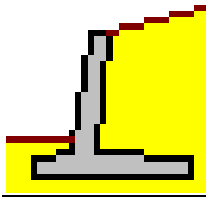


Analysis of cantilever walls

GGU-CANTILEVER

VERSION 5



Last revision:
Copyright:
Technical implementation, layout and sales:

March 2024
Prof. Dr. Johann Buß
Civilserve GmbH, Steinfeld

Contents:

1	Preface.....	7
2	Capabilities	7
3	Licence protection	9
4	Language selection	10
5	Starting the program	10
6	Tips and tricks.....	11
6.1	"?" buttons	11
6.2	Function keys	11
6.3	Keyboard and mouse.....	12
6.4	Calculation functions in input boxes with numbers	14
6.5	"Copy/print area" icon.....	15
7	First steps using a worked example.....	16
7.1	Example system.....	16
7.2	Step 1: Select analysis options	17
7.3	Step 2: Define the excavation.....	18
7.4	Step 3: Define load on cantilever wall	19
7.5	Step 4: Define geometry of cantilever wall	20
7.6	Step 5: Define soils	21
7.7	Step 6: Define type of earth pressure	21
7.8	Step 7: Analyse and design the system.....	22
7.9	Step 8: Optimise system.....	23
7.10	Step 9: Evaluate and visualise the results.....	24
8	Theoretical principles	25
8.1	Cantilever wall	25
8.1.1	Definition.....	25
8.1.2	Determination of wall loads for reinforced concrete design	26
8.2	Soil properties	27
8.3	Active earth pressure.....	27
8.4	At-rest earth pressure.....	27
8.5	Increased active earth pressure	28
8.6	Passive earth pressure.....	28
8.7	Water pressure.....	29
8.7.1	Classical water pressure approach	29
8.7.2	Water pressure approach using flow conduits	30
8.7.3	Recommendation for water pressure approach	31
8.7.4	What is a potential?.....	31
8.8	Berms	33
8.9	Area loads.....	34
8.10	Loads with limited plan dimensions	35
8.11	Bounded surcharges (active side).....	36
8.12	Double-bounded surcharges (active side).....	37
8.13	Bounded surcharges (passive side).....	38
8.14	Double-bounded surcharges (passive side)	38
8.15	Structural system	39

8.16 Design	40
8.17 Action and displacement boundary conditions	40
8.18 Pretensioning	41
8.19 Earth pressure redistribution	41
8.20 Overturning stability	41
8.21 Bearing capacity safety	42
8.22 Sliding safety	43
8.23 General stability	43
8.24 Settlements	44
8.25 Hydraulic heave	44
8.25.1 Hydraulic heave safety using global safety factors	44
8.25.2 Utilisation factor (hydraulic heave) using partial safety factors	45
8.26 Buoyancy	45
8.26.1 Buoyancy safety using global safety factors	45
8.26.2 Utilisation factor (buoyancy) using partial safety factors	46
8.27 Verification of deep-seated stability	46
8.28 Heave of anchor soil	48
8.29 Reinforced concrete design	48
9 Description of menu items	49
9.1 File menu	49
9.1.1 "New" menu item	49
9.1.2 "Load" menu item	50
9.1.3 "Save" menu item	50
9.1.4 "Save as" menu item	50
9.1.5 "Print output table" menu item	51
9.1.5.1 Selecting the output format	51
9.1.5.2 Button "Output as graphics"	52
9.1.5.3 Button "Output as ASCII"	54
9.1.6 "Export to GGU-STABILITY" menu item	55
9.1.7 "Output preferences" menu item	55
9.1.8 "Print and export" menu item	55
9.1.9 "Batch print" menu item	58
9.1.10 "Exit" menu item	58
9.1.11 "1, 2, 3, 4" menu items	58
9.2 Editor 1 menu	59
9.2.1 "Analysis options" menu item	59
9.2.2 "Excavation" menu item	59
9.2.3 "Wall (general)" menu item	60
9.2.4 "Wall (geometry)" menu item	60
9.2.5 "Berms (active side)" menu item	60
9.2.6 "Berms (passive side)" menu item	60
9.2.7 "Soils" menu item	61
9.2.8 "Type of earth pressure" menu item	62
9.2.9 "Active earth pressure" menu item	63
9.2.10 "Passive earth pressure" menu item	64
9.2.11 "At-rest earth pressure" menu item	65
9.2.12 "User-defined earth pressure coefficients" menu item	66

9.2.13	"Groundwater" menu item	66
9.2.14	"Verifications/Safety factors" menu item	67
9.2.15	"Verifications/Partial factors" menu item	68
9.2.16	"Buoyancy + Hydraulic heave" menu item.....	69
9.2.17	"Seismic effects" menu item.....	70
9.2.18	"Deep-seated stability/Heave of anchor soil" menu item.....	71
9.3	Editor 2 menu	72
9.3.1	"Lateral pressures" menu item	72
9.3.2	"Area loads" menu item	73
9.3.3	"Compaction earth pressure" menu item.....	74
9.3.4	"Bounded surcharges" menu item.....	75
9.3.5	"Double-bounded surcharges" menu item	76
9.3.6	"Action boundary conditions" menu item.....	77
9.3.7	"Displacement boundary conditions" menu item.....	77
9.3.8	"Anchors" menu item.....	78
9.3.9	"Struts" menu item.....	79
9.3.10	"Pretensioning" menu item	80
9.3.11	"Groundwater potentials" menu item.....	80
9.3.12	"Settlements" menu item.....	81
9.3.13	"Predeformation info" menu item	82
9.3.14	"Predeformation preferences" menu item	82
9.3.15	"Anchor steel design" menu item.....	83
9.4	System menu	84
9.4.1	"Info" menu item	84
9.4.2	"Special preferences" menu item	84
9.4.3	"Depth subdivisions" menu item.....	84
9.4.4	"Analyse" menu item	85
9.4.4.1	Start dialog box	85
9.4.4.2	"Type of toe pivot" group box	86
9.4.4.3	"Type of redistribution" group box	86
9.4.4.4	"Special preferences" group box	88
9.4.5	"Optimise" menu item.....	89
9.4.6	"Design defaults" menu item	90
9.4.7	"Graph positioning preferences" menu item	91
9.4.8	"Graphics output preferences" menu item	92
9.4.9	"Labelling preferences" menu item.....	93
9.4.10	"Wall pens + colours" menu item	94
9.4.11	"Dimension lines" menu item	94
9.4.12	"Display system" menu item.....	95
9.4.13	"Display results" menu item	95
9.5	Evaluation menu.....	95
9.5.1	General notes	95
9.5.2	"Earth pressure redistribution" menu item	95
9.5.3	"Main output summary" menu item.....	95
9.5.4	"Interim results" menu item	95
9.5.5	"Bearing capacity" menu item	95
9.5.6	"Sliding safety" menu item	95

9.5.7	"Settlements" menu item.....	96
9.5.8	"Deep-seated stability summary" menu item.....	96
9.5.9	"Hydraulic heave FOS summary" menu item	97
9.5.10	"Buoyancy FOS summary" menu item	97
9.5.11	"Anchor and strut summary" menu item.....	97
9.5.12	"Heave of anchor soil" menu item	97
9.5.13	"EQU analysis" menu item	97
9.6	Graphics preferences menu	98
9.6.1	"Refresh and zoom" menu item	98
9.6.2	"Zoom info" menu item	99
9.6.3	"Legend font selection" menu item.....	99
9.6.4	"Pen colour and width" menu item	99
9.6.5	"Mini-CAD toolbar" and "Header toolbar" menu items	100
9.6.6	"Toolbar preferences" menu item	100
9.6.7	"Soil properties legend" menu item	101
9.6.8	"General legend" menu item.....	103
9.6.9	"Design legend" menu item	104
9.6.10	"Settlement legend" menu item.....	105
9.6.11	"Move objects" menu item.....	105
9.6.12	"Save graphics preferences" menu item.....	105
9.6.13	"Load graphics preferences" menu item	105
9.7	Page size + margins menu	106
9.7.1	"Auto-resize" menu item.....	106
9.7.2	"Manual resize (mouse)" menu item.....	106
9.7.3	"Manual resize (editor)" menu item.....	106
9.7.4	"Zoom" menu item.....	106
9.7.5	"Font size selection" menu item.....	106
9.7.6	"Page size and margins" menu item.....	107
9.7.7	"Undo" menu item	108
9.7.8	"Restore" menu item	108
9.7.9	"Preferences" menu item.....	108
9.8	Info menu	109
9.8.1	"Copyright" menu item.....	109
9.8.2	"Maxima" menu item.....	109
9.8.3	"Compare earth pressure coefficients" menu item	109
9.8.4	"Help" menu item	109
9.8.5	"GGU on the web" menu item	109
9.8.6	"GGU support" menu item.....	109
9.8.7	"What's new?" menu item	109
9.8.8	"Language preferences" menu item	109
10	Index.....	110

List of figures:

Figure 1 Illustration of the worked example	16
Figure 2 Base of wall (stem) Example 1	26
Figure 3 Base of wall (stem) Example 2	26
Figure 4 Classical water pressure approach	29
Figure 5 Resultant water pressure	29
Figure 6 Water pressure approach using flow conduits	30
Figure 7 Resultant water pressure with flow conduits	30
Figure 8 Definition of potential	32
Figure 9 Berm on the active side.....	33
Figure 10 Area load.....	34
Figure 11 At-rest earth pressure from area loads.....	35
Figure 12 Bounded surcharge (active side).....	36
Figure 13 Two bounded surcharges.....	37
Figure 14 Double-bounded surcharge	37
Figure 15 Bounded surcharge (passive side).....	38
Figure 16 Logarithmic spiral.....	42
Figure 17 Horizontal equivalent plane	43
Figure 18 Compound "deep slip planes"	46
Figure 19 Compound "deep slip plane", which is not investigated.....	47
Figure 20 Sign rule for spur and wall moments with reinforcement.....	48
Figure 21 Birectangular earth pressure redistribution.....	86
Figure 22 Earth pressure redistribution in a trapezoidal	87
Figure 23 Earth pressure redistribution in a quadrilateral	87
Figure 24 User defined earth pressure redistribution.....	88

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 6.1).

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

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

2 Capabilities

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 40 berms on the active earth pressure side
- Up to 40 berms on the passive earth pressure side
- Analysis with active and increased active earth pressure and with at-rest earth pressure
- Coefficients of active earth pressure to DIN 4085
- Coefficients of passive earth pressure to DIN 4085, Streck, Caquot/Kerisel
- Additional active and passive ep calculation to Culmann is possible.
- 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 pretensioning 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 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

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

We use 3 alternative CmContainer types, which are stored on your PC, the so-called *CodeMeter licence server*:

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

For the CodeMeter protection system, the driver software *CodeMeter Runtime Kit* must be installed on your PC (CodeMeter licence server). The GGU program checks at start-up and during running time whether a licence is present on a CmContainer.

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) (see Section 9.8.8).

5 Starting the program

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

- File
- Info

By going to the "**File**" menu, a previously analysed system can be loaded by means of the "**Load**" menu item, or a new one created using "**New**". After clicking the "**New**" menu item a dialog box opens for specifying general preferences for your new system (see Section 9.1.1). 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.

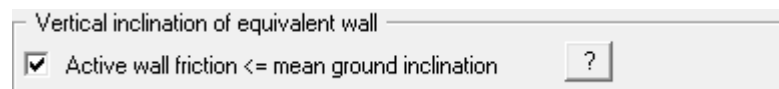
6 Tips and tricks

6.1 "?" buttons

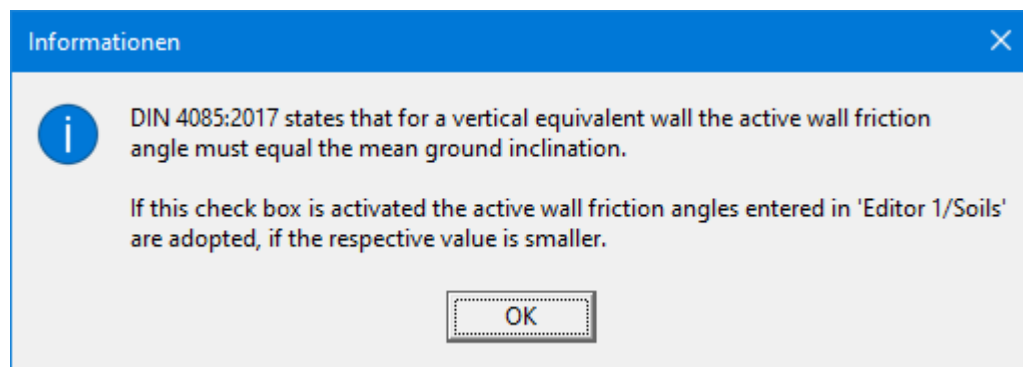
Reading of the manual can mostly be dispensed with, because

"?" buttons 

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



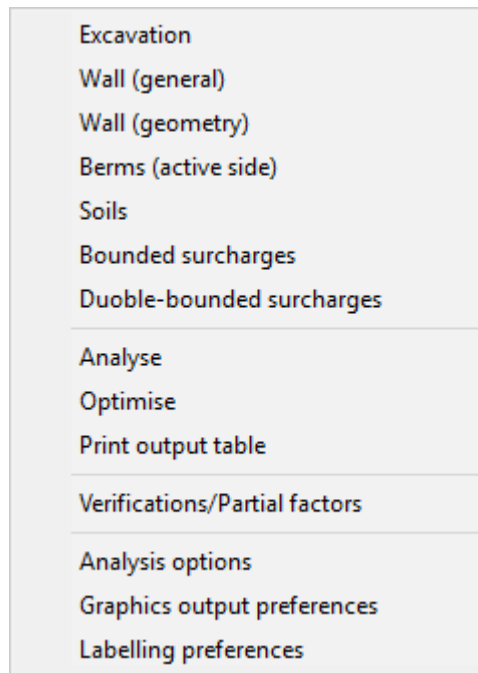
6.2 Function keys

Some of the function keys are assigned program functions. The allocations are noted after the corresponding menu items. The individual function key allocations are:

- [Esc] refreshes the screen contents and sets the screen back to the given format. This is useful if, for example, you have used the zoom function to display parts of the screen and would like to quickly return to a complete overview.
- [F1] opens the manual file.
- [F2] refreshes the screen without altering the current magnification.
- [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"**.

6.3 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.

In most dialog boxes, the buttons for leaving the box or buttons for decisive functions are marked thickly and can be reached by clicking the **[Enter]/[Return]** key.

In dialog boxes in which you have to make entries, e.g. change soil properties, the quickest way to jump to the next input box is to use the **[Tab]** key. The previous value is marked and can be overwritten directly with the new entry. You do not have to move to the box with the mouse and delete the old entry in advance.

In input boxes with numerical values, you can use the calculation operations listed in section 6.4 to change them.

You can scroll the screen with the keyboard using the cursor keys and the **[Page up]** and **[Page down]** keys. After activating the zoom functions, click on an area of the screen to enlarge or reduce the display by a factor of 2. Alternatively, by clicking and dragging the mouse while holding down the **[Ctrl]** key, you define a window section which is then displayed in full screen. By clicking and dragging the mouse while holding down the **[Shift]** and **[Ctrl]** keys, you zoom into your system graphic, i.e. you change the scale of your system representation.

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

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

If you want to change the system coordinates and the scale of your system using the mouse wheel, activate the "**Mouse wheel operation with global coordinates**" check box in the dialog box of the "**Graphics preferences/Zoom info**" menu item (see Section 9.6.2). If you close the program with this setting, the setting is still activated the next time you start it. You can now use the following mouse wheel functions to change the system:

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


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

You can view the changed system coordinates in the menu item "**Page size + margins/Manual resize (editor)**" and, if necessary, adjust them manually by entering more precise numbers (see Section 9.7.3).

6.4 Calculation functions in input boxes with numbers

Task		Input
Add:		
$5 + 12,18$	→	$5 + 12,18$
Subtract:		
$25,74 - 12,18$	→	$25,74 - 12,18$
Multiply:		
$5,23 \cdot 4,18$	→	$5,23*4,18$
$\pi \cdot 2,5^3$	→	$PI*2,5^3$
Divide:		
$5,23 / 4,18$	→	$5,23/4,18$ or $5,23:4,18$
Exponentiate:		
2^5	→	2^5
Radify:		
Root from 27	→	$w(27)$ or $27^{(1/2)}$
5. root from 81,5	→	$81,5^{(1/5)}$
Sine, cosine, tangent, etc.		
$\sin(32^\circ)$	→	$\sin(32)$
$\cos(5,23^\circ)$	→	$\cos(5,23)$
$\tan(45^\circ)$	→	$\tan(45)$
$\arctan(1,0)$	→	$\text{atan}(1,0) = 45^\circ$
Logarithm naturalis		
$\ln(4,53)$	→	$\ln(4,53) = 1,5107$
Exponential function:		
$e^{1,5107}$	→	$\text{ep}(1,5107) = 4,53$

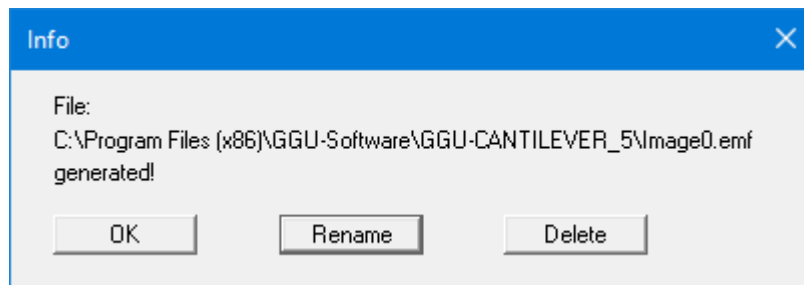
6.5 "Copy/print area" icon

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

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

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

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



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

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

7 First steps using a worked example

7.1 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:

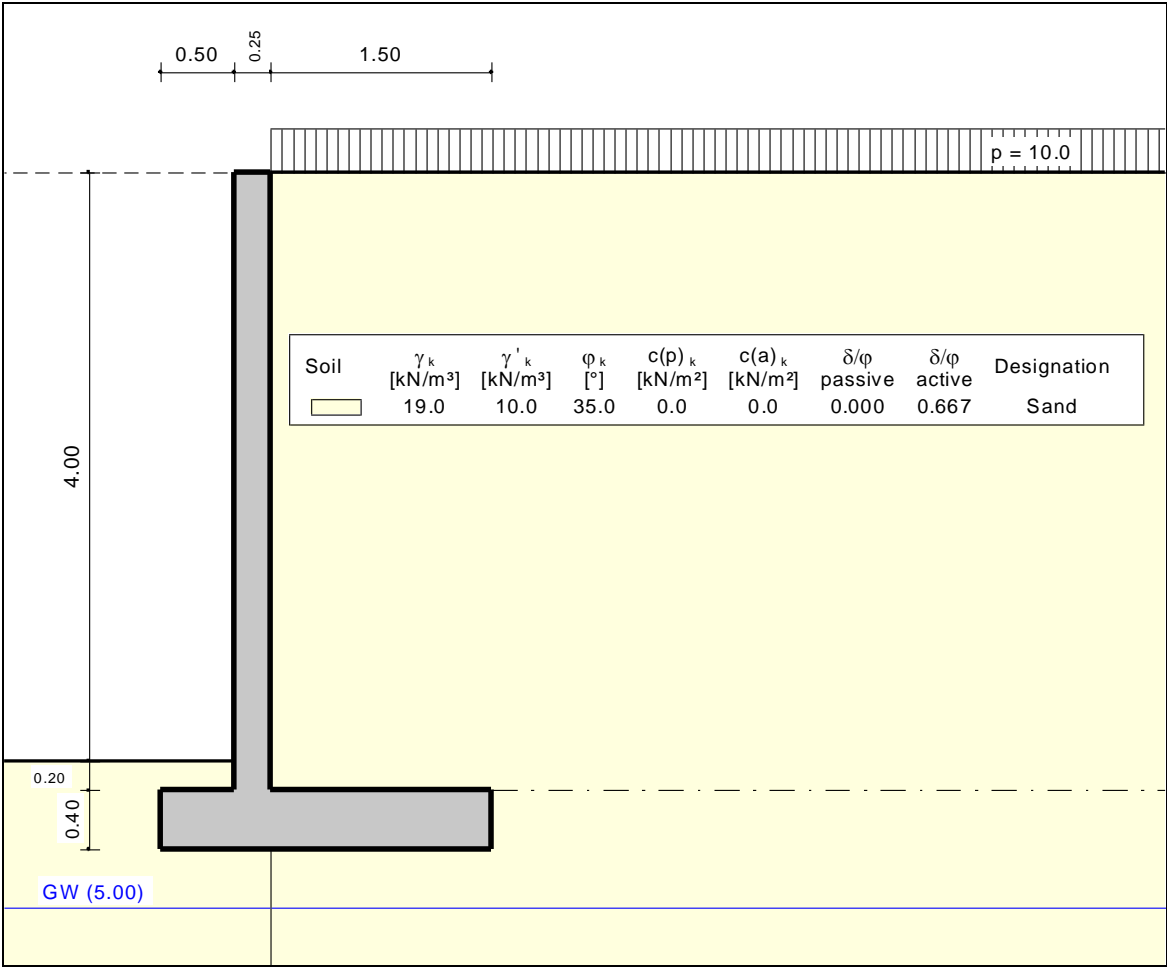


Figure 1 Illustration of the worked example

7.2 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:

Analysis options

Project identification:

Example

Standard:

☒ Partial safety factor concept (EC 7) Info EC 7

☐ Partial safety factor concept (DIN 1054:2005)

☐ Global safety factor concept (DIN 1054 old)

☐ Show excavation on right

☐ Do not investigate overturning

☐ Use absolute heights Design: mNHN

☐ Differentiate active + passive soil properties
(with reference to: phi, gamma + gamma')

"No traffic on spur" load case

☒ Investigate "No traffic on spur" load case ?

Concrete design:

☒ Characteristic normal force ?

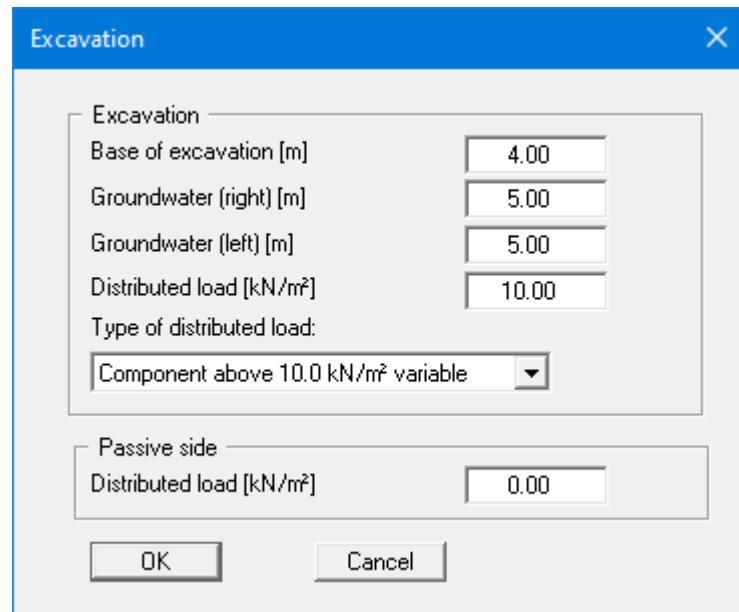
OK Cancel

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 m** (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.

7.3 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" with a blue header bar and a close button (X) in the top right corner. The dialog is divided into two main sections. The first section, titled "Excavation", contains four input fields: "Base of excavation [m]" with a value of 4.00, "Groundwater (right) [m]" with a value of 5.00, "Groundwater (left) [m]" with a value of 5.00, and "Distributed load [kN/m²]" with a value of 10.00. Below these is a dropdown menu labeled "Type of distributed load:" which is currently set to "Component above 10.0 kN/m² variable". The second section, titled "Passive side", contains one input field: "Distributed load [kN/m²]" with a value of 0.00. At the bottom of the dialog are two buttons: "OK" and "Cancel".

Parameter	Value
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
Passive side Distributed load [kN/m²]	0.00

7.4 Step 3: Define load on cantilever wall

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

Cantilever wall

Loads + footing length

	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/m²]:	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

Friction spur

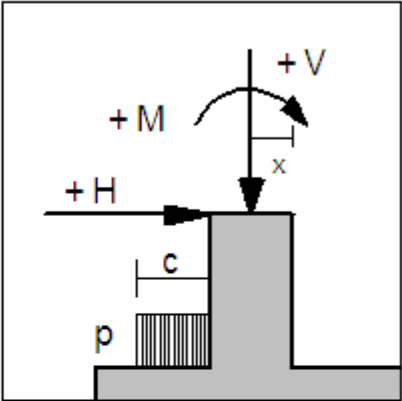
☐ Use friction spur

Depth [m]: 0.40

Thickness [m]: 0.60

Distance from footing edge [m]: 0.00

OK Cancel



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 the "x [m]" value. 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.

Enter the concrete values in the lower section 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.

You can use a friction spur to increase the passive earth pressure and thus increase the sliding safety. With high walls, the active earth pressure is often greater than the passive earth pressure. The effect is then reversed.

7.5 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

Thickness (bottom) for vertical air-side wall

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

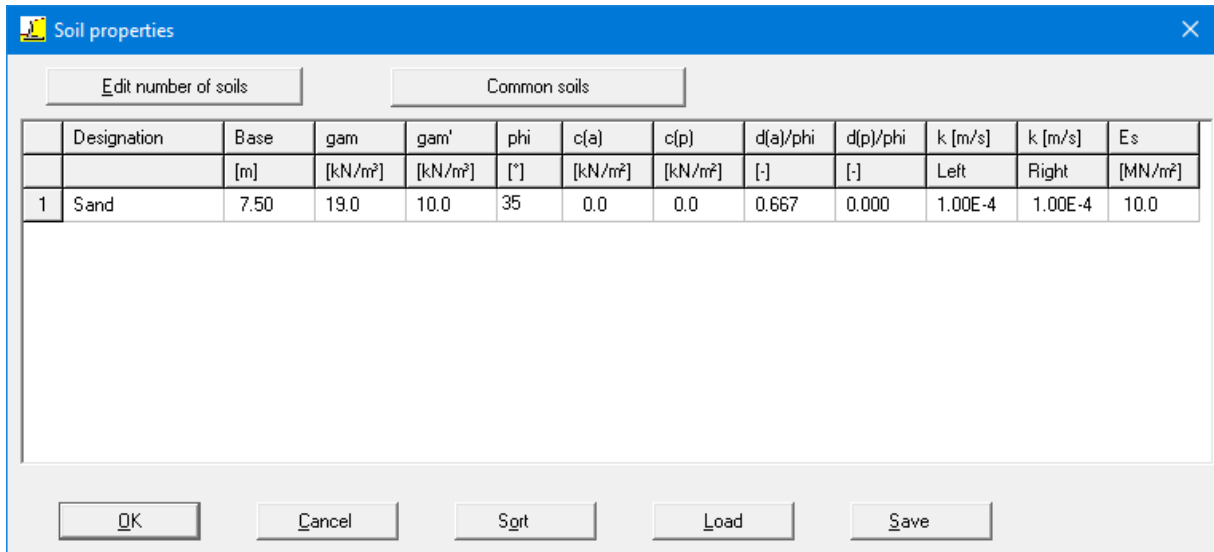
OK Cancel

The diagram illustrates the geometry of a cantilever wall. It shows a vertical wall of height H with a top thickness d_o and a base thickness d_u . The wall is inclined at an angle β from the vertical. The base of the wall is inclined at an angle α from the horizontal. The toe of the wall has a width b_l , a thickness s_l , and a depth d_l below the stem. The heel of the wall has a width b_r , a thickness s_r , and a depth d_r below the stem. The total width of the base is $b_l + b_r$.

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

7.6 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**.



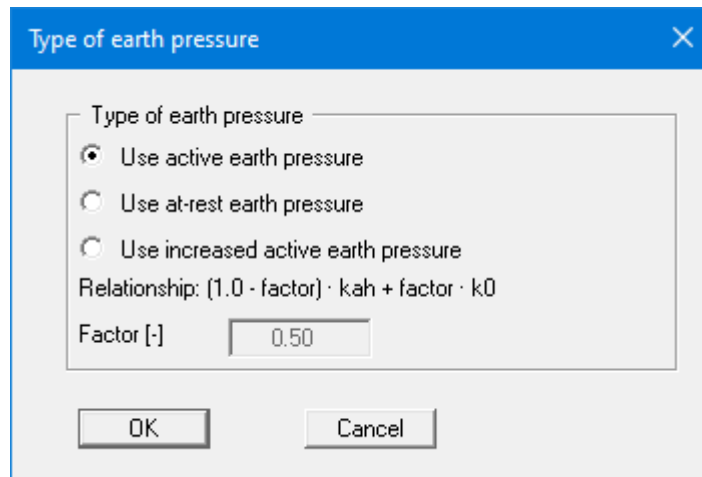
The "Soil properties" dialog box features a blue title bar with a close button. It contains two tabs: "Edit number of soils" and "Common soils". The "Common soils" tab is active, displaying a table with 13 columns: Designation, Base, gam, gam', phi, c(a), c(p), d(a)/phi, d(p)/phi, k [m/s] Left, k [m/s] Right, and Es. The first row of the table is populated with data for "Sand". Below the table is a large empty text area. At the bottom, there are five buttons: OK, Cancel, Sort, 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/m³]	[kN/m³]	[°]	[kN/m²]	[kN/m²]	[-]	[-]	Left	Right	[MN/m²]
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 9.2.7).

7.7 Step 6: Define type of earth pressure

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



The "Type of earth pressure" dialog box has a blue title bar with a close button. It contains a group box labeled "Type of earth pressure" with three radio button options: "Use active earth pressure" (selected), "Use at-rest earth pressure", and "Use increased active earth pressure". Below these options is the text "Relationship: $(1.0 - \text{factor}) \cdot k_{ah} + \text{factor} \cdot k_0$ ". A text field labeled "Factor [-]" contains 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 look at them, in order to familiarise yourself with them.

This completes data input.

7.8 Step 7: Analyse and design the system

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

Analysis options

Type of toe pivot
Toe is fully fixed

Equivalent wall inclination
Vertical ?

☐ Active wall friction angles for concrete wall ?

☐ Active wall friction angles for equivalent wall ?

Type of redistribution

☒ Do not redistribute

☐ EAB 1988

☐ EAB 2012 / 2021

☐ Rectangular

☐ Birectangular Preferences

☐ Triangular Preferences

☐ Trapezoidal Preferences

☐ Quadrilateral Preferences

☐ User-defined Preferences

Special preferences

☐ 2nd order theory

☐ Design reinforced concrete with at-rest ep

☐ Design reinforced concrete using increased active ep

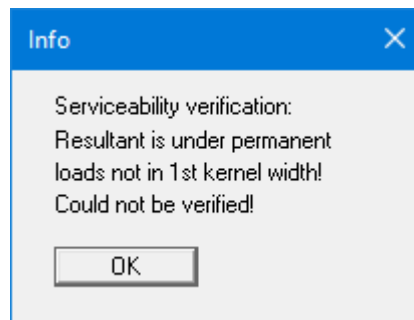
Factor [-] 0.50 ?

Determination of spur normal force (ground side = heel)

☐ = shear force at wall base

OK Cancel

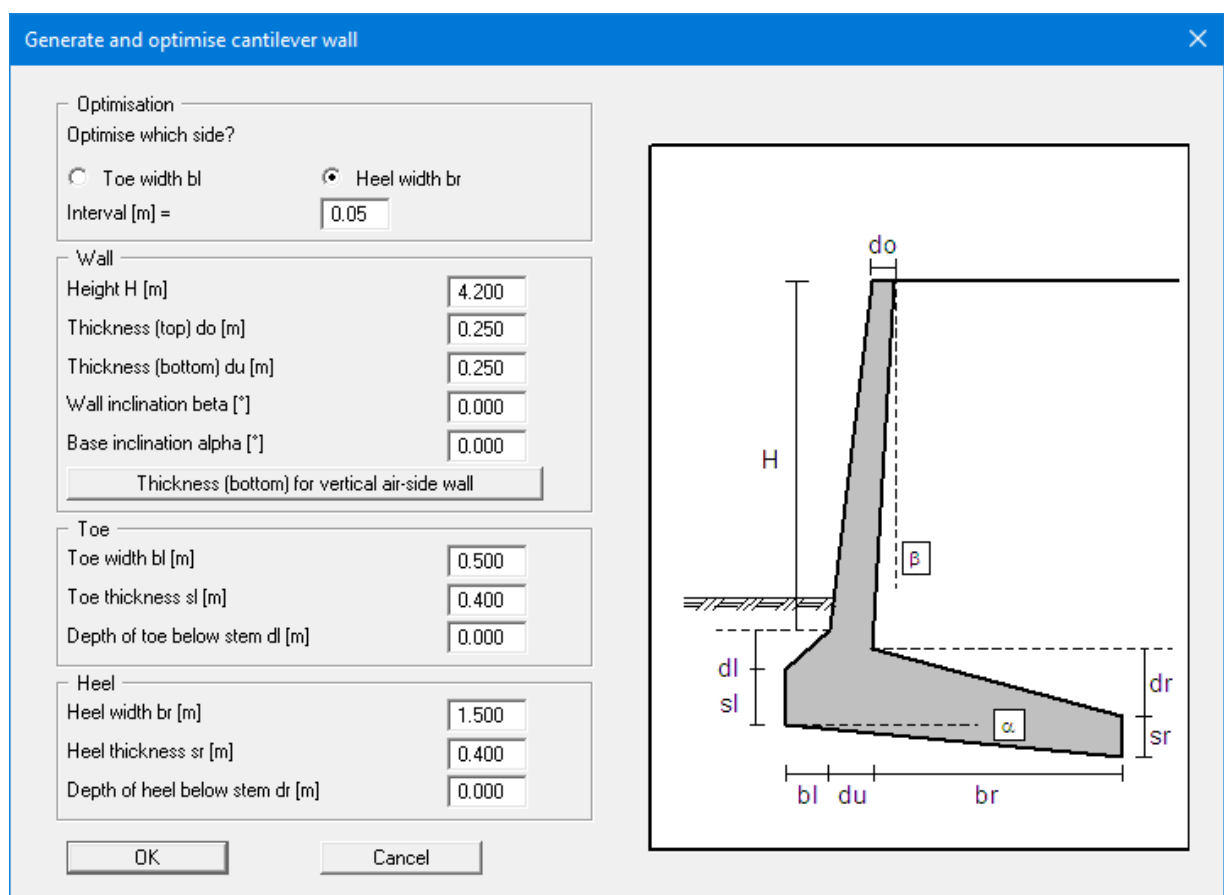
After clicking the "**OK**" button the analysis begins. The program first performs an intensive plausibility check of the data input. Once the analysis is complete, you will receive the following message for the system entered:



After confirming with "OK", the dialog box for reinforced concrete design compliant with EC 2 will be displayed. For the example, accept the proposed specifications in the dialog box. A description of the design can be found in Section 9.4.6. Once design is complete, the results are presented in information boxes and then visualised graphically on the screen.

7.9 Step 8: Optimise system

To correct the displayed error in the serviceability analysis, select the "System/Optimise" menu item. You will first see the dialog box for the analysis, which you accept with the default settings. You can then optimise the cantilever wall:



Accept the proposed specifications and carry out the optimisation. The program then proposes an increase in the spur width (right) from 1.50 m to 1.60 m. Accept the proposal and then carry out design once more.

7.10 Step 9: 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.

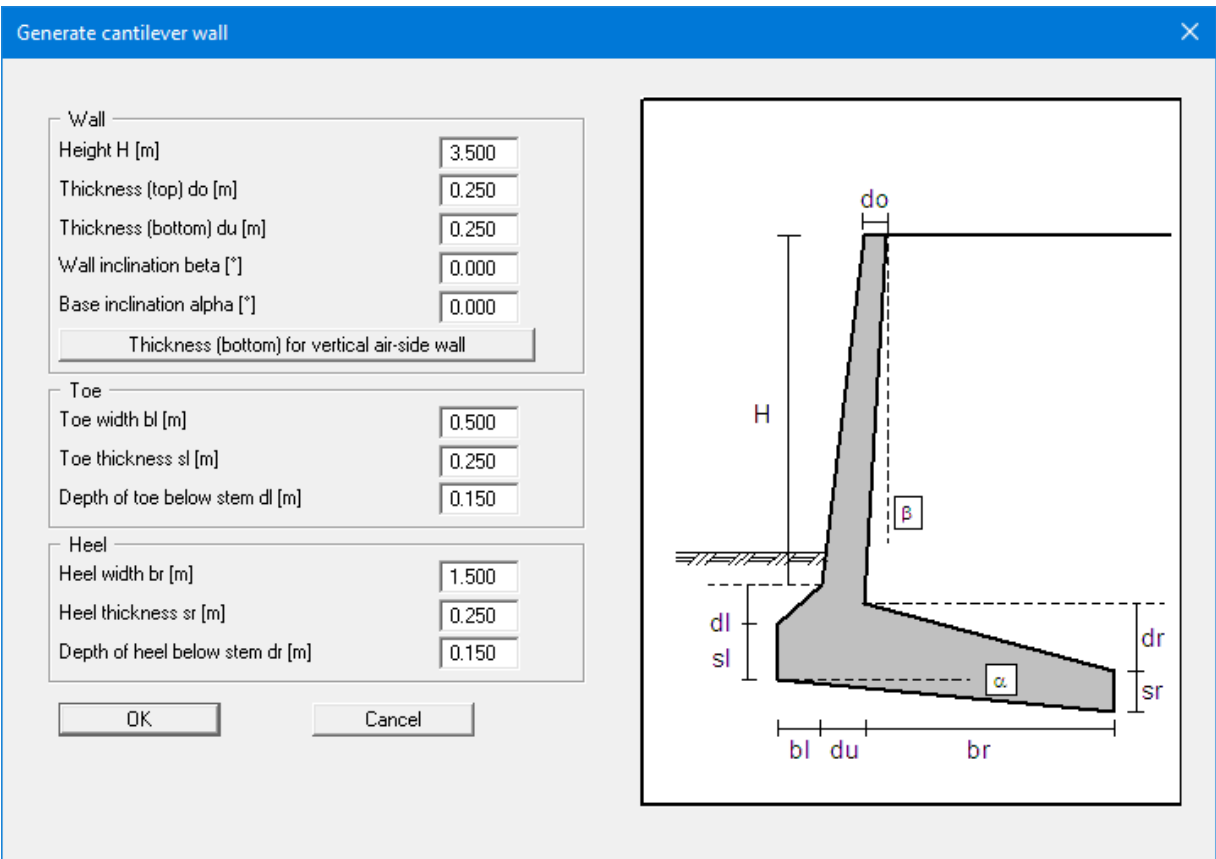
To further clarify the analysis results the graphics can be supplemented with text and graphic elements using the "**Mini-CAD**" module. You can save your work to a file. Go to the "**Save as**" menu item in the "**File**" menu.

8 Theoretical principles

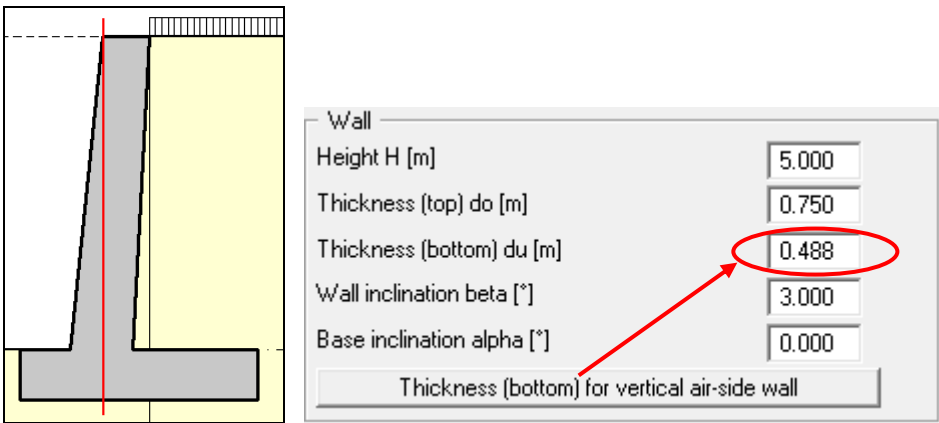
8.1 Cantilever wall

8.1.1 Definition

The cantilever wall is defined via the menu item "Editor 1/Wall (geometry)". The graphics visualise all necessary input.



You can use the "Thickness (bottom) for vertical air-side wall" button to determine the required "Thickness (bottom) du" for an inclined wall. Starting from the upper left corner of the angled cantilever wall, the program extends the air-side wall downwards to determine the lower wall thickness.



8.1.2 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:

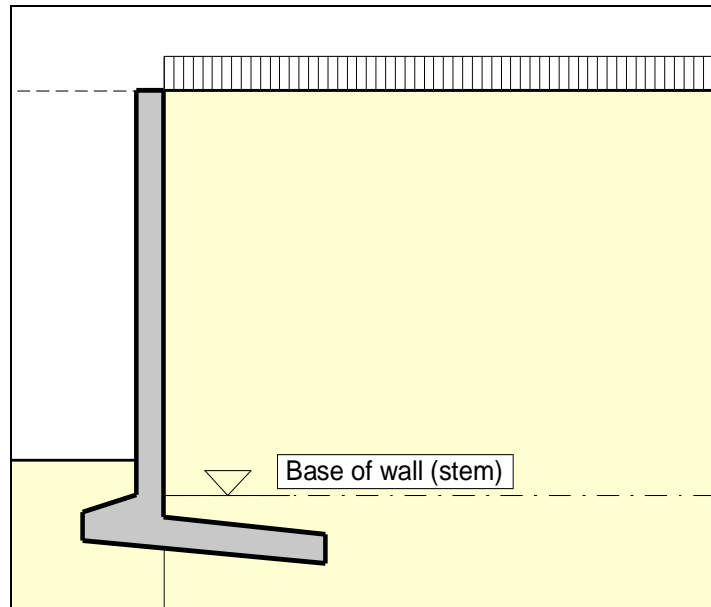


Figure 2 Base of wall (stem) Example 1

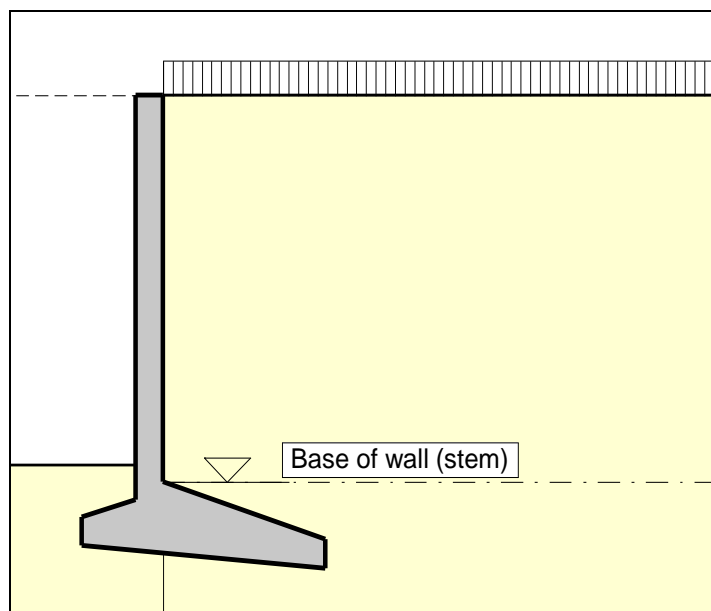


Figure 3 Base of wall (stem) Example 2

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 EC 2 (reinforced concrete design) using this result.

8.2 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 8.7.2).

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.

8.3 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.

In addition, calculation of the active earth pressure after Culmann is possible. The calculations use a slice method with planar slip surfaces.

8.4 At-rest earth pressure

The at-rest earth pressure coefficient is calculated compliant with DIN 4085:2017.

8.5 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:

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

$$0.0 \leq f \leq 1.0$$

8.6 Passive earth pressure

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

- DIN 4085:2017,
- DIN 4085: 2017 planar slip surfaces,
- Streck,
- Caquot/Kerisel,
- DIN 4085:2017 planar slip surfaces/Caquot/Kerisel,
- Culmann.

The earth pressure after Culmann is acquired by varying the slip surface angle (see Piling Handbook 1977). The forces at the earth pressure wedge are calculated using a slice method.

DIN 4085:2017 stipulates that planar slip surfaces may be adopted for analysis up to a friction angle $\leq 35^\circ$ and a wall friction angle $\leq -2/3 \phi$. This returns more favourable earth pressure coefficients.

8.7 Water pressure

8.7.1 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 4:

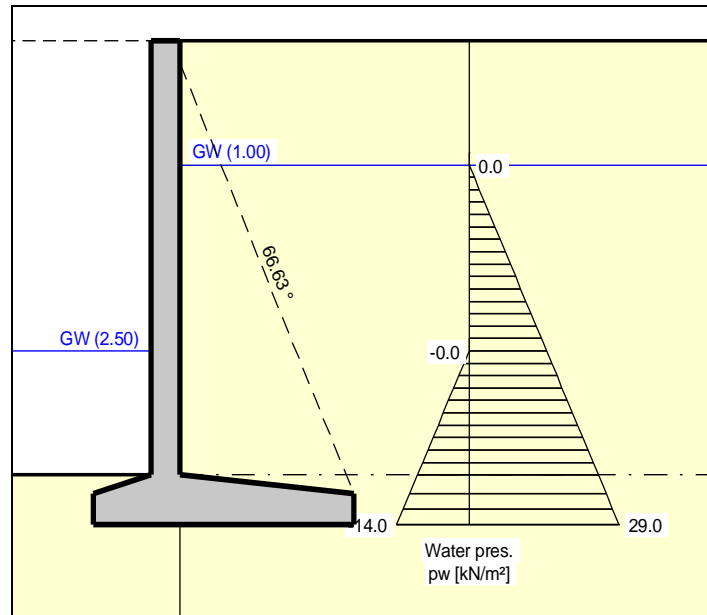


Figure 4 Classical water pressure approach

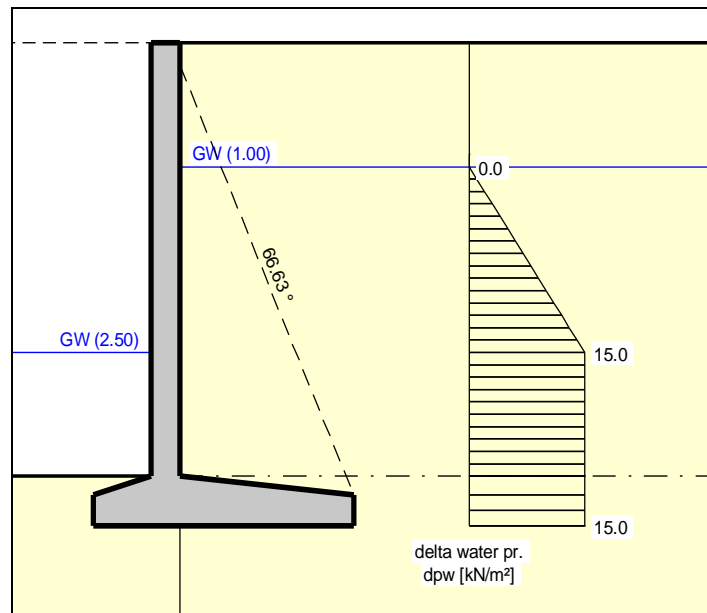


Figure 5 Resultant water pressure

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.

8.7.2 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.

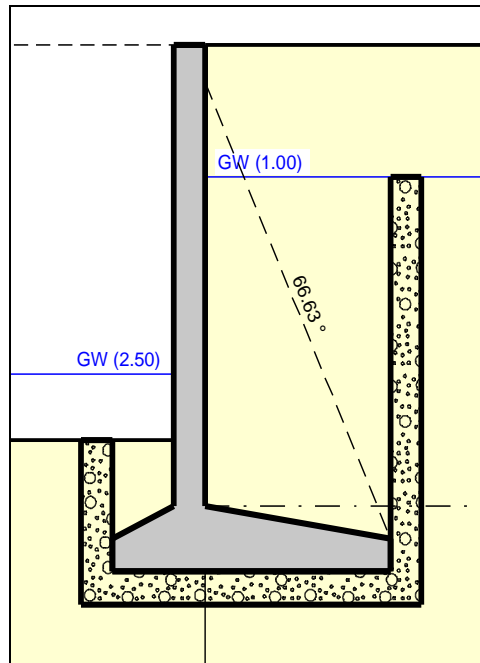


Figure 6 Water pressure approach using flow conduits

The system of the three flow conduits results from the spur widths.

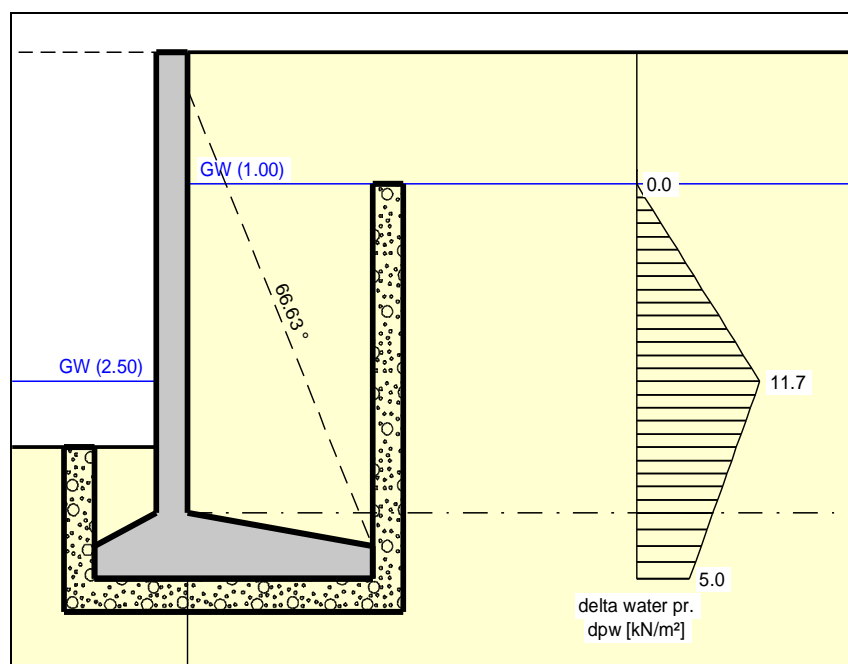


Figure 7 Resultant water pressure with flow conduits

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.

8.7.3 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 4 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.

8.7.4 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:

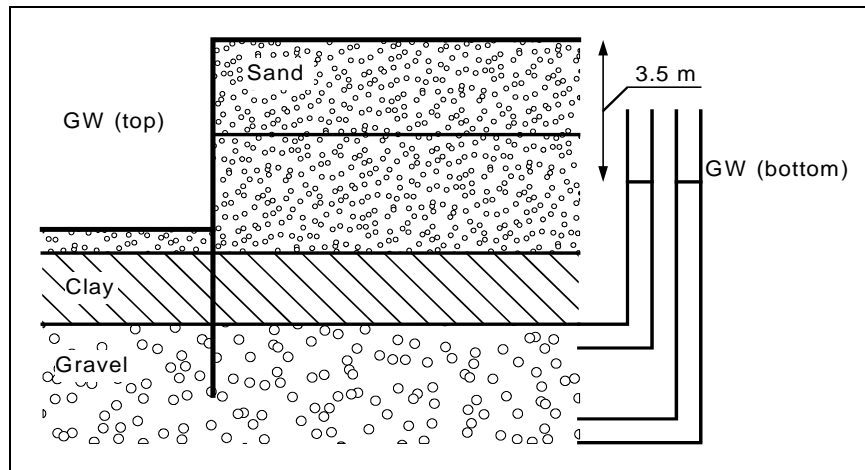


Figure 8 Definition of potential

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.

8.8 Berms

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

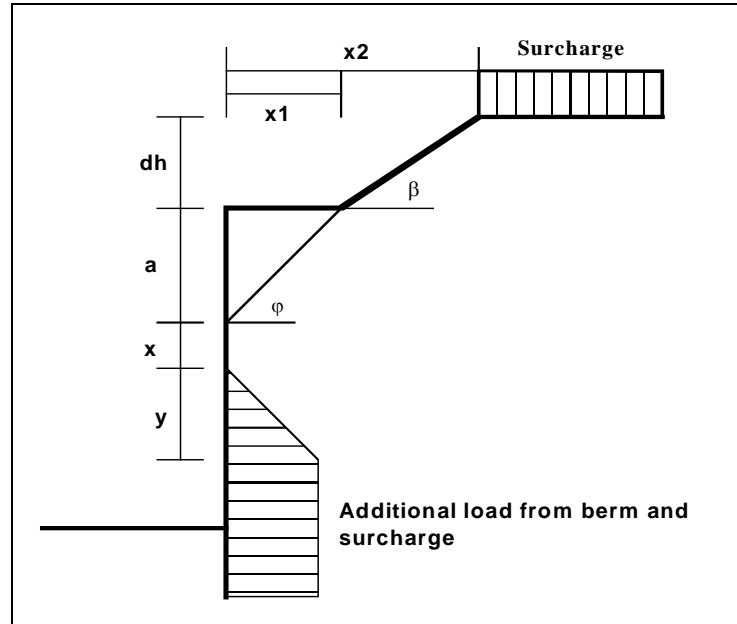


Figure 9 Berm on the active side

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 the same manner.

8.9 Area loads

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

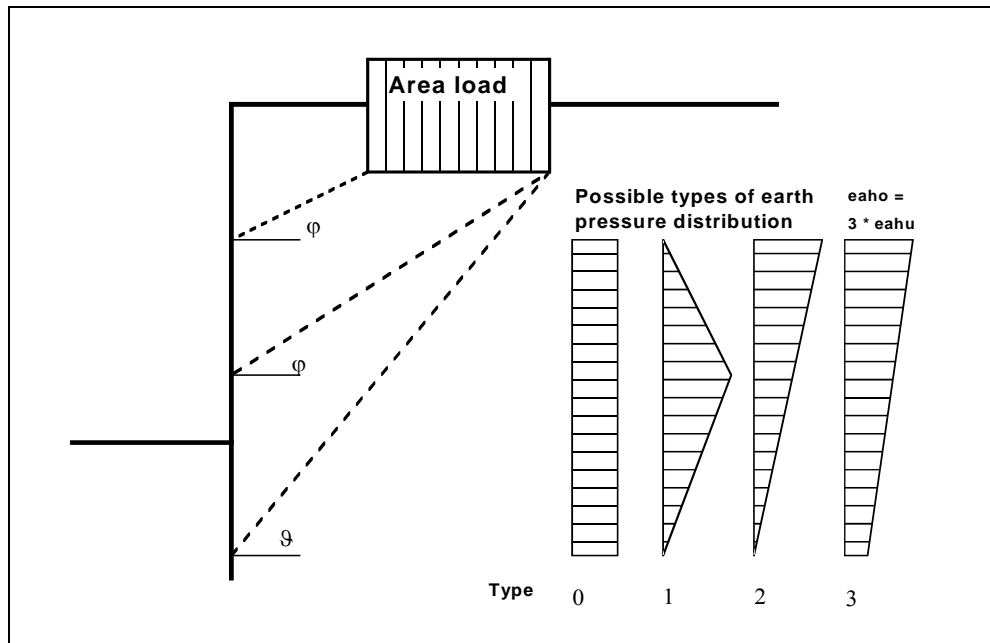


Figure 10 Area load

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 several 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 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 11).

- 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:

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

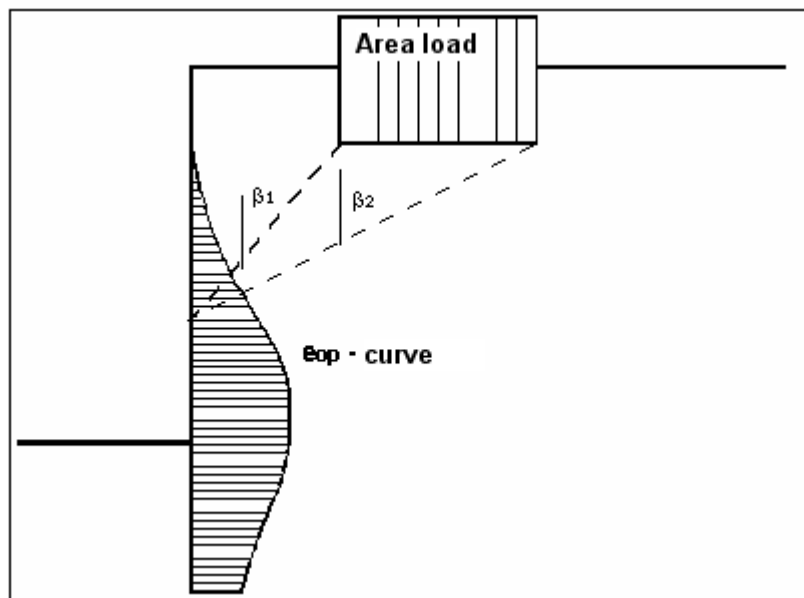


Figure 11 At-rest earth pressure from area loads

8.10 Loads with limited plan dimensions

Loads with limited plan dimensions can be reduced compliant with "Piling Handbook" Figure 4.20 (page 64) or DIN 4085:2017-08 (page 17), see also Section 9.3.2.

8.11 Bounded surcharges (active side)

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

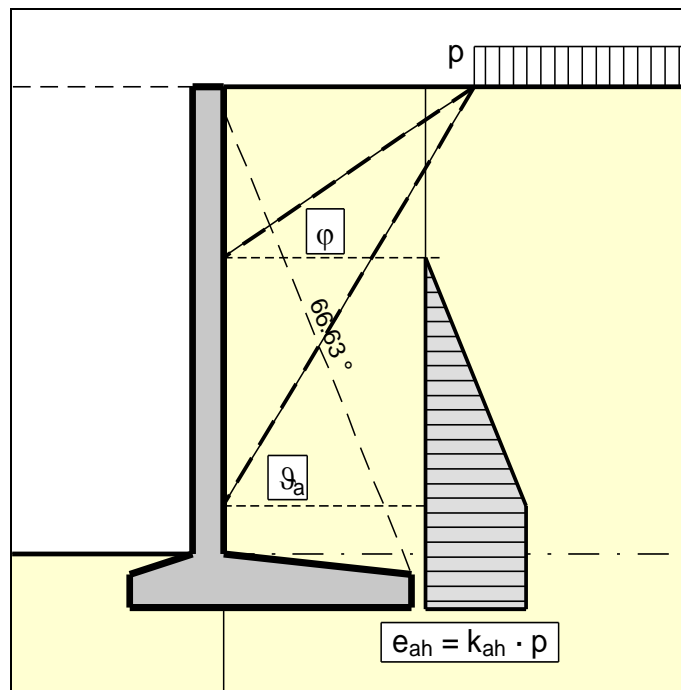


Figure 12 Bounded surcharge (active side)

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

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

8.12 Double-bounded surcharges (active side)

If two bounded surcharges are defined as follows:

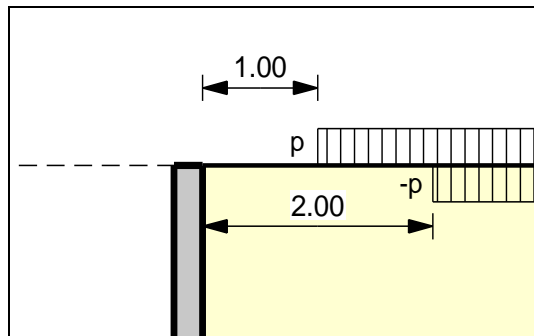


Figure 13 Two bounded surcharges

- 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 8.10).

A double-bounded surcharge, which acts because 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 satisfactory and may be irritating for an examiner. Double-bounded surcharges were therefore introduced.

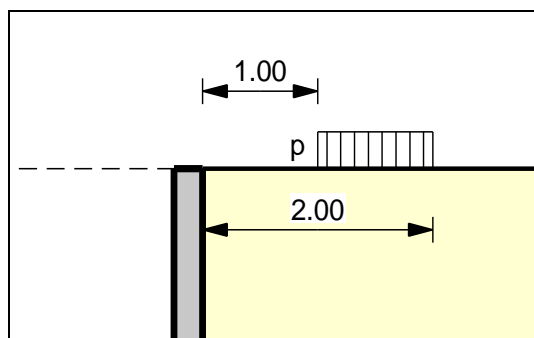


Figure 14 Double-bounded surcharge

The same result is achieved as for the definition provided by Figure 13 but with improved graphics.

The area loads described in Section 8.9 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.

8.13 Bounded surcharges (passive side)

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

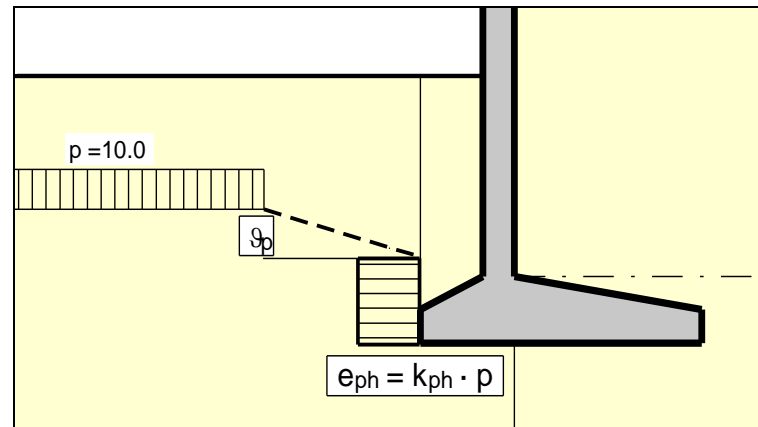


Figure 15 Bounded surcharge (passive side)

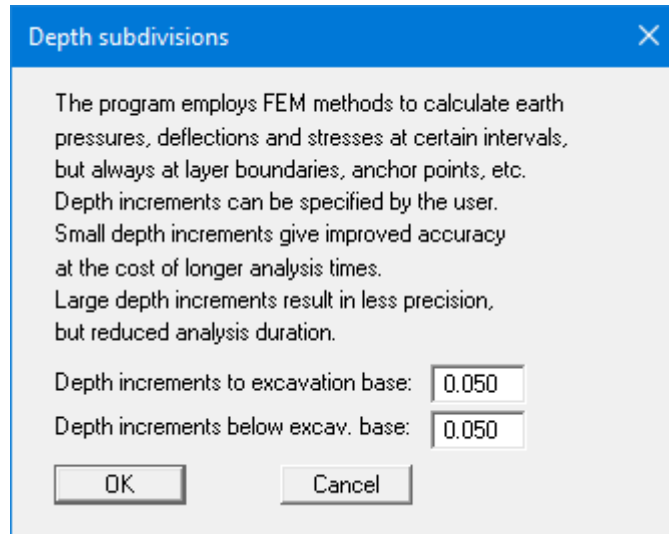
8.14 Double-bounded surcharges (passive side)

Double-bounded surcharges on the passive side can be modelled using two bounded surcharges (also see Section 8.12). Double-bounded surcharges on the passive side were only implemented due to the better graphical representation.

8.15 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 0.05 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 unknowns 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.

8.16 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 calculated for the entire system with footings. 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**".

8.17 Action and displacement boundary conditions

Besides full fixity at the top of the footing in accordance with Section 8.15 ("**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

8.18 Pretensioning

Personal experience has shown that discussions surrounding the necessity for considering pretensioning are never-ending. Independently, the program enables pretensioning for anchors/struts to be taken into consideration using the "**Editor 2/Pretensioning**" menu item.

For anchors, pretensioning must be entered as positive. Faulty input is pointed out before analysis begins. From a structural perspective, the program inserts an action boundary condition equivalent in size and direction to the pretensioning force into the system at the appropriate depth.

Anchors are generally pretensioned at 80% and more of the resulting anchor force. This can be modelled with sufficient accuracy using a high axial rigidity EA without pretensioning. If you wish to analyse more precisely, specify the axial rigidity EA according to the selected anchor and define a pretensioning force. Following analysis, the selected pretensioning force must be examined against the results. This generally requires an iteration process by the user.

The same applies to struts.

8.19 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 many 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.

8.20 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.

8.21 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:

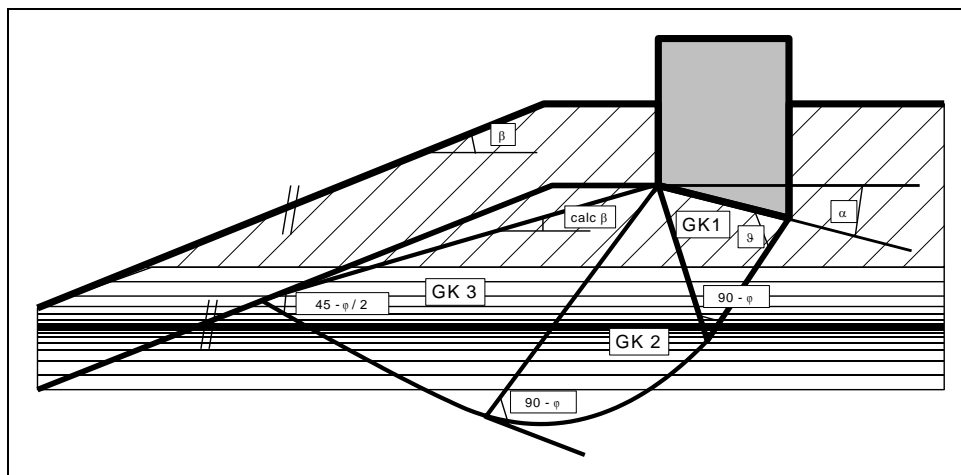


Figure 16 Logarithmic spiral

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

$$\varphi = \frac{\sum \tan \varphi_i \cdot l_i}{\sum l_i}$$

$$\text{cal } c = \frac{\sum c_i \cdot l_i}{\sum l_i}$$

$$\text{cal } \gamma_2 = \frac{\sum \gamma_{2i} \cdot A_i}{\sum A_i}$$

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

8.22 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.

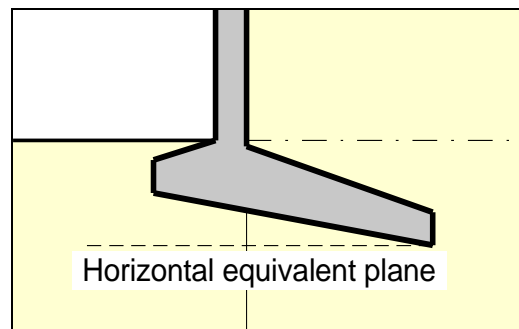


Figure 17 Horizontal equivalent plane

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

Sliding safety

(relative to base inclination of 10.0°)

Governing: g + q
Utilisation factor(sliding) = $H,d / (V,k \cdot \tan(\phi) / \gamma(\text{sliding}) + E_p,d) = 58.4 / (179.6 \cdot \tan(32.5^\circ) / 1.10 + 3.6) = 0.543$
Serviceability analysis to EC 7 (6.6.6)
Utilisation factor = $H,k / (2/3 \cdot V,k \cdot \tan(\phi) + 1/3 \cdot E_p,k) = 35.8 / (2/3 \cdot 175.6 \cdot \tan(32.5^\circ) + 1/3 \cdot 5.1) = 0.469$

(relative to horizontal equivalent plane)

Governing: g + q
Utilisation factor(sliding) = $H,d / (V,k \cdot \tan(\phi) / \gamma(\text{sliding}) + E_p,d) = 88.7 / (166.7 \cdot \tan(32.5^\circ) / 1.10 + 14.3) = 0.800$
Serviceability analysis to EC 7 (6.6.6)
Utilisation factor = $H,k / (2/3 \cdot V,k \cdot \tan(\phi) + 1/3 \cdot E_p,k) = 65.7 / (2/3 \cdot 166.7 \cdot \tan(32.5^\circ) + 1/3 \cdot 20.0) = 0.848$

OK

8.23 General stability

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

8.24 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.

8.25 Hydraulic heave

8.25.1 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.

8.25.2 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.

8.26 Buoyancy

8.26.1 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.

8.26.2 Utilisation factor (buoyancy) using partial safety factors

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

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

- 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
- $G_{k,stb}$ = lower characteristic value of stabilising permanent actions
- $\gamma_{g,stb}$ = 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.

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

- μ_N = utilisation factor of layer N

Utilisation factors ≤ 1.0 mean that sufficient safety is given.

8.27 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.

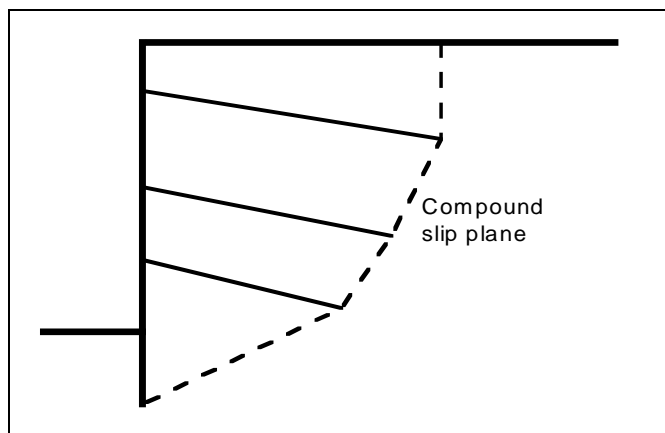


Figure 18 Compound "deep slip planes"

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.

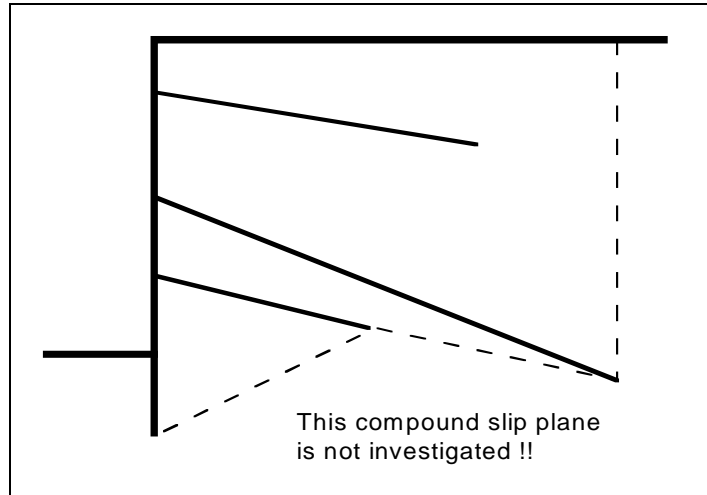


Figure 19 Compound "deep slip plane", which is not investigated

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 regarding a utilisation factor of 1.0 is possible.

8.28 Heave of anchor soil

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

8.29 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 8.15). The output is the section with the largest reinforcement.

The stem area above the footing is designed using the wall internal forces (see Section 8.1.2).

The internal forces M and Q at the stem base/toe and heel intersections are calculated from the soil pressure and any surcharge present. The model concept is a cantilever. The normal force N results from the passive earth pressure.

The following sign rule applies to the spur and wall moments:

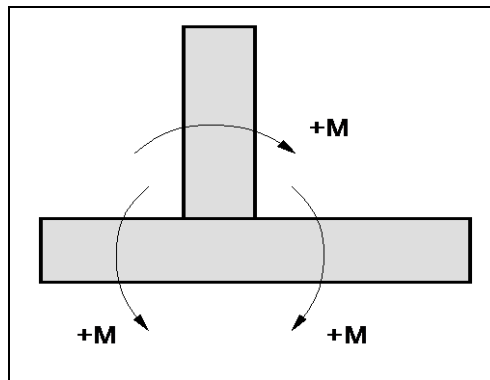


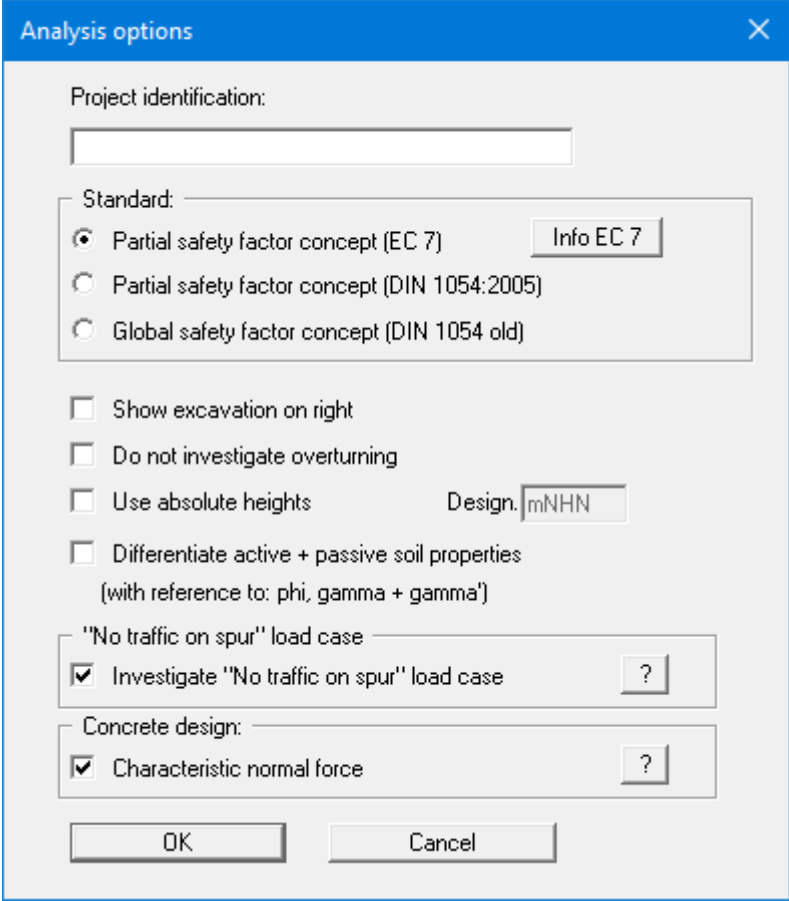
Figure 20 Sign rule for spur and wall moments with reinforcement

9 Description of menu items

9.1 File menu

9.1.1 "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 9.6.8). 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 9.2.7). For better visualisation you can define the soil colours on the active and the passive sides differently using the ***Soil properties legend*** (see Section 9.6.7).

If the normal force is a compressive force, concrete design is not conservative if the design value of the normal force is adopted. Design will adopt the characteristic normal force if the corresponding check box in the "**Concrete design:**" group box is activated.

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 9.2.14) or "**Editor 1/Verifications/Partial factors**" (see Section 9.2.15). Verifications and safety factors can be modified at any time using the above menu items.

9.1.2 "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.

9.1.3 "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.

9.1.4 "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 ".**gg_u_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, ".**gg_u_wst**" will be used automatically.

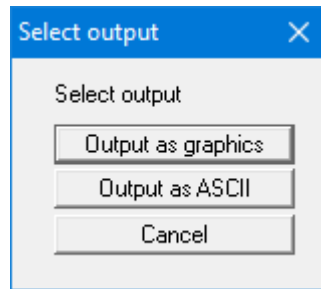
If the current system has been analysed at the time of saving, the analysis results are saved in the file. This is especially necessary if you want to use the results in this file for ***predeformations*** on other systems (see Section 9.3.13). If your current system represents an intermediate construction 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 9.1.1).

9.1.5 "Print output table" menu item

9.1.5.1 *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.

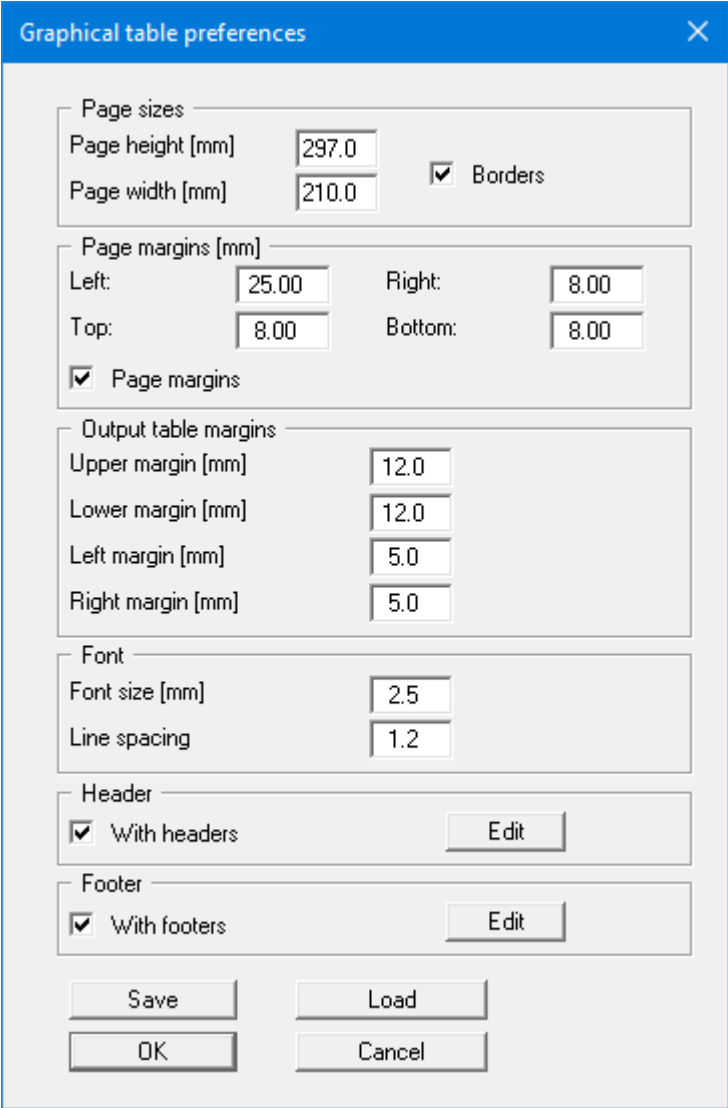
You have the option of designing and printing the output table as an annex to your report within the **GGU-CANTILEVER** application. To do this, select "**Output as graphics**" from the following options.



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.

9.1.5.2 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.

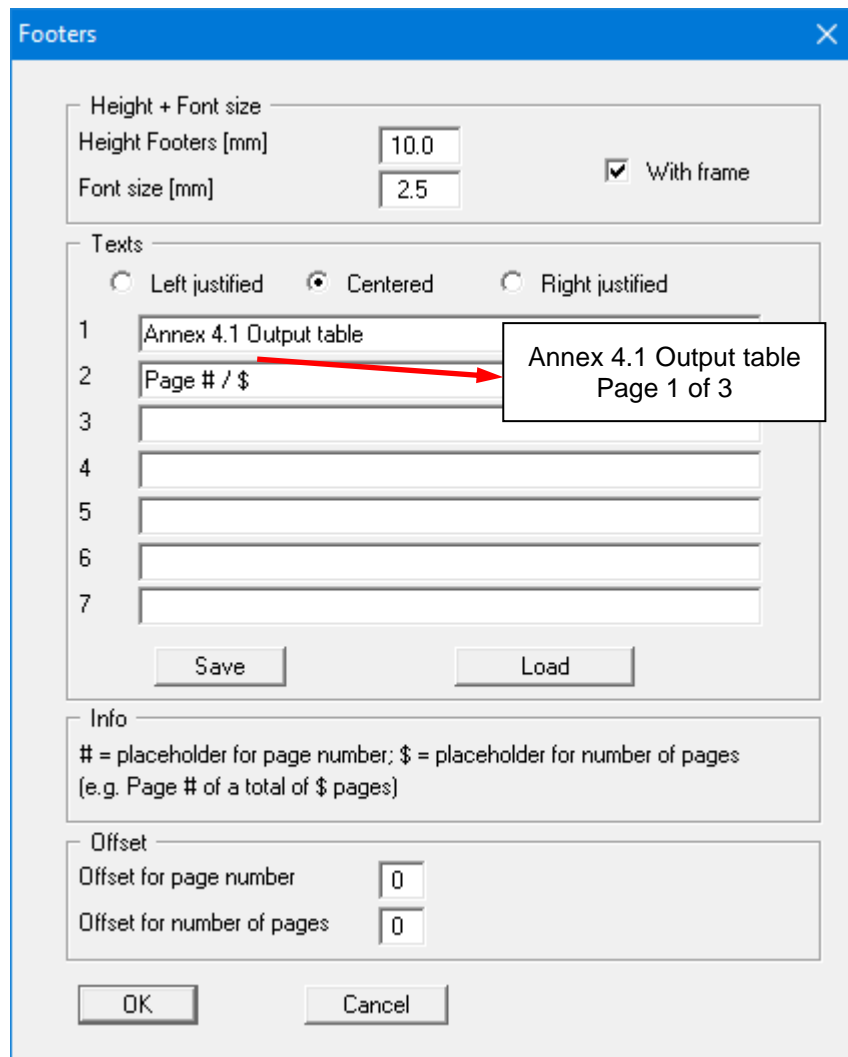


The dialog box titled "Graphical table preferences" contains several sections for configuring output settings:

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

At the bottom of the dialog are four buttons: "Save", "Load", "OK", and "Cancel".

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. You can save your settings for the graphical output table presentation in a "Protokoll.pin_ggu" file at the program level so that they are loaded when the program starts. Using the "Load" button, the output table settings can also be subsequently loaded into an existing file, including that of another GGU program.



The 'Footers' dialog box is used to configure the footer information. It includes sections for 'Height + Font size', 'Texts', 'Info', and 'Offset'.

Height + Font size

- Height Footers [mm]: 10.0
- Font size [mm]: 2.5
- ☒ With frame

Texts

Justification: ☐ Left justified ☒ Centered ☐ Right justified

Text lines (1-7):

- Annex 4.1 Output table
- Page # / \$
-
-
-
-
-

Buttons: Save, Load




Info

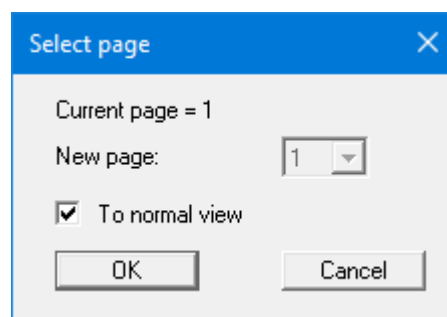
= placeholder for page number; \$ = placeholder for number of pages (e.g. Page # of a total of \$ pages)

Offset

- Offset for page number: 0
- Offset for number of pages: 0

Buttons: OK, Cancel

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:



The 'Select page' dialog box is used to select a page to display. It includes fields for 'Current page' and 'New page', a checkbox for 'To normal view', and 'OK' and 'Cancel' buttons.

Current page = 1

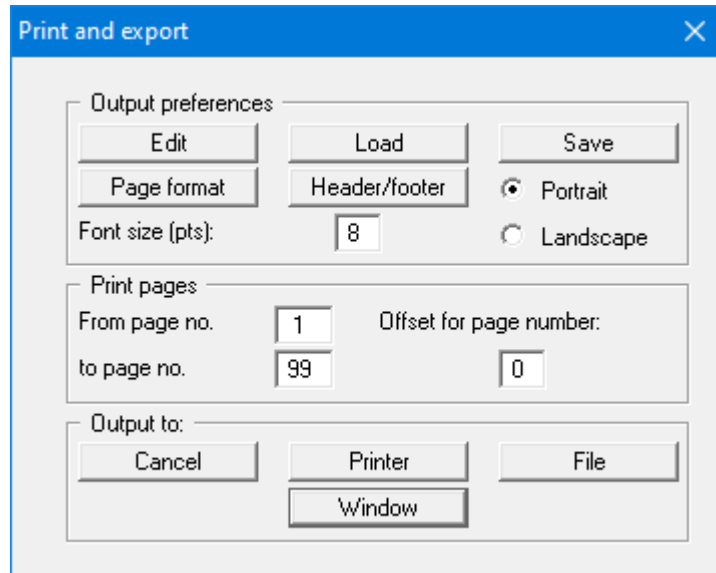
New page: 1

☒ To normal view

Buttons: OK, Cancel

9.1.5.3 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.

- **"Output preferences" group box**
Using the **"Edit"** button the current output preferences can be changed or a different printer selected. Using the **"Save"** button, all preferences from this dialog box can be saved to a file 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.

9.1.6 "Export to GGU-STABILITY" 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.

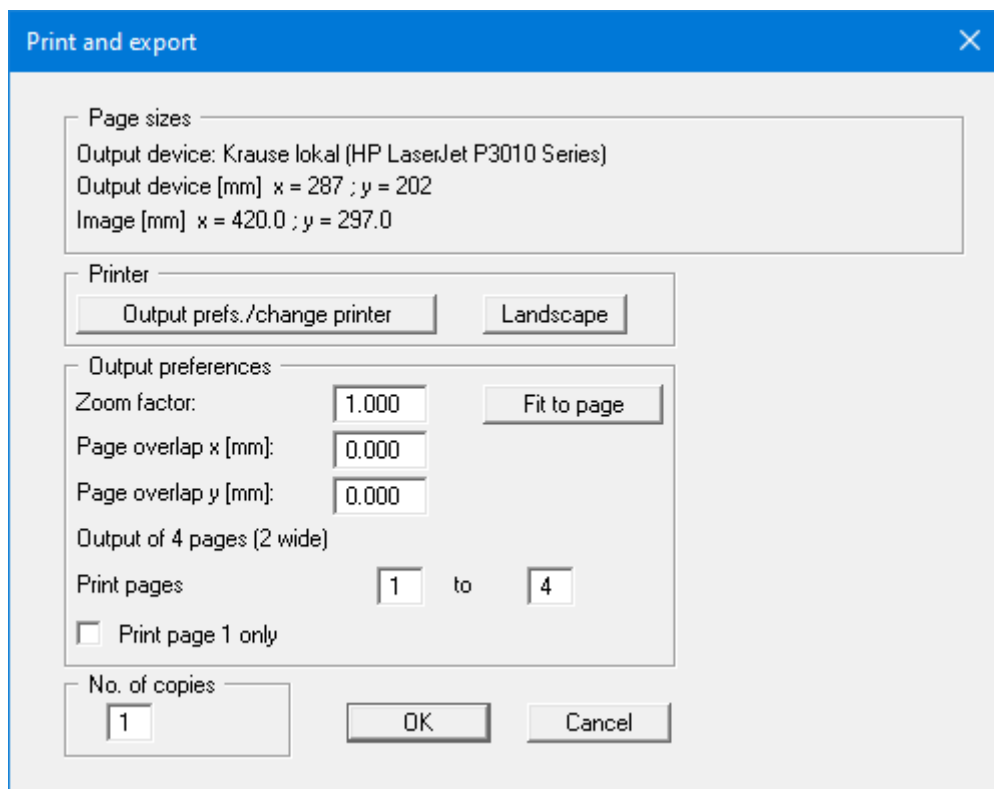
9.1.7 "Output preferences" menu item

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

9.1.8 "Print and export" menu item

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

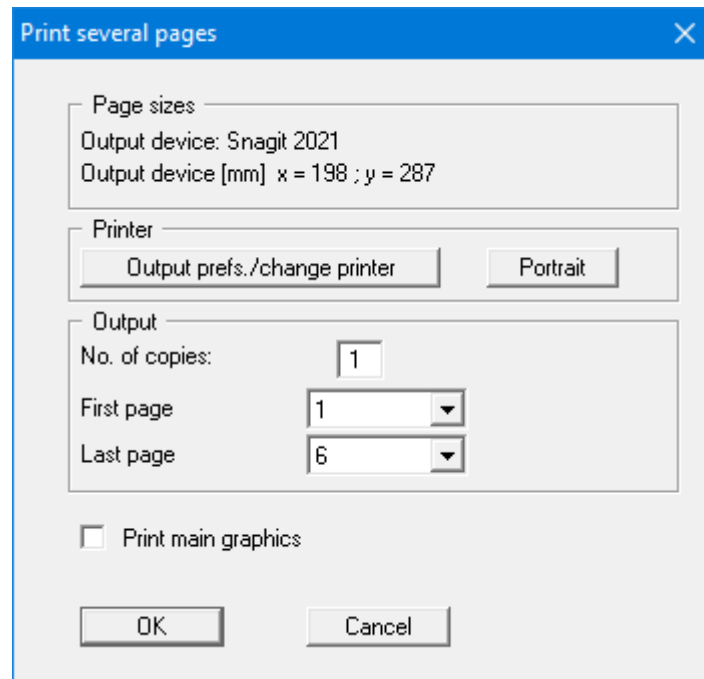
- **"Printer"**
allows graphic output of the current screen contents (*graphical representation*) to the WINDOWS standard printer or to any other printer selected using the menu item **"File/Output preferences"**. But you may also select a different printer in the following dialog box by pressing the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



In the upper group box, the maximum dimensions which the printer can accept are given. Below this, the dimensions of the image to be printed are given. If the image is larger than the output format of the printer, the image will be printed to several pages (in the above example, 4). In order to facilitate better re-connection of the images, the possibility of entering an overlap for each page, in x and y direction, is given.


Alternatively, you also have the possibility of selecting a smaller zoom factor, ensuring output to one page ("**Fit to page**" button). Following this, you can enlarge to the original format on a copying machine, to ensure true scaling. You can also print only the 1st page with zoom factor = 1.0, in which case the bottom left-hand area is printed. Furthermore, you may enter the number of copies to be printed.

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



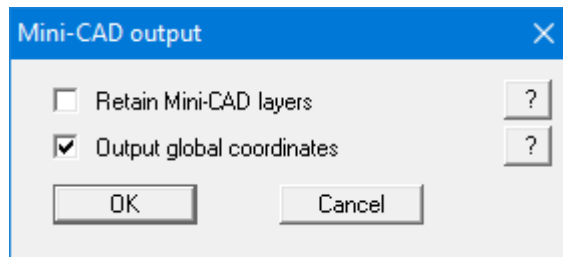
Here, you can select the table pages to be printed. To achieve output with a zoom factor of 1 (button "**Fit in automatically**" is deactivated), you must adjust the page format to suit the size format of the output device. To do this, use the dialog box in "**File/Print output table**" button "**Output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

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

If you select the "**Copy/print area**" tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see "**Tips and tricks**" in Section 6.5).

Using the "**Mini-CAD**" program module you can also import EMF files generated using other GGU applications into your graphics (see Section 9.6.5).

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



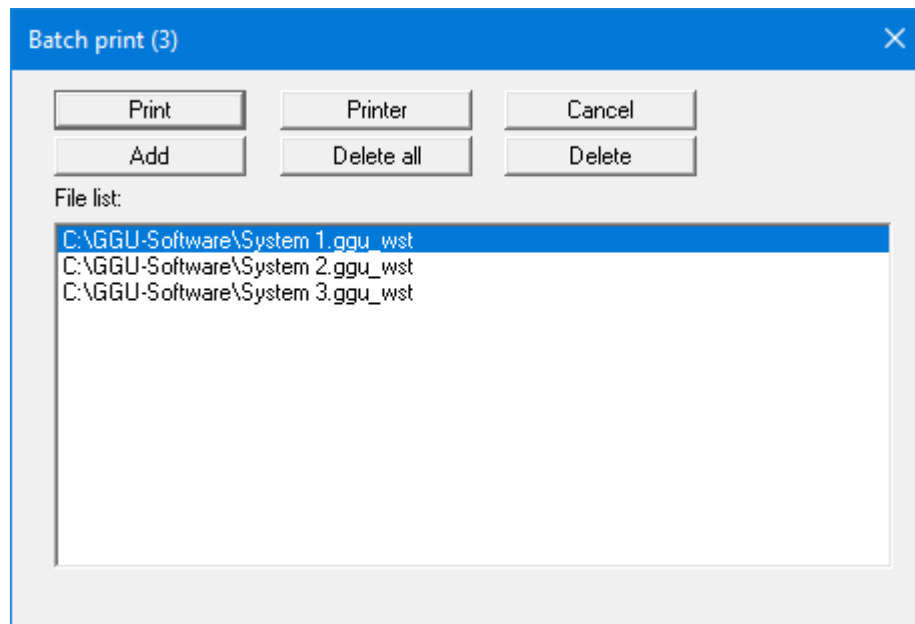
If the "**Retain Mini-CAD layers**" check box is activated, the layer allocations for any existing **Mini-CAD** elements are saved. Otherwise, all **Mini-CAD** elements are saved on Layer 1 and are also inserted into Layer 1 in other GGU programs via the "**Load**" function in the **Mini-CAD** pop-up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

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

- "**GGUMiniCAD**"
allows export of the graphics to a file to enable processing in the **GGUMiniCAD** program.
- "**Cancel**"
Printing is cancelled.

9.1.9 "Batch print" menu item

If you would like to print several annexes at once, select this menu item. You will see the following dialog box:



Create a list of files for printing using "**Add**" and selecting the desired files. The number of files is displayed in the dialog box header. Using "**Delete**" you can mark and delete selected individual files from the list. After selecting the "**Delete all**" button, you can compile a new list. Selection of the desired printer and output preferences is achieved by pressing the "**Printer**" button.

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

9.1.10 "Exit" menu item

After a confirmation prompt, you can quit the program.

9.1.11 "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.

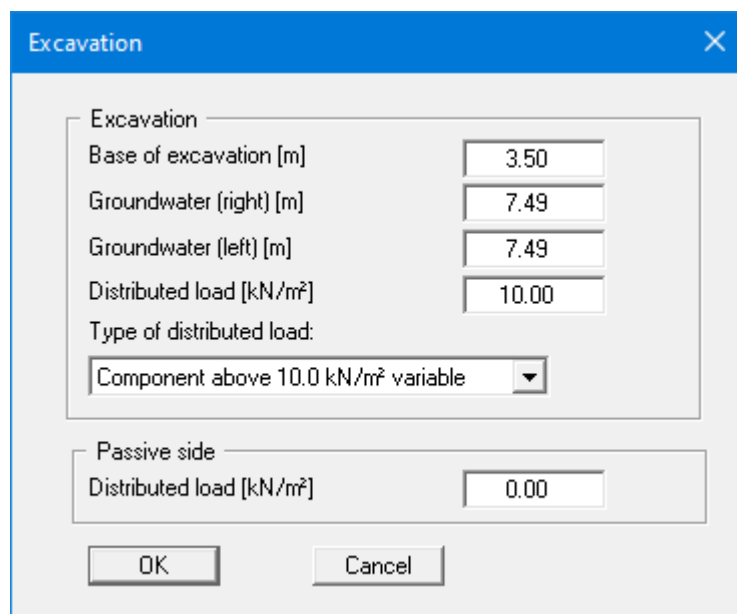
9.2 Editor 1 menu

9.2.1 "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 9.1.1).

9.2.2 "Excavation" menu item

A dialog box opens for defining the excavation:



The image shows a dialog box titled "Excavation" with a blue header bar and a close button (X) in the top right corner. The dialog is divided into two main sections. The first section, labeled "Excavation", contains four input fields: "Base of excavation [m]" with a value of 3.50, "Groundwater (right) [m]" with a value of 7.49, "Groundwater (left) [m]" with a value of 7.49, and "Distributed load [kN/m²]" with a value of 10.00. Below these is a dropdown menu for "Type of distributed load:" with the selected option "Component above 10.0 kN/m² variable". The second section, labeled "Passive side", contains one input field: "Distributed load [kN/m²]" with a value of 0.00. At the bottom of the dialog are two buttons: "OK" and "Cancel".

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.

9.2.3 "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 7.4.

9.2.4 "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 7.5.

9.2.5 "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	0.00	<input type="checkbox"/>

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

If more than one berm is present in the system, click "**x 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.

9.2.6 "Berms (passive side)" menu item

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

9.2.7 "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

Using the "**Common soils**" button, you can easily select the soil properties of many common soils from a database or determine intermediate values. In the dialog box, which you open by pressing the "**Common soils**" button, open the "**Soils_english.gng_ggu**" file when first starting the program in English ("**Edit table**"/"**Load**" buttons). Then save the data set in the "**Soils.gng_ggu**" file on the program level in order to open your modified database file when the program starts. You can also enter your own data ("**Edit table**"/"**x soils to edit**" button) and save it in the "**Soils.gng_ggu**" file.

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 8.7.2).

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).

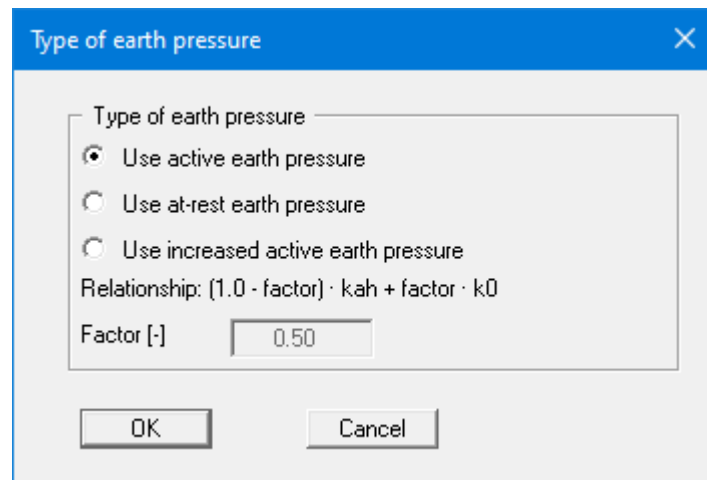
In stratified soils the number of layers must be entered under "**Edit number of soils**". Clicking the "**Sort**" button sorts the soil layers according to depth; however, this is performed automatically when you click "**OK**" to leave the dialog box. This eliminates the possibility of input errors.

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.

9.2.8 "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.



Type of earth pressure

Type of earth pressure

- ☒ Use active earth pressure
- ☐ Use at-rest earth pressure
- ☐ Use increased active earth pressure

Relationship: $(1.0 - \text{factor}) \cdot k_{ah} + \text{factor} \cdot k_0$

Factor [-]

OK Cancel

9.2.9 "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. If the "**Culmann**" check box is activated the active earth pressure is calculated after Culmann using a slice method with straight slip surfaces.


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 $\phi = 40^\circ$ or 45° . This procedure also takes the defined wall friction angle into consideration.

DIN 4085 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. To be able to check calculations performed with such an application, **GGU-CANTILEVER** also offers this possibility.

9.2.10 "Passive earth pressure" menu item

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



The dialog box titled "Passive earth pressure" contains three main sections. The first section, "Passive ep according to:", lists seven calculation methods with radio buttons: DIN 4085:2017, DIN 4085:2017 plan. slip surf. (selected), Streck, Caquot/Kerisel, User-defined (not recommended), DIN 4085:2017 plan. slip surf./Caquot/Kerisel, and Culmann. Each method has a "?" button. The second section, "Partial safety factor, ...", contains three input fields: "Partial factor passive ep (sliding)" with value 1.40, "Factor passive ep (bearing cap.)" with value 0.50, and "Passive ep calibration factor" with value 1.00. A note below states "(calibration factor <= 1.0 see DIN 1054 10.6.3 (4))". The third section, "Special preferences", has two checkboxes: "Limit passive earth pressure" (unchecked) and "Passive ep <= FACTOR · active ep" (unchecked). The first checkbox has a "Max. passive earth pressure [kN/m²]" input field with value 100.0. The second checkbox has a "FACTOR [-]" input field with value 1.000. At the bottom are "OK" and "Cancel" buttons.

Section	Option / Label	Value / State
Passive ep according to:	<input type="radio"/> DIN 4085:2017	?
	<input checked="" type="radio"/> DIN 4085:2017 plan. slip surf.	?
	<input type="radio"/> Streck	
	<input type="radio"/> Caquot/Kerisel	
	<input type="radio"/> User-defined (not recommended)	?
	<input type="radio"/> DIN 4085:2017 plan. slip surf./Caquot/Kerisel	?
	<input type="radio"/> Culmann	?
Partial safety factor, ...	Partial factor passive ep (sliding)	1.40
	Factor passive ep (bearing cap.)	0.50
	Passive ep calibration factor	1.00
	(calibration factor <= 1.0 see DIN 1054 10.6.3 (4))	
Special preferences	<input type="checkbox"/> Limit passive earth pressure	
	Max. passive earth pressure [kN/m²]:	100.0
	<input type="checkbox"/> Passive ep <= FACTOR · active ep	
	FACTOR [-]:	1.000

In the upper group box you specify the type of passive earth pressure calculation. If the "**Culmann**" check box is activated the passive earth pressure is calculated after Culmann using a slice method with straight slip surfaces.

9.2.11 "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 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.

At-rest earth pressure

Handling area loads for at-rest earth pressure

☐ Area loads using theory of elastic half-space

☒ Area loads by increasing using the factor: k_0/k_{ah}

Area loads using theory of elastic half-space

Concentration factor for area loads

☐ = 3 ☒ = 4

'3' for overconsolidated cohesive soils

'4' for non-cohesive and non-overconsolidated cohesive soils

☐ Constant at-rest pressure below lowest anchor

OK Cancel

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. To do this, activate the check box at the bottom of the dialog box.

9.2.12 "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 8.8).

No.	kagh	kach	kpgh	kpch	phi (a/p) [°]
1	0.3333	1.1547	3.0000	3.4641	32.5/32.5

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 9.2.9 and 9.2.10).

9.2.13 "Groundwater" menu item

Regarding adopting horizontal water pressure you are referred to the explanations in Section 8.7.

Horizontal water pressure approach: ☒ Classical ☐ With flow conduits

Consider hydraulic gradients i on: ☐ Active side ☒ Passive side ☒ Incl. bearing capacity

According to EAB R 63, the hydraulic gradient on the passive side must be considered. It may be ignored on the active side.

Condition: $\gamma(\text{soil}) \cdot i \cdot \gamma(w) \geq 0.0$

☒ Adhere to

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.

9.2.14 "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 dialog box titled "Safety factors" contains 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 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.

9.2.15 "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

Design situation:

Permanent actions:

Permanent actions (at-rest pressure):

Variable actions:

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

Passive earth pressure (sliding):

Factor passive ep (bearing cap.):

Passive ep calibration factor:

Bearing capacity

☒ Verify

gamma(bearing cap.):

☐ Bearing cap. with depth factor

☒ Check and correct 5° condition

Decrement [°]:

☒ Top of bearing cap. failure body parallel to slope

Sliding

☒ Verify

gamma(sliding):

Friction angle base plane [°] =

Passive wall friction(sliding safety) / phi =

Serviceability verification:

☐ Sliding safety without passive EP

☒ Serviceability verification:

EQU analysis

☒ Verify

gamma(G,dst):

gamma(G,stab): gamma(Q,dst):

Default values

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 or compliant with Swiss standards using the "**SIA 267**" button.

9.2.16 "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.

9.2.17 "Seismic effects" menu item

Seismic loads are taken into account according to EC 8. In addition to the modification of the earth pressure coefficients according to EC 8, a horizontal load from a rigid structure or from the dead weight of the wall can be considered. It is based on the equations from Annex E of DIN EN 1998-5, December 2010, p. 33 ff.

Seismic effects

Consideration of seismics

- ☒ Use DIN EN 1998-5 Dec. 2010
- ☐ Horizontal loading from rigid structure
- ☒ Horizontal loading from $k_h \cdot \text{self-weight (wall)}$
- ☒ Modify earth pressure coefficients (EC 8)
- ☐ Modify earth pressure coefficients (EAU 1990)

Input

$k_h = a_h/g$ (horizontal) [-]:

$k_v = a_v/g$ (vertical) [-]:

Partial safety factor γ_{ϕ} [-]:

(a_h = horizontal seismic acceleration in m/s^2)
(a_v = vertical seismic acceleration in m/s^2)
(g = gravitational acceleration = 9.81 m/s^2)

Info

$k_h = 0,1 \Rightarrow$ Building damage
 $k_h = 0,2 \Rightarrow$ Serious building damage
 $k_h = 0,8 \Rightarrow$ Devastating
 $k_h = 1,0 \Rightarrow$ Complete devastation

OK Cancel

9.2.18 "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.

Deep-seated stability/heave of anchor soil preferences

Intersection with wall

☐ Intersection with wall 'manually'

Depth of intersection [m]: 2.000

Analysis for increased earth pressure/at-rest pressure

☒ For increased earth pressure/at-rest pressure with active ep

☒ use partial factors DS-P/T ?

Heave of anchor soil

☒ Verify

OK Cancel

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.

9.3 Editor 2 menu

9.3.1 "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/m²]	e (bottom) [kN/m²]	Live
1	1.000	2.000	10.00	20.00	<input type="checkbox"/>

The number of lateral pressures can be modified using the "**x lateral 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.

9.3.2 "Area loads" menu item

Using this menu item you define area loads and loads with limited plan dimensions.

No.	p(v) [kN/m²]	p(h) [kN/m²]	x(Left) [m]	x(Right) [m]	Depth [m]	Wall pressure shape	Live
1	20.00	0.00	0.500	1.000	0.000	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 8.9).

Use the "**Loads with limited plan dimensions**" button, to define such loads (see also Section 8.10):

Loads with limited plan dimensions can be reduced compliant with "Piling Handbook" Figure 4.20 (page 64) or DIN 4085:2017-08 (page 17).

Whereby: $p(\text{new}) = p \cdot b / (2 \cdot a + b)$

b = width parallel to wall

Piling Handbook 1977: a = distance of front of surcharge from the wall

DIN 4085:2017-08: a = distance of rear of surcharge from the wall

A number > 1 can be advantageous for a long strip load.

p(v) [kN/m²]: 100.000

p(h) [kN/m²]: 0.000

x(left) [m]: 1.000

x(right) [m]: 5.000

b [m]: 0.500

Depth [m]: 1.000

No. of subsections: 4

Type of load: Rectangle

☐ Delete existing loads

Analyse compliant with: DIN 4085:2017-08

9.3.3 "Compaction earth pressure" menu item

The compaction earth pressure to DIN 4085:2017 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 references.

9.3.4 "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/m²]	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.

9.3.5 "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/m²]	x(Left) [m]	x(Right) [m]	Depth [m]	Live
1	0.00	0.000	1.000	0.000	<input type="checkbox"/>

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 12 in Section 8.10) may be adopted at its full value for the bounded surcharge with the negative sign has not yet been conclusively clarified. This can be influenced using the control parameter:

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

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

9.3.6 "Action boundary conditions" menu item

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

Point loads

Done Forw. Back Cancel

1 point load(s) to edit

No.	Depth [m]	M [kN·m/m]	H [kN/m]	V [kN/m]	Live
1	0.00	0.00	-15.00	0.00	<input checked="" type="checkbox"/>

Diagram: A vertical wall with a downward arrow labeled +V, a curved arrow labeled +M, and a horizontal arrow pointing left labeled +H.

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.

9.3.7 "Displacement boundary conditions" menu item

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

Displacement boundary conditions

Forw. Back Cancel Done

1 displacement BC(s) to edit

No.	Depth [m]	Size [m or radians]	
1	2.00	0.00000	phi

Diagram: A vertical wall with a curved arrow labeled +phi, a downward arrow labeled +wy, and a horizontal arrow pointing right labeled +wx.

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.

9.3.8 "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.

9.3.9 "Struts" menu item

Struts are defined in this dialog box.

No.	Depth [m]	Inclin. [°]	Length [m]	EA [kN/m]	EI [kN·m²/m]	V load [kN/m²/m]	Hinge	No. of rods
1	0.50	5.0	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 several rods (finite elements, see Section 8.15). The maximum number of rods that can be specified is 20.

9.3.10 "Pretensioning" menu item

Personal experience has shown that discussions surrounding the necessity for considering pretensioning are never-ending. Independently, the program enables pretensioning for anchors/struts to be taken into consideration

Anchor no.	Depth [m]	Pretensioning [kN/m]
1	0.50	0.00

For anchors, pretensioning must be entered as positive. Faulty input is pointed out before analysis begins. From a structural perspective, the program inserts an action boundary condition equivalent in size and direction to the pretensioning force into the system at the appropriate depth.

Anchors are generally pretensioned at 80% and more of the resulting anchor force. This can be modelled with sufficient accuracy using a high axial rigidity EA without pretensioning. If you wish to analyse more precisely, specify the axial rigidity EA according to the selected anchor and define a pretensioning force. Following analysis, the selected pretensioning force must be examined against the results. This generally requires an iteration process by the user.

The same applies to struts.

9.3.11 "Groundwater potentials" menu item

You can define additional potentials anywhere along the wall (see Section 8.7.2).

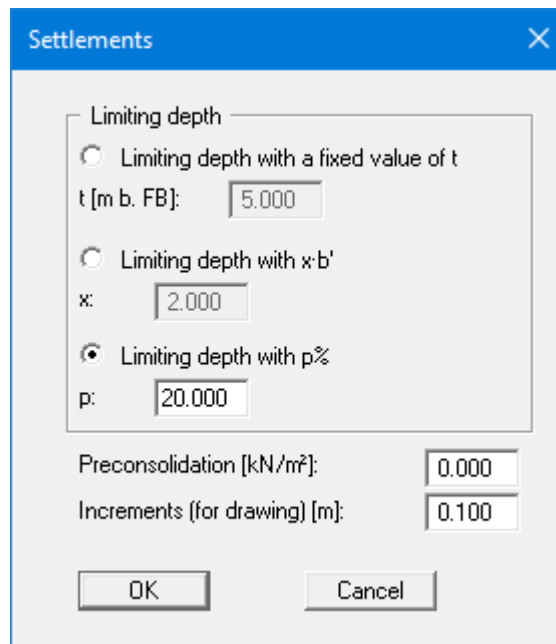
No.	Depth [m]	Value [m]	Left/Right
1	2.000	4.000	<input checked="" type="checkbox"/> Right
2	2.000	4.000	<input type="checkbox"/> Right

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 9.2.13).

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

9.3.12 "Settlements" menu item

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



The screenshot shows a software dialog box titled "Settlements". It contains three radio button options for "Limiting depth":
1. "Limiting depth with a fixed value of t" (selected): The input field "t [m b. FB]:" has the value "5.000".
2. "Limiting depth with x:b": The input field "x:" has the value "2.000".
3. "Limiting depth with p%": The input field "p:" has the value "20.000".
Below these options are two more input fields:
- "Preconsolidation [kN/m²]:" with the value "0.000".
- "Increments (for drawing) [m]:" with the value "0.100".
At the bottom of the dialog are "OK" and "Cancel" buttons.

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.

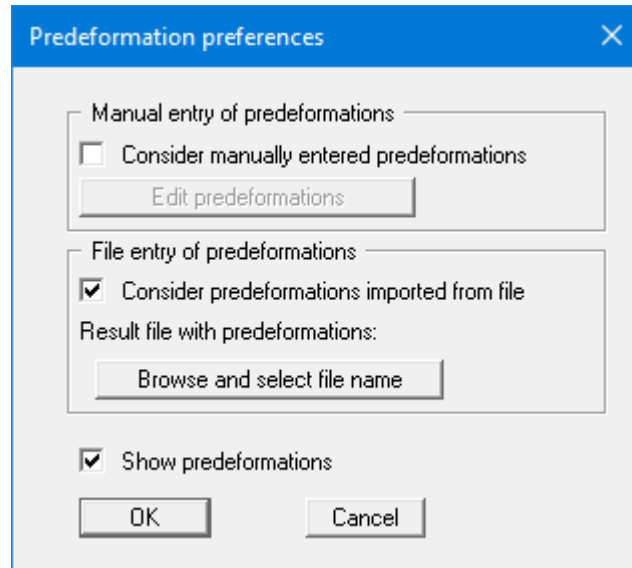
The "**Increments**" are only important for the graphical visualisation of the stress diagram.

9.3.13 "Predeformation info" menu item

You will see information on predeformations.

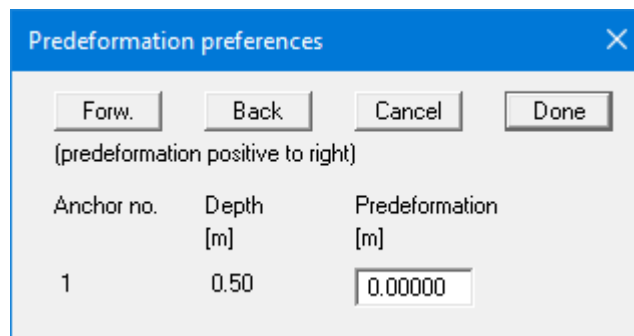
9.3.14 "Predeformation preferences" menu item

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.

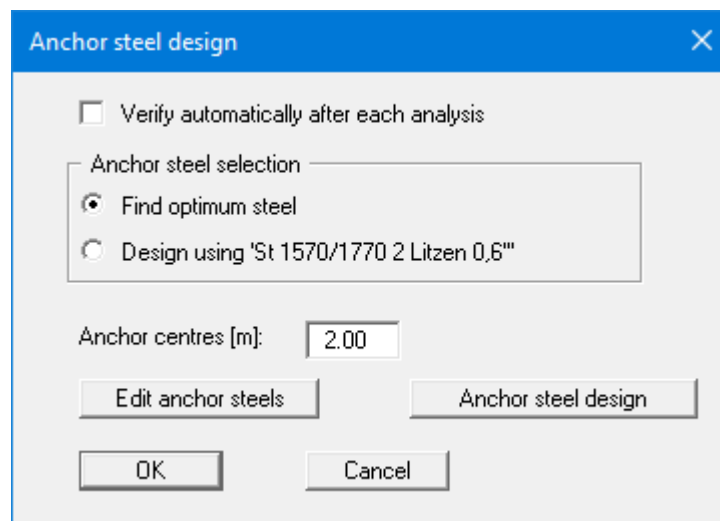
- **File entry of predeformations**
Alternatively, predeformation data can be automatically imported from an existing **GGU-CANTILEVER** file. Select "**Consider predeformations imported from file**" and "**Browse and select file name**". A dialog box will open for selecting the appropriate file, which 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 during 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.

9.3.15 "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 pretensioned 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!

9.4 System menu

9.4.1 "Info" menu item

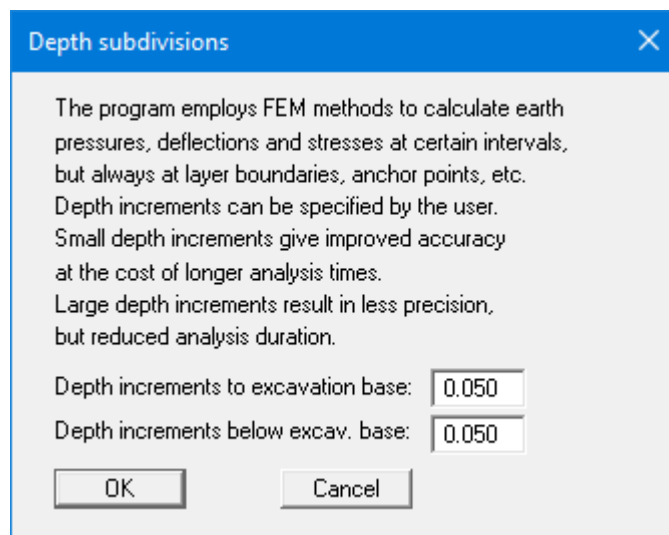
You will see information on the current system in a message box.

9.4.2 "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.

9.4.3 "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 8.15). You can specify the size of these depth increments for the region above and below the excavation base.



9.4.4 "Analyse" menu item

9.4.4.1 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 9.4.4.2 to 9.4.4.4 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, shown here for a system with anchors.

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

Vertical ?

☐ Active wall friction angles for concrete wall ?

☐ Active wall friction angles for equivalent wall ?

Type of redistribution

- ☒ Do not redistribute
- ☐ EAB 1988
- ☐ EAB 2012 / 2021
- ☐ Rectangular
- ☐ Birectangular Preferences
- ☐ Triangular Preferences
- ☐ Trapezoidal Preferences
- ☐ Quadrilateral Preferences
- ☐ User-defined Preferences

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 9.4.6 (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 9.1.4).

9.4.4.2 "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. This creates a torsion spring at the base of the wall.

9.4.4.3 "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 / 2021"**
Redistribution diagrams are given as a function of anchor positioning in EAB 1988 and EAB 2012 or 2021. 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 (eaho/eahu), as well as depth of the subdivision x , can be specified.

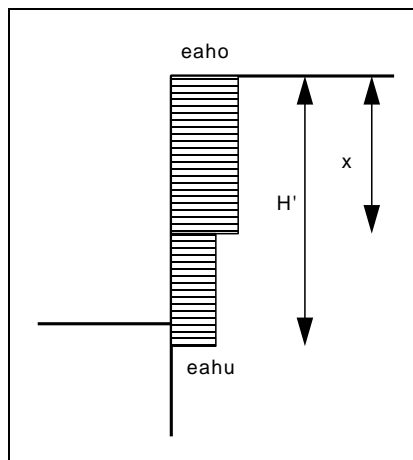


Figure 21 Birectangular earth pressure redistribution

- **"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.

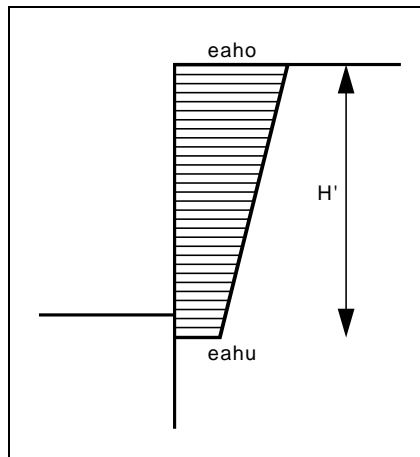


Figure 22 Earth pressure redistribution in a trapezoidal

- **"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} .

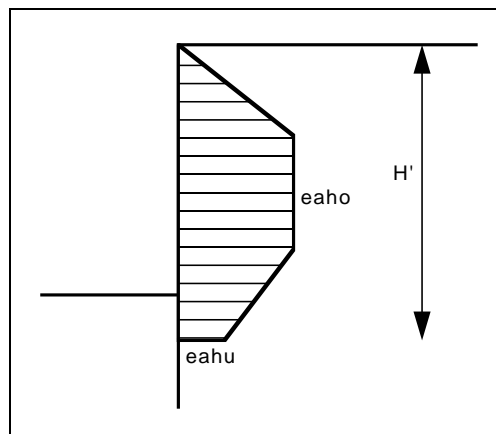


Figure 23 Earth pressure redistribution in a quadrilateral

- **"User-defined"**

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

No.	Depth [m]	eah' [-]
1	Wall top	1.0000
2	1.0000	3.0000
3	3.0000	3.0000
4	4.5000	6.0000
5	5.5000	6.0000
6	Excavation base	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:

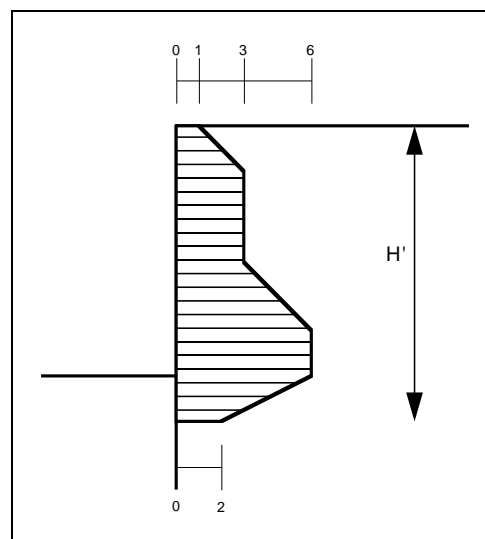


Figure 24 User defined earth pressure redistribution

9.4.4.4 "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.

9.4.5 "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 9.4.4). The following dialog box then opens:

Generate cantilever wall

Wall

Height H [m]	3.500
Thickness (top) do [m]	0.250
Thickness (bottom) du [m]	0.250
Wall inclination beta [°]	0.000
Base inclination alpha [°]	0.000
Thickness (bottom) for vertical air-side wall	

Toe

Toe width bl [m]	0.500
Toe thickness sl [m]	0.250
Depth of toe below stem dl [m]	0.150

Heel

Heel width br [m]	1.500
Heel thickness sr [m]	0.250
Depth of heel below stem dr [m]	0.150

OK Cancel

The diagram illustrates the cross-section of a cantilever wall. The wall has a vertical stem of height H. The top thickness is do, and the bottom thickness is du. The wall is inclined at an angle beta from the vertical. The base is inclined at an angle alpha from the horizontal. The toe (left spur) has a width bl, thickness sl, and depth dl below the stem. The heel (right spur) has a width br, thickness sr, and depth dr below the stem. The total width at the base is bl + du + br.

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 9.4.6).

9.4.6 "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 B500

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 $Q_d < V_{Rd,c}$
Otherwise, minimum reinforcement displayed. see EC 2 Section 6.2.2

☒ For wall shear verification

☒ For toe and heel shear verification

☒ Reduce shear force for changeable height

Type of output table for graphics legend

Only reinforcement

(comprehensive data in output table)

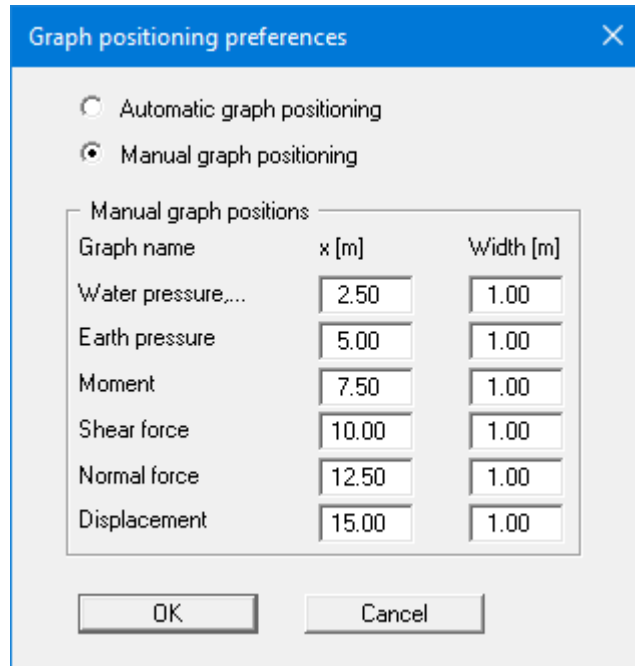
OK Cancel

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 9.1.5).

9.4.7 "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.



Graph name	x [m]	Width [m]
Water pressure,...	2.50	1.00
Earth pressure	5.00	1.00
Moment	7.50	1.00
Shear force	10.00	1.00
Normal force	12.50	1.00
Displacement	15.00	1.00

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.

9.4.8 "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.

Graphics output preferences

Water presentation

☐ None ☐ Diff. water pressure

☒ Pw pressure ☐ Potential

☐ Gradient ☐ Ep + pw

Show system

Show results

Cancel

Earth pressure and state variable hatching

☒ Fixed spacing Spacing [mm] = 2.0

Presentation internal forces etc.

☒ Earth pressure ☒ Moment ☒ Shear force

☒ Displacement ☒ Normal force ☐ Water uplift

☒ Area loads ☒ Deep-seated stab. ☒ Soil pressures

☐ Settlements ☐ Logarithmic spiral

Display (earth pressure):

☐ d ☒ (g+q).k ☐ g.k ☐ q.k

Display (state variables):

☐ d ☐ g.d ☒ (g+q).k ☐ g.k ☐ q.k

Display (displacement):

☐ d ☐ g.d ☒ (g+q).k ☐ g.k ☐ q.k

☒ With footing rotation

Load visualisation

Visualisation height [m]: 0.300 Load 'hatching': Hatched

☐ Same height for all

Hatch spacing [mm]: 2.0

Overheightening passive/active ep [-]: 1.0

State variable for strut: Moment

☒ Display water pressure only when water pressure <> 0.0

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.

9.4.9 "Labelling preferences" menu item

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

Labelling preferences

☐ Label active berm ☒ Show active berms inclination

☐ Label passive berms ☒ Show passive berms inclination

☐ Berm inclination in degrees

☐ Label loads ☐ Label wall

☐ Label logarithmic spiral ☒ Label stress distribution

☒ Show anchor steels ☒ Show anchor lengths/inclinations

☒ Show loads (area of influence)

Decimal places (displacement):

Distributed load labelling: ☐ Left ☒ Central ☐ Right

Groundwater labelling (left):

Groundwater labelling (right):

Anchor labelling:

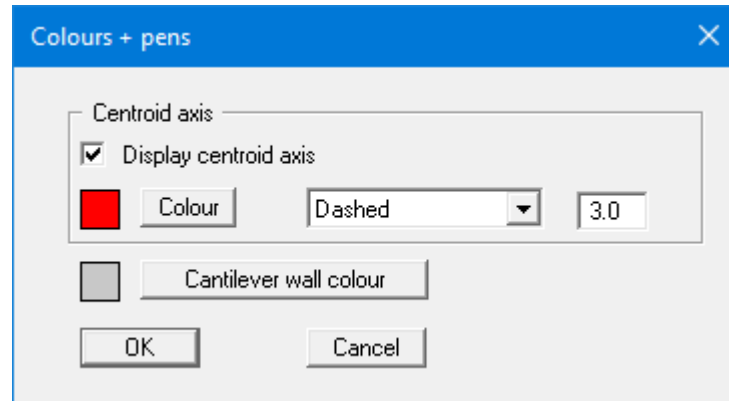
Font size [mm] Force arrow factor [-]

Width of grouted sections [m]:

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.

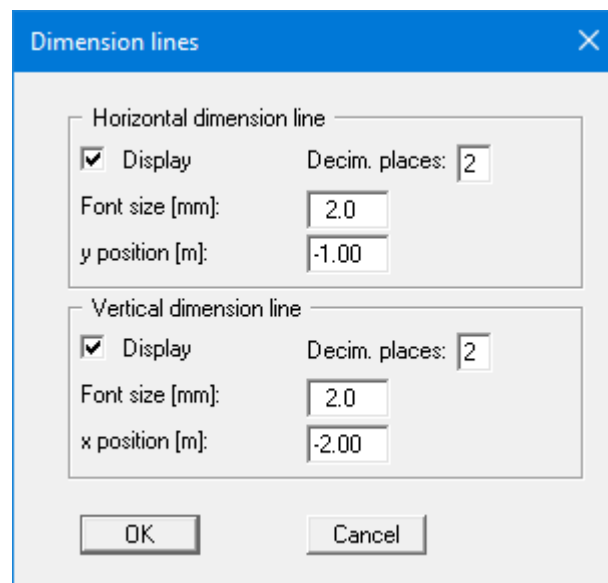
9.4.10 "Wall pens + colours" menu item

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.



9.4.11 "Dimension lines" menu item

You can define a vertical and/or horizontal dimension line for the graphics 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 9.7.3). 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.

9.4.12 "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.

9.4.13 "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.

9.5 *Evaluation menu*

9.5.1 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.

9.5.2 "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.

9.5.3 "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.

9.5.4 "Interim results" menu item

A message box containing the results described in Section 8.1 is displayed. This function can also be accessed by pressing [F6].

9.5.5 "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.

9.5.6 "Sliding safety" menu item

The calculated sliding safety is displayed in a message box.

9.5.7 "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.

9.5.8 "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.

Deep-seated stability factors of safety

OK Forw. Back

Starting point of slip plane in wall area = 3.50 m

$A_{h,g,d} = A_{h,g,k} \cdot \gamma_{m,g}$ and $A_{h,g+q} = A_{h,g,k} \cdot \gamma_{m,g} + A_{h,q,k} \cdot \gamma_{m,q}$

Poss. $A_{h,g} = \text{Poss. } A_{h,g,k} / \gamma_{m,g}$ and Poss. $A_{h,g+q} = \text{Poss. } A_{h,g+q,k} / \gamma_{m,g}$

μ = utilisation factor ≤ 1.0

No.	Depth [m]	Length [m]	$A_{h,g+q}$ [kN/m]	Poss. $A_{h,g+q}$ [kN/m]	$\mu(g+q)$ [-]	$A_{h,g,d}$ [kN/m]	Poss. $A_{h,g}$ [kN/m]	μ,g [-]
1	0.10	8.00	15.5	123.7	0.125	15.5	123.7	0.125

Optimise

In addition to the information with regard to the anchors, the critical utilisation factor for deep-seated stability for each anchor is given. In the example shown here, the anchor has an utilisation factor 0.14 which is below the required value of 1.0. Select the "**Optimise**" button to optimise the length of the anchor.

Optimise anchor length

Anchor length increment [m]: 0.050

Min. anchor length [m]: 4.000

Max. anchor length [m]: 35.000

Current anchor length = 8.00

☒ Search completely

Optimisation can be stopped during analysis by pressing the right mouse button.

OK Cancel

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*.

9.5.9 "Hydraulic heave FOS summary" menu item

You obtain all data relating to verification of hydraulic heave safety displayed in a message box (see Section 8.25).

9.5.10 "Buoyancy FOS summary" menu item

You obtain all data relating to verification of buoyancy safety displayed in a message box (see Section 8.26).

9.5.11 "Anchor and strut summary" menu item

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

9.5.12 "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 9.2.18).

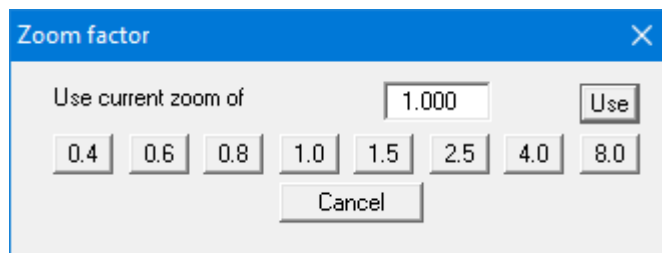
9.5.13 "EQU analysis" menu item

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

9.6.1 "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.

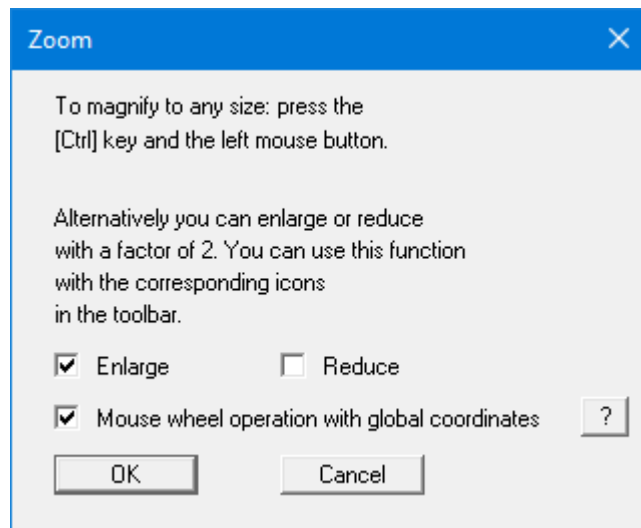


A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "Use" to exit the box the current factor is accepted. By clicking on the "0.4", "0.6", etc. buttons, the selected factor is used directly, and the dialog box closed.

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.

9.6.2 "Zoom info" menu item

You can enlarge a section of the screen by clicking on two diagonally opposite points to see details better. An info box informs you about the activation and possibilities of the zoom functions.



A quicker way to zoom in and out of the system graphics is to use the mouse wheel. When the program is started for the first time, the default setting for mouse wheel operation is activated according to Windows conventions.

If you want to change the system coordinates and the scale of your system using the mouse wheel, activate the "**Mouse wheel operation with global coordinates**" check box. If you close the program with this setting, the setting is still activated the next time you start it.

The functions via the mouse wheel can be found in the chapter "**Tips and Tricks/Keyboard and Mouse**" (see Section 6.3, page 13).

9.6.3 "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.

9.6.4 "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.

9.6.5 "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.). PDF files can also be imported as images. The same pop-up menu opens for both menu items, the icons and functions used are described in more detail in the **Mini-CAD** manual saved in the 'C:\Program Files (x86)\GGU-Software\Manuals' folder during installation. The differences between the Mini-CAD and Header CAD are as follows:

- Objects created with **Mini-CAD** are based on the coordinate system (generally in metres), in which the drawing is produced, and are shown accordingly. You should use the "**Mini-CAD toolbar**" when you wish to add information to the system (for example, labelling of slope inclinations or the location of any footings).
- Objects created with the **Header CAD** are based on the page format (in mm). This makes you independent of the coordinate system and keeps you in the same position on the page. You should select the "**Header toolbar**" if you wish to place general information on the drawing (company logo, report numbers, plan numbers, stamp etc.). Once you have saved the header information to disk (see **Mini-CAD** user manual), you can load it into completely different systems (with different system coordinates). The saved header information will appear in the same position on the page, which greatly simplifies the creation of general page information.

9.6.6 "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 6.5).



"Colour on/off"

If you need to remove the colour from the system presentation, to create a black and white printout, for example, use this on/off switch.



"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.

9.6.7 "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.

Soil properties legend

Position soil properties legend
relative to bottom left page margin in mm

☒ Show legend

x [mm]: 60.00

y [mm]: 72.00

Font size [mm]: 2.5

☐ Background colour

☐ With depths

☐ Soil designations in main graphics

☒ Legend coloured

☐ With permeabilities

☐ With constrained modulus

Soil colours (passive side)

OK Soil colours Cancel

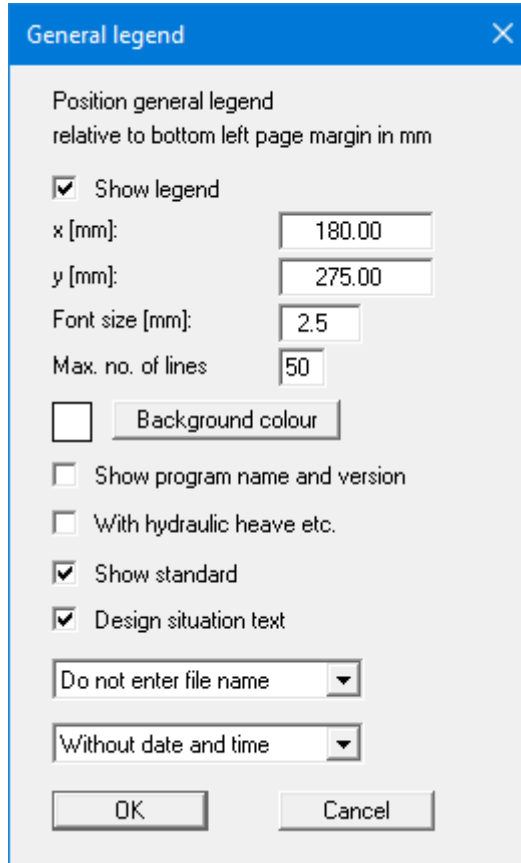
You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**".

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.

- "**Background colour**"
Using this button you can define the background colour for the legend.
- "**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.
- "**With constrained modulus**"
The constrained modulus 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.

9.6.8 "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.



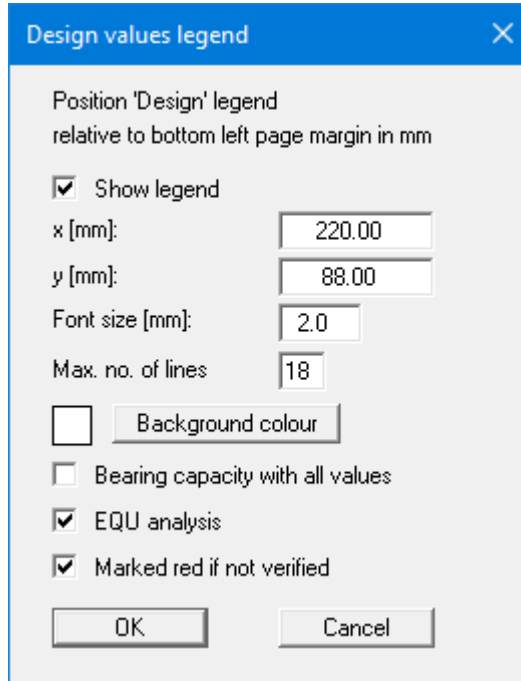
You can define and edit the position of the legend using the values "**x value**" and "**y value**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used. Additionally, you can define a background colour for the legend.

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.

9.6.9 "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. Additionally, you can define a background colour for the legend.

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 9.1.5).

9.6.10 "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 also display the footing rotation.

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. Additionally, you can define a background colour for the legend.

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.

9.6.11 "Move objects" menu item

Select this menu item 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 9.4.7), the result graphs can also be repositioned using this function.

9.6.12 "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"**.

9.6.13 "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 re-freshed.

9.7 Page size + margins menu

9.7.1 "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.

9.7.2 "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.

9.7.3 "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 9.7.6).

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

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

9.7.4 "Zoom" menu item

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

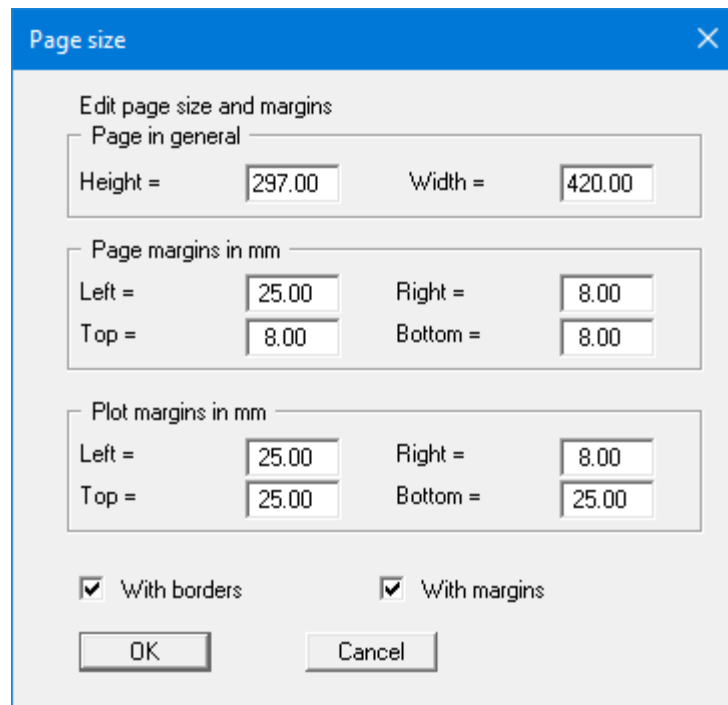
9.7.5 "Font size selection" menu item

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.

9.7.6 "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 size

Edit page size and margins

Page in general

Height = 297.00 Width = 420.00

Page margins in mm

Left = 25.00 Right = 8.00
Top = 8.00 Bottom = 8.00

Plot margins in mm

Left = 25.00 Right = 8.00
Top = 25.00 Bottom = 25.00

☒ With borders ☒ With margins

OK Cancel

- **"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 margins"** define 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 margins"** define a set distance between the page margin and the actual *drawing area* in which the graphical evaluation of your input is presented.

9.7.7 "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 9.6.6).

9.7.8 "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 9.6.6).

9.7.9 "Preferences" menu item

You can activate or deactivate the undo functions.

9.8 Info menu

9.8.1 "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**.

9.8.2 "Maxima" menu item

Here you can check the defaults for maximum values.

9.8.3 "Compare earth pressure coefficients" menu item

Here you can calculate the earth pressure coefficients for given values of ϕ , δ and β .

9.8.4 "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.

9.8.5 "GGU on the web" menu item

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

Check the page of your GGU program module at regular intervals for information about updates and changes. On the "**Changelogs**" subpage, you can also subscribe to an email notification that informs you of all changes monthly.

9.8.6 "GGU support" menu item

This menu item takes you to the GGU support portal. You can enter your support request there and upload your edited GGU program file.

9.8.7 "What's new?" menu item

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

9.8.8 "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 dialogues, menus)**" und "**Graphiktexte übersetzen (translate graphics)**".

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

10 Index

2

2nd order theory, select for analysis..... 88

A

Absolute heights, use 17, 50

Action boundary conditions,

define..... 77

equivalent for pretensioning 41

Active earth pressure,

analysis principles 27

calculation after Culmann 27

edit preferences 63

select for analysis 62

Anchor length, optimise..... 96

Anchor steel design 83

Anchor wall 78

Anchors,

consider pretensioning..... 41

define..... 78

define pretensioning 80

display design values in a message box..... 97

Angle of wall friction,

enter/adopt from soil database 61

Area loads,

consider in analysis 34

define..... 73

select ep analysis method 65

to DIN 4085 35

At-rest earth pressure,

coefficient to DIN 4085..... 27

keep constant..... 65

select for analysis 62

B

Bearing capacity failure plane, determine..... 42

Bearing capacity safety,

analysis principles 42

display results in a message box..... 95

Bending line, display with predeformations 83

Berms,

activate labelling 93

define on active side..... 60

define on passive side..... 60

Bounded surcharges,

consider in analysis 36, 38

define..... 75

Buoyancy safety,

analysis using global safety factors 45

analysis using partial safety factors 46

display results in a message box..... 97

C

Cantilever wall,

define colour 94

define geometry..... 20

define loads 19

define rotation/displacement 77

optimise geometry 89

Caquot/Kerisel,

passive ep analysis 28

select for passive ep analysis..... 64

Centroid axis/line, activate display 94

Characteristic point,

display settlements 96

limiting depth determination 81

Cholesky 39

Clipboard 56

Cohesion,

coefficient determination method 27

enter/adopt from soil database 61

Colour,

define for cantilever wall..... 94

define for soils..... 102

switch on/off..... 101

Colour/hatching, define for loads 93

Colour/pen,

define for centroid axis/line..... 94

define for graphical elements 99

Compaction earth pressure,

adopt to DIN 4085-2011/Franke 74

Company letterhead, add via Mini-CAD 100

Concrete, values for structural analysis 19

Constrained modulus,

display in legend..... 102

for settlement analysis 44

Context menu, open..... 12

Coordinates,

alter mouse wheel..... 13

alter via editor 106

alter with mouse 106

optimise/reset 106

save/load..... 106

Copy/print area 15, 56, 101

Crack width, activate verification 90

Culmann,

active ep calculation 27

select for active ep analysis 63

select for passive ep analysis..... 64

Culmann, passive ep calculation..... 28

Cutting borders, switch on/off 107

D

Database,

for soil properties of common soils 61

Dataset description,

display 103

enter..... 49

Deep-seated stability,

define intersection with wall 71

safety/utilisation factor for anchors 96

verification principles..... 46

Delete, soil layers 61

Design,

display results in legend 104

edit defaults 90

principles 40, 48

Dimension lines, define 94

DIN 4017 42

DIN 4019	44
DIN 4085,	
consider in analysis	27
passive ep analysis	28
select for passive ep analysis	64
Displacement boundary conditions, define	77
Displacement, define for cantilever wall	77
Distributed load, define	59
Double-bounded surcharges,	
consider in analysis	37, 38
define	76
Drawing area, define	107
DXF file,	
export	56
import via Mini-CAD	7

E

Earth pressure coefficients,	
be calculated	109
calculate	66
user-defined	66
Earth pressure redistribution,	
display in a message box	95
to EAB	86
user-defined	88
Earth pressure,	
preferences for active	63
select for analysis	62
Earth quake, as design situation to EC 7	69
EC 7, load cases description	69
Editor window, output table	54
Elevation head, for potential definition	31
EMF format	56
Equation solution to Cholesky	39
Equivalent earth pressure coefficient,	
activate use	63
Excavation,	
change alignment	49
define base	59

F

File,	
display name in legend	103
load/save	50
Finite element module,	
for structural system	39
for water pressure determination	31
Flow force,	
characteristic	45
for hydraulic heave safety	44
influence on earth pressure	66
Font selection	99
Font size,	
design legend	104
dimension lines	94
edit for graphical elements	106
general legend	103
settlement legend	105
soil properties legend	102
system/result graphics	93
Footer, output table	52
Footing length, define	19
Footing rotation, display in settlement legend	105

Force boundary conditions,	
equivalent for pretensioning	41
Friction angle,	
enter/adopt from soil database	61
Friction spur, define	19
Function keys	11

G

General page information,	
add via Mini-CAD	100
General stability,	
verification using file export	43, 55
GGU-CAD file, export	56
GGUMiniCAD file, export	57
GGU-STABILITY file, export	55
Global coordinates,	
use for Mini-CAD output	57
Global safety factors/verifications, define	67
Graph positions,	
alter via numerical input	91
alter with mouse	105
Graphics, add via Mini-CAD	100
Groundwater level, enter	59
Groundwater,	
activate labelling	93
consider storeys/confined aquifers	31
define colour	99
enter additional potentials	80
Grouted section,	
enter length for graphics	78
enter width for graphics	93

H

Hatching, define for loads	93
Header CAD, application explanations	100
Header, output table	52
Heave of anchor soil,	
activate analysis	71
present results in a message box	97
Hydraulic gradient,	
calculation	31, 32
determine for analysis	66
Hydraulic heave safety,	
analysis using global safety factors	44
analysis using partial safety factors	45
display results in a message box	97
display results in legend	103

I

Increased active earth pressure,	
coefficient	28
select for analysis	62
Interim results, display in a message box	95

L

Labelling, system/result graphics	93
Language preferences	10, 109
Lateral pressures, define	72
Layout, output table	52
Legends, move with mouse	105
Licence protection	9

Limiting depth,	
defining methods	44
select determination	81
Live load, consider distributed load as	59
Load cases,	
adopt to DIN 1054-2010/EC 7	69
adopt to ÖNORM EN 1997-1	69
adopt to SIA 267	69
Load concentration factor,	
consider	35
define	65
Loads with limited plan dimensions,	
define	73
reduce	35, 73
Loads, edit graphical output preferences	93
Logarithmic spiral,	
display below wall	92
in bearing capacity failure plane	42

M

Manual, open as PDF file	109
Maximum values, display for system	109
Mean,	
friction angle	42
soil properties	42
Metafile, export	56
Mini-CAD,	
application explanations	100
export file	57
output with global coordinates	57
Moment load, sign rule for reinforcement	48
Mouse click functions	12
Mouse wheel functions,	
apply to global coordinates	13, 99

N

Navigation, output table	100
--------------------------------	-----

O

Objects, move with mouse	105
ÖNORM EN 1997-1,	
select partial safety factors	69
Output preferences	54, 55
Output table,	
change to system graphics	100
edit output as ASCII	54
edit output as graphics	52
save/load settings for graphical display	52
select output format	51
Output table, navigation	53
Overburden stress,	
consider in settlement analysis	81
Overturning stability,	
analysis principles	41
deactivate analysis	49

P

Page,	
copy/print section	15, 101
define format	107
switch margins on/off	107

Pagination, automatic	53, 54
Partial safety factors,	
define	68, 69
select from ÖNORM EN 1997-1	69
select from SIA 267	69
Passive earth pressure,	
analysis methods for coefficients	28
calculation after Culmann	28
specify calculation	64
PDF file, import via Mini-CAD	7, 100
Pen preferences, edit for graphical elements	99
Permeabilities,	
consider in analysis	27
display in legend	102
enter/adopt from soil database	61
using flow conduits	30
Piling Handbook,	
consider loads with limited dimensions	35
effect on earth pressure	33
Plausibility checks,	
show during analysis	86
view results	84
Plot margins, define	107
Potential, definition	31
Preconsolidation,	
consider in settlement analysis	81
Predeformations, define	82
Presentation height, define for loads	93
Pretensioning,	
consider in analysis	41
define for anchors/struts	80
Print,	
graphics	55
output table	56
section	15, 56, 101
several files	58
Program,	
display name in legend	103
save/load preferences	105
show improvements	109
show information	109
Project data, add via Mini-CAD	100
Project identification,	
display	103
enter	49

R

Redistribution figures,	
available	41
select	86
Reinforced concrete,	
activate design using at-rest earth pressure ..	88
design principles	48
edit design defaults	90
Reinforcement, sign rule for moments	48
Representation, change to system graphics	53
Result graphics,	
display	95
edit preferences	92
label	93
position	91, 105
Results, display for selected depth	12, 95
Rod construction module	39

Rods,	
define for struts	79
define for system	84
Rotation, define for cantilever wall	77

S

Safety concept, select	49
Safety factors, define	67
Scale,	
alter via factor input	106
alter via mouse wheel	13
alter with mouse	106
define manually	106
determine automatically	106
Scroll the screen	12
Seismic effects,	
as design situation to EC 7	69
consider loads	70
Settlement analysis,	
display results in a message box	96
display results in legend	105
edit preferences	81
limiting depth definition	44
subtraction of preconsolidation loading	44
Settlements, display share in legend	105
Shear reinforcement, edit output preferences	90
SIA 267,	
select partial safety factors	69
Sign rule, moments	48
Sliding safety,	
analysis principles	43
display results in a message box	95
increase using a friction spur	19
Slip planes, compound	46
Slope stability application	55
Smart icons, for menu items	100
Soil colours/numbers, activate display	102
Soil designations,	
activate presentation in system graphics	102
define	61
Soil layers,	
adapt to absolute heights	59
define/delete	61
maximum number	27
Soil properties,	
activate separate input act./pass. values	50
display in legend	101
enter/adopt from soil database	61
Standard, display in legend	103
Status bar main program, activate	100
Stiffness of wall, determination	39, 40
Streck,	
passive ep analysis	28
select for passive ep analysis	64
Stress analysis, for limiting depth	81
Struts,	
consider pretensioning	41
define	79
define pretensioning	80
display design values in a message box	97

System coordinates,	
alter via mouse wheel	13
System,	
analyse	85
display	95
display properties in legend	103
division into rods	84
edit graphics output preferences	92
show information	84, 109
structural	39

T

Theory of elastic half-space	35
Toolbar, edit for menu items	100
Torsion spring at wall base, create	86
Translation, activate	109
True-type font	99

U

Unit weight,	
enter/adopt from soil database	61
User-defined ep coefficients, enter	66
Utilisation factor,	
buoyancy	46
deep-seated stability	96
hydraulic heave	45

V

Verifications,	
activate to DIN 1054 old	67
activate to DIN 1054-2010/EC 7	68, 69
Version number,	
display in a message box	109
display in legend	103

W

Wall toe, select bedding conditions	86
Water pressure approach,	
classical	29
recommendation	31
select for analysis	66
using flow conduits	30
Water pressure,	
enter additional wp	80
user-defined distribution	31
What you see is what you get	98

Y

Young's modulus, define	19
-------------------------------	----

Z

Zoom factor, define for full-screen display	98
Zoom function, activate	99, 100