-STABILITY**, for quick determination of safety factor against general failure
- Analysis of static equilibrium in the EQU limit state
- 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. In contrast to many other applications, the influence of inclined anchors or struts and their interactions are thus directly considered in the analysis approach. Analysis can even be performed using 2nd order theory, making the generally onerous buckling length investigation of struts and wall unnecessary. In addition, it is possible to consider loads on the struts (e.g. for auxiliary bridges additionally acting as struts) during analysis.
- 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
- 20 anchor and strut sets. In defining the anchors and struts, axial stiffness and bending stiffness can be specified, so that, for example, passive *soft* anchors can also be incorporated.
- Consideration of prestressing/pretensioning of anchors and struts
- Definition of up to 20 additional potentials for the analysis of flow conduits to the left and right of the wall for complex groundwater conditions
- Automatic computation of earth pressure redistribution in accordance with EAB recommendations
- Other earth pressure redistribution options:
 - No redistribution
 - Rectangle
 - 2 Rectangles
 - Triangle (maximum can be optionally placed at top, middle or bottom)
 - Trapezoid
 - Quadrilateral with maximum at anchor locations or any other location.
 - User-defined polygon
- Following computation of the system the earth pressure, pore water pressure, moment, shear, normal force and bending line are displayed on the screen. The visualisation can be varied within wide limits. For example, the distributions of the potential and gradient, etc., can also be displayed.

- At anchor points, predeformations that are already present can be defined as boundary conditions.
- Legends can be displayed on screen indicating soil characteristics, general computation data, data needed for settlement analysis and the main design data. Thus, virtually all the raw data necessary for the computation are shown on screen.
- **GGU-CANTILEVER'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 insert 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-CANTILEVER** 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-CANTILEVER** 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-CANTILEVER 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). You then see eight menus in the menu bar:

- File
- Editor 1
- Editor 2
- 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-CANTILEVER** 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 using a worked example

6.1 Example system

Knowing from experience that having to work one's way through a software manual can be very tiresome, the following sections provides a short description of **GGU-CANTILEVER**'s main functions, which will quickly enable you to carry out a cantilever wall analysis. Details, when needed, will be found in the appropriate chapter of this manual.

The following cantilever wall will be analysed:

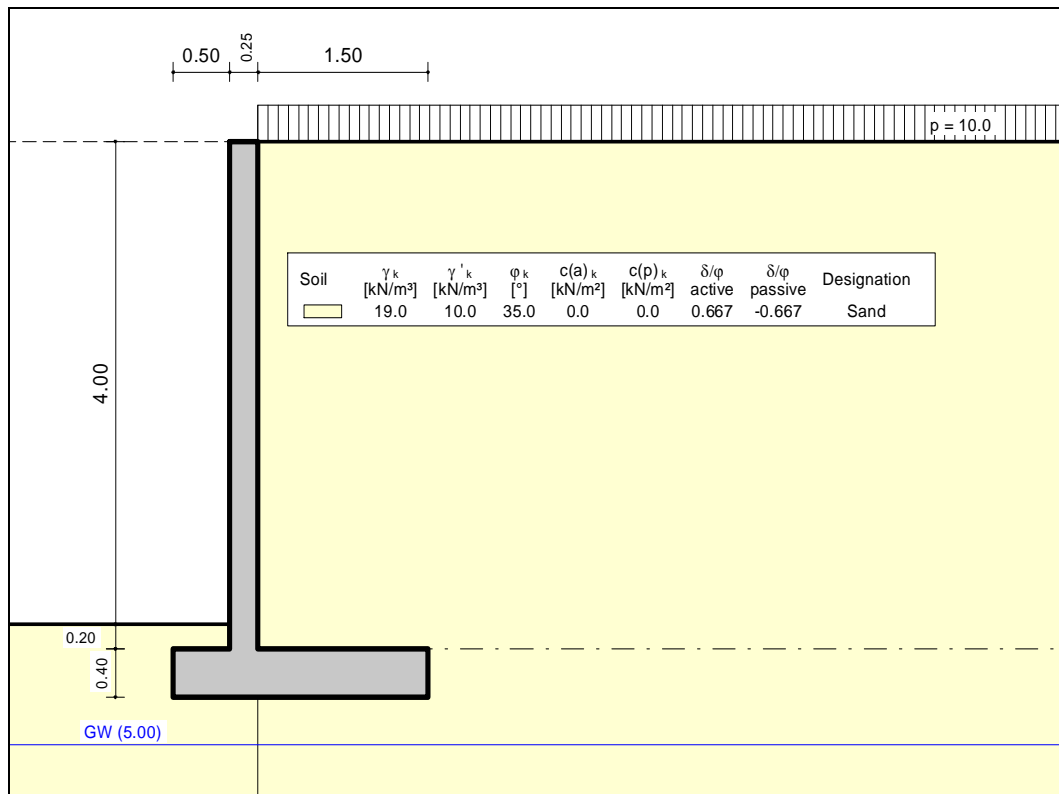
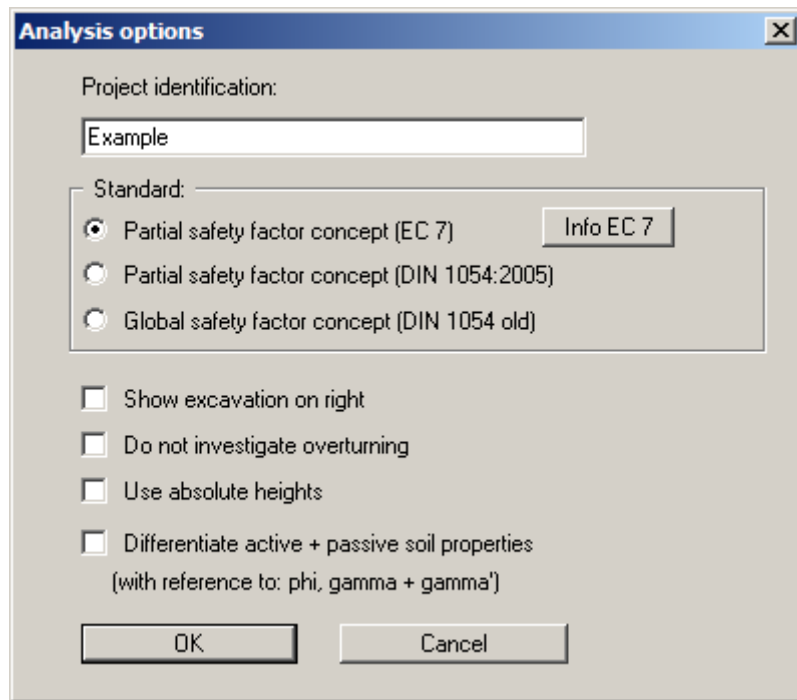


Figure 1 Illustration of the worked example

6.2 Step 1: Select analysis options

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

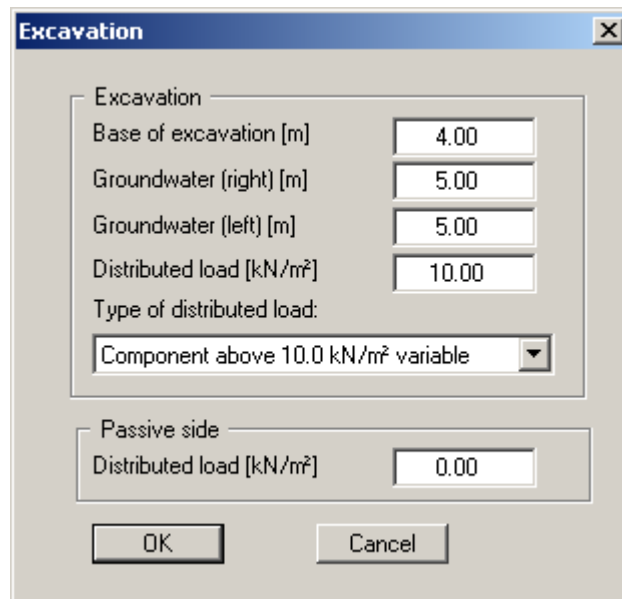


If you select the "**Use absolute heights**" check box you can enter all depths or heights in **m AD** (heights are positive upwards). If you leave this box unselected, the top of the wall is assumed at **0.0** (height/depth) and all input of layer depths etc. is positive downwards. If, however, you 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.

Select the buttons shown in the above dialog box and click "**OK**". A new system is displayed on the screen and the complete menu bar is activated.

6.3 Step 2: Define the excavation

Go to "Excavation" in the "Editor 1" menu. The following dialog box will appear. Enter the figures shown below:



The image shows a software dialog box titled "Excavation". It contains two main sections: "Excavation" and "Passive side".

Excavation section:

- Base of excavation [m]: 4.00
- Groundwater (right) [m]: 5.00
- Groundwater (left) [m]: 5.00
- Distributed load [kN/m²]: 10.00
- Type of distributed load: Component above 10.0 kN/m² variable (selected from a dropdown menu)

Passive side section:

- Distributed load [kN/m²]: 0.00

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

6.4 Step 3: Define load on cantilever wall

From the "Editor 1" menu select "Wall (general)".

	Permanent(g)	Changeable(q)
Vertical load [kN/m]:	0.0	0.0
x [m]:	-0.125	
Horizontal load [kN/m]:	0.0	0.0
Moment [kN·m/m]:	0.0	0.0
Toe surcharge p [kN/m ²]:	0.00	
Width c [m]:	0.100	
Footing length [m]: (for bearing capacity + settlement)	10.00	

Concrete parameters

gamma [kN/m³]: 25.000

Young's mod. [kN/m²]: 3.0000E+7

In the dialog box, define the size of any existing "Vertical load" or "Horizontal load" acting on the top of the wall. When using the *partial safety factor concept*, the loads are entered separately as "Permanent(g)" and "Changeable(q)" loads. Define the location of the vertical load from the top right edge of the cantilever wall using "x(load)". It is also possible to define a moment acting on the wall and an additional load on the toe. Input of a "Footing length" influences bearing capacity failure and settlement analyses only.

The concrete parameters are entered in the concrete parameters group box at the bottom of the dialog box. "Young's mod." designates the Young's modulus of the concrete. This value is required for the structural analysis of the cantilever wall.

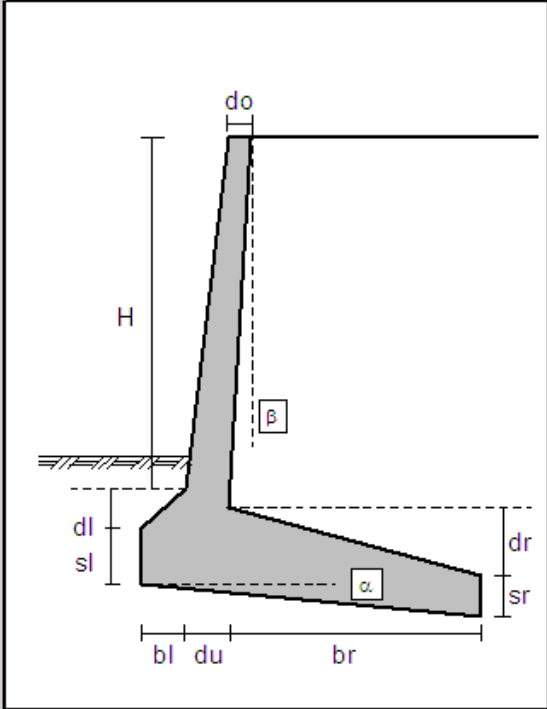
6.5 Step 4: Define geometry of cantilever wall

From the "Editor 1" menu select "Wall (geometry)".

Generate cantilever wall

Wall	
Height H [m]	4.200
Thickness (top) d_o [m]	0.250
Thickness (bottom) d_u [m]	0.250
Wall inclination beta [°]	0.000
Base inclination alpha [°]	0.000
Toe	
Toe width b_l [m]	0.500
Toe thickness s_l [m]	0.400
Depth of toe below stem d_l [m]	0.000
Heel	
Heel width b_r [m]	1.500
Heel thickness s_r [m]	0.400
Depth of heel below stem d_r [m]	0.000

OK Cancel

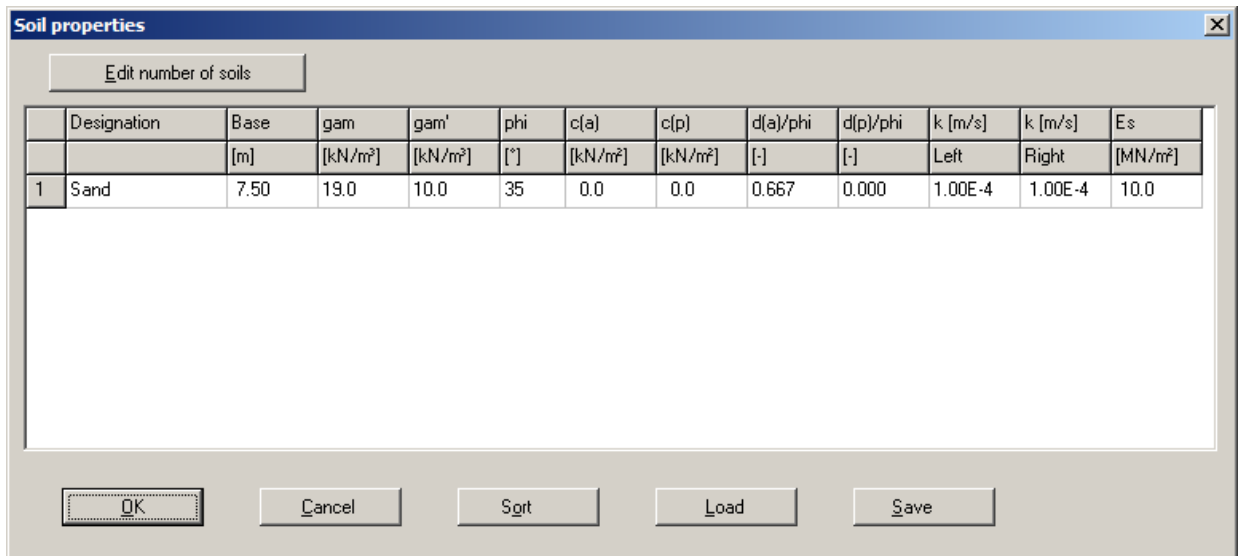


The diagram shows a cross-section of a cantilever wall. The wall has a height H . The top thickness is d_o and the bottom thickness is d_u . The wall is inclined at an angle β from the vertical. The base is inclined at an angle α from the horizontal. The toe has a width b_l , thickness s_l , and depth d_l below the stem. The heel has a width b_r , thickness s_r , and depth d_r below the stem.

The graphics visualise all necessary input, so no further descriptions are necessary here.

6.6 Step 5: Define soils

Go to "**Editor 1**" menu and select "**Soils**". Change the number of soils by clicking "**Edit number of soils**" and enter **1**.



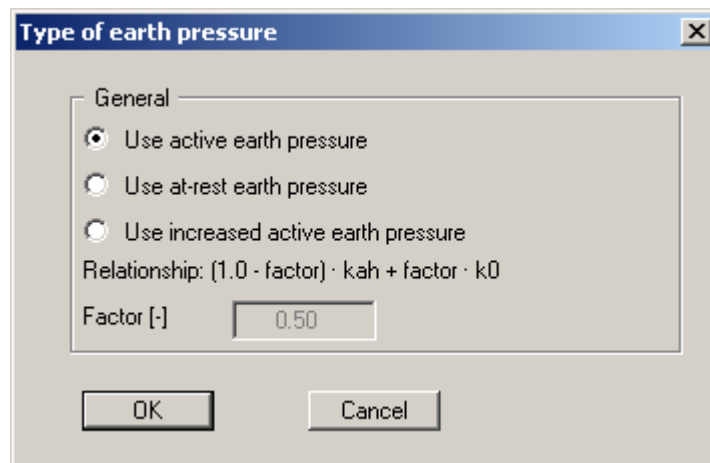
The dialog box titled "Soil properties" contains a button "Edit number of soils" at the top. Below it is a table with 13 columns: Designation, Base, gam, gam', phi, c(a), c(p), d(a)/phi, d(p)/phi, k [m/s] (Left), k [m/s] (Right), and Es. The first row of data is for soil 1, designated as "Sand", with a base of 7.50, gam of 19.0, gam' of 10.0, phi of 35, c(a) of 0.0, c(p) of 0.0, d(a)/phi of 0.667, d(p)/phi of 0.000, and Es of 10.0. At the bottom of the dialog are buttons for "OK", "Cancel", "Sgnt", "Load", and "Save".

	Designation	Base	gam	gam'	phi	c(a)	c(p)	d(a)/phi	d(p)/phi	k [m/s]	k [m/s]	Es
		[m]	[kN/m ²]	[kN/m ²]	[°]	[kN/m ²]	[kN/m ²]	[-]	[-]	Left	Right	[MN/m ²]
1	Sand	7.50	19.0	10.0	35	0.0	0.0	0.667	0.000	1.00E-4	1.00E-4	10.0

Enter the values given in the above dialog box. You can enter differing cohesion and friction angles for the active (a) and the passive sides (p) (see Section 8.2.7).

6.7 Step 6: Define type of earth pressure

Go to the "**Editor 1**" menu and select "**Type of earth pressure**".



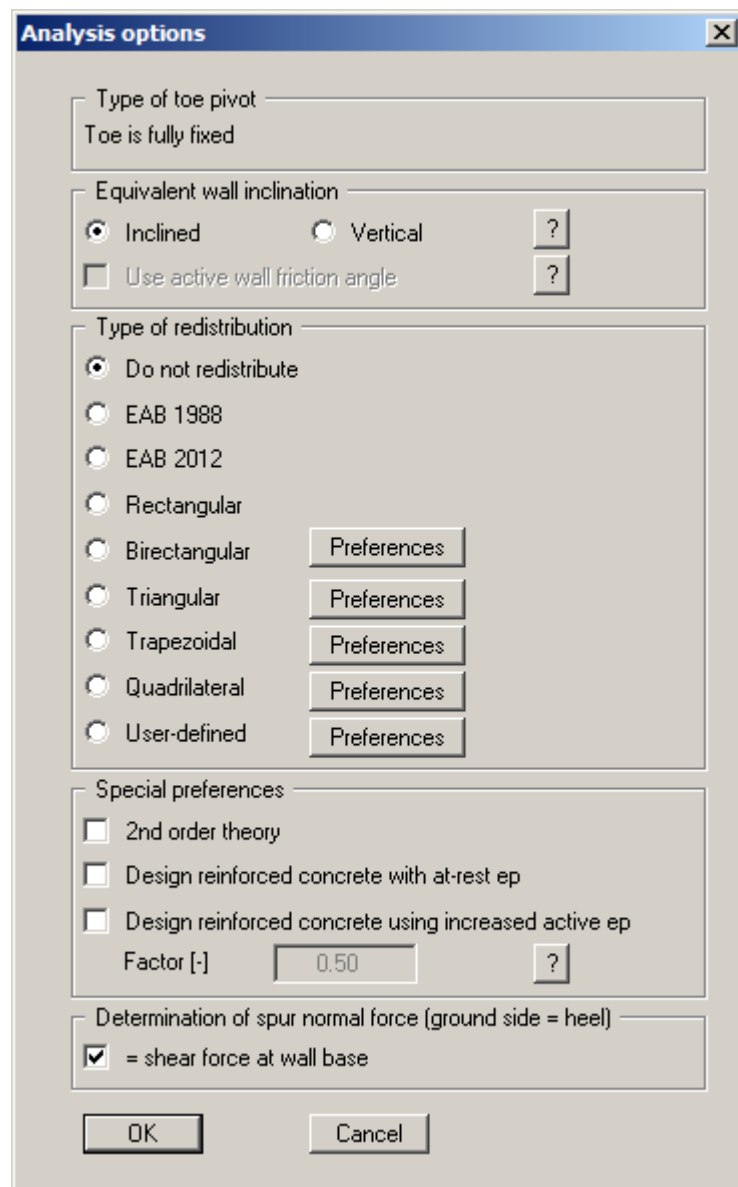
The dialog box titled "Type of earth pressure" has a "General" section with three radio button options: "Use active earth pressure" (selected), "Use at-rest earth pressure", and "Use increased active earth pressure". Below these is the text "Relationship: (1.0 - factor) · kah + factor · k0" and a text input field for "Factor [-]" containing the value "0.50". At the bottom are "OK" and "Cancel" buttons.

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.

6.8 Step 7: Analyse and design the system

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



Anchors or struts are not included in the example. It is therefore not necessary to use earth pressure redistribution and is rejected by the program.

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 design can follow; accept the proposed setting of the "System/Design defaults" dialog box. The design phase is more closely described in Section 8.4.6. Following completion of design, the results are presented in message boxes and then visualised graphically on the screen.

6.9 Step 8: Evaluate and visualise the results

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

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.

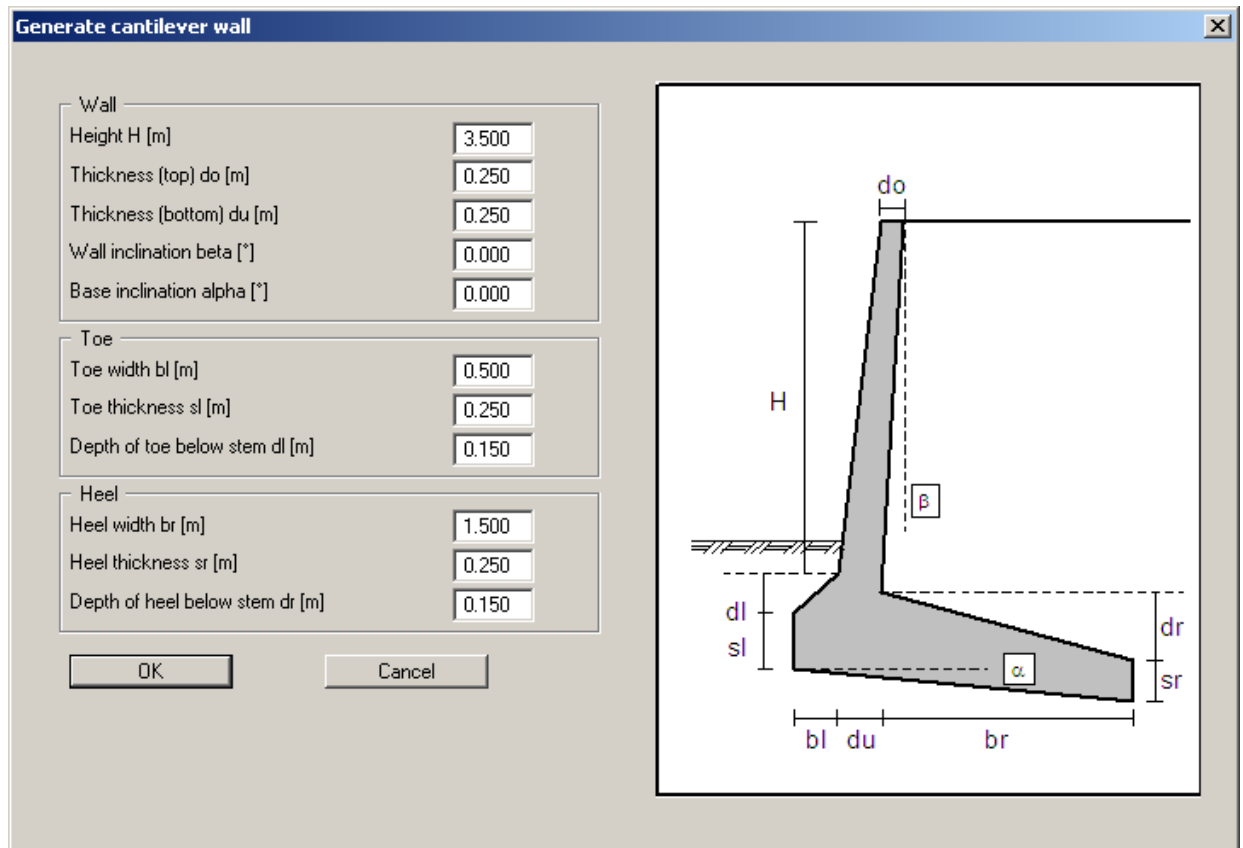
To further clarify the analysis results the graphics can be supplemented with text and graphic elements using the "**Mini-CAD**" module. You can save your work to a file. Go to the "**Save as**" menu item in the "**File**" menu.

7 Theoretical principles

7.1 Cantilever wall

7.1.1 Definition

The cantilever wall is defined via the menu item "Editor 1/Wall (geometry)":



The graphics visualise all necessary input, so no further descriptions are necessary here.

7.1.2 Determination of wall loads for reinforced concrete design

The program first analyses the loads on the wall above the footing. The base of the wall (stem) is marked by a dashed line. This is shown on two examples below:

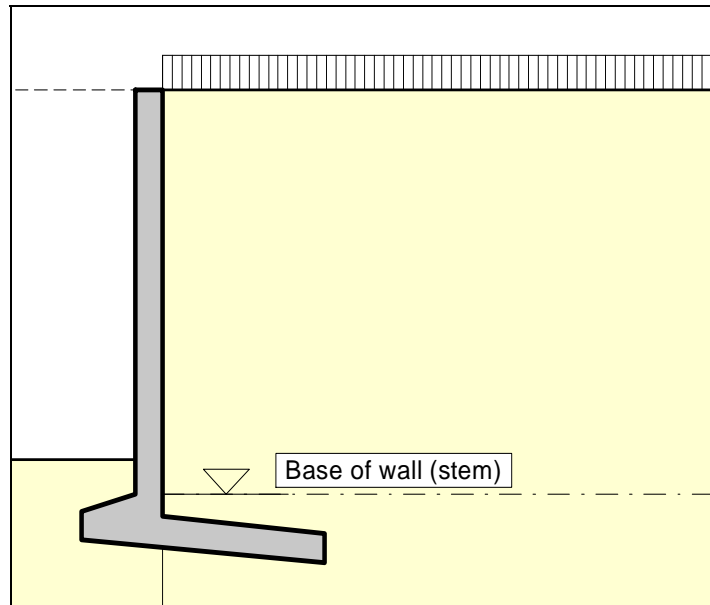


Figure 2 Base of wall (stem) Example 1

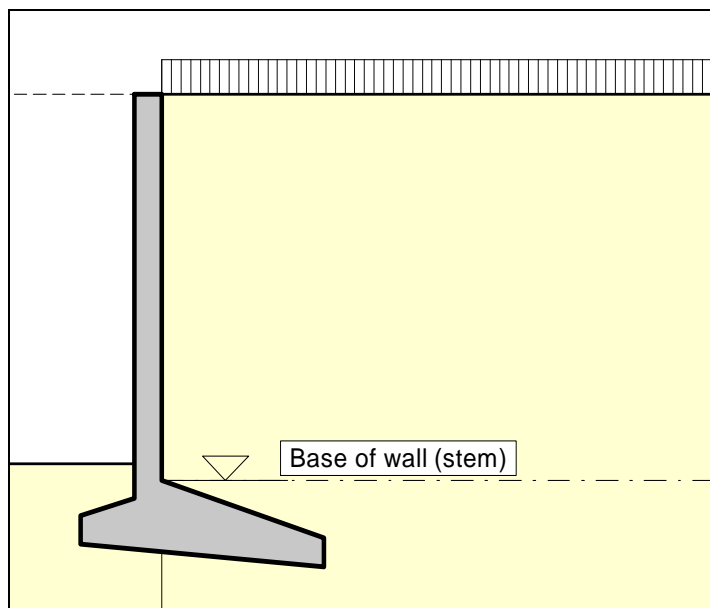


Figure 3 Base of wall (stem) Example 2

Earth pressure is calculated to the stem as shown in the figures. The result is the load on the wall above the footing zone. The wall is then designed to DIN 1045 (reinforced concrete design) using this result.

7.1.3 Analysis of stability of overall system

In addition to the loads on the wall the stability of the system as a whole (wall and footing) must be analysed (sliding, overturning and bearing capacity). The earth pressure adopted for stability analysis is calculated to the base of the heel and requires an equivalent wall to be defined. Starting at the top of the heel the inclination of the equivalent wall is given by:

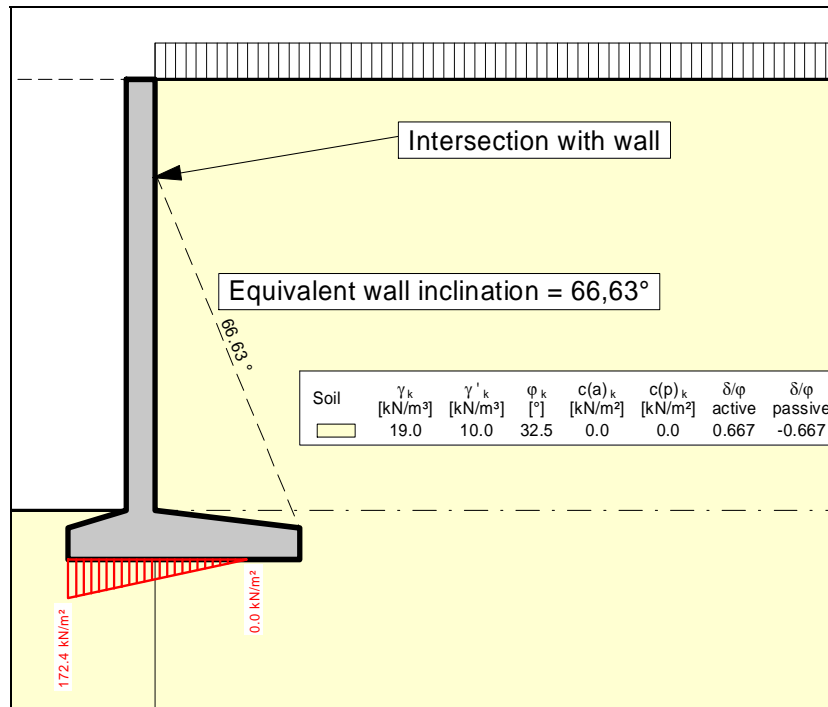


Figure 4 Equivalent wall inclination

$$\vartheta'_a = \pi - \vartheta_a - (\pi/2 - \varphi) = \text{Equivalent wall inclination}$$

$$\vartheta_a = \varphi + \text{arc cot} \left[\tan(\alpha + \varphi) + \frac{1}{\cos(\alpha + \varphi)} \sqrt{\frac{\sin(\varphi + \delta) \cdot \cos(\alpha + \beta)}{\sin(\varphi - \beta) \cdot \cos(\delta - \alpha)}} \right]$$

φ = friction angle

α = wall inclination

δ = angle of skin friction

β = ground slope

If the given geometry leads to an intersection with the stem the earth pressure below this intersection is calculated using an angle of skin friction φ on the inclined equivalent wall. If increased active earth pressure is adopted, φ is reduced accordingly (e.g.: a factor for increased earth pressure of 0.25 results in a reduction to 0.75). The earth pressure acting on the possibly inclined stem above the intersection is calculated using the angle of skin friction δ . When using increased active earth pressure, δ is reduced in accordance with the data given above.

The angle β is given as follows for sloping ground:

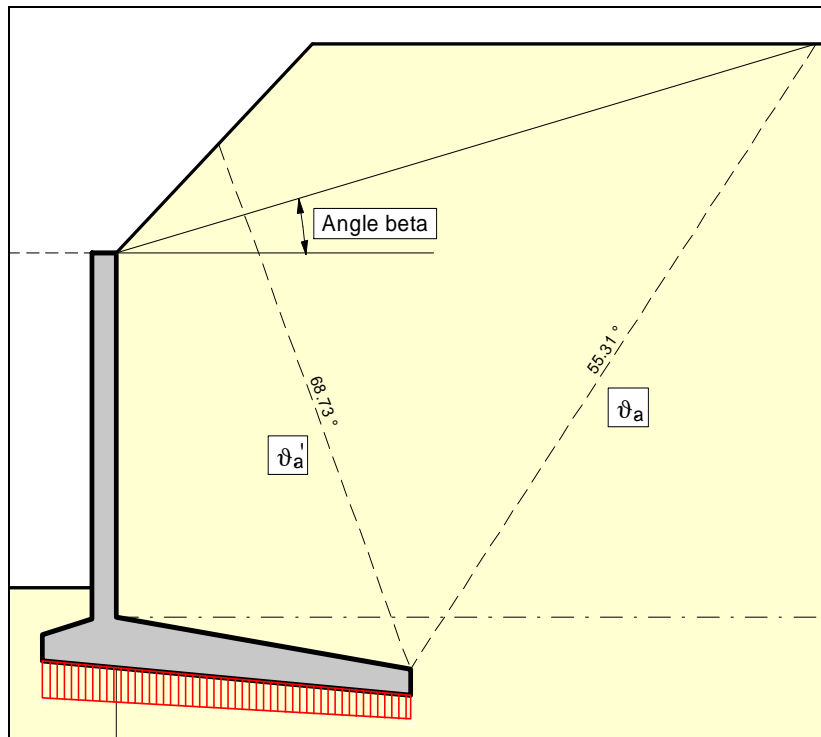


Figure 5 Determination of angle β

All load parameters necessary for stability analysis can now be determined using this data. All data can be viewed by going to the menu item "**Evaluation/Interim Results**".



These data are included in the output table. The stability of the wall (bearing capacity, sliding, overturning) is thus verified and its settlements modelled.

7.2 Soil properties

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

- depth in metres below top of the wall, or absolute depth;
- unit weight [kN/m³] of moist soil γ ;
- unit weight [kN/m³] of buoyant soil γ' ;
- friction angle [°];
- cohesion (active and passive) [kN/m²];
- active angle of wall friction as the ratio δ_a/φ ;
- passive angle of wall friction δ_p/φ ;
- permeability to the left and right of the wall [m/s];
- constrained modulus E_s .

Permeability is taken into consideration in a calculation of subsurface hydraulic conditions via flow conduits (see Section 7.7.2).

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.

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:

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

$$0.0 \leq f \leq 1.0$$

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.

7.7 Water pressure

7.7.1 Classical water pressure approach

Water pressure can affect the active and passive sides of the cantilever wall. The classical water pressure approach is illustrated in the following Figure 6:

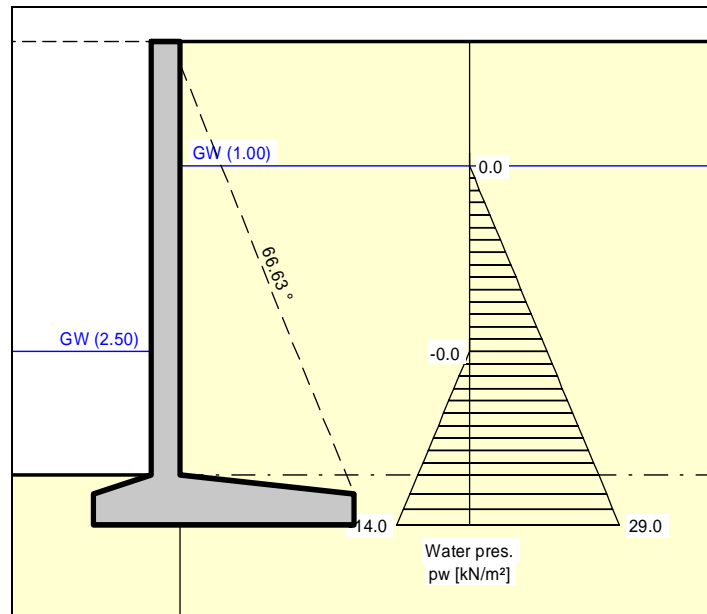


Figure 6 Classical water pressure approach

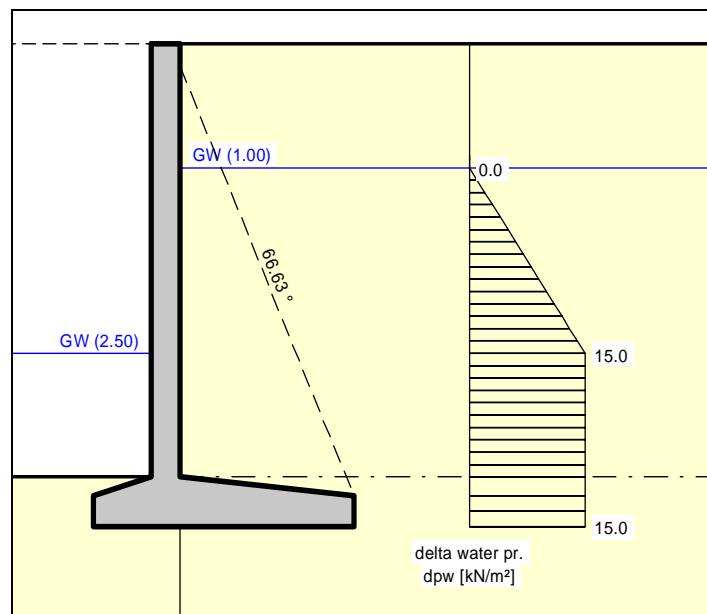


Figure 7 Resultant water pressure

However, the classical approach does not take the permeability of the soil into consideration. The gradients on the active and passive sides are obtained by assuming a linear reduction in pressure.

7.7.2 Water pressure approach using flow conduits

Beside the classical water pressure approach, **GGU-CANTILEVER** also allows analysis of walls around which groundwater can percolate. To facilitate this, a flow conduit is calculated on the active and on the passive side. The permeabilities within the flow conduit can be specified separately for each soil layer.

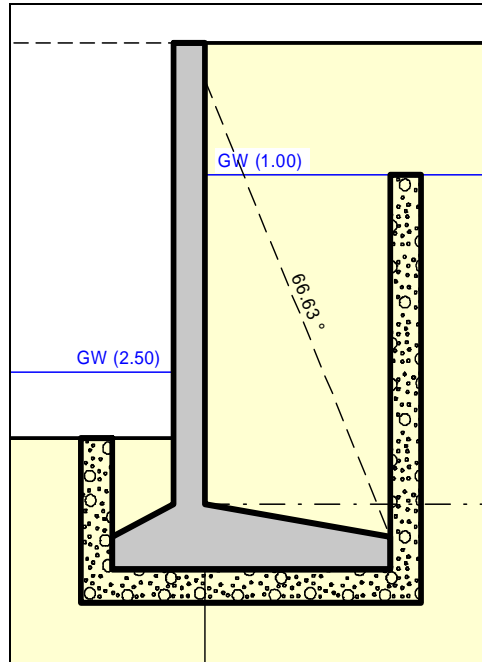


Figure 8 Water pressure approach using flow conduits

The three flow conduit system is given by the spur widths.

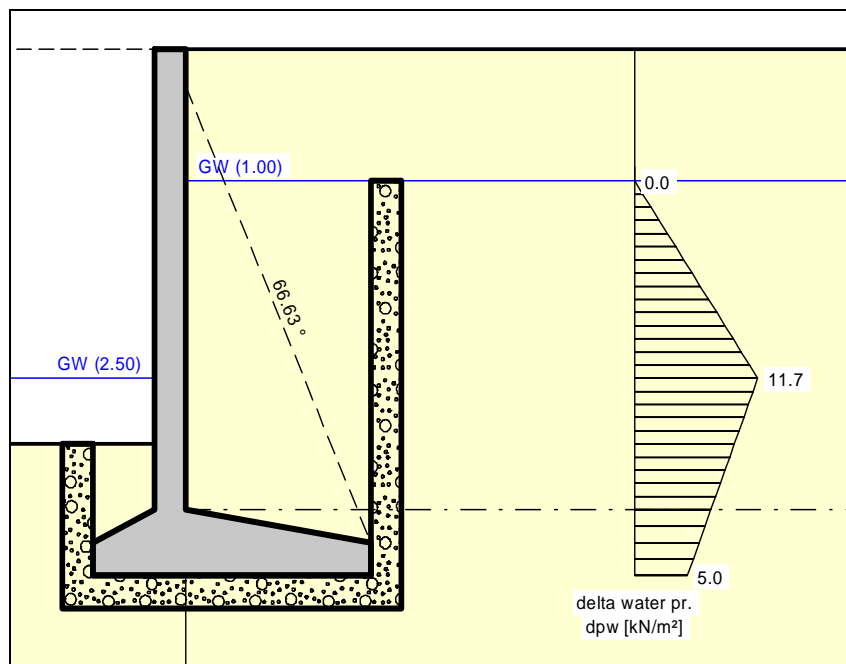


Figure 9 Resultant water pressure with flow conduits

If you select the water pressure approach using flow conduits, the potentials h at the height of both water levels will be automatically adopted as boundary conditions. In the simplest case of a global permeability value the result is linear pressure dissipation along the flow conduit. Besides the water pressures, the hydraulic gradients will also be calculated and can be selected for consideration in the calculations of active and passive earth pressures.

An approach using flow conduits is, however, much more interesting for use with variable permeabilities. In this case, the water pressures are correctly determined in accordance with potential theory by employing a small finite element module within **GGU-CANTILEVER**. You can still define additional potentials at any location to the left or right of the cantilever wall. This allows correct, simple consideration of several groundwater storeys or confined aquifers. In addition, employing user-defined potential definitions allows you to create or model any kind of water pressure distribution.

7.7.3 Recommendation for water pressure approach

The classical water pressure approach is valid when the toe of the wall is embedded in a layer of *low permeability*. Otherwise the difference in water pressure at the wall toe (see Figure 6 Classical water pressure approach) has no physical meaning, but nevertheless provides conservative design values.

The classical water pressure approach and the approach using flow conduits (under uniform conditions of permeability) both assume a linear pressure drop along the wall. Given *uniform permeability*, this can lead to an underestimate of the hydraulic gradient, since the two-dimensional flow to which the wall is subjected is not taken into consideration (see also EAU). Using flow conduits, with additionally defined potentials at the toe of the wall, this can be corrected.

For soils with *non-uniform permeability* above the wall toe, the water pressure approach using flow conduits should always be preferred. If doubts remain, you will have to carry out a two-dimensional groundwater analysis according to the EAU. The potentials determined at the wall can be incorporated into flow conduits.

If an *impermeable* layer occurs at the base of the wall there will be no hydraulic gradient along it. Nevertheless, when using the classical approach, a linear reduction in water pressure is assumed in the literature, which really amounts to unnecessarily doing the same thing twice.

7.7.4 What is a potential?

In subsurface hydraulics the potential (h) is composed of:

- the elevation head (y) of the point under consideration (in metres) and
- the water pressure (u) at the point under consideration (in metres)

If p is the water pressure in kN/m^2 at a particular point, then water pressure, u (in metres) = p/γ_w (γ_w = unit weight of water $\cong 10.0 \text{ kN/m}^3$).

$$h = p/\gamma_w + y$$

$$\begin{aligned} h &= \text{potential [m]} \\ p &= \text{water pressure [kN/m}^2\text{]} \\ y &= \text{elevation head [m]} \end{aligned}$$

An example:

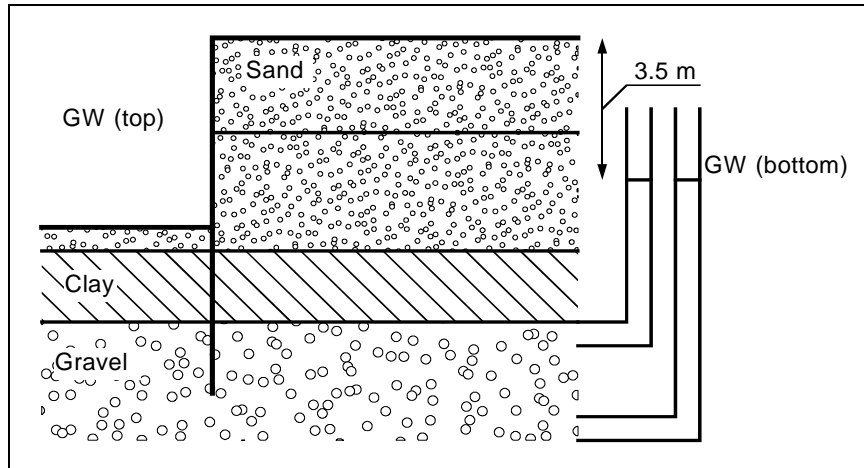


Figure 10 Definition of potential

In this example, two permeable soil layers are separated by a low-permeability layer (clay). In the excavation, groundwater is kept at the base level. The groundwater in the gravel layer below the clay is under pressure and rises to 3.5 m below the top of the wall (represented in the diagram by the water levels in the two standpipes on the right). The lowest point in the right-hand standpipe is at a higher water pressure than that in the left-hand standpipe, but both have the same potential of 3.5 m below the top of the wall. Because potential is a function of the elevation head and the water pressure [m], the left-hand standpipe makes up for its lower water pressure by its greater elevation head.

If you want to calculate such a system using flow conduits, it is sufficient to enter a potential of 3.5 m below the top of the wall for any point within the layer of gravel. The potential above the layer of clay is defined by groundwater levels to the left and right of the wall. **GGU-CANTILEVER** automatically applies these conditions, which it uses, together with the permeabilities, to calculate the potentials along the flow conduit. From the calculated potentials (h), water pressure (p) and also the gradient (i) can now be determined. Applying the above formula for p:

$$p = \gamma_w \cdot (h - y)$$

For the gradient (i) we have:

$$i = \Delta h / \Delta L$$

Δh = potential difference between two points

ΔL = distance between the two points

Using flow conduits, the load-decreasing effect of the upwardly directed hydraulic gradient on passive earth pressure can now be taken precisely into consideration.

7.8 Berms

GGU-CANTILEVER can handle 10 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).

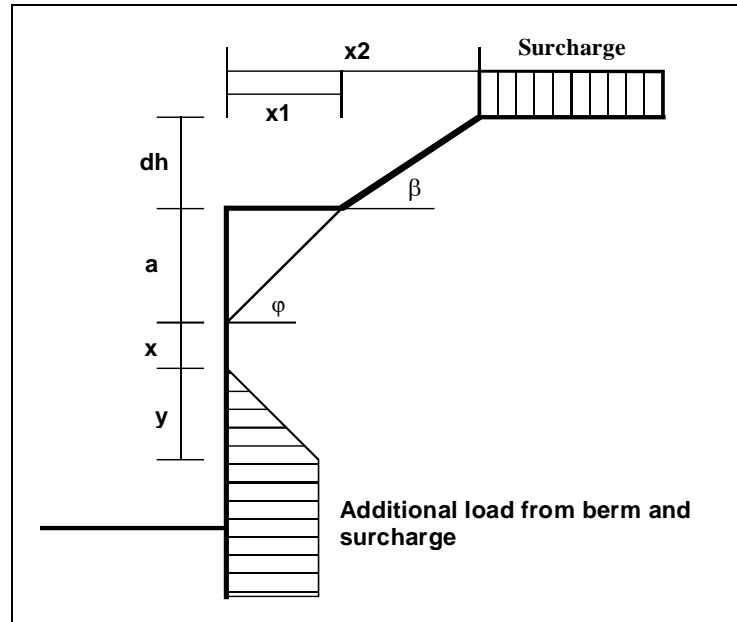


Figure 11 Berm on the active side

The following relationships apply for the parameters x and y :

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

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

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

γ = unit weight of soil in the berm area

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

7.9 Area loads

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

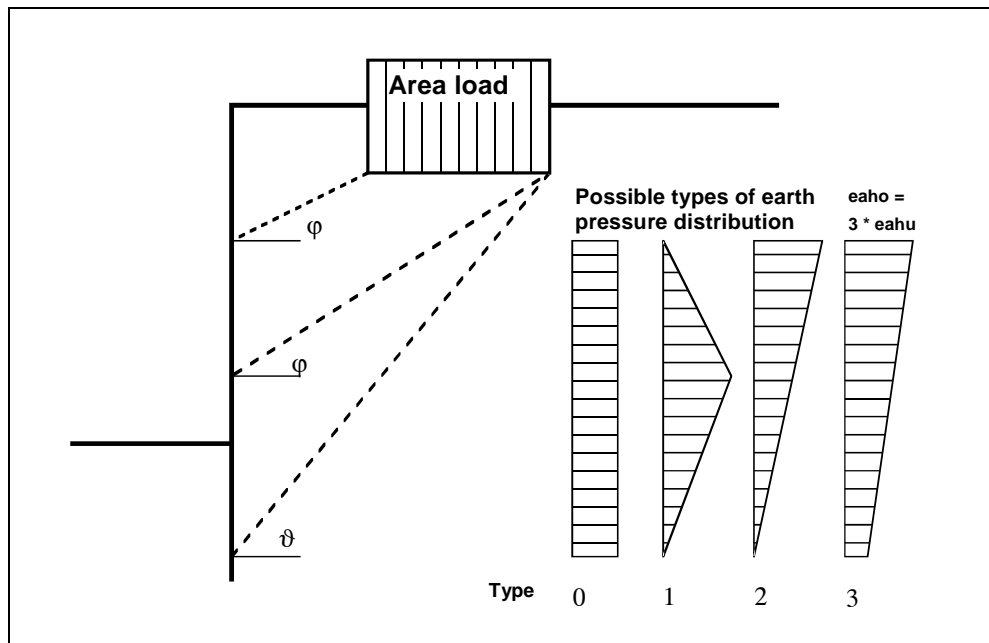


Figure 12 Area load

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-CANTILEVER** 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 13).

- For over consolidated, cohesive soils the concentration factor "3" applies, where:

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

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

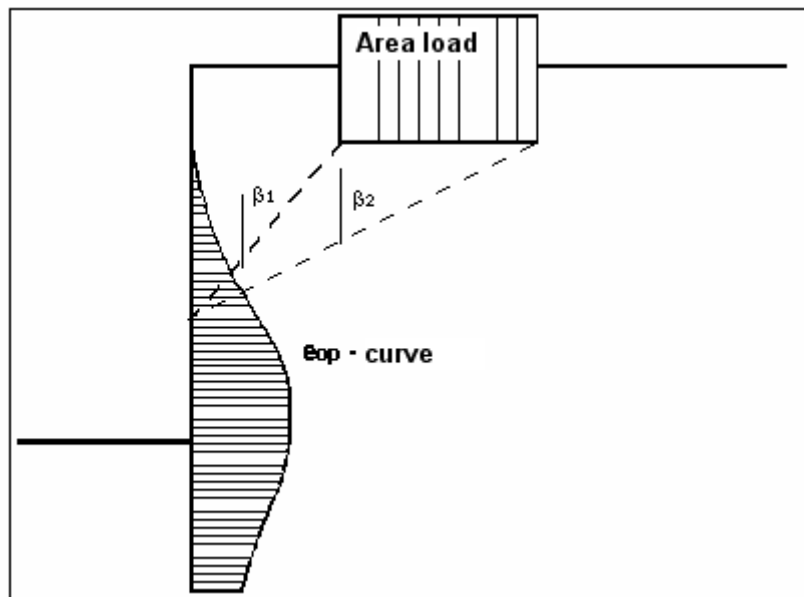


Figure 13 At-rest earth pressure from area loads

7.10 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.

These line loads can be very easily defined in the menu item "**Editor 2/Area and line loads**".

7.11 Bounded surcharges (active side)

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

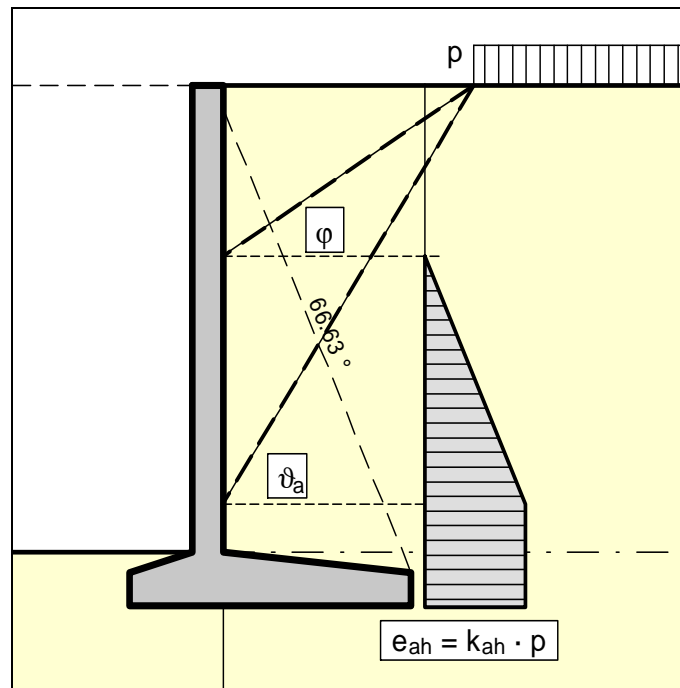


Figure 14 Bounded surcharge (active side)

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

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

7.12 Double-bounded surcharges (active side)

If two bounded surcharges are defined as follows:

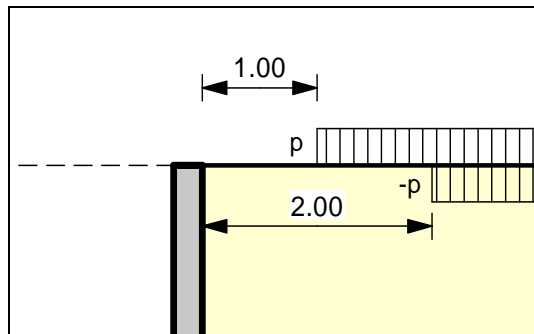


Figure 15 Two bounded surcharges

- 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 7.11).

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

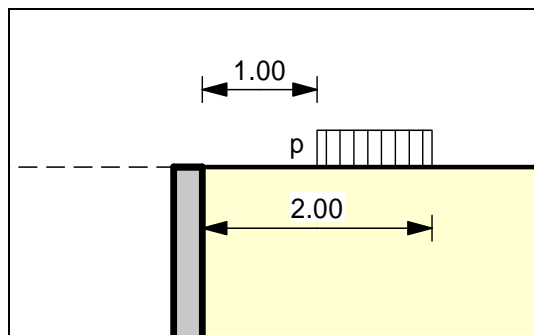


Figure 16 Double-bounded surcharge

The same result is achieved as for the definition provided by Figure 15 but with improved graphics.

The area loads described in Section 7.9 are based on different assumptions for the resulting earth pressure, so the definition of a double-bounded surcharge does not provide the same result as an equivalent area load.

7.13 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:

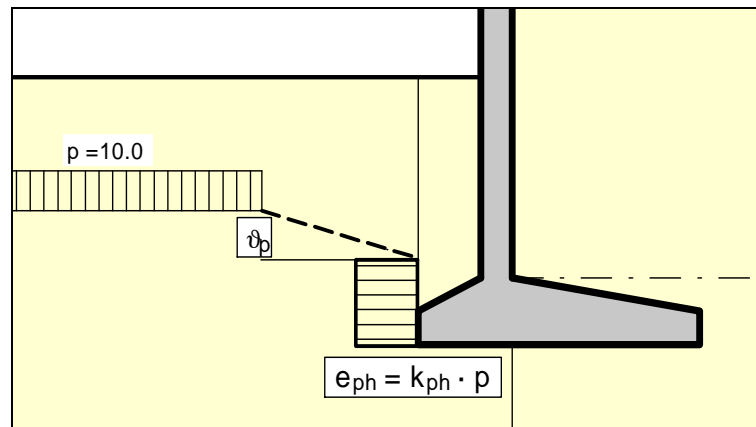


Figure 17 Bounded surcharge (passive side)

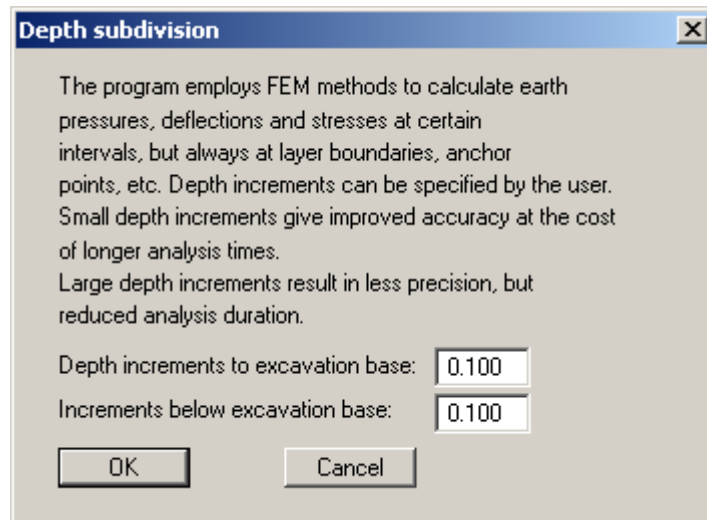
7.14 Double-bounded surcharges (passive side)

Double-bounded surcharges on the passive side can be modelled using two bounded surcharges (also see Section 7.12). Double-bounded surcharges on the passive side were only implemented due to the better graphical representation.

7.15 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 cantilever wall and any anchors and struts as a single structural system. The wall is taken into consideration as a rod construction, running along the centroid of the cantilever wall. The base of the structural system corresponds to the top of the footing as shown in Figure 2 or Figure 3 (Base of wall (stem) Example 1/Example 2). Full fixation is assumed.

The wall is analysed using the finite-element method. The size of the finite elements (rods) can be defined in the menu item "**System/Depth subdivisions**".



The setting in the dialog box assumes subdivision into 0.10 m long rods. Empiricism indicates that this produces sufficiently accurate results. A reduction in these values is not necessary.

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. The remaining values of any anchors or struts can be specified before beginning the analysis. If anchors or struts exist in the system, the point of acting is automatically placed in the centroid axis.

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-CANTILEVER** using Cholesky's method, which is also employed in other GGU programs and is numerically highly stable.

7.16 Design

When you begin an analysis, the program requires the stiffnesses of the cantilever wall and any anchors and struts present. 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. The remaining values of any anchors or struts can be specified before beginning the analysis.

The program first computes the internal forces of the cantilever wall above the footing (see Figure 2 or Figure 3 - Base of wall (stem) Example 1/Example 2). The wall is then designed using the identified internal forces.

The earth and water pressures are then computed for the system as a whole, including footing. Together with the self weights and area loads, bounded or double-bounded surcharges, stability analysis can now be performed:

- calculation of bearing capacity safety to DIN 4017;
- calculation of sliding safety;
- calculation of overturning safety
- calculation of settlements to DIN 4019;
- calculation of hydraulic heave safety;
- calculation of buoyancy safety;
- calculation of heave of anchor soil.

By going to the menu item "**Editor 1/Verifications/Safety factors**", or selecting the *partial safety factors* in "**Editor 1/Verifications/Partial factors**" and "**Editor 1/Buoyancy + Hydraulic heave**", you can select the verification to be performed.

The settlement analysis can be activated or deactivated by going to "**Editor 2/Settlements**".

7.17 Action and displacement boundary conditions

Besides full fixity at the top of the footing in accordance with Section 7.15 ("**Theoretical principles/Structural system**"), additional boundary conditions can be defined at 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 Prestressing

Personal experience has shown that discussions on the necessity of considering prestressing are never-ending. Regardless of these discussions, the program allows prestressing to be taken into consideration using the menu item "**Editor 2/Prestressing**".

Positive prestressing must be entered for anchors, negative for struts. Faulty input is pointed out before analysis begins. From a structural perspective, the program inserts a force boundary condition equivalent in size and direction to the prestress into the system at the appropriate depth. Following analysis the prestress is added to the anchor force (strut force). This ensures that deformations only result from forces greater than the prestressing force.

7.19 Earth pressure redistribution

Classical earth pressure redistribution is not appropriate for braced or tied-back cantilever walls. Earth pressure calculated according to classical theory must be redistributed. **GGU-CANTILEVER** includes a large number of pre-programmed 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;
- quadrilateral with maximum at anchor locations or at any point;
- user-defined redistribution figure by means of a polygon;
- all EAB redistribution figures.

7.20 Overturning stability

Overturning stability analysis is performed using characteristic values! The following applies:

- The resultant must remain within the 2nd kernel width under both permanent and changeable loads. Base tilt is permissible.
- The resultant must remain within the 1st kernel width under permanent loads.

7.21 Bearing capacity safety

The program analyses the bearing capacity safety to DIN 4017.

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:

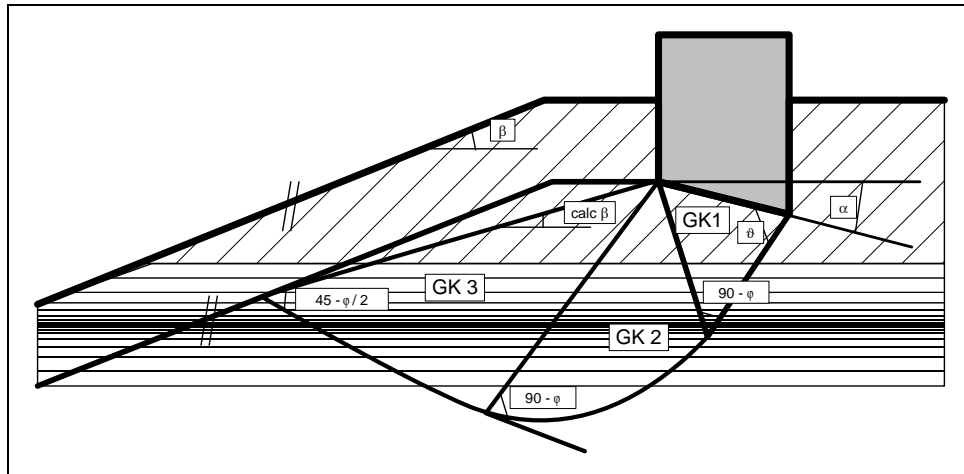


Figure 18 Logarithmic spiral

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° 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.

7.22 Sliding safety

Sliding safety is computed to EC 7. In addition, the sliding safety of an equivalent horizontal plane is computed for an inclined base plane.

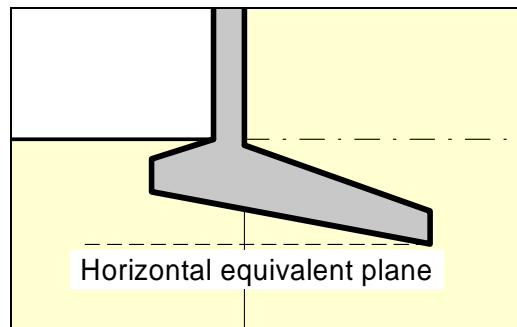
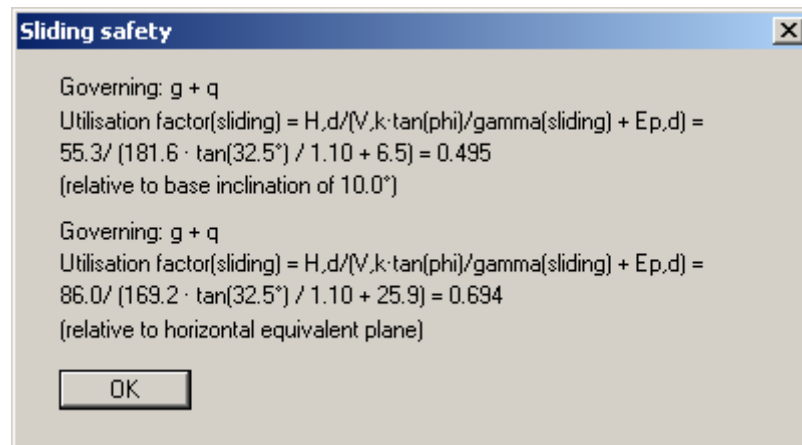
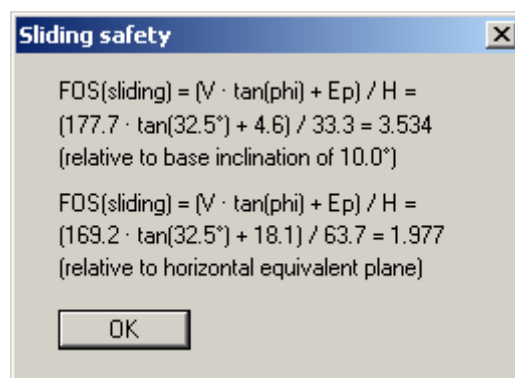


Figure 19 Horizontal equivalent plane

Sliding safety is calculated as follows in accordance with the partial safety factor concept:



Sliding safety is calculated as follows in accordance with the partial safety factor concept:



7.23 General stability

General stability can be simply verified by exporting the data from **GGU-CANTILEVER** to **GGU-STABILITY** (GGU slope stability application).

7.24 Settlements

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

The limiting depth can be defined in three different ways:

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

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

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

7.25 Hydraulic heave

7.25.1 Hydraulic heave safety using global safety factors

The hydraulic heave safety for each layer below the excavation base is determined via a comparison of the soil weights to the flow forces at the respective layer bases.

$$\eta_N = \frac{\sum_{i=1}^N G'_i}{S_N}$$

- η_N = hydraulic heave safety of the layer N
- G'_i = buoyant self-weight of layer i
- S_N = flow force of layer N
- layer 1 ($i = 1$) is the uppermost layer

The minimum value of all η_N is the hydraulic heave safety of the system.

7.25.2 Utilisation factor (hydraulic heave) using partial safety factors

Using the partial safety factor concept the following must be verified:

$$S'_k \cdot \gamma_H \leq \sum_{i=1}^N G_k^i \cdot \gamma_{g, stb}$$

- S'_k = characteristic flow force on the percolated soil mass
- γ_H = partial factor for the flow force in favourable or unfavourable subsoil in the HYD (EC 7) limit state
- G'_k = characteristic dead load of the buoyant percolated soil mass
- $\gamma_{g, stb}$ = partial factor for stabilising permanent actions in the HYD (EC 7) limit state
- layer 1 ($i = 1$) is the uppermost layer

The so-called utilisation factor μ can also be calculated from this relationship.

$$\mu_N = \frac{S'_k \cdot \gamma_H}{\sum_{i=1}^N G_k^i \cdot \gamma_{g, stb}}$$

- μ_N = utilisation factor of layer N

Utilisation factors ≤ 1.0 mean that sufficient safety is given.

7.26 Buoyancy

7.26.1 Buoyancy safety using global safety factors

The buoyancy safety for each soil layer within the excavation is determined via a comparison of the soil weights to the water pressures at the respective layer bases. The self-weights of site plant and structures, frictional forces etc. are not included.

$$\eta_N = \frac{\sum_{i=1}^N G_i}{P_N}$$

- η_N = buoyancy safety of layer N
- G_i = self-weight of layer i
- P_N = water pressure at base of layer N
- layer 1 ($i = 1$) is the uppermost layer

The minimum value of all η_N is the buoyancy safety of the system.

If the same permeability has been defined for the whole system, safety against buoyancy is not determined. In certain cases, for example, when permeability on the passive side is much greater than on the active side, the calculation of safety factors for hydraulic uplift is meaningless. If, in such, or similar cases, the message "**Buoyancy safety could not be demonstrated**" appears, you can either ignore it or set the "**Safety against buoyancy**" to **1.0**, which suppresses the message.

7.26.2 Utilisation factor (buoyancy) using partial safety factors

Using the partial safety factor concept the following must be verified:

$$A_k \cdot \gamma_{g,dst} \leq \sum_{i=1}^N G_{k,stab}^i \cdot \gamma_{g,stab}$$

- A_k = the characteristic hydrostatic buoyant force acting on the lower surface of the complete structure, the soil layer in question or the excavation structure
- $\gamma_{g,dst}$ = partial factor for destabilising permanent actions in the UPL (EC 7) limit state
- $G_{k,stab}$ = lower characteristic value of stabilising permanent actions
- $\gamma_{g,stab}$ = partial factor for stabilising permanent actions in the UPL (EC 7) limit state
- layer 1 ($i = 1$) is the uppermost layer

The so-called utilisation factor μ can also be calculated from this relationship.

$$\mu_N = \frac{A_k \cdot \gamma_{g,dst}}{\sum_{i=1}^N G_{k,stab}^i \cdot \gamma_{g,stab}}$$

- μ_N = utilisation factor of layer N

Utilisation factors ≤ 1.0 mean that sufficient safety is given.

7.27 Verification of deep-seated stability

Verification of deep-seated stability is required for anchored cantilever walls. This primarily serves to determine the necessary anchor lengths. Verification uses the method described by Ranke/Ostermayer (*Bautechnik* 1968 (Construction Engineering), Issue 10). When **verifying deep-seated stability** each anchor is first investigated (including the influence of the remaining anchors on the slip plane). Compound slip planes, which are determined by connecting the end points of the anchors involved, are then analysed.

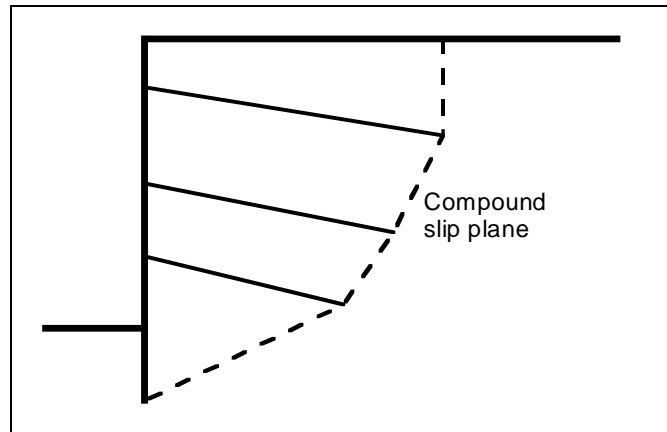


Figure 20 Compound "deep slip planes"

All possible combinations are analysed. For example, when there are four anchors:

- Slip plane passes through anchor end points:
1,2 and 1,3 and 1,4 and 1,2,3 and 1,2,4 and 1,3,4 and 1,2,3,4 and 2,3 and 2,4 and 2,3,4 and 3,4

The only condition is that the next anchor end point is always to the right of and above the preceding one.

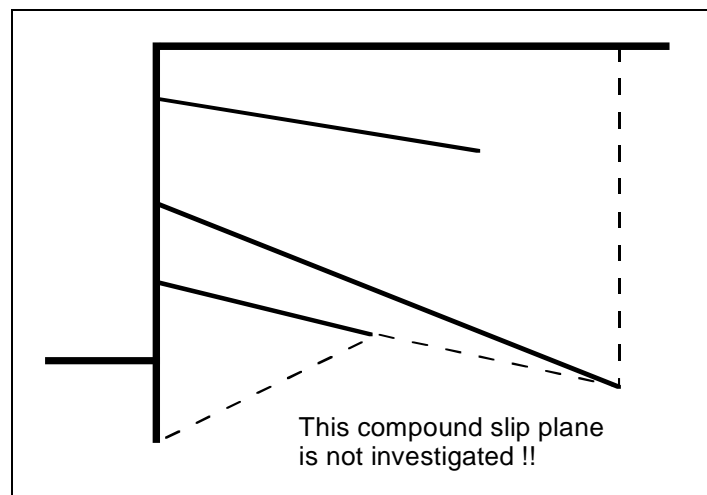


Figure 21 Compound "deep slip plane", which is not investigated

These slip planes are not critical. The most unfavourable slip plane associated with each anchor is displayed on the screen with the corresponding safety factor. A safety factor of 1.5 is generally required when adopting **global safety factors**. If this safety factor cannot be achieved or is exceeded heavily, the program can optimise individual anchor lengths.

Using **partial safety factors** the possible anchor force is acquired in complete analogy to **global safety factors**, but is divided by the passive earth pressure partial safety factor. The deep-seated stability is deemed as verified if:

$$A_{g,k} \cdot \gamma_g \leq A_{poss,k} / \gamma_{Ep} ,$$

where $A_{poss,k}$ is determined from the force polygon with permanent loads only, and

$$A_{g,k} \cdot \gamma_g + A_{q,k} \cdot \gamma_q \leq A_{poss,k} / \gamma_{Ep} ,$$

where $A_{poss,k}$ is determined from the force polygon with permanent and changeable loads. Where:

- $A_{g,k}$ = characteristic anchor force from permanent loads
- $A_{q,k}$ = characteristic anchor force from changeable loads

Here, too, optimisation with regard to a utilisation factor of 1.0 is possible.

7.28 Heave of anchor soil

Verification of heave of anchor soil is performed similar to the method described in Section 7.3.4 of the Piling Handbook 1977 (*Spundwand-Handbuch 1977*).

7.29 Reinforced concrete design

Reinforced concrete is designed to EC 2 or, alternatively, to DIN 1045 (old) depending on the selected safety factor concept. Reinforcement is calculated for all wall element sections (see Section 7.15). The output is the section with the largest reinforcement.

The stem area above the footing is designed using the wall internal forces (see Section 7.1.2).

The internal forces M and Q at the stem base/toe intersection are calculated from the soil pressure below the toe and any surcharge present. The normal force N is given by the passive earth pressure.

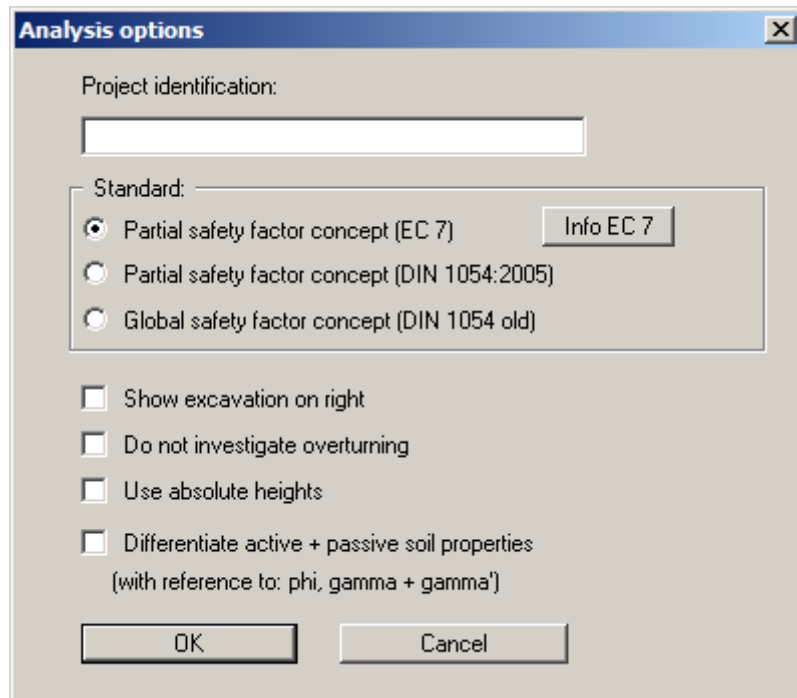
The internal forces M and Q at the stem base/heel intersection are given by adding the internal forces of the toe and the internal forces at the base of the stem. The normal force N of the heel corresponds to the normal force of the toe.

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 8.6.8). This can be particularly useful when you are working with the menu item "**Editor 2/Predeformation preferences**". Using these program functions previously saved datasets are appended to the current dataset. The existing project identifications are also shown in the appropriate dialog boxes to aid file characterisation.

In the first group box you decide which safety concept to use for your analysis and design. Activate the standard required with the corresponding command button. 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 (see Section 8.2.7). For better visualisation you can define the soil colours on the active and the passive sides differently using the *Soil properties legend* (see Section 8.6.7).

If the safety concept has been altered using the "**File/New**" dialog box, the dialog box for specifying either the safety factors or the partial factors always opens after leaving the dialog box by clicking "**OK**". These dialog boxes can also be accessed via the menu items "**Editor 1/ Verifications/Safety factors**" (see Section 8.2.14) or "**Editor 1/Verifications/Partial factors**" (see Section 8.2.15). Verifications and safety factors can be modified at any time using the above menu items.

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. The file contains all system information. Loading again later creates exactly the same visualisation 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 ".**ggu_wst**" 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, ".**ggu_wst**" will be used automatically.

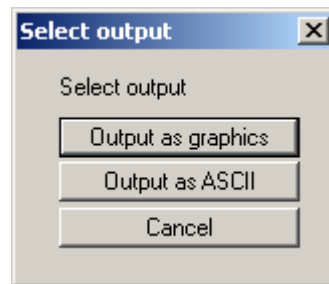
If the current system has been analysed at the time of saving, the analysis results are saved in the file. This is especially necessary if you want to use the results in this file for *predeformations* on other systems (see Section 8.3.13). If your current system represents an intermediate construction stage, it may be expedient to enter a further description of the file in "**Project identification**", in the "**File/New**" or "**Editor 1/Analysis options**" dialog box (see Section 8.1.1).

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.

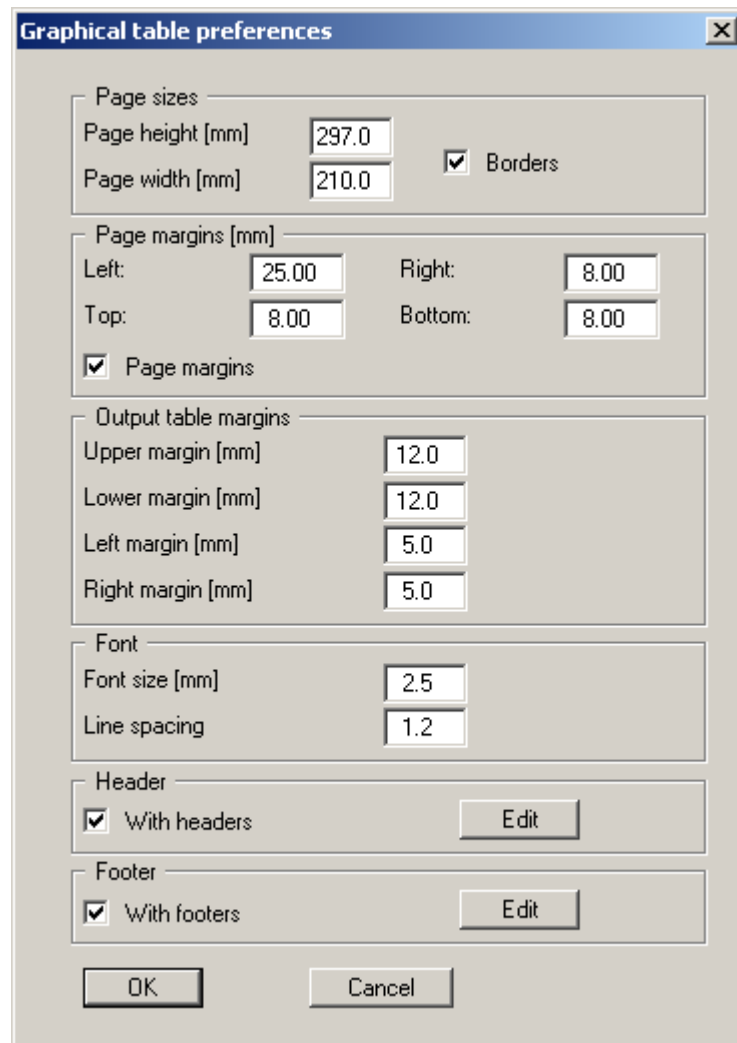
You have the option of designing and printing the output table as an annex to your report within the **GGU-CANTILEVER** application. To do this, select "**Output as graphics**" from the following options.



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

8.1.5.2 Button "Output as graphics"

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

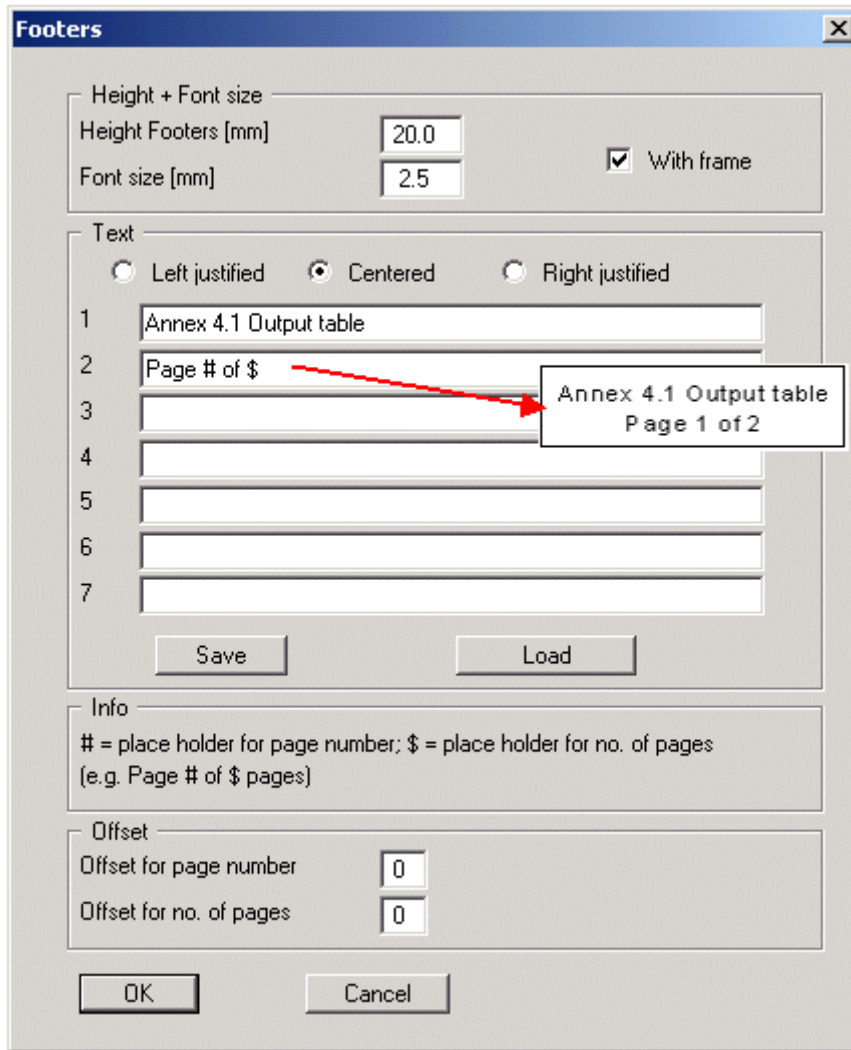





The dialog box titled "Graphical table preferences" contains several sections for configuring output settings:

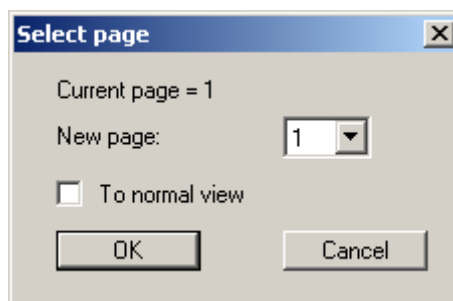
- Page sizes:** Page height [mm] is 297.0, Page width [mm] is 210.0, and the "Borders" checkbox is checked.
- Page margins [mm]:** Left is 25.00, Right is 8.00, Top is 8.00, and Bottom is 8.00. The "Page margins" checkbox is checked.
- Output table margins:** Upper margin [mm] is 12.0, Lower margin [mm]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

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. 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 then click on the "Edit" button. You can then edit as required in a further dialog box.

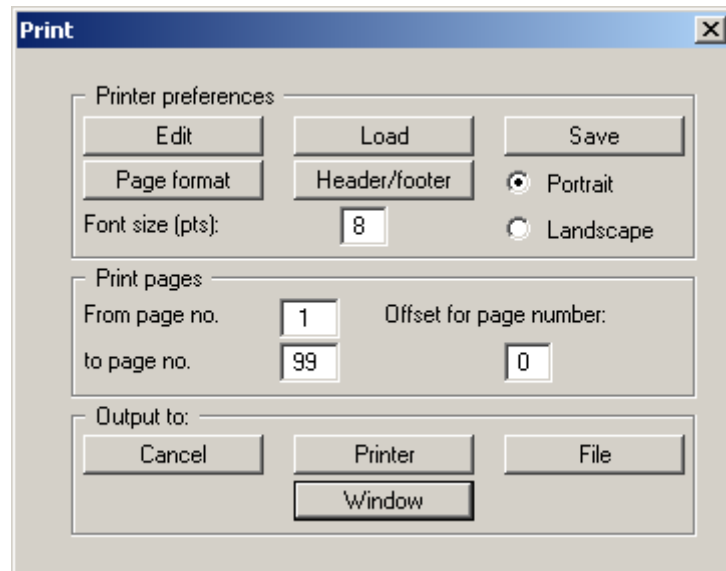


Automatic pagination can also be employed here if you work with the placeholders as described. After closing the dialog box using "OK", define the state variables and displacements to be represented in the output table in a further dialog box. The output table is then displayed page by page on the screen.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



8.1.5.3 Button "Output as ASCII"

You can have your analysis data sent to the printer, without further work on the layout, or save it to a file for further processing using a different program, e.g. a word processing application. First, define the state variables and displacements to be represented in the output table in a dialog box. You will then see the following dialog box:



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-CANTILEVER.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-CANTILEVER** 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. In addition, the wall can be exported as structural element.

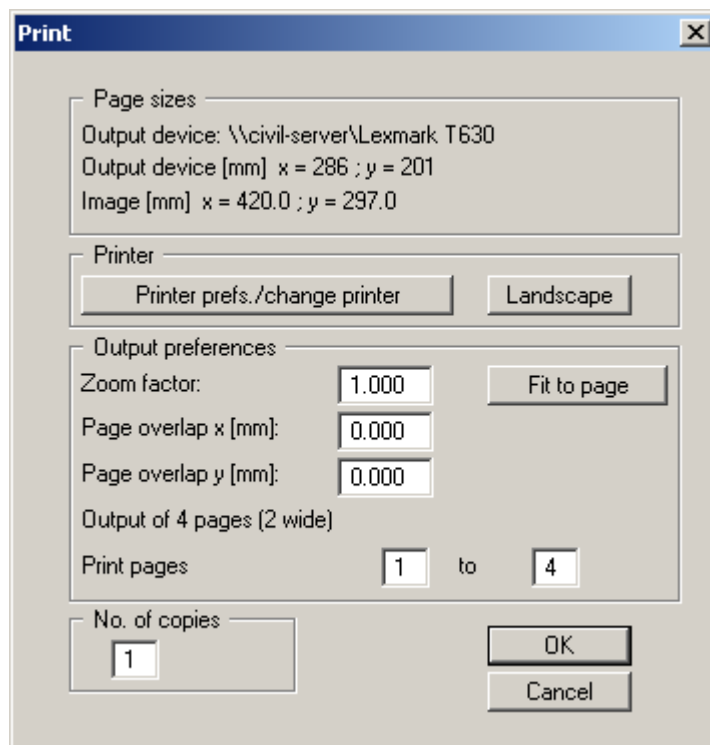
8.1.7 "Printer preferences" menu item

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

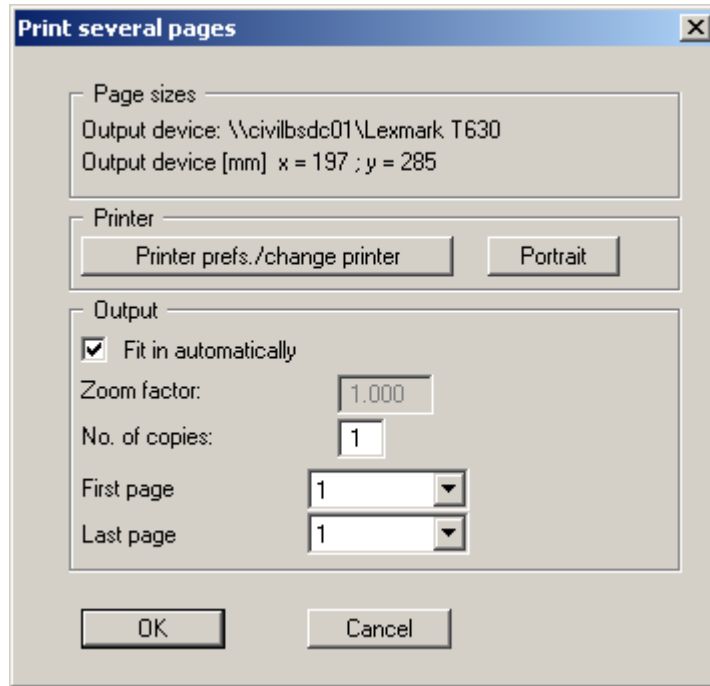
You can select your output format in a dialog box. You have the following options:

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



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

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

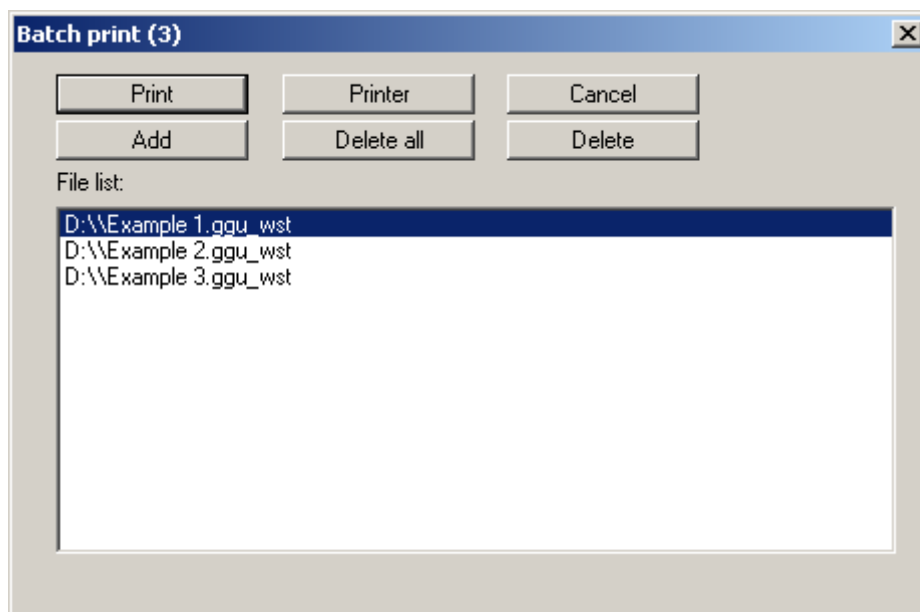
If you select the "Copy/print area" tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see "Tips and tricks" in Section 9.4).

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	
Top of wall [m]	0.00
Base of excavation [m]	-3.50
Groundwater (right) [m]	-7.49
Groundwater (left) [m]	-7.49
Distributed load [kN/m ²]	10.00
Type of distributed load:	Component above 10.0 kN/m ² variable

Passive side	
Distributed load [kN/m ²]	0.00

In the upper group box, enter the depth of the excavation base 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 additional data must be relative to this value, i.e. the corresponding [m AD] data must be entered.

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. Additionally, you can define a distributed load on the passive side.

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

8.2.3 "Wall (general)" menu item

Enter the loads acting on the cantilever wall and the length of the cantilever wall. A more detailed description can be found in "**First steps using a worked example**", Section 6.4.

8.2.4 "Wall (geometry)" menu item

Enter the geometry of the cantilever wall. A more detailed description can be found in "**First steps using a worked example**", Section 6.5.

8.2.5 "Berms (active side)" menu item

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

No.	Left [m]	Right [m]	Delta h [m]	Surcharge [kN/m ²]	Live load
1	1.000	2.000	0.50	5.00	<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 berm(s) 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.

8.2.6 "Berms (passive side)" menu item

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

8.2.7 "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	k [m/s]	k [m/s]	Es
		[m]	[kN/m ³]	[kN/m ³]	[°]	[kN/m ²]	[kN/m ²]	[-]	[-]	Left	Right	[MN/m ²]
1	Sand	-7.50	19.0	10.0	32.5	0.0	0.0	0.667	0.000	1.00E-4	1.00E-4	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. It is only necessary to enter permeabilities if hydrostatic pressures must be considered, and any hydraulic gradients, using flow conduits (see Section 7.7.2).

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 (see dialog box above).

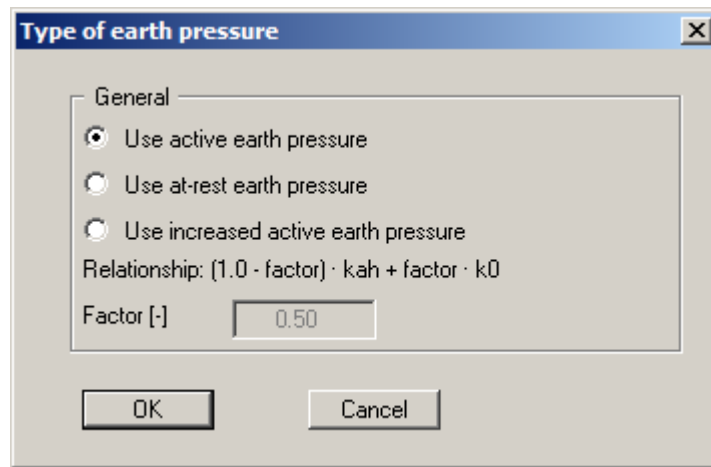
In stratified soils the number of layers must be entered under "Edit number of soils". Clicking the "Sort" button sorts the soil layers according to depth; however, this is performed automatically when you click "OK" to leave the dialog box. This eliminates the possibility of input errors.

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.

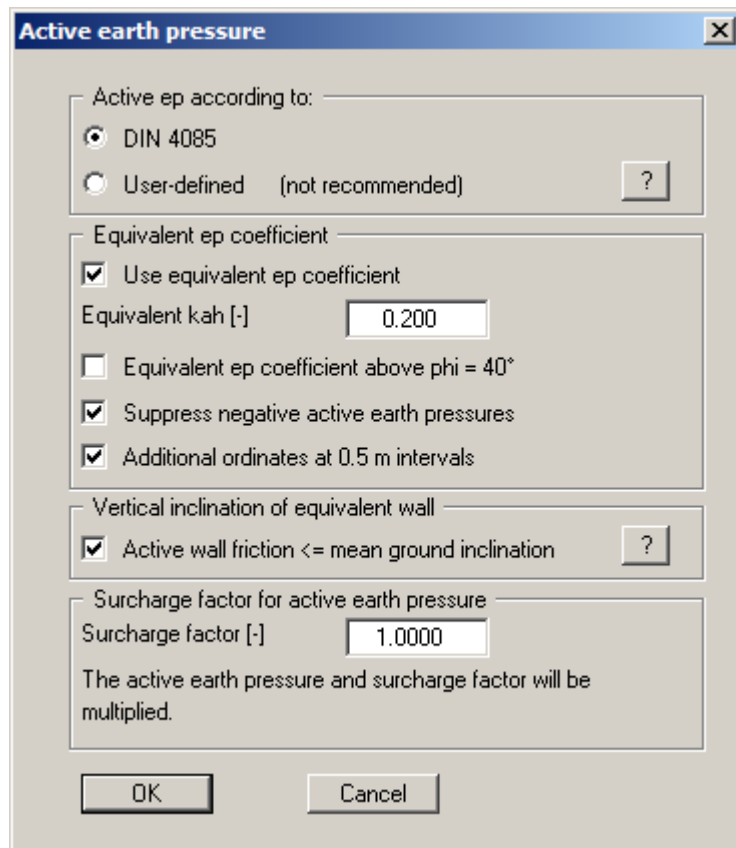
8.2.8 "Type of earth pressure" menu item

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



8.2.9 "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 method is only of interest if you wish to analyse an example from older literature sources or check certain results.

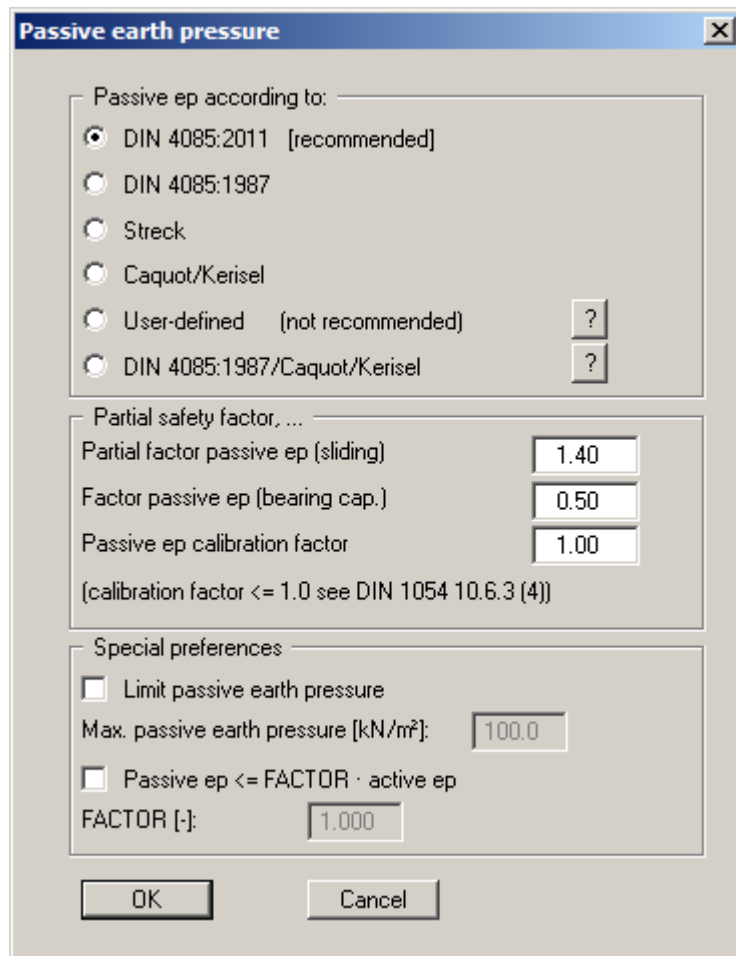
The "**Use equivalent ep coefficient**" check box should only be deactivated in exceptional circumstances (see EAB R 4). The equivalent earth pressure coefficient can only be smaller than 0.2 in special circumstances (see EAB R 4). It only makes sense to deactivate this check box when re-examining existing 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.

DIN 4085:2011 states that for a vertical equivalent wall the active wall friction angle must equal the mean ground inclination. If the check box "**Active wall friction <= mean ground inclination**" is activated, the data entered in "**Editor 1/Soils**" are adopted.

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-CANTILEVER** also offers this possibility.

8.2.10 "Passive earth pressure" menu item

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

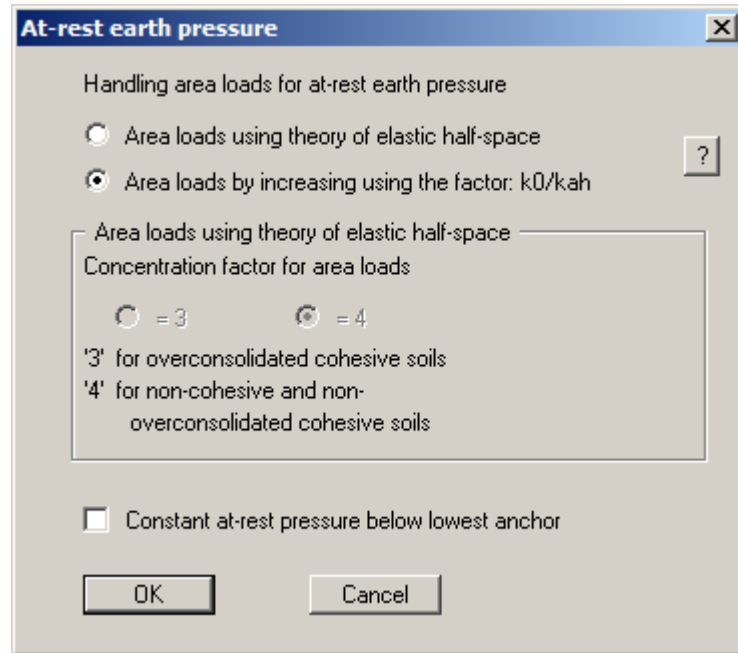


In the upper group box you specify the type of passive earth pressure calculation. The method of choice is "**DIN 4085:2011**". Only if you want to check examples from older literature or examine certain analysis results does it make sense to select a different method.

8.2.11 "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.

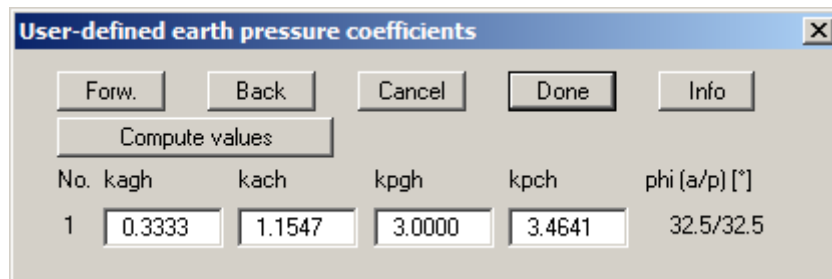
Alternatively, it is possible to determine the demands placed on the wall by area loads according to the theory of elastic half-space. The required concentration factor is specified in the following dialog box, if the upper check box is activated.



According to the EAB, R 23 (Paragraph 2), for systems with at least two rows of anchors or struts, at-rest earth pressure can be kept constant from the lowest support. This is effected by selecting the check box at the bottom of the dialog box.

8.2.12 "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-CANTILEVER** will convert them for sloping ground using equation values for k_{ah0} and $k_{ah\beta}$ (see "**Theoretical principles/Berms**", Section 7.8).

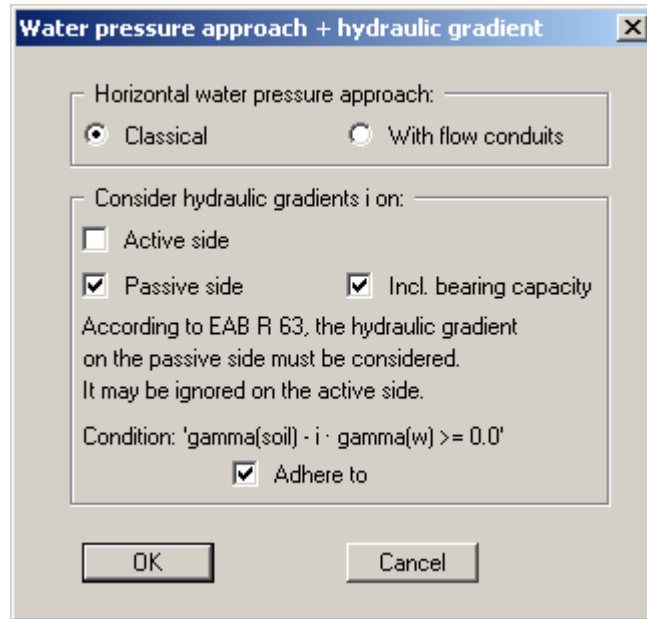


If the "**Compute values**" button is pressed the earth pressure coefficients can be calculated by the program in accordance with the required standard and slope angle.

The earth pressure coefficients entered here are only adopted if the "User-defined" check box is activated in "Editor 1/Active earth pressure" and/or in "Editor 1/Passive earth pressure" menu items (see Sections 8.2.9 and 8.2.10).

8.2.13 "Groundwater" menu item

With regard to adopting horizontal water pressure you are referred to the explanations in Section 7.7.



The cantilever walls of an excavation in flowing groundwater will experience increased earth pressure on the active side and decreased earth pressure on the passive side. In accordance with EAB R 63, the effect on the active side can generally be ignored, while the effect on the passive side must always be taken into consideration. You can make the appropriate selection in the bottom dialog group box.

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:

- Hydraulic heave:** FOS = 1.50, Verify
- Buoyancy:** FOS = 1.10, Verify
- 'Deep-seated stability':** FOS = 1.50
- Sliding safety:** FOS = 1.50, Verify, Sliding safety without passive EP (Info), Friction angle in base plane [°] = 30.00, Use
- Bearing capacity:** FOS = 2.00, Verify, Bearing cap. with depth factor (Info), Check and correct 5° condition, Decrement [°]: 0.10 (Info), Top of bearing cap. failure 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 a dialog box for defining the partial factors.

Verifications/Partial factors preferences

Actions/Resistance

Permanent actions: 1.35

Permanent actions (at-rest pressure): 1.20

Variable actions: 1.50

Always design struts for DS-P (1.35/1.50)

Passive earth pressure (sliding): 1.40

Factor passive ep (bearing cap.): 0.50 ?

Passive ep calibration factor: 1.00 ?

Bearing capacity

Verify

gamma(bearing cap.): 1.40

Bearing cap. with depth factor ?

Check and correct 5° condition

Decrement [°]: 0.10 ?

Top of bearing cap. failure body parallel to slope

sliding

Verify

gamma(sliding): 1.10

Friction angle in base plane [°] =

30.00 Use ?

Serviceability verification:

Sliding safety without passive EP ?

Serviceability verification: ?

EQU analysis

Verify

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

OK Cancel

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

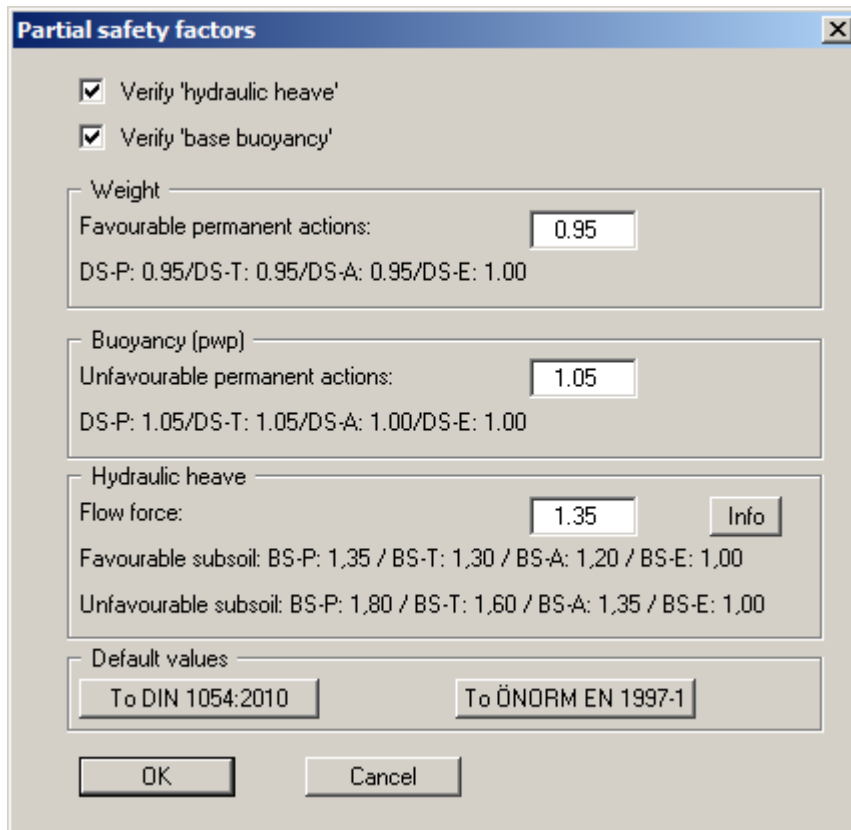
In the "**Default values**" group box the partial factors for the various load cases and subsoil conditions given in the DIN 1054:2010 and in the EC 7 can be selected by means of the dialog box reached by clicking the "**To DIN 1054:2010**" button. The load case designations were altered for the EC 7 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 "Buoyancy + Hydraulic heave" menu item

If you have selected the *partial safety factor concept*, you will see a dialog box, using this menu item, in which you can activate the verifications for hydraulic heave and the buoyancy of the excavation base.

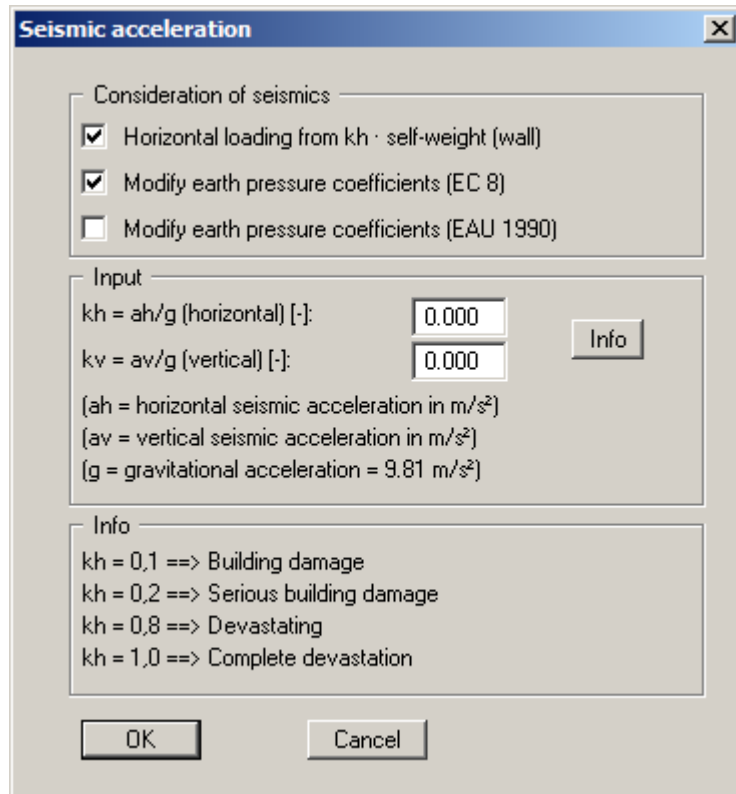


If you press the "**Info**" button, you will see further information on the subsoil situation (favourable/unfavourable). In the "**Default values**" group box the partial factors for the various load cases and subsoil conditions given in the DIN 1054:2010 and in the EC 7 can be selected by means of the dialog box reached by clicking the "**To DIN 1054:2010**" button. In addition, it is possible to select the partial safety factors compliant with Austrian standards using the "**To ÖNORM EN 1997-1**" button.

8.2.17 "Seismic acceleration" menu item

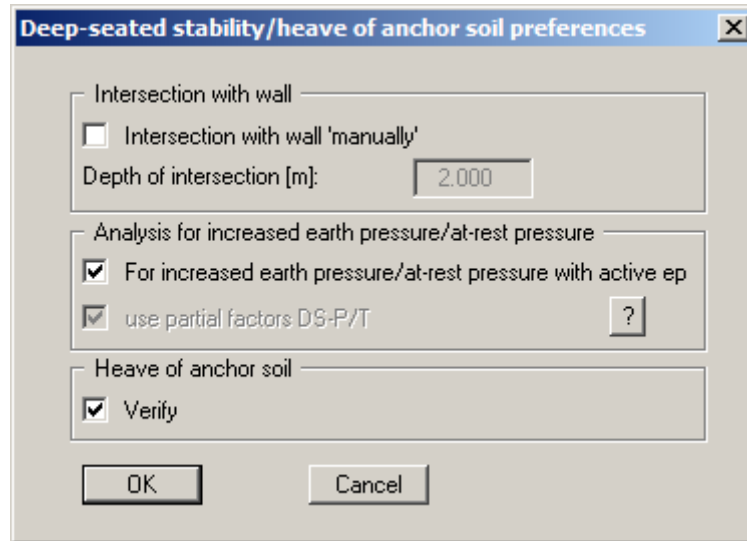
Seismic loads are given in multiples of gravitational acceleration. The following options, which can be selected individually or together, are available for taking seismic loads into consideration:

- 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 8 or EAU 1990, Section 2.14, by increasing the active earth pressure coefficients and reducing the passive earth pressure coefficients.



8.2.18 "Deep-seated stability/Heave of anchor soil" menu item

Using this menu item you can select the intersection with the wall manually and enter an appropriate depth.



In the "**Analysis for increased earth pressure/at-rest pressure**" group box you can define whether the deep-seated stability analysis should be done with *active earth pressure* or alternatively, the *partial factors DS-P/T* should be used.

EAB R 44 stipulates that this must also be used for increased or at-rest earth pressure to analyse deep-seated stability. In addition, the partial factors for DS-P/T (LC1/2) must be adopted. In the DS-P/T load case the partial factor for passive earth pressure is 1.35. Only when the 'normal' partial factor for the passive earth pressure is greater than 1.35 DS-P/T is adopted.

The partial factors for the actions to DS-P/T are around 5% greater than those to DS-T. Simplified, but conservatively, the anchor forces are multiplied by the factor '1.05'. R 44 does not always return higher utilisation factors.

Verification of *heave of anchor soil* can be activated in the lower group box of the dialog box. If a value unequal to '0' is entered for the height of the dead man when entering the anchors, and this check box is activated, verification is performed similar to the method described in Section 7.3.4 of the Piling Handbook 1977.

8.3 Editor 2 menu

8.3.1 "Lateral pressures" menu item

If, in addition to the diverse options for determining earth pressure on the wall, you also need to take surcharges on the active side into account, define them here.

No	top [m]	bottom [m]	e (top) [kN/m ²]	e (bottom) [kN/m ²]	Live
1	1.000	3.250	10.00	20.00	<input type="checkbox"/>

The number of lateral pressures can be modified using the "**x lateral pressure(s) 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 the new *partial safety factors* you will also see the "**Live**" check box because of the differentiation between permanent and changeable lateral pressures.

8.3.2 "Area and line loads" menu item

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

No	p(v) [kN/m ²]	p(h) [kN/m ²]	x(left) [m]	x(right) [m]	Depth [m]	Wall pressure shape	Live
1	150.00	0.00	2.000	3.000	1.200	Triangle (max. at top)	<input type="checkbox"/>

The "**x area load(s) to edit**" button allows you to determine the number of area loads to be considered. Subsequently you can enter the sizes "**p(v)**" (= vertical) and "**p(h)**" (= horizontal), the ordinates and the "**Depth**" of the area loads. You must also enter the "**Wall pressure shape**" of the resultant horizontal forces on the wall (see also Section 7.9).

Using the "**Generate line loads**" button, line loads acting vertically on the wall may be considered adopted as area loads (see Section 7.10).

8.3.3 "Compaction earth pressure" menu item

The compaction earth pressure to DIN 4085:2011 can be analysed using this menu item. The data required are described in the DIN and can be entered into the dialog box shown below. Pressing the "?" buttons, you will see further information.

Compaction earth pressure

Adopt compaction earth pressure

Compaction earth pressure also for stability analysis

Surface loading

Distributed load as surface load ?

Compaction plane [m] 0.00 ?

Compaction

Intense compaction Light compaction ?

Wall

Yielding wall Non-yielding wall

Width of backfill space [m] 1.00

Soil properties

Use uppermost soil layer properties

Use user-defined soil properties

Friction angle $\phi_{i,k}$ [°] 32.50

$\gamma_{i,k}$ [kN/m³] 19.00

OK Cancel

Two files containing analyses taken from *Geotechnische Nachweise nach EC 7 und DIN 1054* (Geotechnical analyses to EC7 and DIN 1054), 3rd edition 2012, by Martin Ziegler, can be found in the program's example folder. The analyses refer to pages 70 and 72 of the literature reference.

8.3.4 "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 ²]	x(left) [m]	Depth [m]	Live
1	25.00	2.000	0.000	<input type="checkbox"/>

Using "**x bounded surcharge(s) 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.

8.3.5 "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 ²]	x(Left) [m]	x(Right) [m]	Depth [m]	Live
----	------------------------------	----------------	-----------------	--------------	------

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

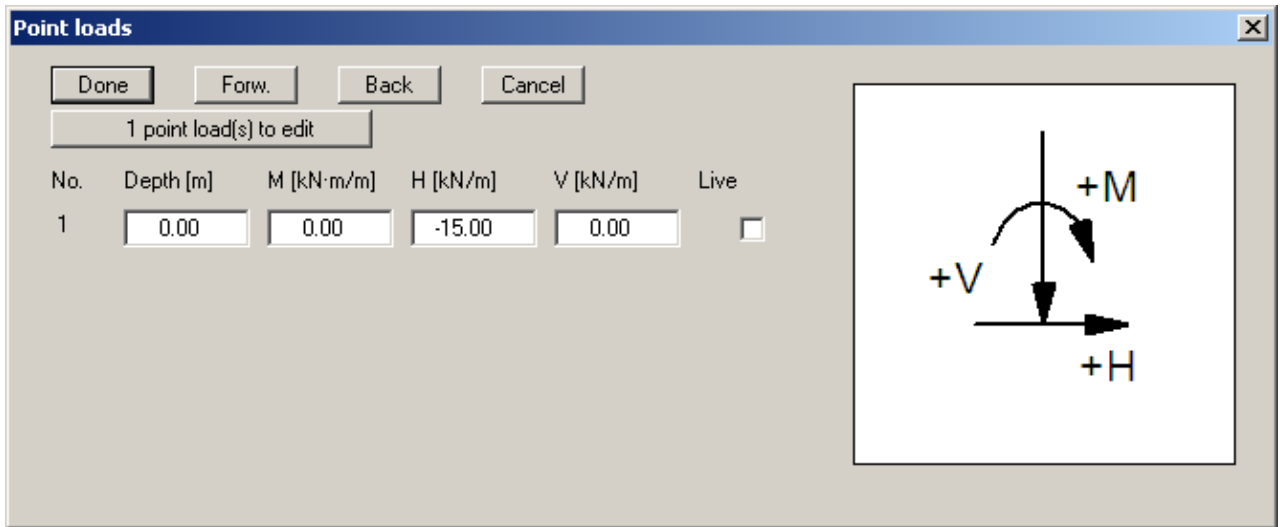
Double-bounded surcharges consist of 2 bounded surcharges with opposing signs. Whether or not the linear component of the resulting earth pressure (see Figure 14 in Section 7.11) may be adopted at its full value for the bounded surcharge with the negative sign has not yet been conclusively clarified. This can be influenced using the control parameter:

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

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

8.3.6 "Action boundary conditions" menu item

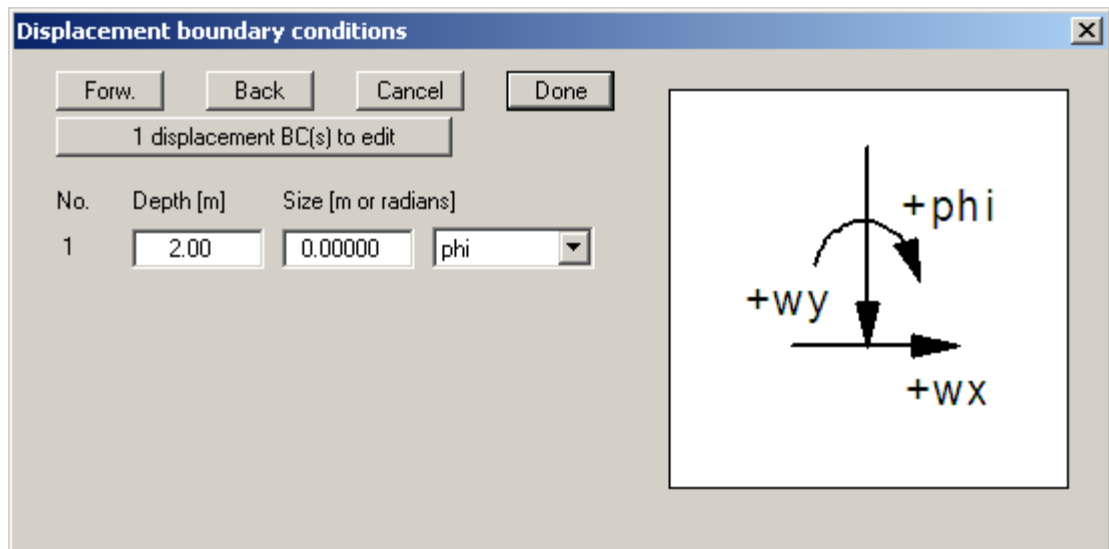
You can introduce additional action boundary conditions anywhere along the wall.



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

8.3.7 "Displacement boundary conditions" menu item

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



In the example above a rotation "**phi**" of the wall of 0.0 has been entered 2.0 m below the top of the wall. "**wx**" and "**wy**" in the pull down menu stand for the horizontal and vertical displacements; the direction is defined by means of the sign.

8.3.8 "Anchors" menu item

Anchors are defined in the dialog box of this menu item.

No.	Depth [m]	Inclin. [°]	Length [m]	EA [kN/m]	H DM [m]	L GR [m]
1	0.50	5.0	8.00	2.100E+7	0.00	5.00

It is necessary to enter the following information:

- Depth [m] = position of anchor head
- Inclination [°] of anchor
- Length [m] of anchor
- EA [kN/m] = Axial stiffness of the anchor steel
- H DM [m] = Height of deadman (if present)
This value is only necessary for analysis of deep-seated stability.
- L GR [m] = Length of grouted section!!!
This value is of no importance for the analysis itself, but for the graphical visualisation.
The length labelling of the anchor includes half the length of the grouted section.

8.3.9 "Struts" menu item

Struts are defined in this dialog box.

No.	Depth [m]	Inclin. [°]	Length [m]	EA [kN/m]	EI [kN·m ² /m]	V load [kN/m ² /m]	Hinge	No. of rods
1	0.00	10.0	8.00	2.100E+7	2.100E+7	0.00	<input checked="" type="checkbox"/> Yes	10

It is necessary to enter the following information:

- Depth [m] = position of the strut head
- Inclination [°] of the strut
- Length [m] of the strut
- EA [kN/m] = Axial stiffness of the strut [kN/m]
- EI [kN·m²/m] = Bending stiffness of the strut [kN·m²/m]
- V load [kN/m²/m] = Vertical load on the strut [kN/m²/m]
- Hinge
Here you specify whether the strut is hinged or rigidly fixed to the cantilever wall.
- No. of rods
If you have defined a vertical load on the strut, you will also get its distribution of moments. Of course, this only works if the strut is divided into a number of rods (finite elements, see Section 7.15). The maximum number of rods that can be specified is 20.

8.3.10 "Prestressing" menu item

Personal experience has shown that discussions on the necessity of considering prestressing are never-ending. Regardless of these discussions, the program allows prestressing to be taken into consideration.

Anchor no.	Depth [m]	Prestressing [kN/m]
1	0.50	10.00

Positive prestressing must be entered for anchors, negative for struts. Faulty input is pointed out before analysis begins. From a structural perspective, the program inserts a force boundary condition equivalent in size and direction to the prestress into the system at the appropriate depth. Following analysis the prestress is added to the anchor force (strut force). This ensures that deformations only result from forces greater than the prestressing force.

8.3.11 "Groundwater potentials" menu item

You can define additional potentials anywhere along the wall (see Section 7.7.2).

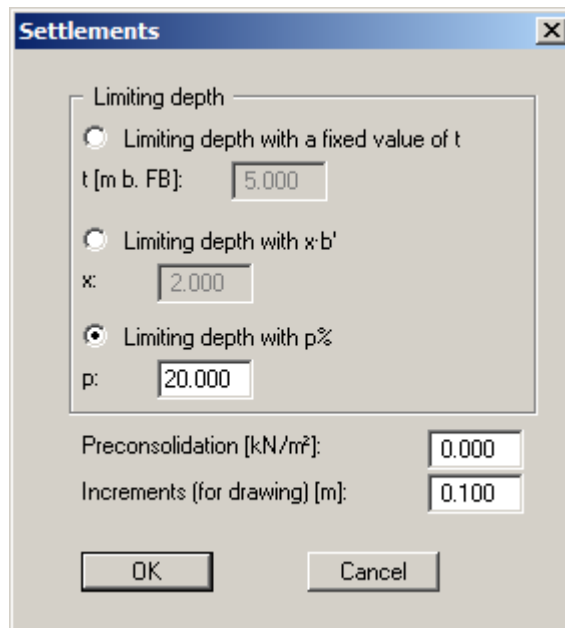
No.	Depth [m]	Value [m]	Left/Right
1	2.000	4.000	<input checked="" type="checkbox"/> Right
2	2.000	4.000	<input type="checkbox"/> Right

In the above example a potential of 4.0 m below the top of wall at the left and right of the cantilever wall has been entered at a depth of 2.0 m below the top of wall. Additional potentials will only be considered if the water pressure approach using *flow conduits* has been selected (see Section 8.2.13).

After clicking "**Switch to: Enter water pressure**", enter the water pressure in kN/m² instead of the potentials in the adapted dialog box.

8.3.12 "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 wall. The stress analysis for determining the limiting depth is performed for the mean soil pressure in the characteristic point. The lateral stresses on the active side of the wall are adopted for this limiting depth calculation. If the limiting depth lies below the depth of the lowest soil layer for this type of calculation, the values of this lowest layer are adopted for the settlement analysis.

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

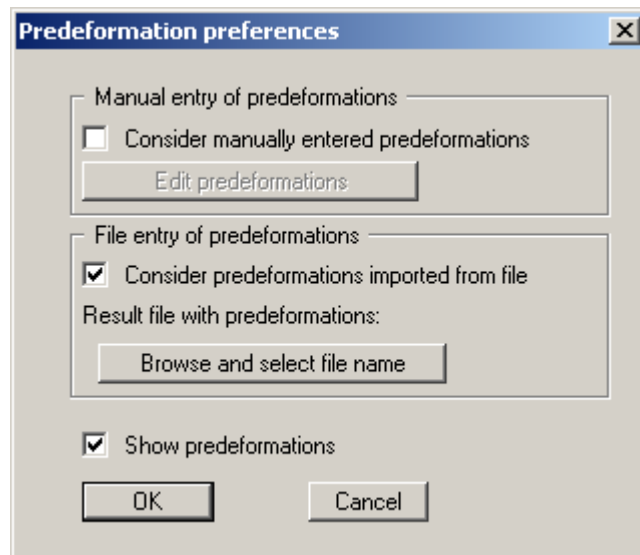
The "**Increments**" are only important for the graphical visualisation of the stress diagram.

8.3.13 "Predeformation info" menu item

You will see information on predeformations.

8.3.14 "Predeformation preferences" menu item

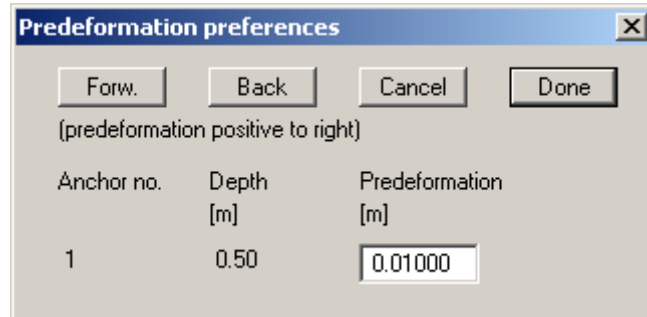
If predeformations from previous construction phases must be taken into consideration as boundary conditions, the necessary settings are made here.



Predeformations can be defined in two different ways:

- **Manual entry of predeformations**

If you know the size of the predeformations, select "**Consider manually entered predeformations**", and then click the "**Edit predeformations**" button.



In the dialog box that appears you can enter a predeformation for every currently defined anchor.

- **File entry of predeformations**

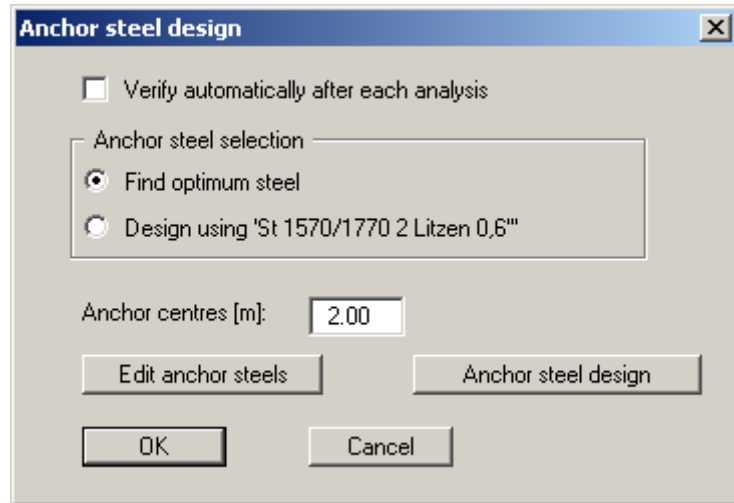
Alternatively, predeformation data can be automatically imported from an existing **GGU-CANTILEVER** file. Select "**Consider predeformations imported from file**" and "**Browse and select file name**". A dialog box will open for selecting the appropriate file, which must have been saved with the results of the analysis; otherwise you will receive an error message. The program imports the appropriate data from this file. The file name is then shown on the button and below this the project identification of the imported file.

You can view the loaded values by clicking "**Edit predeformations**". If you need to alter the height of the anchors during further processing, these values will be automatically adjusted. This means that whenever you begin an analysis, the selected file will be loaded and the values recomputed. You may also want to recompute the current system in the course of a later session, although the file with the predeformation data has altered in the meantime. In this case, the new predeformation data will be loaded automatically.

In the bottom section of the dialog box you can choose whether or not to have the loaded predeformation data shown on screen. If you choose to have it shown, the system graphics show the system's bending line with predeformations.

8.3.15 "Anchor steel design" menu item

Using this menu item you can specify preferences for designing the anchor steel and carry out the design for an analysed system by means of the "**Anchor steel design**" button. If you activate the "**Verify automatically after each analysis**" check box, you will be automatically presented with a query for anchor steel design when analysis is complete.



You can choose to design with a specified anchor steel or to search for the optimum steel from a list of existing steels.

For an analysis using *global safety factors* you must then enter the factor for the anchor steels which for active earth pressure = 0.0 and for loading solely by at-rest pressure = 1.0. Intermediate values can be linearly interpolated. FOS equals 1.0 for prestressed anchors.

When using the *partial safety factors* only the anchor centres still need to be defined, as the other data is taken into consideration via the partial factor for permanent actions resulting from at-rest earth pressure.

Using the "**Edit anchor steels**" button you arrive at a dialog box listing the existing anchor steels with their names and the allowable anchor forces for active and at-rest earth pressures. By marking the selection check box in front of the desired anchor steel and clicking the "**Selected steel as design steel**" button, it can be transferred to the above dialog box. New anchor steels can be added to the list after clicking "**x anchor steel(s) to edit**".

The "**Anchor steel design**" button only appears if the system is already analysed!

8.4 System menu

8.4.1 "Info" menu item

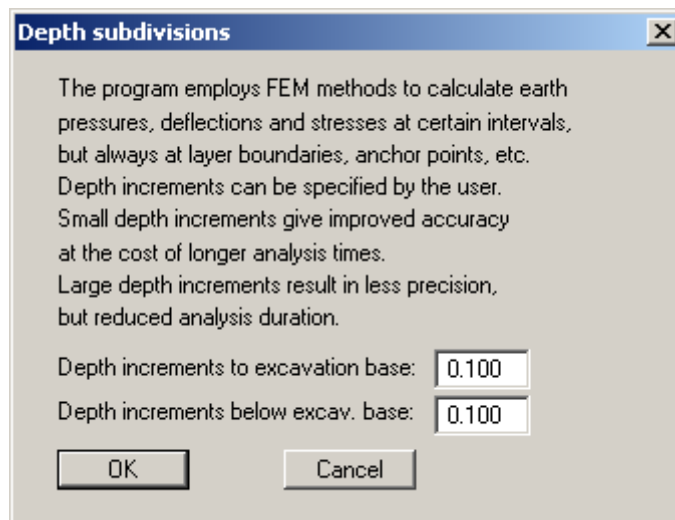
You will see information on the current system in a message box.

8.4.2 "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 a message 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.

8.4.3 "Depth subdivisions" menu item

GGU-CANTILEVER uses the finite element method, which requires the system to be divided into a number of finite elements (rods) (see Section 7.15). You can specify the size of these depth increments for the region above and below the excavation base.



8.4.4 "Analyse" menu item

8.4.4.1 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 Sections 8.4.4.2 to 8.4.4.4 for descriptions). You can also initiate the analysis using the [F5] function key and see the same start dialog box. The available options and check boxes vary depending on the analysis principles adopted.

Analysis options

Type of toe pivot

- Toe is fully fixed
- Toe freely pivoted
- Base is embedded

Subgrade reaction modulus k_s [kN/m²]:

1.00E+4

Equivalent wall inclination

- Inclined
- Vertical

Use active wall friction angle

Type of redistribution

- Do not redistribute
- EAB 1988
- EAB 2012
- Rectangular
- Birectangular
- Triangular
- Trapezoidal
- Quadrilateral
- User-defined

Special preferences

- 2nd order theory
- Design reinforced concrete with at-rest e_p
- Design reinforced concrete using increased active e_p

Factor [-]: 0.50

Determination of spur normal force (ground side = heel)

- = shear force at wall base

OK Cancel

After confirming your preferences start the analysis by clicking "**OK**". The program first performs comprehensive plausibility checks on your input and notifies you of any inconsistencies and special preferences you made.

Once analysis is complete the design dialog box immediately opens. Explanations of design aspects can be found in Section 8.4.6 (menu item "**System/Design defaults**"). The analysis and design results are first displayed in info boxes and then visualised graphically on the screen. The system data can be saved in a file, together with the results (menu item "**File/Save as**", Section 8.1.4).

8.4.4.2 "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 anchors or struts. In such cases, the toe is always *fully fixed*.

In systems with anchors or struts, 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.

8.4.4.3 "Type of redistribution" group box

In the "**Type of redistribution**" group box the following options are available:

- **Do not redistribute**
The analysis is performed using classical earth pressure redistribution.
- **"EAB 1988"** and **"EAB 2012"**
Redistribution diagrams are given as a function of anchor positioning in EAB 1988 and EAB 2012. The program selects the appropriate redistribution from these diagrams. If no agreement is found an appropriate error message is shown.
- **"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 (e_{ah0}/e_{ahu}), as well as depth of the subdivision x , can be specified.

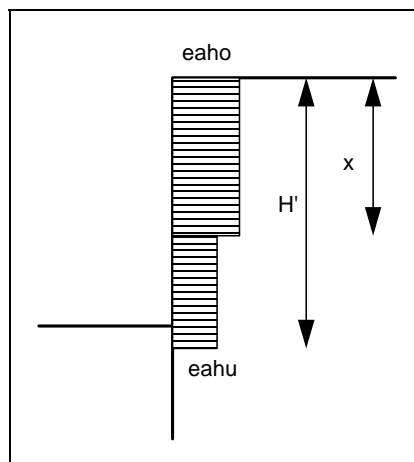


Figure 22 Birectangular earth pressure redistribution

- **"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.

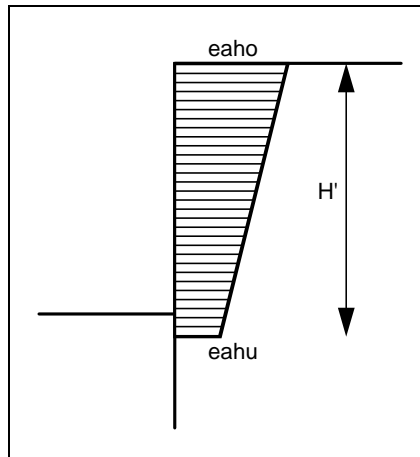


Figure 23 Earth pressure redistribution in a trapezoidal

- **"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 base of the wall is defined by the ratio e_{aho}/e_{ahu} .

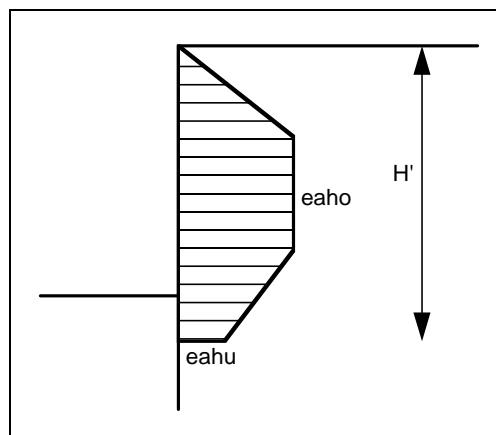


Figure 24 Earth pressure redistribution in a quadrilateral

- **"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	Top of wall	1.0000
2	1.0000	3.0000
3	3.0000	3.0000
4	4.5000	6.0000
5	5.5000	6.0000
6	Wall toe	2.0000

You can define a number of depths between the top of the wall and the wall toe, 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:

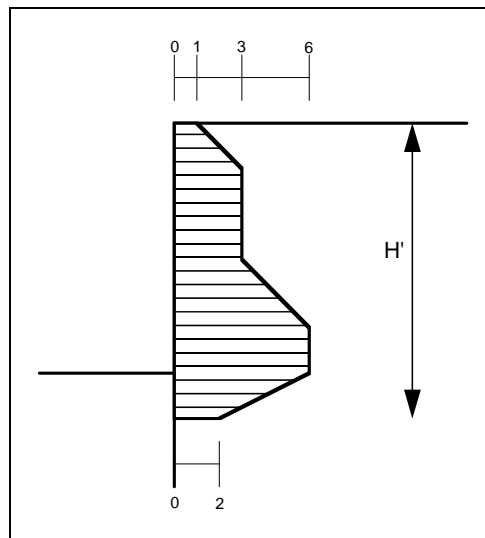


Figure 25 User defined earth pressure redistribution

8.4.4.4 "Special preferences" group box

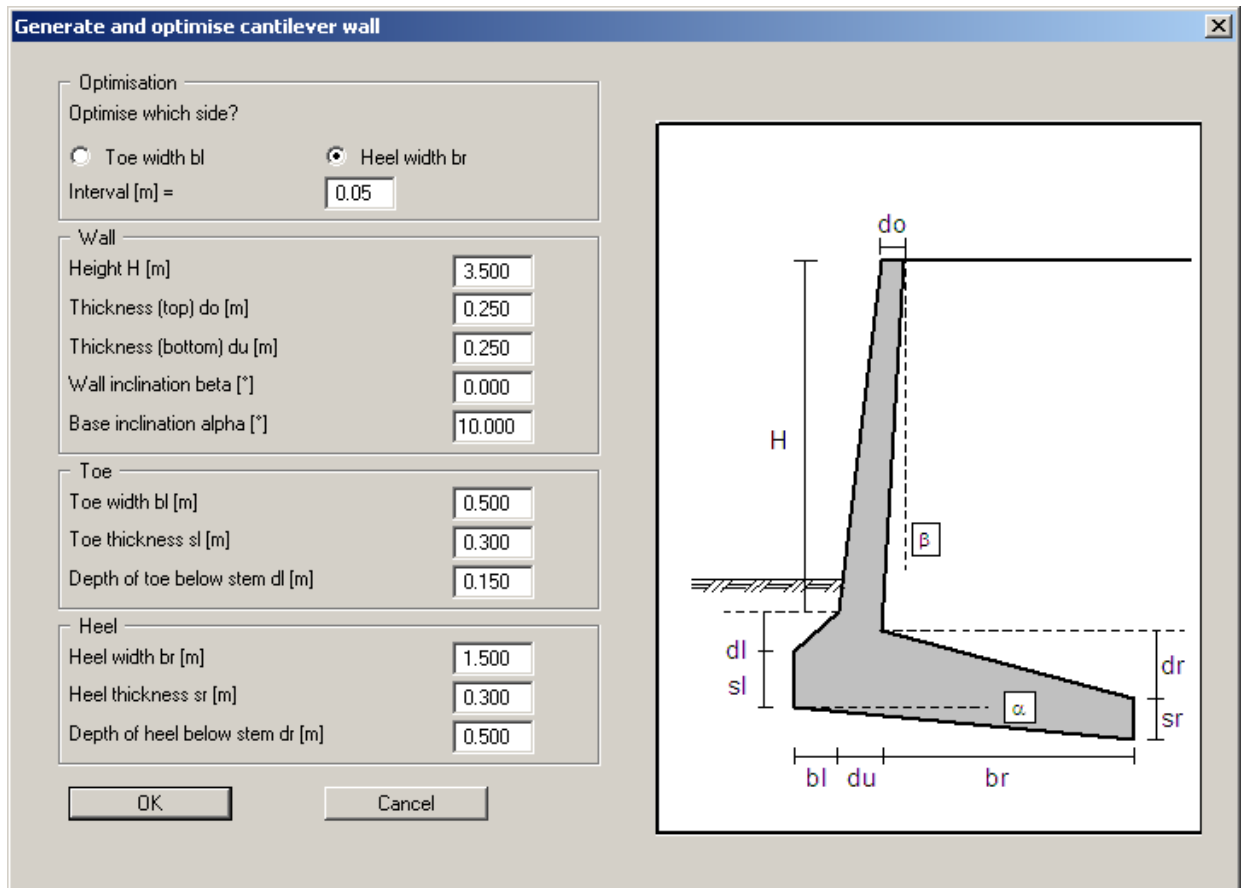
In the **"Special preferences"** group box it is possible to specify that analysis be performed using **"2nd order theory"**. To do this, the program must perform an iteration process, which prolongs analysis.

It is usual to design the reinforced concrete using the at-rest earth pressure. In addition, you can design the reinforced concrete using increased active earth pressure. Wall stability analyses (bearing capacity, sliding, ...), in contrast, are generally performed using the active earth pressure.

8.4.5 "Optimise" menu item

The geometry of the cantilever wall can be optimised via this menu item or by pressing [F8].

The analysis dialog box opens first, allowing the required earth pressure redistribution to be selected (see "System/Analyse" menu item, Section 8.4.4). The following dialog box then opens:



The existing cantilever wall geometry can be viewed and edited by entering data directly into the input boxes. Specify whether the toe or heel (left and right spurs) are to be optimised in the upper group box. Enter an appropriate interval.

Once optimisation is complete the optimum value is shown in a message box and can be selected for further use or rejected. If this value is accepted the program continues the analysis and a further dialog box opens for reinforced concrete design (see Section 8.4.6).

8.4.6 "Design defaults" menu item

This menu item can be used for reinforced concrete design using different settings for a previously analysed system. The following dialog box opens:

Reinforced concrete design to EC 2

Concrete/Steel
Concrete C 30/37
Steel BSt 500/550

Partial safety factors for materials
γ (Concrete) 1.50
γ (Steel) 1.15

Reinf. spacing to structural element boundary
d1 [m] (wall) 0.050
d1 [m] (toe and heel) 0.050

Minimum longitudinal reinforcement
 not considered

Crack width verification
 Analyse wall
 Analyse toe and heel

Shear reinforcement
Message 'Shear reinforcement unnecessary' displayed if $Q_d < VR_{d,c}$
Otherwise, minimum reinforcement displayed. see EC 2 Section 6.2.2
 For wall shear verification
 For toe and heel shear verification

Type of output table for graphics legend
Only reinforcement
(comprehensive data in output table)

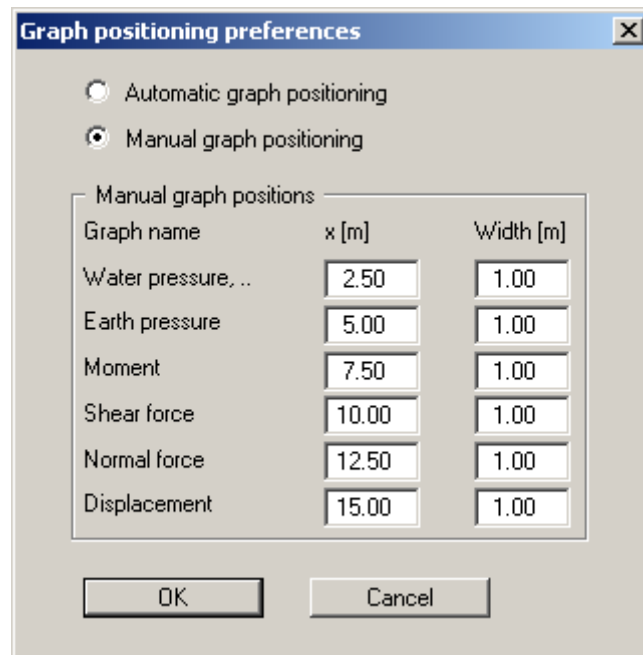
OK Cancel

The spacing of the reinforcement for the wall, toe and heel is defined, besides the concrete and steel qualities. By activating the appropriate check boxes it is possible to perform crack width verification and define preferences for shear reinforcement output for the stem, toe and heel.

"Only reinforcement" is the default setting to stop the design legend from appearing too large in the system visualisation. However, all results are shown in the output table (see menu item **"File/Print output table"** in Section 8.1.5).

8.4.7 "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.

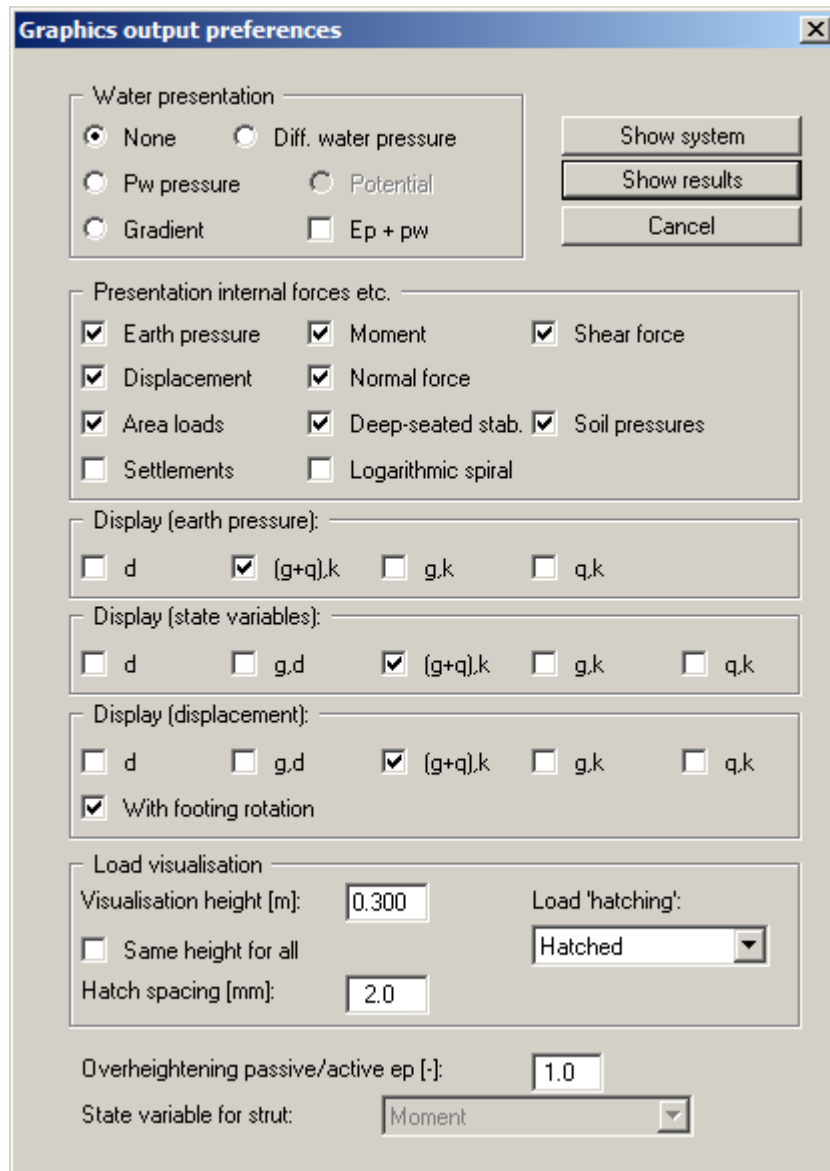


The graphs will then be shown at position "**x**" (central) with the specified "**Width**".

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

8.4.8 "Graphics output preferences" menu item

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



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

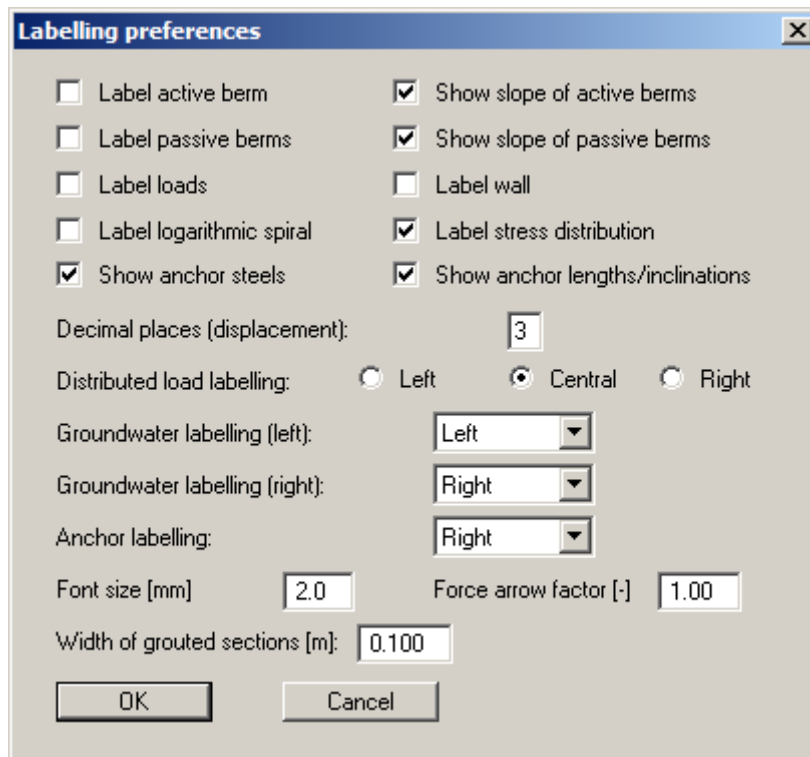
When analysing using *partial safety factors* you will also see the group boxes "Display (earth pressure):", "Display (state variables):" and "Display (displacement):", in which you can activate display of the permanent (g) and/or live loads (q). In addition, the design values (d) can be displayed.

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

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

8.4.9 "Labelling preferences" menu item

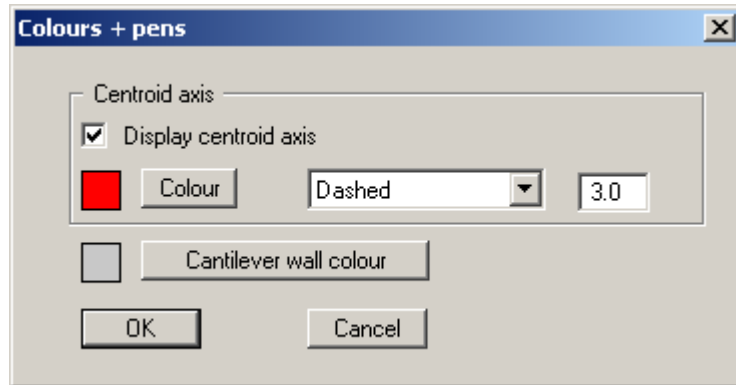
This menu item allows you to specify labelling preferences for the system visualisation and the result graphics.



In the dialog box, you activate the required check boxes and select the preferences for alignment or font sizes. In addition, groundwater labelling and the width of the graphical visualisation of any grouted section can be edited.

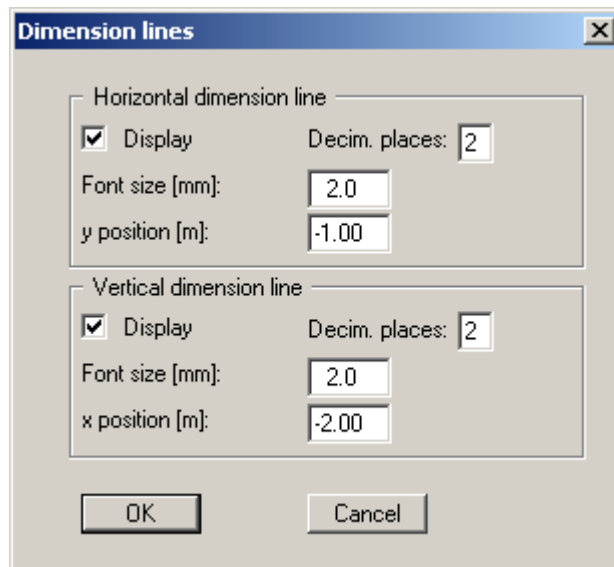
8.4.10 "Wall pens + colours" menu item

The cantilever wall can be visualised together with the centroid axis. The colour and line type of the centroid axis can be defined if the check box is activated. The cantilever wall colour can be altered to suit requirements using the "**Cantilever wall colour**" button.



8.4.11 "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 cantilever wall 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 cantilever wall. All values are in metres in the scale selected (see the menu item "**Page size + margins/Manual resize (editor)**" in Section 8.7.3). The fastest way to modify the position of the dimension lines is to press the [**F11**] function key and then to pull the legend to the new position with the left mouse button pressed.

8.4.12 "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.

8.4.13 "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.

8.5 Evaluation menu

8.5.1 General notes

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.

8.5.2 "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.

8.5.3 "Main output summary" menu item

A message box appears containing the main system analysis parameters. This function can also be initialized pressing the [F4] function key.

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

8.5.4 "Interim results" menu item

A message box containing the results described in Section 7.1 is displayed. This function can also be accessed by pressing [F6].

8.5.5 "Bearing capacity" menu item

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

8.5.6 "Sliding safety" menu item

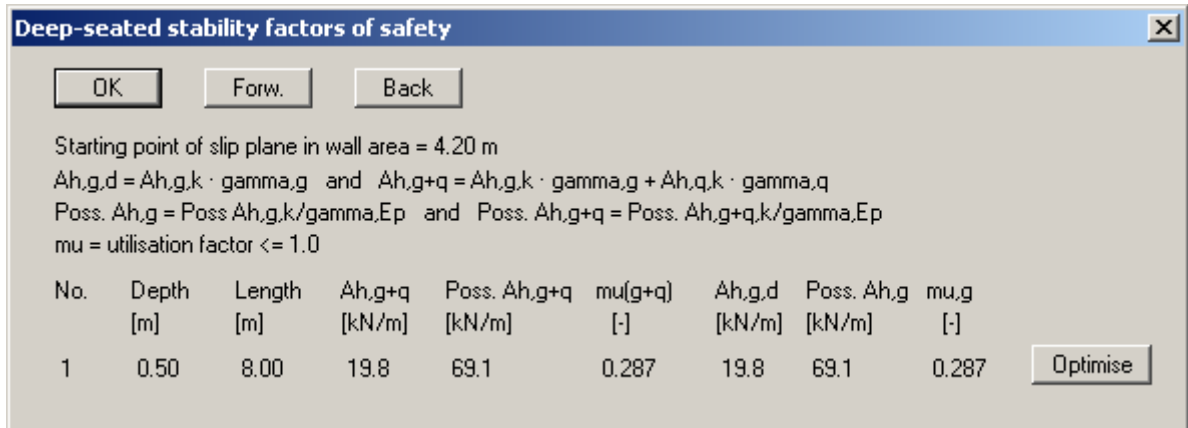
The calculated sliding safety is displayed in a message box.

8.5.7 "Settlements" menu item

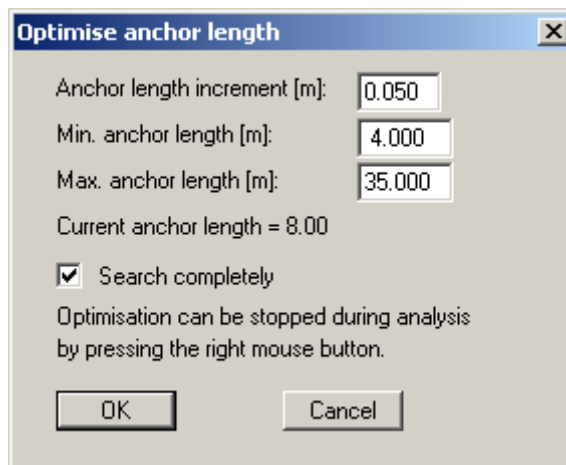
Here you will obtain the results of the settlement analysis displayed in a message box. *CP* means that the settlements are calculated for the characteristic points. This function can also be initialized pressing the [F8] function key.

8.5.8 "Deep-seated stability summary" menu item

After the state variables have been calculated by the program, and depending on the safety factors selected, the deep-seated stability safety factor or the utilisation factor of any anchor is automatically determined.



In addition to the information with regard to the anchors, the critical safety factor or the utilisation factor for deep-seated stability for each anchor is given. In the example shown here, the anchor has an utilisation factor 0.29 which is below the required value of 1.0. Select the "**Optimise**" button to optimise the length of the anchor.



By activating the "**Search completely**" button the optimum is searched for between the minimum and the maximum anchor length (*recommended setting!*). Otherwise, an optimum is searched for based on the current neighbouring anchor lengths. Because interaction occurs between anchors if more than one anchor is involved, it is possible for other anchors to fall below the required safety factor during the optimisation. These anchors must then be *subsequently optimised*.

8.5.9 "Hydraulic heave FOS summary" menu item

You obtain all data relating to verification of hydraulic heave safety displayed in a message box (see Section 7.25).

8.5.10 "Buoyancy FOS summary" menu item

You obtain all data relating to verification of buoyancy safety displayed in a message box (see Section 7.26).

8.5.11 "Anchor and strut summary" menu item

Here the design values for all the system's anchors and struts are shown in a message box.

8.5.12 "Heave of anchor soil" menu item

This menu item displays the results in a message box. Verification must be activated, of course, (see menu item "**Editor 1/Deep-seated stability/Heave of anchor soil**", Section 8.2.18).

8.5.13 "EQU analysis" menu item

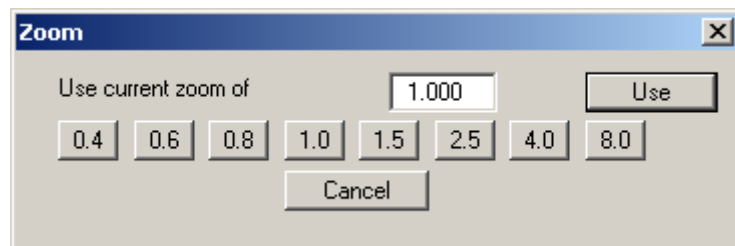
All the necessary data on the analysis of the static equilibrium limit state are displayed in a message box.

8.6 Graphics preferences menu

8.6.1 "Refresh and zoom" menu item

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

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



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

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

8.6.2 "Zoom info" menu item

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

8.6.3 "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.

8.6.4 "Pen colour and width" menu item

In order to enhance the clarity of the graphics you can edit the pen settings for various graphic elements (e.g. 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.

8.6.5 "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, 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.

8.6.6 "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-CANTILEVER.alg_wst" 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.



"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**" in 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 input or change or movement of graphical elements (e.g. made using [F11]) can be undone.

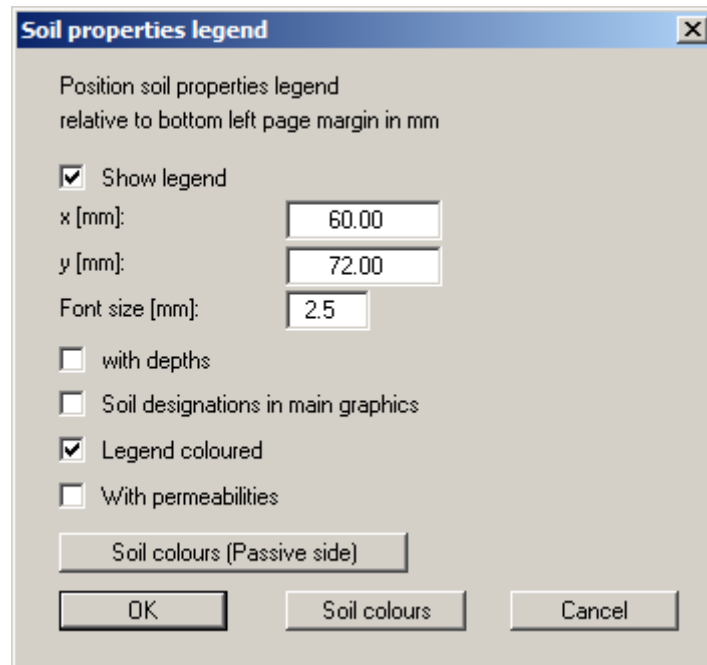


"Restore "

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

8.6.7 "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.



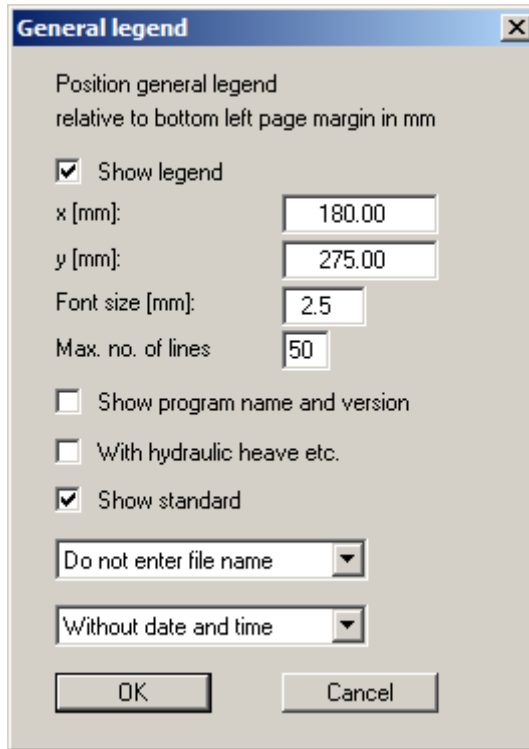
You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**".

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.
- "**With permeabilities**"
The k-values of the individual soil layers will be shown in the soil properties legend.
- "**Soil colours**"/"**Soil colours (Passive side)**"
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.

8.6.8 "General legend" menu item

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



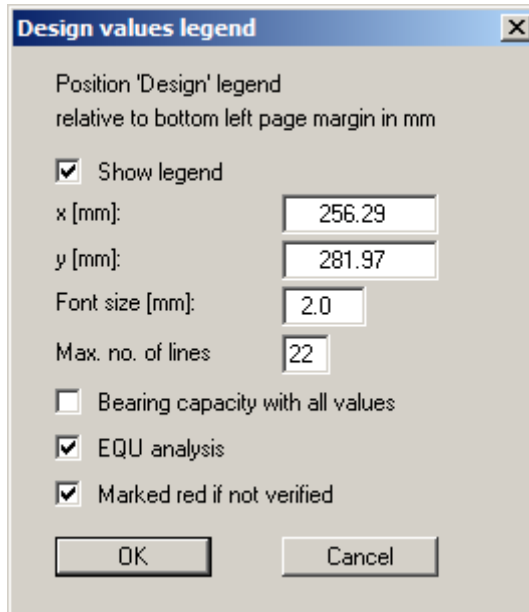
You can define and edit the position of the legend using the values "**x value**" and "**y value**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used.

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*, as well as the hydraulic heave if the appropriate check box is activated.

8.6.9 "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.

The number of values displayed in the graphical design legend can be influenced using the lower check boxes. 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).

8.6.10 "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.

8.6.11 "Move objects" menu item

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

If "**Manual graph positioning**" has been selected in the "**System/Graph positioning preferences**" menu item (see Section 8.4.7), the result graphs can also be repositioned using this function.

8.6.12 "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-CANTILEVER.alg_wst**" 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**".

8.6.13 "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.

8.7 Page size + margins menu

8.7.1 "Auto-resize" menu item

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

8.7.2 "Manual resize (mouse)" menu item

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

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

8.7.3 "Manual resize (editor)" menu item

You can alter the current 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 8.7.6).

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

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

8.7.4 "Zoom" menu item

You can enlarge or reduce the graphical representation by a fixed value by entering the required factor in the dialog box.

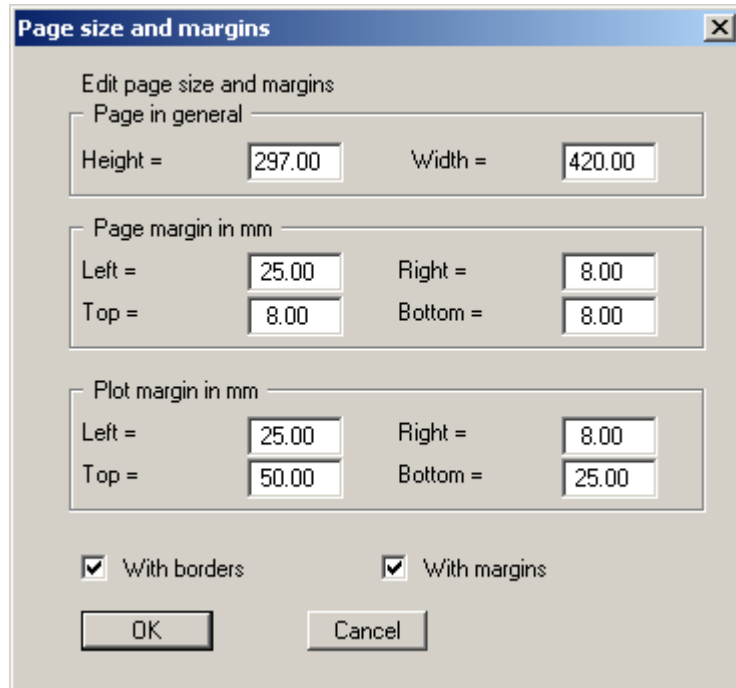
8.7.5 "Font size selection" menu item

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

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

8.7.6 "Page size and margins" menu item

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



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

8.7.7 "Undo" menu item

If you have carried out any changes to dialog boxes or moved objects to a different position on the screen after selecting the "**Graphics preferences/Move objects**" menu item or using the [F11] function key, this menu item will allow you to undo the movements. This function can also be reached by using the key combination [Alt] + [Back] or the appropriate tool in the toolbar (see Section 8.6.6).

8.7.8 "Restore" menu item

When this menu item is selected the last change made in a dialog box or the last change in the position of objects, which you undid using the menu item "**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 8.6.6).

8.7.9 "Preferences" menu item

You can activate or deactivate the undo functions.

8.8 Info menu

8.8.1 "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-CANTILEVER**.

8.8.2 "Maxima" menu item

Here you can check the defaults for maximum values.

8.8.3 "Compare earth pressure coefficients" menu item

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

8.8.4 "Help" menu item

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

8.8.5 "GGU on the web" menu item

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

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

8.8.6 "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.

8.8.7 "What's new?" menu item

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

8.8.8 "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 dialogs, 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

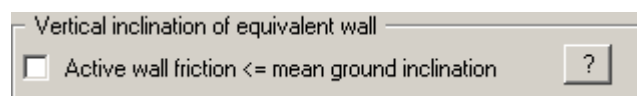
9.1 "?" buttons

Reading of the manual can mostly be dispensed with, because

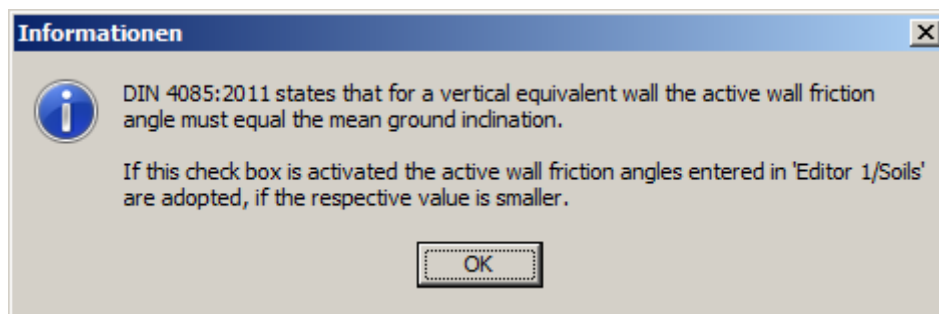
"?" buttons 

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/Active earth pressure**" dialog box:

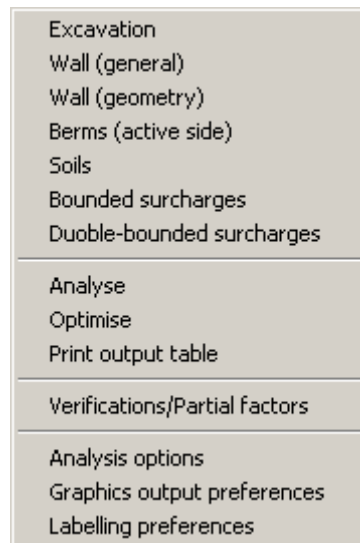


If you click on the question mark, you will see the following message box:



9.2 Keyboard and mouse

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



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

If you double-click in the system graphics with the left mouse button you will see a message 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.
- [F4] opens the menu item "Evaluation/Main output summary".
- [F5] opens the menu item "System/Analyse".
- [F6] opens the menu item "Evaluation/Interim results".
- [F7] opens the menu item "Evaluation/Bearing capacity".
- [F8] opens the menu item "System/Optimise".
- [F9] opens the menu item "Page size + margins/Auto-resize".
- [F11] activates the menu item "Graphics preferences/Move objects".

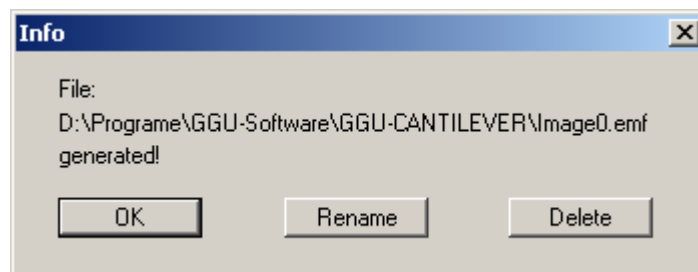
9.4 "Copy/print area" icon

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

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

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

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



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

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

10 Index

2

2nd order theory, select for analysis 85

A

Absolute heights, use 13, 47
Action boundary conditions, define 73
Action boundary conditions, equivalent for
 prestressing 39
Active earth pressure, analysis principles 25
Active earth pressure, edit preferences 60
Active earth pressure, select for analysis 59
Anchor length, optimise 94
Anchor steel design 80
Anchor wall 74
Anchors, define 74
Anchors, define prestressing 76
Anchors, display design values in
 a message box 95
Angle of wall friction, enter 58
Area loads, consider in analysis 32
Area loads, define 69
Area loads, to DIN 4085 33, 62
At-rest earth pressure, coefficient to
 DIN 4085 25
At-rest earth pressure, keep constant 62
At-rest earth pressure, select for analysis 59

B

Bearing capacity failure plane, determine 40
Bearing capacity safety, analysis principles 40
Bearing capacity safety, display analysis
 results in a message box 93
Bending line, display with predeformations 79
Berms, activate labelling 90
Berms, define on active side 57
Berms, define on passive side 57
Bounded surcharges, consider in analysis .. 34, 36
Bounded surcharges, define 71
Buoyancy safety, analysis using global
 safety factors 43
Buoyancy safety, analysis using partial
 safety factors 44
Buoyancy safety, display analysis results
 in a message box 95

C

Cantilever wall, define colour 91
Cantilever wall, define geometry 16
Cantilever wall, define loads 15
Cantilever wall, define rotation/displacement .. 73
Cantilever wall, optimise geometry 86
Caquot/Kerisel, passive ep analysis 26
Caquot/Kerisel, select for analysis 61

Centroid axis/line, activate display 91
Characteristic point, display settlements 93
Characteristic point, limiting depth
 determination 77
Cholesky 37
Clipboard 54
CodeMeter stick 10
Cohesion coefficient, determination methods... 25
Cohesion, enter 58
Colour, define for cantilever wall 91
Colour, define for soils 99
Colour, switch on/off 98
Colour/hatching, define for loads 90
Colour/pen, define for centroid axis/line 91
Colour/pens define for graphical elements 97
Compaction earth pressure, adopt to
 DIN 4085-2011/Franke 70
Company letterhead, add via Mini-CAD 97
Concrete, values for structural analysis 15
Constrained modulus, for
 settlement analysis 42
Context menu, open 108
Coordinates, alter via editor 103
Coordinates, alter with mouse 103
Coordinates, optimise/reset 103
Coordinates, save/load 103
Copy/print area 54, 98, 110
Crack width, activate verification 87
Cutting borders, switch on/off 104

D

Dataset description, display 100
Dataset description, enter 47
Deep-seated stability, define intersection
 with wall 68
Deep-seated stability, safety/utilisation
 factor for anchors 94
Deep-seated stability, verification principles... 45
Delete, soil layers 58
Design, display results in legend 101
Design, edit defaults 87
Design, principles 38, 46
Dimension lines, define 91
DIN 4017 40
DIN 4019 42
DIN 4085, consider in analysis 25, 26
DIN 4085, select for analysis 60, 61
Displacement boundary conditions, define 73
Displacement, define for cantilever wall 73
Distributed load, define 56
Double-bounded surcharges, consider
 in analysis 35, 36
Double-bounded surcharges, define 72
Drawing area, define 104
DXF file, export 54
DXF file, import 7

E

Earth pressure coefficients, be calculated.....	106
Earth pressure coefficients, calculate	62
Earth pressure coefficients, user-defined.....	62
Earth pressure redistribution, display in a message box	93
Earth pressure redistribution, to EAB.....	83
Earth pressure redistribution, user-defined.....	85
Earth pressure, preferences for active.....	60
Earth pressure, select for analysis.....	59
Earth quake, as design situation to EC 7	66
EC 7, load cases description	66
Editor window, output table	52
Elevation head, for potential definition	29
EMF format	54
Equation solution to Cholesky.....	37
Equivalent earth pressure coefficient, activate use	60
Excavation base, define	56
Excavation, change alignment.....	47

F

File, display name in legend.....	100
Finite element module, for water pressure determination	29
Flow force, characteristic	43
Flow force, for hydraulic heave safety	42
Flow force, influence on earth pressure.....	63
Font selection	96
Font size, define for drawing elements.....	103
Font size, design legend.....	101
Font size, dimension lines	91
Font size, general legend	100
Font size, settlement legend.....	102
Font size, soil properties legend	99
Font size, system/result graphics	90
Footer, output table.....	50
Footing length, define.....	15
Force boundary conditions, equivalent for prestress	39, 76
Friction angle, enter.....	58
Function keys	109

G

General page information, add via Mini-CAD	97
General stability, verification using file export	42, 53
GGUCAD file, export.....	54
GGUMiniCAD file, export.....	55
GGU-STABILITY file, export	53
Global safety factors/verifications define	64
Graph positions, alter via numerical input.....	88
Graph positions, alter with mouse	102
Graphics, add via Mini-CAD.....	97
Groundwater level, enter	56
Groundwater, activate labelling.....	90
Groundwater, consider storeys/ confined aquifers	29
Groundwater, define colour	97
Groundwater, enter additional potentials.....	76

Grouted section, enter length for graphics.....	74
Grouted section, enter width for graphics.....	90

H

Hatching, define for loads	90
Header CAD, use.....	97
Header, output table.....	50
Heave of anchor soil verification, present results in a message box	95
Heave of anchor soil, activate analysis.....	68
Hydraulic gradient, calculation.....	29, 30
Hydraulic gradient, determine for analysis.....	63
Hydraulic heave safety, analysis using global safety factors	42
Hydraulic heave safety, analysis using partial safety factors	43
Hydraulic heave safety, display analysis results in a message box	95
Hydraulic heave, show results in legend	100

I

Increased active earth pressure, coefficient	26
Increased active earth pressure, select for analysis.....	59
Installation	10
Interim results, display in a message box	93

K

Knowledge Base, access.....	106
-----------------------------	-----

L

Labelling, system/result graphics	90
Language preferences	10, 106
Lateral pressures, define	69
Layout, output table.....	50
Legends, move with mouse	102
Licence protection	10
Limiting depth, defining methods.....	42
Limiting depth, select determination	77
Line loads, define	69
Line loads, perpendicular to wall axis	33
Live load, consider distributed load as	56
Load cases, adopt in accordance with DIN 1054-2010/EC 7	66
Load cases, adopt in accordance with ÖNORM EN 1997-1	66
Load concentration factor, consider.....	33
Load concentration factor, define	62
Loads, edit graphical output preferences	90
Logarithmic spiral, display below wall.....	89
Logarithmic spiral, in bearing capacity failure plane.....	40

M

Manual, open as PDF file	106
Maximum values, display for system	106
Mean friction angle.....	40
Mean soil properties	40
Metafile, export	54

Mini-CAD file, export	55
Mini-CAD, use	97
Mouse click functions	108
Mouse wheel functions	108

N

Navigation, output table	98
--------------------------------	----

O

Objects, move with mouse	102
ÖNORM EN 1997-1, select partial safety factors	66
Output table, change to system graphics	98
Output table, edit output as ASCII	52
Output table, edit output as graphics	50
Output table, navigation	51
Output table, select output format	49
Overburden stress, consider in settlement analysis	77
Overturning analysis, deactivate	47
Overturning stability, analysis principles	39

P

Page format, define	104
Page margins, switch on/off	104
Page section, copy/print	98, 110
Pagination, automatic	51, 52
Partial factors, define	65, 66
Partial safety factors, select from ÖNORM EN 1997-1	66
Passive earth pressure, analysis methods for coefficients	26
Passive earth pressure, specify calculation	61
Pen preferences, edit for graphical elements	97
Permeabilities, consider in analysis	25
Permeabilities, display in legend	99
Permeabilities, enter	58
Permeabilities, using flow conduits	28
Piling Handbook, effect on earth pressure	31
Plausibility checks, show during analysis	83
Plausibility checks, view results	81
Plot margins, define	104
Potential, definition	29
Preconsolidation, consider in settlement analysis	77
Predeformations, define	79
Presentation height, define for loads	90
Prestressing, consider in analysis	39
Prestressing, define for anchors/struts	76
Print, graphics	53
Print, output table	54
Print, section	54, 98, 110
Print, several files	55
Printer preferences	52, 53
Program, display name in legend	100
Program, save/load preferences	102
Program, show improvements	106
Program, show information	106
Project data, add via Mini-CAD	97
Project identification, display	100
Project identification, enter	47

R

Redistribution figures, available	39
Redistribution figures, select	83
Reinforced concrete design, principles	46
Reinforced concrete, activate design using at-rest earth pressure	85
Reinforced concrete, edit design defaults	87
Representation, change to system graphics	51
Result graphics, display	92
Result graphics, edit preferences	89
Result graphics, label	90
Result graphics, position	88, 102
Results, display for selected depth	93, 108
Rod construction module	37
Rods, define for struts	75
Rods, define for system	81
Rotation, define for cantilever wall	73

S

Safety concept, select	47
Safety factors, define	64
Scale, alter via factor input	103
Scale, alter with mouse	103
Scale, define manually	103
Scale, determine automatically	103
Scroll the screen	108
Seismic loads, consider	67
Settlement analysis, display results in legend	102
Settlement analysis, edit preferences	77
Settlement analysis, limiting depth definition	42
Settlement analysis, subtraction of preconsolidation loading	42
Settlement, display results in a message box	93
Settlements, display share in legend	102
Shear reinforcement, edit output preferences	87
Sliding safety, analysis principles	41
Sliding safety, display analysis results in a message box	93
Slip planes, compound	45
Slope stability application	53
Smart icons, for menu items	98
Soil colours/numbers, activate display	99
Soil designations, activate presentation in system graphics	99
Soil designations, define	58
Soil layers, adapt to absolute heights	56
Soil layers, define/delete	58
Soil layers, maximum number	25
Soil properties, activate separate input active/passive values	48
Soil properties, define	58
Soil properties, display in legend	99
Standard, display in legend	100
Status bar main program, activate	98
Stiffness, wall	37, 38
Stiffnesses, consider in analysis	37, 38
Streck, passive ep analysis	26
Streck, select for analysis	61
Stress analysis, for limiting depth	77
Struts, define	75

Struts, define prestressing	76
Struts, display design values in a message box	95
System properties, display in legend	100
System, analyse	82
System, display	92
System, division into rods	81
System, edit graphics output preferences	89
System, show information	81, 106
System, structural	37

T

Theory of elastic half-space	33
Toolbar, edit for menu items	98
Translation, activate	106
True-type font	96

U

Unit weight, enter	58
User-defined ep coefficients, enter	62
Utilisation factor, buoyancy	44
Utilisation factor, deep-seated stability	94
Utilisation factor, hydraulic heave	43

V

Verifications, activate to DIN 1054 old	64
Verifications, activate to DIN 1054-2010/ EC 7	65, 66
Version number, display in a message box	106
Version number, display in legend	100

W

Wall toe, select bedding conditions	83
Water pressure approach, classical	27
Water pressure approach, recommendation	29
Water pressure approach, select for analysis	63
Water pressure approach, using flow conduits	28
Water pressure, enter additional wp	76
Water pressure, user-defined distribution	29
What you see is what you get	96

Y

Young's modulus, define	15
-------------------------------	----

Z

Zoom factor, define for full-screen display	96
Zoom function, activate	96, 98, 108