

GGU-RETAIN manual

Analysis of retaining walls (EAB/EAU)

Exported on 1.9.2026

Table of contents

- 1 Preface
- 2 Capabilities
- 3 Licence protection
- 4 Language selection
- 5 Starting the program
- 6 Tips and tricks
 - 6.1 "?"- and "Info" buttons
 - 6.2 Function keys
 - 6.3 Keyboard and mouse
 - 6.4 Mathematical functions in input boxes with numbers
 - 6.5 "Copy/print area" icon

7 Theoretical principles

7.1 General notes to types of retaining wall

7.2 Soil properties

7.3 Active earth pressure

7.4 At-rest earth pressure

7.5 Increased active earth pressure

7.6 Passive earth pressure

7.7 Water pressure

7.7.1 Approach for sheet pile walls and in-situ concrete walls

7.7.1.1 Classical water pressure approach

7.7.1.2 Water pressure approach using flow conduits

7.7.1.3 Recommendation for water pressure approach

7.7.1.4 Possible conceptual error using flow conduits

7.7.2 Approach for soldier pile walls

7.7.3 What is a potential?

7.8 Berms

7.9 Area loads

7.10 Loads with limited plan dimensions

7.11 Bounded surcharges (active side)

7.12 Double-bounded surcharges (active side)

7.13 GGu-RETAIN: Bounded surcharges (passive side)

7.14 Double-bounded surcharges (passive side)

7.15 Length surcharge

7.16 Structural system

7.17 Design

7.18 2nd order theory

7.19 Bedding conditions of wall toe

7.20 Bedded systems

7.20.1 General information on types

7.20.2 Variant "Section length fixed and bedded toe"

7.20.3 Variant "Automatically determined section length and bedded toe"

7.21 Action and displacement boundary conditions

7.22 GGU-Retain: Predeformations

7.23 Pretensioning

- 7.24 Modulus of subgrade reaction
- 7.25 Excavations and waterfront structures using the p-y method
- 7.26 Earth pressure redistribution
- 7.27 Base heave safety
- 7.28 General stability
- 7.29 Hydraulic heave
 - 7.29.1 Hydraulic heave safety using global safety factors
 - 7.29.2 Utilisation factor (hydraulic heave) using partial safety factors
 - 7.29.3 Analysis of hydraulic heave after Aulbach/Ziegler
- 7.30 Buoyancy
 - 7.30.1 Buoyancy safety using global safety factors
 - 7.30.2 Utilisation factor (buoyancy) using partial safety factors
- 7.31 Analysis of sum H
- 7.32 Analysis of mobilised passive earth pressure
 - 7.32.1 General note on mobilised passive earth pressure
 - 7.32.2 Analysis of mobilised passive earth pressure using global safety factors
 - 7.32.3 Analysis of mobilised passive earth pressure using partial safety factors
- 7.33 Analysis of vertical capacity
- 7.34 Verification of deep-seated stability
- 7.35 Heave of anchor soil
- 7.36 Construction phases

8 File menu

- 8.1 "New" menu item
- 8.2 "Load" menu item
- 8.3 "Save" menu item
- 8.4 "Save as" menu item
- 8.5 "Print output table" menu item
 - 8.5.1 Selecting the output format (Print output table)
 - 8.5.2 "Output as graphics" button (Print output table)
 - 8.5.3 "Output as ASCII" button (Print output table)
- 8.6 "Export to GGU-STABILITY" menu item
- 8.7 "Output preferences" menu item
- 8.8 "Print and export" menu item
- 8.9 GGu-RETAIN: "Batch print" menu item
- 8.10 "Exit" menu item
- 8.11 "1, 2, 3, ..." menu items

9 Editor 1 menu

- 9.1 "Analysis options" menu item
- 9.2 "Excavation" menu item
- 9.3 "Berms (active side)" menu item
- 9.4 "Berms (passive side)" menu item
- 9.5 "Soils" menu item
- 9.6 "Type of earth pressure" menu item
- 9.7 "Active earth pressure" menu item
- 9.8 "Passive earth pressure" menu item
- 9.9 "At-rest earth pressure" menu item
- 9.10 "User-defined earth pressure coefficients" menu item
- 9.11 "Groundwater" menu item
- 9.12 "Seismic effects" menu item
- 9.13 "Verifications/Safety factors" menu item
- 9.14 "Verifications/Partial factors" menu item
- 9.15 "Deep-seated stability/Heave of anchor soil" menu item
- 9.16 "Buoyancy + Hydraulic heave" menu item
- 9.17 "Pull-out resistance" menu item
- 9.18 "Common systems" menu item

10 Editor 2 menu

10.1 "Lateral pressures" menu item

10.2 "Area loads" menu item

10.3 "Bounded surcharges" menu item

10.4 "Double-bounded surcharges" menu item

10.5 "Action boundary conditions" menu item

10.6 "Displacement boundary conditions" menu item

10.7 "Anchors" menu item

10.8 "Struts" menu item

10.9 "Pretensioning" menu item

10.10 "Groundwater potentials" menu item

10.11 "Subgrade reaction moduli" menu item

10.12 "Compaction earth pressure" menu item

10.13 "Predeformation information" menu item

10.14 "Predeformation preferences" menu item

10.15 "Soldier pile"/"Sections"/"Bored pile wall"/"Diaphragm wall"/ "Bored soldier pile wall"/"Girder"/"Section data"/"Inserted beams" menu items

10.15.1 General note to soldier piles and sections

10.15.2 "Soldier piles" menu item

10.15.3 "Sections" menu item

10.15.4 "Bored pile wall"/"Diaphragm wall"/"Bored soldier pile wall" menu items

10.15.5 "Girder" menu item

10.15.6 "Section data" menu item

10.15.7 "Inserted beams" menu item

10.16 "Young's modulus/Specific weight"/"Specific weight" menu items

10.17 "Anchor steel design" menu item

10.18 "Waling design" menu item

11 System menu

11.1 "Info" menu item

11.2 "Special preferences" menu item

11.3 "Depth subdivisions" menu item

11.4 "Length surcharge" menu item

11.5 "Analyse" menu item

11.5.1 Start dialog box (Analyse)

11.5.2 "Embedment depth via:" group box (Analyse)

11.5.3 "Special preferences" group box (Analyse)

11.5.4 Type of redistribution (Analyse)

11.6 "Design defaults" menu item

11.7 "Graph positioning preferences" menu item

11.8 "Graphics output preferences" menu item

11.9 "Labelling preferences" menu item

11.10 "Graph grid preferences" menu item

11.11 "Dimension lines" menu item

11.12 "Display system" menu item

11.13 "Display results" menu item

12 Evaluation menu

12.1 General note on evaluation

12.2 "Earth pressure redistribution" menu item

12.3 "Main output summary" menu item

12.4 "Maximum reaction summary" menu item

12.5 "Anchor and strut summary" menu item

12.6 "Deep-seated stability summary" menu item

12.7 "Sum V FOS summary" menu item

12.8 "Sum H FOS summary" menu item

12.9 "Hydraulic heave FOS summary" menu item

12.10 "Buoyancy FOS summary" menu item

12.11 "Heave of anchor soil" menu item

12.12 "Heave FOS summary" menu item

12.13 "Verification of pull-out resistance" menu item

12.14 "Earth support analysis" menu item

12.15 "Grouting points analysis" menu item

12.16 "Welding lengths analysis" menu item

13 Construction phases menu

- 13.1 General notes on construction phases
- 13.2 "Info" menu item (Construction phases)
- 13.3 "Select files" menu item
- 13.4 "Display summary" menu item

14 Graphics preferences menu

- 14.1 "Refresh and zoom" menu item
- 14.2 "Zoom info" menu item
- 14.3 "Legend font selection" menu item
- 14.4 "Pen colour and width" menu item
- 14.5 "Mini-CAD toolbar" and "Header toolbar" menu items
- 14.6 "Toolbar preferences" menu item
- 14.7 "Soil properties legend" menu item
- 14.8 "General legend" menu item
- 14.9 "Design legend" menu item
- 14.10 "Subgrade modulus legend" menu item
- 14.11 "p-y curves legend" menu item
- 14.12 "Retaining wall diagram" menu item
- 14.13 "Move objects" menu item
- 14.14 "Save graphics preferences" menu item
- 14.15 Menüeintrag "Einstellungen laden"

15 Page size + margins menu

- 15.1 "Auto-resize" menu item
- 15.2 "Manual resize (mouse)" menu item
- 15.3 "Manual resize (editor)" menu item
- 15.4 "Zoom" menu item
- 15.5 "Font size selection" menu item
- 15.6 "Page size and margins" menu item
- 15.7 GGu-RETAIN: "Undo" menu item
- 15.8 "Restore" menu item
- 15.9 "Preferences" menu item

16 Info menu

- 16.1 "Copyright" menu item
- 16.2 "GGU on the web" menu item
- 16.3 "GGU support" menu item
- 16.4 "Maxima" menu item
- 16.5 "Active wall friction angle" menu item
- 16.6 "Compare earth pressure coefficients" menu item
- 16.7 "Vertical capacity" menu item
- 16.8 "Active earth pressure (constrained slip surface)" menu item
- 16.9 "Steel design to DIN EN 1993" menu item
- 16.10 "Grouting points analysis (U sections)" menu item
- 16.11 "Welding lengths analysis (U sections)" menu item
- 16.12 "Help" menu item
- 16.13 "What's new?" menu item
- 16.14 "Language preferences" menu item



Theorie. Und Praxis.

Geotechnische Softwarelösungen können so einfach sein. Denn Theorie und Praxis lassen sich mit **GGU-Software** und den neuen Angeboten der **civilserve Academy** prima kombinieren: Knackige theoretische Aufgaben lösen und als Sahnehäubchen Ihr

**Know-how durch
Praxisseminare
vertiefen!**

Civilserve GmbH
Exklusivvertrieb GGU-Software
Weuert 5 · D-49439 Steinfeld
Tel. +49(0)5492 6099996
info@ggu-software.com

Infos und Termine zu unseren Präsenz- und
Online-Seminaren jetzt unter

www.ggu-software.com

1 GGU-RETAIN: Preface

GGU-RETAIN allows the analysis of sheet pile walls, soldier pile walls and in-situ concrete walls (bored pile walls, diaphragm walls and bored soldier pile walls) as well as FMI (cut, mix, inject) walls and combined sheet pile walls.

The application is essentially based on the Recommendations of the Working Group for Excavations (German: *EAB*) and the Recommendations of the Committee for Waterfront Structures (*EAU 2012*). Virtually all the cases considered by the Working Groups are included as examples. A special feature of the application is automatically finding the earth-pressure redistribution values suggested by the EAB. For analysis and design both the **global safety factor concept** to DIN 1054 (old) and the **partial safety factor concept** to EC 7 may be taken into consideration.

The program includes convenient data input. The system is permanently displayed on the screen. Every change to the data is displayed on the screen, so that optimum control of the input data is guaranteed.

There are "?" and "Info" buttons in the dialog boxes for many technical and program-specific questions. Click on the "?" buttons or "Info" buttons to obtain the necessary information.

The graphic output supports the true type fonts provided by WINDOWS, so that an excellent layout is guaranteed. Colour output and numerous graphic formats (BMP, TIF, JPG etc.) are supported. PDF and DXF files can also be imported via the integrated **Mini-CAD** system (see "**Mini-CAD**" manual).

The program has been extensively tested. No errors 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 GGU-RETAIN: Capabilities

GGU-RETAIN has the following characteristics and maximum capabilities

- Up to 50 soil layers
- Transfer of soil properties from **GGU-CONNECT**
- Up to 40 berms on the active earth pressure side
- Up to 40 berms on the passive earth pressure side
- Analysis with active and increased active earth pressure and with at-rest earth pressure
- Coefficients of active earth pressure pursuant to DIN 4085
- Coefficients of passive earth pressure pursuant to DIN 4085, Streck, Caquot/Kerisel
- Additional calculation of active and passive earth pressure after Culmann possible.
- Classical water pressure approach for impermeable retaining walls and, alternatively, by means of flow conduits 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 passive earth pressure sides
- Verification of safety against deep-seated failure with optimisation of anchor lengths
- Analysis of hydraulic heave 'classical' or after Aulbach/Ziegler
- Verification of buoyancy safety
- Verification of base heave safety
- Verification of pull-out resistance of driven steel tubular piles and injection piles
- Verification of sum H
- Analysis of mobilised passive earth pressure
- Analysis of vertical capacity
- Convenient interface to the stability analysis application, **GGU-STABILITY**, for quick determination of safety factor against general failure
- Up to 50 additional earth pressure distributions
- Up to 20 area loads at any depth

- Structural analysis of the retaining wall by means of a two-dimensional rod construction module based on finite-element methods. In contrast to many other retaining wall analysis 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 retaining wall unnecessary. In addition, it is possible to consider loads on the struts (e.g. for auxiliary bridges additionally acting as struts) during analysis.
- Buckling analysis to DIN EN 1993-1-1 using 2nd order theory
- Elastic subgrade reaction at the wall toe with user-defined values. Option for automatic selection of an elastic modulus of subgrade reaction profile consistent with the specified passive earth pressure distribution
- 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 anchors can be incorporated.
- Consideration of prestressing of anchors and struts possible
- Definition of up to 20 additional potentials by the calculation of flow conduits to the left and right of the wall for complex groundwater conditions
- Expandable database of default sections for soldier piles, sheet piles, combined sheet piles and tubular sections with which automatic design, including automatic searches for the optimum section, can be carried out.
- Reinforced concrete design to EC 2 for circular and rectangular cross-sections
- Modelling corrosion for all sections
- Design of horizontal lagging for soldier pile walls
- Automatic computation of section weights for analysis of sum V
- Variable bending resistance of the retaining wall
- Automatic computations of earth pressure redistributions according to the EAB Recommendations
- Other earth pressure distribution options:
 - no redistribution
 - rectangle
 - 2 rectangles
 - triangle (maximum can be optionally placed at top, middle or bottom)
 - trapezoidal
 - quadrilateral with maximum at anchor locations or any other location
 - user-defined polygon
- EAU 2012

- Soil pressure can be redistributed either to the wall toe or to the transition point.
- Passive earth pressure can be superimposed or located in front of the wall.
- Calculation of the transition point with or without porewater pressure
- Structural analysis can be performed in 4 different ways:
 - section length determined automatically after entering degree of fixation for the wall toe
 - determination of degree of fixation for fixed section length
 - section length automatically determined with elastic bedding of wall toe
 - bedding of wall toe with fixed section length
- Following computation of the system, earth pressure, porewater pressure, moment, shear, normal force and bending line are displayed on the screen. The visualisation can be varied within wide limits. For example, the distribution of the modulus of subgrade reaction as well as the distributions of the potential and the gradient, etc. can also be displayed.
- For soldier pile walls it is necessary to demonstrate the equilibrium of horizontal forces below the excavation level. This is carried out by the application. If necessary, the section length is automatically increased.
- At anchor points, existing predeformations can be defined as boundary conditions.
- Previously calculated datasets can be combined to investigate the additive deformation of individual advancing and retreating states. A Mohr's envelope showing the distribution of moments, shear and normal forces can also be depicted. Visualisation of the reinforcement envelope is also possible for diaphragm walls, bored pile walls and bored soldier pile walls.
- Legends can be displayed on screen showing soil properties and general computation data. Thus, virtually all the raw data necessary for the computation and the results are shown on screen.
- **GGU-RETAIN'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. 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).
- **GGU-RETAIN** includes virtually all the examples contained in the Krupp Hoesch Stahl "Piling Handbook" ("*Spundwand-Handbuch*") and in Weißenbach's "Excavations" III ("*Baugruben* III", 1977) as datasets.
- By clicking the "**Copy/print area**" icon on the toolbar you can copy any part of the graphics to the clipboard, save it as an EMF file (Enhanced Metafile Format) or print it directly on your printer. Using the "**Mini-CAD toolbar**" or "**Header toolbar**" modules, you can insert EMF files into your graphics. Thus, the results of a slope stability analysis or of a grading analysis, for example, can easily be imported into the current graphics.

3 GGU-RETAIN: Licence protection

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

To be able to use the GGU licences in a CmContainer, a runtime environment, the **CodeMeter Runtime Kit**, must be installed on your computer via driver software. You can obtain the current version from [Software for Users - Wibu-Systems](#) (CodeMeter User Runtime for Windows).

You can work with the following CmContainer types on your PC with the CodeMeter Runtime Kit installed (= **CodeMeter licence server**):

- **CmStick**
The licence for your GGU program is stored in a USB dongle.
- **CmActLicense** (soft licence, not for virtual PC/servers)
The licence for your GGU program is stored in a licence file that is linked to the hardware of a computer.
- **CmCloudContainer**
The licence is located on a CmCloud server of WIBU-Systems and is mirrored to your CodeMeter licence server. The advantages of this type of licence binding are described on the following website: [Licence binding | GGU Software](#).

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

4 GGU-RETAIN: Language selection

The GGU programs can be used in German and English. The program always starts with the language in which it was terminated.

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

5 GGU-RETAIN: 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 menu item "**File/New**"). You then see nine menus in the menu bar:

- File
- Editor 1
- Editor 2
- System
- Evaluation
- Construction phases
- 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 screen is not refreshed after every alteration.

If you would like to refresh the screen contents, press either [F2] or [Esc]. The [Esc] key also resets the screen display to your current image zoom, which is usually preset to 1.0, which corresponds to an A3 or A4 sheet in most GGU programs (see menu item "**Graphics preferences/Refresh and zoom**").

6 Tips and tricks

6.1 GGU-RETAIN: "?"- and "Info" buttons

Reading of the manual can mostly be dispensed with, because

"?" buttons or "Info" buttons

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

6.2 GGU-RETAIN: 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:

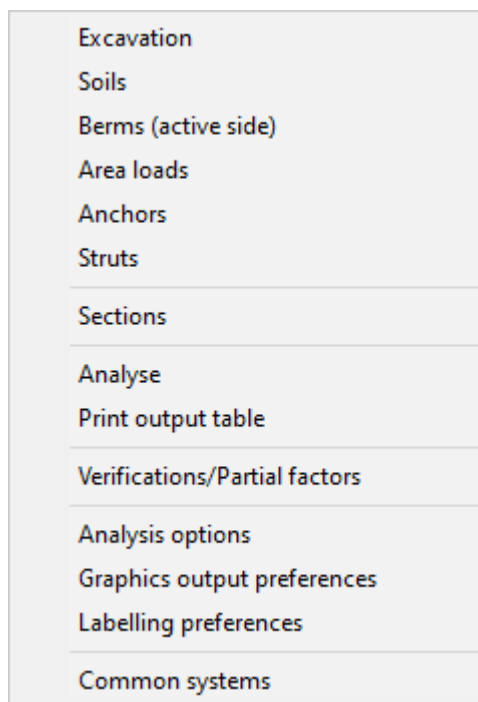
- **[Esc]** refreshes the screen content and resets the screen section to your current image zoom, which is preset to 1.0. This is useful, for example, if you have used the magnifying glass function to display sections of the drawing on the screen and want to quickly return to the overall view.
- **[F1]** opens the manual file.
- **[F2]** refreshes the screen without altering the current magnification.
- **[F11]** activates the menu item "**Move objects**", which appears in the GGU programs under various menu titles, e.g. "**Page size + margins**", "**Graphics preferences**" or "**Output preferences**".

In this GGU program, the following function keys are also assigned to program functions:

- **[F5]** opens the menu item "**System/Analyse**".
- **[F6]** opens the menu item "**Evaluation/Main output summary**".
- **[F9]** opens the menu item "**Page size + margins/Auto-resize**".

6.3 GGU-RETAIN: Keyboard and mouse

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



You can quickly design various new systems using the "**Common systems**" menu. Without having to work through the various menu items one after the other, you can enter the essential elements such as excavation geometry, groundwater, anchors, soil properties, etc. in just one dialog box (see menu item "**Editor 1/Common systems**").

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.

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 "**Tips and tricks: Mathematical functions in input boxes with numbers**" 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
- **[Strg]** + mouse wheel up = enlarge screen image (zoom in)

[Strg] + mouse wheel down = reduce screen image (zoom out)

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

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

6.4 GGU-RETAIN: Mathematical functions in input boxes with numbers

In input boxes for numerical values, you can use the following arithmetic operations (also in combinations) to determine the desired value or to adjust the existing value. After pressing the **[Return]** or **[Enter]** key, the calculated value is inserted.

Function	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:	
25	2^5
Radify:	
Root of 27	$w(27)$ or $27^{(1/2)}$
5th root of 81,5	$81,5^{(1/5)}$
Sine, cosine, tangent etc.	
$\sin(32^\circ)$	$\sin(32)$
$\cos(5,23^\circ)$	$\cos(5,23)$
$\tan(45^\circ)$	$\tan(45)$
$\arctan(1,0)$	$\text{atan}(1,0) = 45^\circ$
Natural logarithm:	
$\ln(4,53)$	$\ln(4,53) = 1,5107$
Exponential function:	
$e^{1,5107}$	$\text{ep}(1,5107) = 4,53$

6.5 GGU-RETAIN: "Copy/print area" icon

A dialog box opens when the **"Copy/print area"** icon

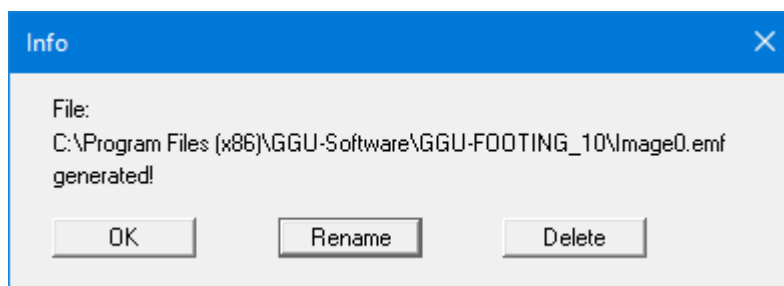


in the menu toolbar is clicked, describing the options available for this function. For example, using this icon it is possible to either copy areas of the screen graphics and paste them into the report, or send them directly to a printer.

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 appears after defining the area, shown here as an example for the **GGU-FOOTING** program:



The default location of the file is the folder from which the program is started (here: GGU-FOOTING_10) 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.

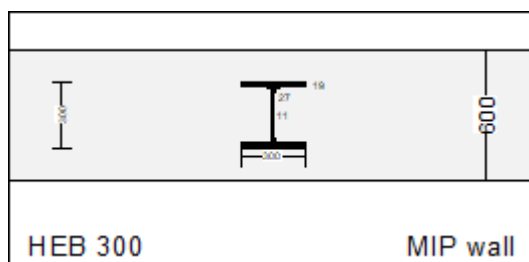
7 Theoretical principles

7.1 GGU-RETAIN: General notes to types of retaining wall

The following types of retaining wall can be analysed:

- soldier pile walls,
- sheet pile walls,
- bored pile walls,
- diaphragm walls,
- bored soldier pile walls,
- FMI/MIP wall,
- combined sheet pile walls.

Sheet pile walls, bored pile walls and diaphragm walls differ only in respect to the design of their sections, i.e. design of the reinforced concrete to EC 2 (bored pile wall = circular cross section; diaphragm wall = rectangular cross section). Soldier pile walls and bored soldier pile walls differ only with respect to the design of the steel section (soldier pile wall) and the circular cross section (bored soldier pile wall). The FMI (cut, mix, inject) wall, which is often also referred to as a MIP (mixed-in-place) wall, represents a special case.



7.2 GGU-RETAIN: Soil properties

7.2.1 Soil properties

A maximum of 50 soil layers can be taken into consideration. The following parameters must be given for each:

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 retaining wall [m/s],
- skin friction $q_{s,k}$ [kN/m²],
- cone resistance from CPT q_c ,
- shear strength of the undrained soil $c_{u,k}$.

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.

Permeability is taken into consideration in a calculation of subsurface hydraulic conditions via flow conduits parallel to the wall (see "**Theoretical principles: Water pressure**" and there "**Water pressure approach for sheet pile walls and in-situ concrete walls using flow conduits**").

If you want to analyse the pull-out resistance of anchors, activate the "**Verification with $q_{s,k}$** " check box in the "**Editor 2/Anchors**" dialog box.

To analyse the vertical capacity to EAU, EAB and EAP, enter the cone resistance q_c and the shear strength of the undrained soil $c_{u,k}$.

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

If you are already working with **GGU-CONNECT**, you can also import soil properties from your subsoil layer model that you have created in **GGU-CONNECT**.

Further explanations can be found under the menu item "**Editor 1 / Soils**".

7.3 GGU-RETAIN: 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.

7.4 GGU-RETAIN: At-rest earth pressure

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

7.5 GGU-RETAIN: Increased active earth pressure

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

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

$$0.0 \leq f \leq 1.0$$

7.6 GGU-RETAIN: Passive earth pressure

7.6.1 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.

The Mohr/Coulomb (root) approach is obsolete and should no longer be used!

The passive earth pressure acting on soldier piles is calculated after Weißenbach (EAB R 14). When the soldier piles are so close together that the effects of passive earth pressure overlap, the calculated values should be reduced. The passive earth pressure with and without overlapping must be determined for this purpose. Passive earth pressure acting on soldier piles without an overlap effect takes the shape of a parabolic curve and depends, amongst other things, on pile width. **GGU-RETAIN** calculates both values (with and without overlap effect) for all subdivisions. The lower of the two values obtained is then used in the subsequent analysis. Thus, in a graphic visualisation of passive earth pressure acting on a soldier pile, part of the curve may be parabolic, while another part is linear.

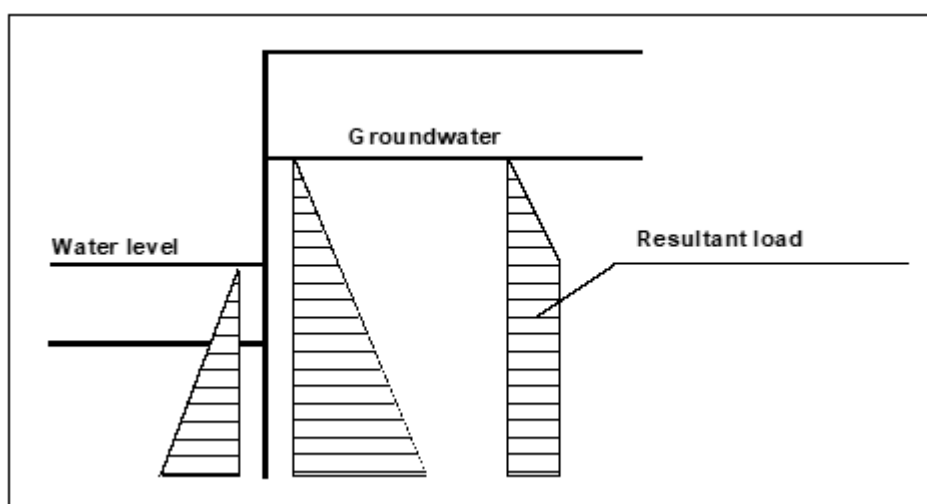
DIN 4085:2017 stipulates that planar slip surfaces may be adopted for analysis up to a friction angle $\leq 35^\circ$ and a wall friction angle $\leq -2/3 \phi$. This returns more favourable earth pressure coefficients.

7.7 Water pressure

7.7.1 Approach for sheet pile walls and in-situ concrete walls

7.7.1.1 GGU-RETAIN: Classical water pressure approach

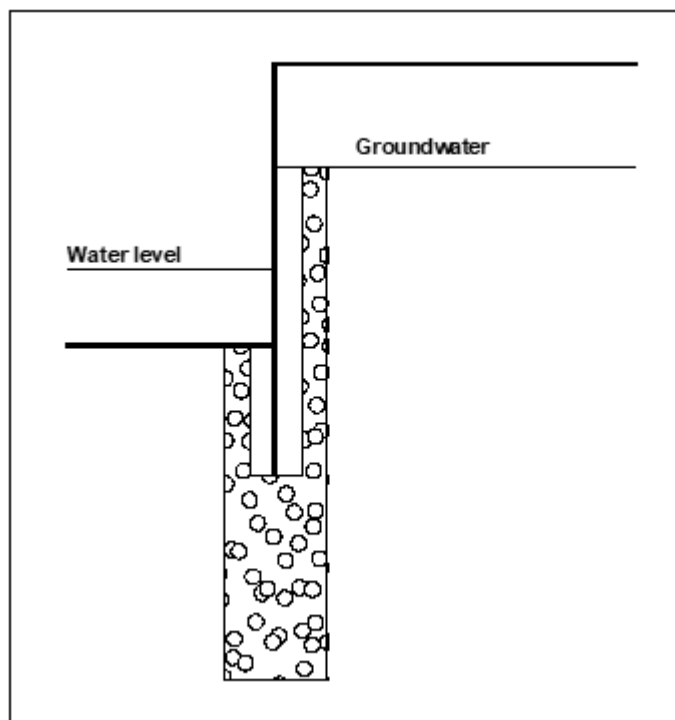
For sheet pile walls and in-situ concrete walls, water pressure can affect the active and passive sides of the wall. The classical water pressure approach is illustrated in the following figure:



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 around the retaining wall.

7.7.1.2 GGU-RETAIN: Water pressure approach using flow conduits

Beside the classical water pressure approach, **GGU-RETAIN** 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.



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-RETAIN**. You can still define additional potentials at any location to the left or right of the retaining wall. This allows correct, simple consideration of several groundwater storeys or confined aquifers. In addition, employing user-defined potential definitions allows you to create or model any kind of water pressure distribution.

7.7.1.3 GGU-RETAIN: 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 for Classical water pressure approach) has no physical meaning, but nevertheless provides conservative design values.

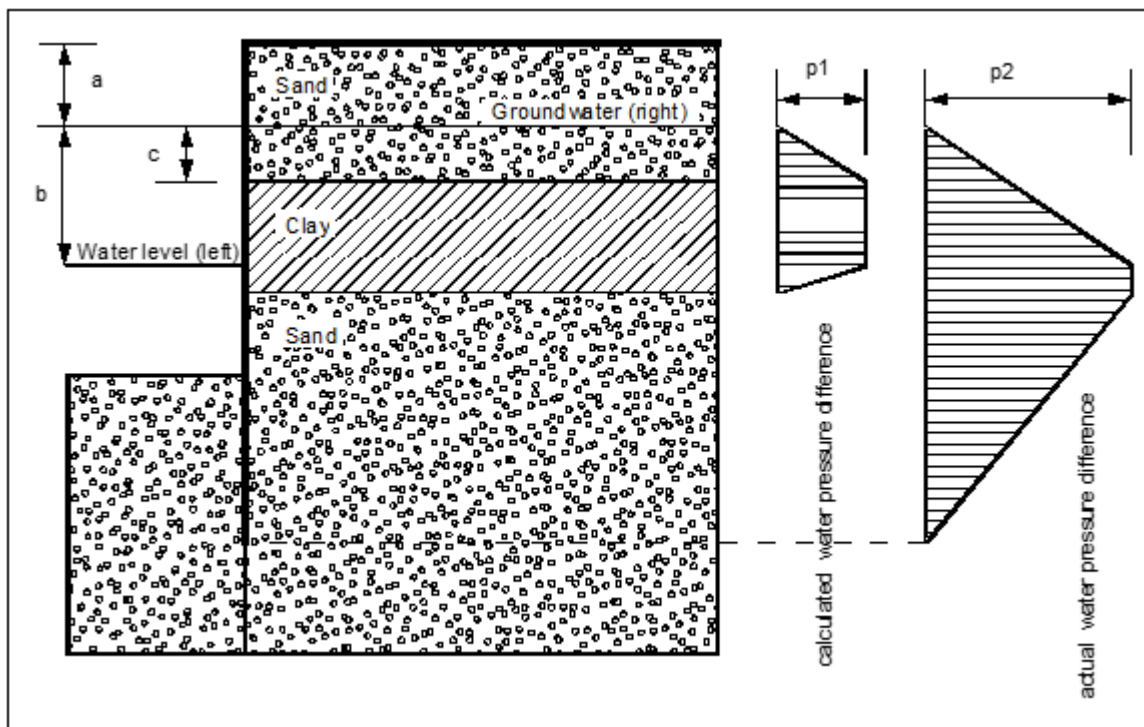
The classical water pressure approach and the approach using flow conduits (under uniform conditions of permeability) both assume a linear pressure drop along the wall. Given **uniform permeability**, this can lead to an underestimate of the hydraulic gradient, since the two-dimensional flow to which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

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

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

7.7.1.4 GGU-RETAIN: Possible conceptual error using flow conduits

On the other hand, the following conceptual error can easily be made using flow conduits when dealing with the following system:



$$p1 = \gamma_w \cdot c$$

$$p2 = \gamma_w \cdot b$$

The permeabilities of the clay and sand layers are $1 \cdot 10^{-8}$ m/s and $1 \cdot 10^{-4}$ m/s, respectively. Although the program conforms completely to potential theory, the calculated difference in water pressure, p_1 [kN/m²], does not correspond to the distribution of pressure that would be expected. Based on what has already been described, the application assumes a current of water in a flow conduit parallel to the wall from the water level on the right to the water level on the left of the wall. At the top right-hand end of the flow conduit the water level corresponds to the right-hand water level, while at the top left-hand end of the flow conduit it corresponds to the left-hand water level. The reduction in water pressure between these two points is now determined according to potential theory. The resulting difference in water pressure between the left and right sides corresponds to the distribution p_1 shown in the figure above, and according to the applied method is perfectly correct.

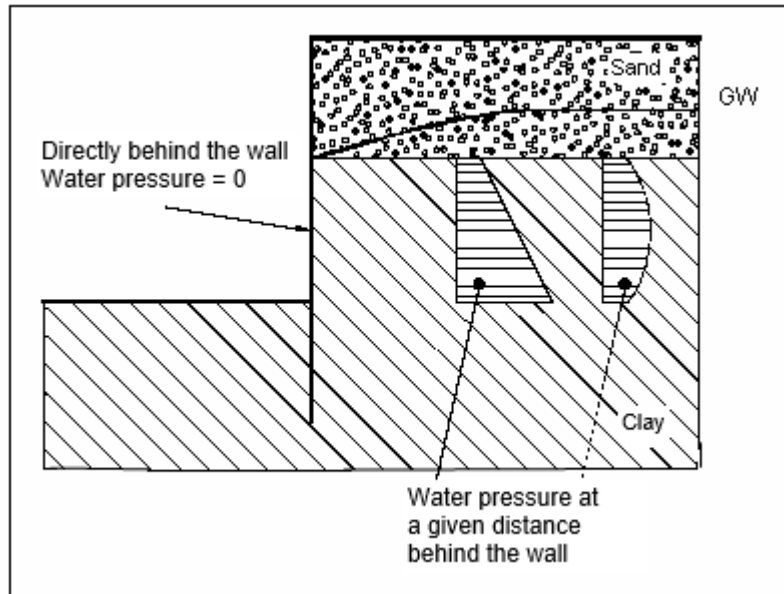
However, if the groundwater level below the layer of undisturbed clay is the same as that above it, this approach will produce an erroneous value for the water pressure distribution. In the above example, the water pressure below the layer of clay will be determined almost entirely by the water pressure in the excavation. If the water pressure below the layer of clay is to correspond with that above it, using flow conduits, a potential of

$$h = a$$

must be defined on the right side, below the layer, to achieve the desired distribution of water pressure differentials, p_2 , as shown in the figure above. Although the program conducts plausibility checks, there are nevertheless situations in which an analysis without additional definition of a potential below the clay layer is useful. Whether or not the computed difference in water pressure is what you had in mind can be easily checked by selecting the "**Differential water pressure**" command button in the dialog box that appears when you click "**Graphics output preferences**" in the "**System**" menu. Then the program graphically displays the difference in water pressure between the left and right sides of the retaining wall.

7.7.2 GGU-RETAIN: Approach for soldier pile walls

In general, adopting water pressures for soldier pile walls does not make sense, because they are usually permeable to water. But even when using timber infill, water pressure can build up behind the wall in certain circumstances (see figure).



The program therefore allows for adopting water pressures with soldier pile walls.

7.7.3 GGU-RETAIN: 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 $\cong 10.0 \text{ kN/m}^3$).

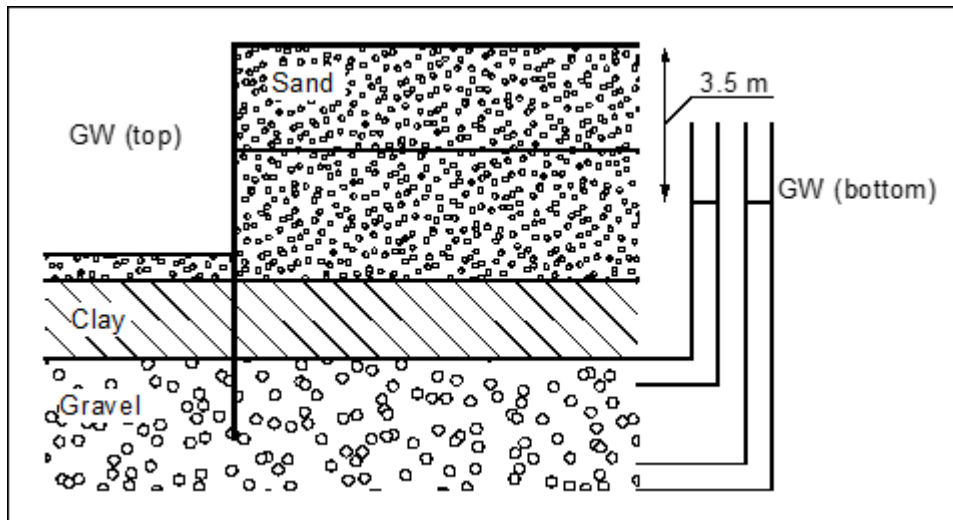
$$h = p/\gamma_w + y$$

h = potential [m]

p = water pressure [kN/m^2]

y = elevation head [m]

An example:



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. This boundary condition is used automatically by the program. Together with the permeabilities, the program calculates the potentials along the flow conduit resulting from these boundary conditions. From the calculated potentials (h), water pressure (p) and also the gradient (i) can now be determined. Applying the above formula for p :

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

For the gradient (i) we have:

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

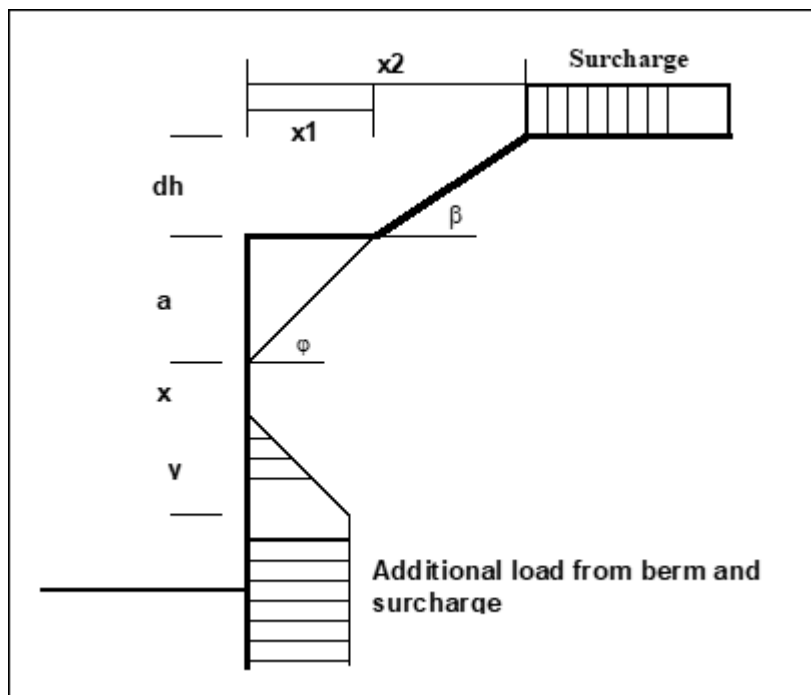
Δh = potential difference between two points

ΔL = distance between the two points

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

7.8 GGU-RETAIN: Berms

The program can handle 40 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).



The following relationships apply for the parameters x and y :

$$x = \frac{kah_0}{(kah\beta - kah_0)} \cdot a$$

$$y = \frac{kah_0}{(kah\beta - kah_0)} \cdot dh$$

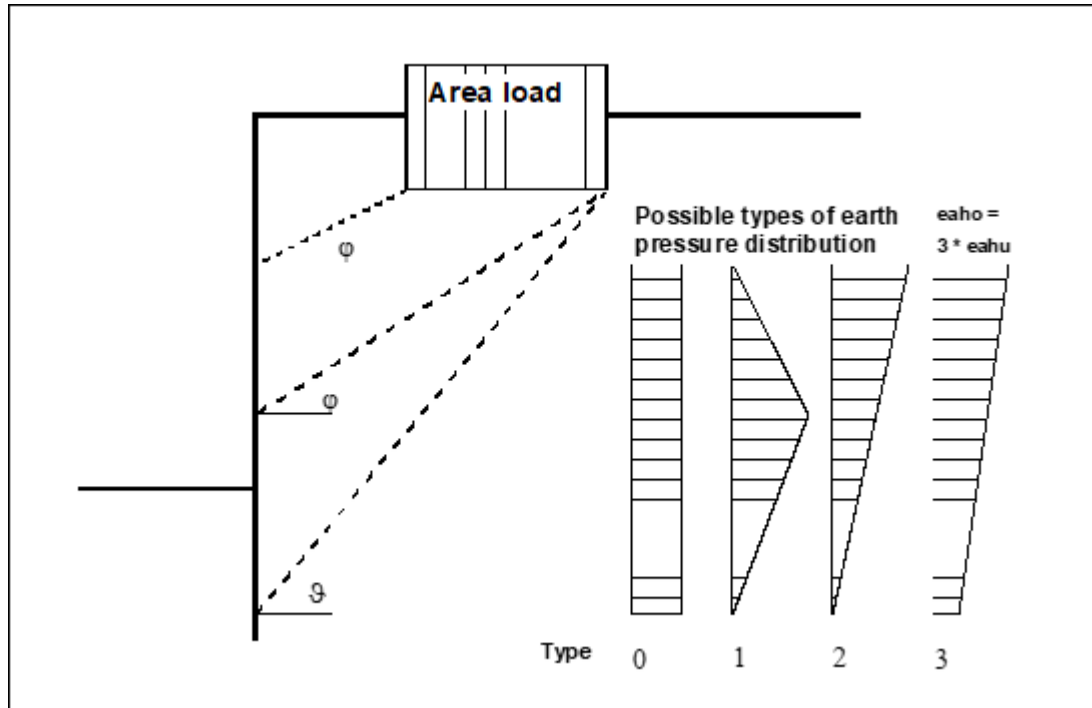
$$\Delta e_{ahu} = \gamma \cdot dh + \text{surcharge}$$

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.

7.9 GGU-RETAIN: Area loads

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



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, the program 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 the following Figure).

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

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

$$eop = q/4 \cdot (\sin^3\beta_2 - \sin^3\beta_1)$$

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

It is possible to enter a horizontal load $p(h)$ when defining area loads ("**Editor 2**" menu).

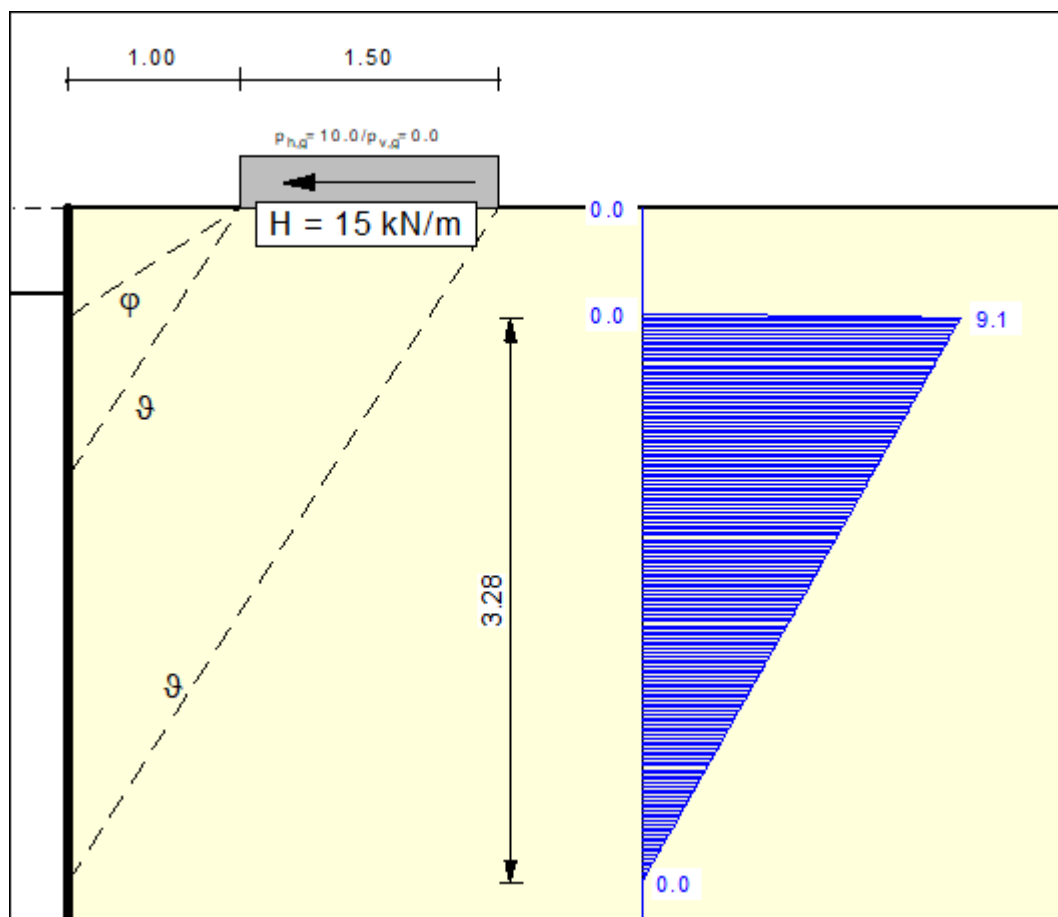
Area loads

Done Forw. Back Cancel Sort

1 area load(s) to edit Loads with limited plan dimensions

No.	$p(v)$ [kN/m ²]	$p(h)$ [kN/m ²]	$x(\text{left})$ [m]	$x(\text{right})$ [m]	Depth [m]	Type	Live
1	0.00	10.00	1.000	2.500	0.000	Triangle (max. at top)	☐

In this case, a horizontal load of 10 kN/m² was defined with a width of 1.50 m. This corresponds to a horizontal load H of 15 kN/m. According to the Piling Handbook (*Spundwand-Handbuch*) this results in the active earth pressure shown in the following figure for homogeneous ground:



The area covered by the active earth pressure resulting from horizontal loads corresponds to a horizontal load of 15 kN/m.

In homogeneous ground, the resulting earth pressure is independent of unit weight and cohesion. The earth pressure distribution with depth is acquired from the friction angle φ and from ϑ , which results from it. The earth pressure area is independent of the angle of friction and always corresponds to the adopted horizontal force H. In stratified ground the beginning and end of the earth pressure's zone of influence is acquired by averaging φ and ϑ .

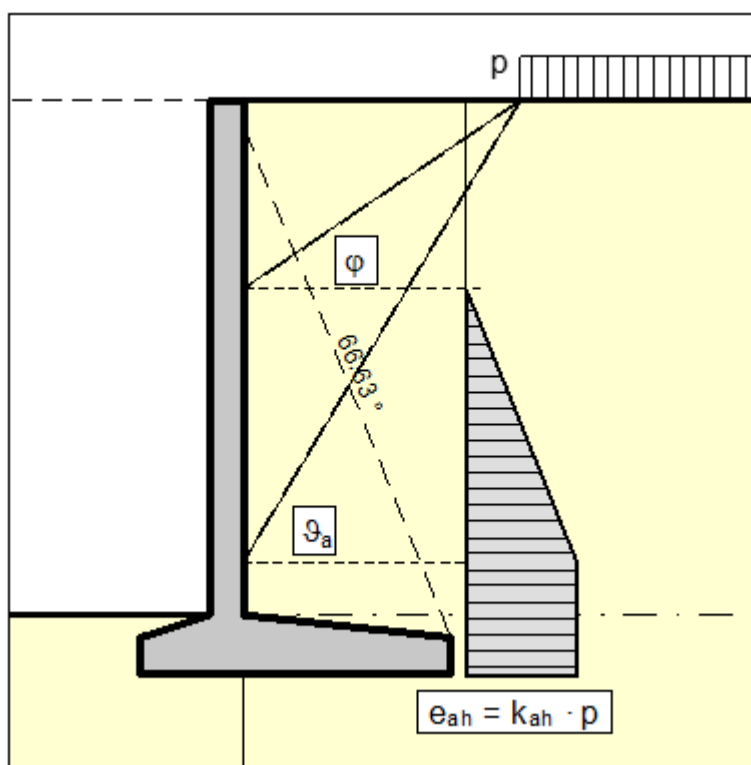
If at-rest earth pressure or increased active earth pressure are adopted the horizontal forces from the resulting earth pressure cannot be greater than the acting horizontal force H for equilibrium reasons. The same earth pressure distribution is therefore adopted in **GGU-RETAIN** for at-rest earth pressure and increased active earth pressure as for active earth pressure.

7.10 GGU-RETAIN: 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 menu item "**Editor 2/Area loads**".

7.11 GGU-RETAIN: Bounded surcharges (active side)

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

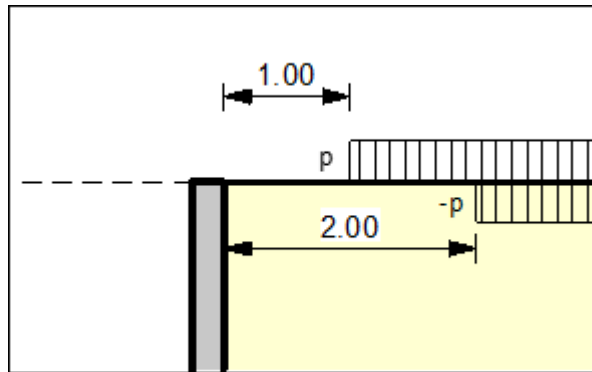


The earth pressure coefficient k is acquired from k_{ah} for active earth pressure and from k_0 for at-rest earth pressure. If this option is activated, the resulting earth pressure is then redistributed.

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

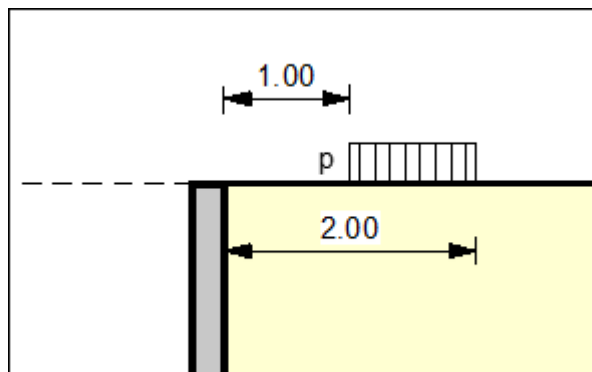
7.12 GGU-RETAIN: Double-bounded surcharges (active side)

If two bounded surcharges are defined as follows:



- 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 under "**Theoretical principles: Bounded surcharges (active side)**").

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.

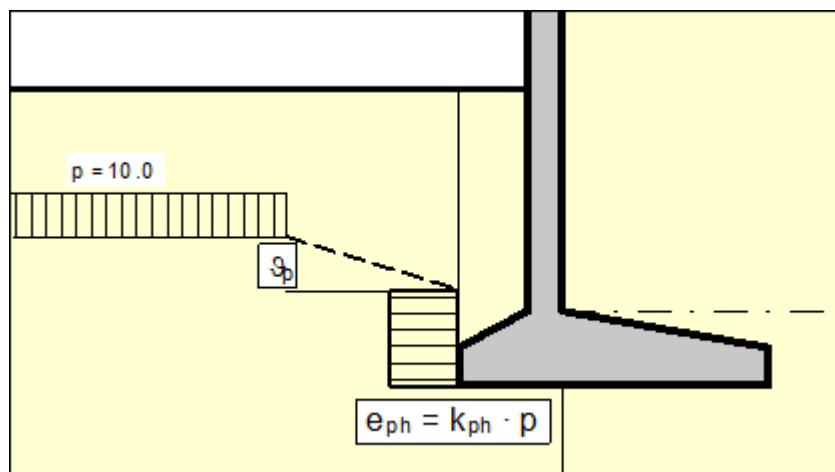


The same result is achieved as for the definition provided by the first figure above but with **improved graphics**.

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

7.13 GGu-RETAIN: 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:



7.14 GGU-RETAIN: Double-bounded surcharges (passive side)

Double-bounded surcharges on the passive side can be modelled using two bounded surcharges (also see “**Theoretical principles: Double-bounded surcharges (active side)**”). Double-bounded surcharges on the passive side were only implemented due to the **better graphical representation**.

7.15 GGU-RETAIN: Length surcharge

In the case of full or partial fixity of the retaining walls, a length surcharge for the theoretically determined embedment depth is required. In accordance with the EAB, this surcharge can be adopted at 20% of the theoretically determined embedment depth for full fixity. For partial fixity the value is acquired by linear interpolation between 0% and 20%. Alternatively, the length multiplier x can also be calculated in accordance with the EAU (*Empfehlungen des Arbeitsausschusses "Ufereinfassungen"* - Recommendations of the Committee for Waterfront Structures, Harbours and Waterways):

$$\Delta x \geq Ch / e_{phC}$$

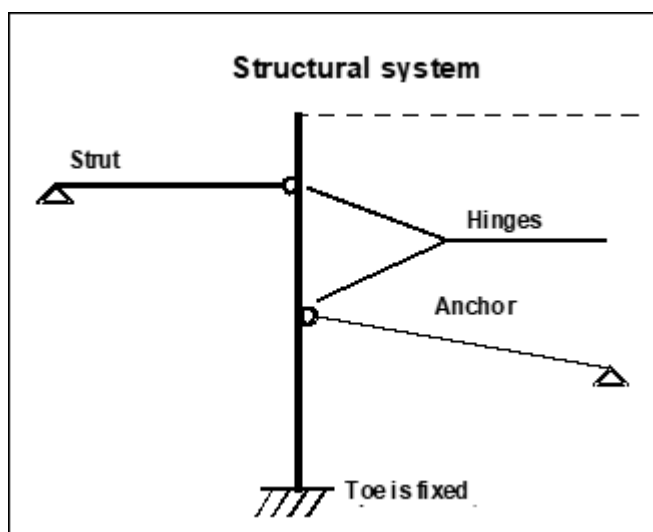
Ch = equivalent force after Blum (horizontal component)

e_{phC} = passive earth pressure stress on the equivalent force side below the theoretical toe TF

The program can use either method. When calculating with the above equation, the program always adopts the coefficient of passive earth pressure after Streck. After determining x , the program checks whether the additional length extends into soil with a lower coefficient of passive earth pressure, in which case it uses the lower value. The output table will provide you with a detailed presentation of the results.

7.16 GGU-RETAIN: Structural system

The structural analysis required to determine the state variables (displacement, moment, shear force and normal force) is not carried out by means of a continuous beam analysis, as is usual in retaining wall applications, but by means of a rod construction module, which treats the retaining wall and any anchors and struts as a single structural system. For inclined anchors in particular, the interactions between the anchor and the retaining wall can thus be correctly processed in a single computation. In a system with one anchor and one strut, for example, the following schematic would represent the governing structural system:



In addition, a vertical load can be defined for a strut and, if necessary, a rigid connection to the retaining wall be taken into consideration. For anchors and struts, the axial rigidity must be specified so that its influence on the action effects can be correctly recorded. The rod construction module can also compute using 2nd order theory. This means that buckling length investigations for struts, for example, can be dispensed with.

The theoretical principles of the rod construction module can be traced back to an article by Duddeck/Ahrens (*Betonkalender* 1976 (Concrete Calendar), Volume 2). Basically, this is a finite element method based on the displacement method.

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

7.17 GGU-RETAIN: Design

When you start an analysis, the program requires the stiffnesses of the retaining wall and any anchors and struts present. This data can be specified before going ahead with the analysis. The GGU-RETAIN program also provides an additional option for determining the optimum section for the computed action effects, using a list of sections loaded with the program. This list can also be edited and expanded. To do this, the program also requires you to enter the maximum allowable stress. If the newly determined section does not coincide with the section on which the analysis was based, **GGU-RETAIN** alters the calculated displacements as a function of the moments of inertia of the two sections.

However, a new analysis would normally be performed, as the stiffness conditions have changed, because the axial rigidity of the anchors and struts, and any subgrade reaction moduli, do not alter. In most cases, nevertheless, renewed analysis is unnecessary, because the axial rigidity of the anchors/struts is generally so great that appreciable support displacements do not occur. In the appropriate program situations, you will still be warned.

Stress determination can be by one of three different means:

- working $\sigma = N/A + (N \cdot w + M)/W$;
- working $\sigma = N/A + M/W$;
- working $\sigma = M/W$.

where

N = greatest normal force (absolute)

A = cross-sectional area

w = greatest displacement (absolute)

M = greatest moment (absolute)

W = section modulus.

Shear design is also performed.

In-situ concrete walls can be designed to EC 2. Shear design is also possible here.

7.18 GGU-RETAIN: 2nd order theory

Analysis of sheet pile walls under buckling loads is performed in **GGU-RETAIN** compliant with DIN EN 1993-1-1.

The differential equation for a **normal** flexural member is:

$$EI w''''(x) = q(x)$$

The normal force N is taken into consideration for a buckling member:

$$EI w'''(x) + N w''(x) = q(x)$$

For the sake of completeness, the differential equation for an additionally bedded system is shown here since the program also allows parallel processing of systems with 2nd order theory and elastic bedding. The equation is:

$$EI w'''(x) + N w''(x) + k_s w(x) = q(x)$$

Analysis is performed on the deformed system. In

- **DIN EN 1993-5**

Design of Steel Structures,
Part 5: Piling

analysis using 2nd order theory is recommended for analysis of sheet pile walls under buckling loads. DIN EN 1993-5 refers to

- **DIN EN 1993-1-1**

Design of Steel Structures,
Part 1-1: General Rules and Rules for Buildings

for these analyses. Analysis using 2nd order theory produces more accurate results than the usual and simplified equivalent member method.

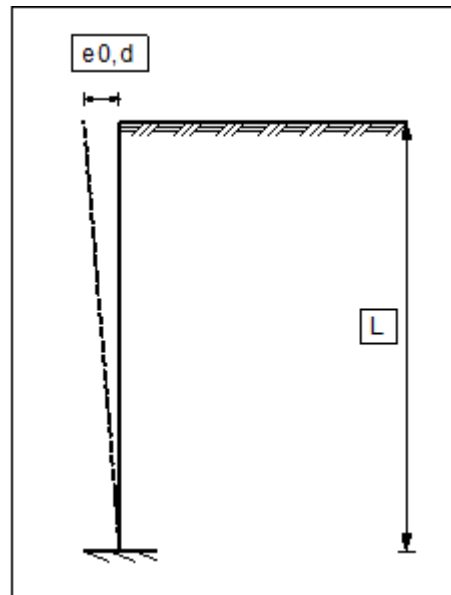
Analysis using 2nd order theory requires a predeformation or pre-curvature of the underlying system. Pre-curvature values are given in Table 5.1 of DIN EN 1993-1-1.

Table 5.1 – Pre-curvature design values $e_{0,d}/L$ for structural elements

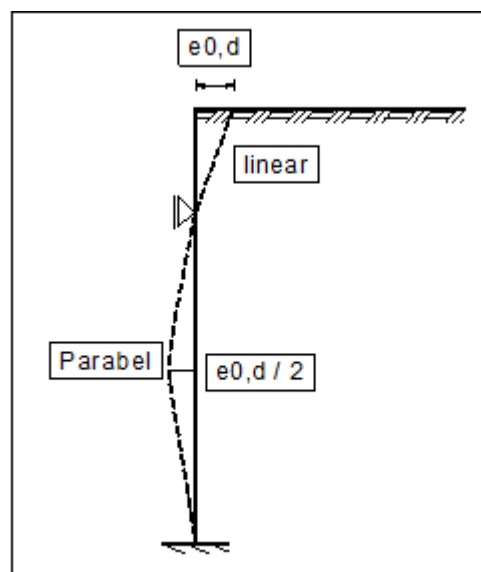
Buckling line from Table 6.1	Elastic analysis	Plastic analysis
	$e_{0,d}/L$	$e_{0,d}/L$
a ₀	1/350	1/300
a	1/300	1/250
b	1/250	1/200
c	1/200	1/150
d	1/150	1/100

Pre-curvatures are given as a function of the buckling line. In simplification, sheet pile walls can be analysed with a pre-curvature of $e_{0,d}/L = 1/150$.

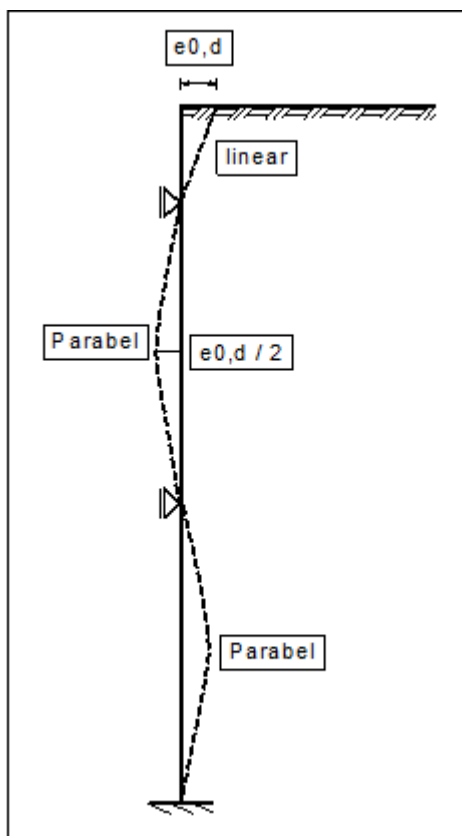
In the case of embedded, non-anchored walls, the deformed system results from an inclined position of the wall as shown in the following figure.



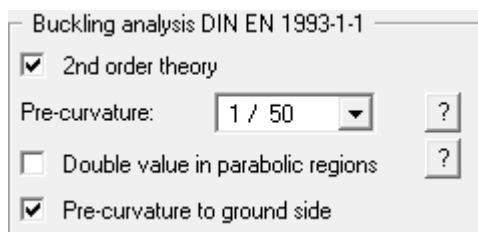
In the case of embedded, simply back-anchored walls or a freely supported, simply back-anchored wall, the deformed system results from a linear pre-curvature from the support point to the wall head and a parabolic pre-curvature between the support points and the wall base (see following figure).



The length L is given per field. The procedure for two rows of anchors is shown in the following figure.



The settings are defined in **GGU-RETAIN** in the analysis menu under "**System/Analyse**".



The size and direction of the pre-curvature can be specified. Whether a pre-curvature towards the ground side or the atmosphere side provides the more unfavourable design values is system-dependent: the following message will therefore be displayed once analysis begins:



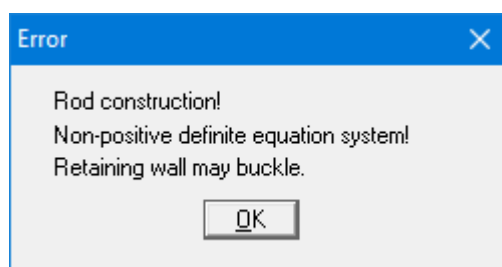
You must therefore deactivate the "**Pre-curvature to ground side**" check box following successful analysis and check, in a new analysis, whether pre-curvature to the atmosphere side delivers less favourable values.

In an analysis using 2nd order theory the necessary iteration process in terms of displacement is carried out using the design normal force N_d . The analysis is based on a frame system, such that axial stiffnesses, and anchor and strut inclinations are taken into consideration correctly.

Subsequent design is based on a comparison of stresses

$$\sigma_d \leq f_{y,k} / \gamma_M = f_{y,k} / 1,1 = f_{y,d}$$

The 'Example' folder contains 4 files, which deal with the classical Euler cases 1 to 4. If the vertical load V is increased slightly using the "**Editor 2/Action boundary conditions**" menu item and the system analysed, the following error message appears:



The normal force given in the data files thus corresponds to the buckling force determined after Euler.

7.19 GGU-RETAIN: Bedding conditions of wall toe

Classical methods of computation recognise two limit states for bedding of the wall toe:

- **Limit state 1:**

Wall toe freely embedded on resultant of passive earth pressure.

Embedment depth is determined through iterative displacement of a horizontal support. Embedment depth is found when support force is zero.

- **Limit state 2:**

Wall toe fully fixed.

Determination of the embedment depth is achieved through iterative displacement of a fixation point at the bottom of the system. The embedded depth is found when the moment is zero at the point of fixation.

For full fixation in accordance with the EAB, **GGU-RETAIN** increases the theoretical embedment depth by 20%. Between the two limit states, intermediate conditions of partial fixation between 0 % and 100 % are possible. If you want to compute partial fixations, **GGU-RETAIN** will first calculate embedment depths for the two limit states. The necessary embedment depth is then obtained through linear interpolation between the two extremes. In the same way, for a given section length, the degree of fixation is determined from the two extreme embedment depths. The necessary longitudinal extension (length addition) of between 0 % (for free earth support) and 20% (for complete fixation) is also determined by linear interpolation in compliance with the EAB.

Thus, there are two different ways in which section length and the degree of fixation can be determined:

- determination of section length for a given degree of fixation of wall toe,
- determination of degree of fixation of wall toe for a given section length.

7.20 Bedded systems

7.20.1 GGU-RETAIN: General information on types

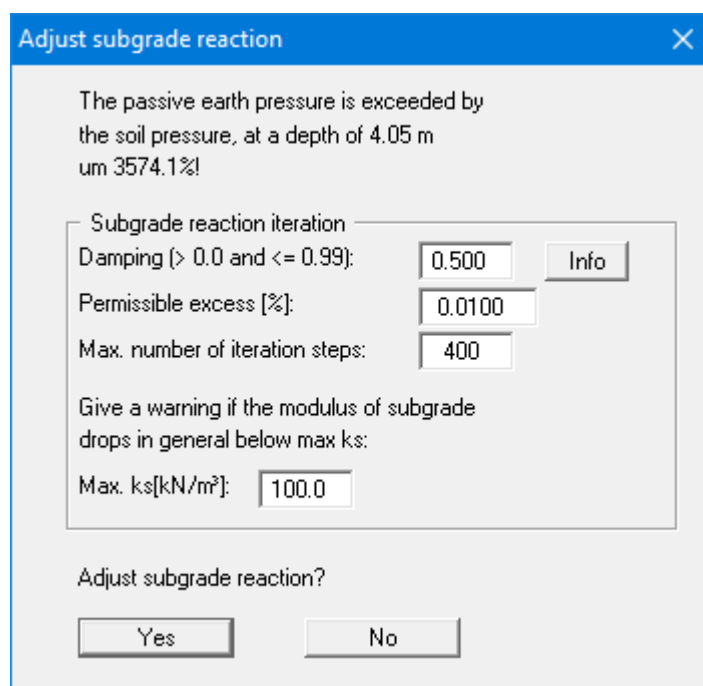
When a retaining wall is bedded there is always a degree of toe fixation, whereby the degree of fixation cannot be meaningfully given. Thus, for a bedded wall toe we have the following two variations:

- bedded wall toe with given section length,
- bedded wall toe with automatically determined section length.

7.20.2 GGU-RETAIN: Variant "Section length fixed and bedded toe"

In this variation you enter a fixed height for the wall.

The elastic calculation results in a soil pressure that occurs in front of the wall and can be determined from the product of the modulus of subgrade reaction and the deformation along the wall. This **elastic pressure** may not be greater than the passive earth pressure acting in front of the pile. Following analysis, this condition will generally not be met, so that iterative reduction of the modulus of subgrade reaction, along with the resulting soil pressure, will be necessary to meet it. The following dialog box will appear, with which you can control the iteration process.



The dialog box titled "Adjust subgrade reaction" contains the following information and controls:

- Message: "The passive earth pressure is exceeded by the soil pressure, at a depth of 4.05 m um 3574.1%!"
- Section: "Subgrade reaction iteration" with a minus sign icon.
- Fields and buttons:
 - "Damping (> 0.0 and <= 0.99):" with a text box containing "0.500" and an "Info" button.
 - "Permissible excess [%]:" with a text box containing "0.0100".
 - "Max. number of iteration steps:" with a text box containing "400".
 - "Give a warning if the modulus of subgrade drops in general below max ks:" with a text box containing "100.0".
- Question: "Adjust subgrade reaction?"
- Buttons: "Yes" and "No".

The **default values** have proven themselves in practice and will generally not need to be altered.

"**Damping**" prevents strong fluctuations around the actual solution during the iteration process. A value of 0.0 produces no damping, while a value of 0.99 produces very strong damping and long computation times.

Using "**Permissible excess**" you specify the percentage by which the elastic stress may exceed the passive earth pressure before the iteration is ended.

The number of iteration steps can also be specified. If, after the set number of steps, soil pressure is still greater than passive earth pressure, iteration is ended without result.

During iteration, the modulus of subgrade reaction is reduced. If the analysis requires a reduction over the entire wall length to a value below "**max ks**", iteration is aborted. This usually happens when the wall is not long enough to take the load.

7.20.3 GGU-RETAIN: Variant "Automatically determined section length and bedded toe"

Using this option, you can determine the optimum height of the wall.

First you must define the range of section lengths within which **GGU-RETAIN** is to find the ideal value.

The dialog box titled "Iterative determination of section length" contains the following settings:

- Section length iteration:**
 - Minimum section length [m]: 5.00
 - Maximum section length [m]: 12.00
 - Delta section length [m]: 0.10
- Iteration conditions:**
 - Damping (> 0.0 and ≤ 0.99): 0.500
 - Permissible excess [%]: 0.0100
 - Go to next section length if the modulus of subgrade drops in general below max ks:
 - Max. ks[kN/m²]: 100.0
 - Max. no. iteration steps/section length: 400
- Additional iteration criterion:**
 - ☒ No additional criterion
 - ☐ Max. wall top displacement [mm]: 50.00
 - ☐ Max. wall top rotation [°]: 1.00

Buttons: OK, Cancel

Enter the maximum and minimum section lengths, as well as a delta section length, in the top group box of the dialog box shown above. This defines the range of lengths from which the optimum length should be selected. The central group box deals with iteration (see explanations in "**Theoretical principles: Bedded systems - Variant "Section length fixed and bedded toe"**"). In the bottom group box, you can define additional iteration criteria, such as maximum displacement of the wall top.

7.21 GGU-RETAIN: Action and displacement boundary conditions

The program automatically sets the boundary conditions shown in the figure under "**Theoretical principles: Structural system**". However, it is also possible to specify additional boundary conditions at any point on the wall. All six state variables can be defined as boundary conditions:

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

7.22 GGU-Retain: Predeformations

When a number of excavation phases are under study, deformations from a previous phase can be implemented as lowering of supports at the new anchor points for the next phase. According to the EAB (R 11), however, such studies are not usually necessary.

In a continuous beam analysis, it is a simple matter to introduce the corresponding support displacements. In a rod framework application that does not consider anchors and struts as mere supports (see Figure 19 in "**Theoretical principles: Structural system**"), the same lowering of the support must be introduced at the anchor toe as at the anchor head, otherwise a large proportion of the resultant load would be transferred to the anchor. Thus, apart from being able to define displacement boundary conditions, the program also offers a menu item dealing with predeformations. If you wish to enter predeformations **by hand**, or need to, this must be done as displacements under predeformations and not as displacement boundary conditions. Of course, once this is done, the program takes care of the rest. You also have the option of automatically adopting predeformations from a previous analysis.

Designing a retaining wall involves a multitude of different adjustments, making it all too easy to lose one's bearings. The program is thus designed to ask numerous questions, which, apart from controlling correct entry of data, also check plausibility, special preferences, and compatibility with the EAB. When a data entry error is recognised, a warning box appears, and analysis is put on hold. Responses to inconsistencies in respect to plausibility or special preferences always offer two options:

- accept warning and start analysis,
- accept warning and do not start analysis.

When the program draws your attention to special preferences, the dialog box is always designed so that simply pressing **[Return]** will not terminate the analysis.

The program is designed for a high degree of user interaction, with the aim of avoiding erroneous and unwanted preferences (see above). Batch processing for the background processing of several files simultaneously is incompatible with this concept. Therefore, the program does not create a single dataset when dealing with several construction phases. All phases must be dealt with in separate datasets. The only disadvantage of this is the treatment of predeformations. If you want predeformations from a previous construction phase to be automatically taken into consideration, this previous phase must already have been analysed (obviously) and be available as a dataset (on your hard disk). This dataset must be given as the dataset from which the predeformations are to be determined for the new construction phase. Since there is no re-analysis of the old phase, this dataset and the results obtained with it must be saved. If you should mistakenly select a dataset that contains no results, the program will inform you of this. Otherwise, there are no restrictions. The name of the file with the predeformations is saved with the current dataset, where it is available for future use.

7.23 GGU-RETAIN: 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.

7.24 GGU-RETAIN: Modulus of subgrade reaction

According to the EAB (R 11), elastic analysis with subgrade reaction moduli can be employed to determine internal forces. For this purpose, you can define linearly variable subgrade reaction modulus curves for the wall toe area, which can vary from area to area, making it possible to reproduce any subgrade reaction modulus curve you choose. Soil stresses result from analyses with subgrade reaction moduli, as the product of displacement and subgrade reaction moduli, which must not be greater than the passive earth pressure permitted at the point under consideration. Meeting this requirement generally requires iteration, which is performed by the program.

The subgrade reaction modulus curve is shown on the screen, but only when one of the following two variations is selected (see **"Theoretical principles: Bedded systems - General information on types"**):

- fixed section length and bedded toe,
- automatic determination of section length and bedded toe.

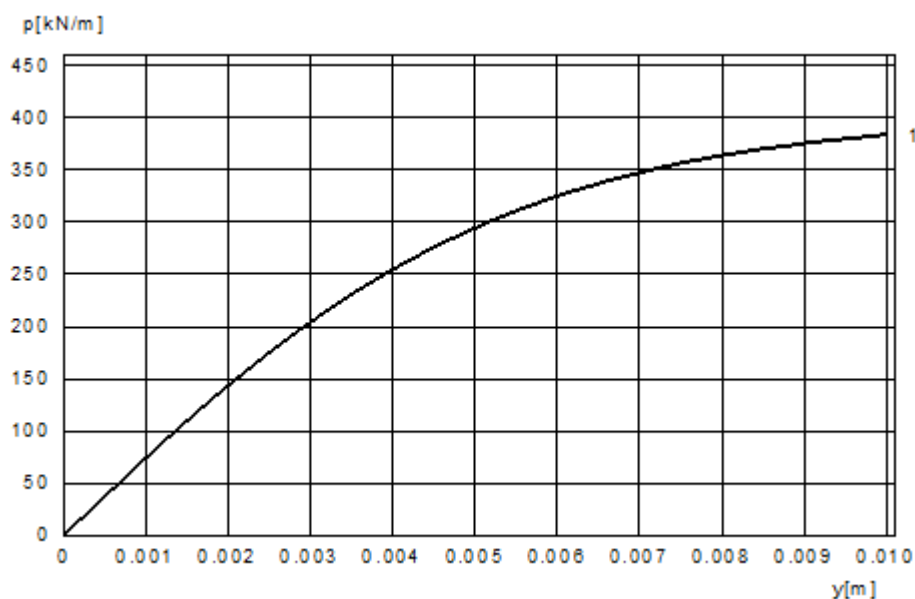
7.25 GGU-RETAIN: Excavations and waterfront structures using the p-y method

Generally, retaining walls and waterfront structures are calculated according to Blum's method.

The calculation of dolphins presents a comparable problem. Blum's method can also be used here. For dolphins, the so-called p-y method is preferred, especially on an international level.

This paper examines whether the use of the p-y method is possible and reasonable for excavations and waterfront structures.

The p-y method is a non-linear modulus of subgrade reaction method and can be used for pile diameters < 2.5 m. In the literature, p-y curves can be found for non-cohesive soils (without cohesion) and for cohesive soils (without friction angle). The figure below shows a p-y curve for sand.



The curve describes the magnitude of the earth resistance (p) in front of the pile as a function of the pile displacement (y).

Sample calculations show good agreement between Blum and p-y method.

Advantage of the p-y method:

- Realistic deformations with comparable state variables.

Disadvantages p-y method:

- No curves for soils with friction angle and cohesion.
- No curves for inclined ground.

7.26 GGU-RETAIN: Earth pressure redistribution

Classical earth pressure redistribution is not appropriate for braced or tied-back retaining walls. Earth pressure calculated according to classical theory must be redistributed.

A large number of redistribution figures are programmed into the program. In addition, any form of redistribution figure can be created by defining any polygon. 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.
- EAU 2012.

For **normal** retaining walls, earth pressure redistribution according to the EAB is certainly the simplest way to find the correct redistribution figure. If you select this option, you only need to wait for the analysis to be completed. The program automatically selects the correct redistribution figure from the possible redistribution figures according to EAB (R 69 for soldier pile walls and R 71 for sheet pile or in-situ concrete walls). If **GGU-RETAIN** is unable to find a suitable EAB figure because of **abnormal** anchor or strut positions, analysis is terminated, and a warning message is displayed. You then have to think about the redistribution figure yourself and select one of the other figures offered.

You can specify whether block loads are to be included in the redistribution. Water pressure is never redistributed.

7.27 GGU-RETAIN: Base heave safety

Safety against base heave (see EAB R 10; Paragraph 1) is demonstrated using a bearing capacity failure analysis after Weißenbach (Baugruben III, 1977), which compares weight forces (including area loads, berms etc.) with the bearing capacity load, whereby a total of 50 vertical failure planes are investigated. The first failure plane is 0.2 times the excavation depth behind the retaining wall, the last failure plane is 5.0 times the excavation depth behind the retaining wall. For soldier pile walls, the bottom edge of the failure plane is the excavation bottom, while for sheet pile and in-situ concrete walls it is the base of the section.

Analysis of the safety against base heave of **heavy foundations** in the vicinity of excavations (see EAB R 10, Paragraph 2) is not performed.

7.28 GGU-RETAIN: General stability

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

7.29 Hydraulic heave

7.29.1 GGU-RETAIN: Hydraulic heave safety using global safety factors

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

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

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

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

7.29.2 GGU-RETAIN: Utilisation factor (hydraulic heave) using partial safety factors

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

$$S'_k \cdot \gamma_H \leq \sum_{i=1}^N G'^i_k \cdot \gamma_{g,stab}$$

- 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
- $\gamma_{g,stab}$ = 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.

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

- μ_N = utilisation factor of layer N

Utilisation factors ≤ 1.0 mean that sufficient safety is given.

7.29.3 GGU-RETAIN: Analysis of hydraulic heave after Aulbach/Ziegler

In 2013 Aulbach performed extensive investigations and derived a relationship for homogeneous ground conditions (Aulbach, Benjamin: Hydraulischer Grundbruch - Zur erforderlichen Einbindetiefe bei Baugruben in nichtbindigem Baugrund, dissertation, RWTH Aachen, 2013, URN: urn:nbn:de:hbz:82-opus-46909). The hydraulic heave safety can thus be directly determined.

The relationship for the three-dimensional case is shown below (abstract from Aulbach & Ziegler (2013)¹):

$$\frac{T}{H} = Be \cdot \left[0,32 \cdot A + (1,244 - 0,32 \cdot A) \cdot e^{\left(\frac{\frac{B}{H}}{U \cdot \left(0,541 + 0,395 \cdot \left(1 - e^{\left(1 - \frac{S}{H} \right)} \right)} \right) \cdot \left(1 + \left(\frac{B}{L} - 0,3 \right) \cdot (3,156 - 1,564 \cdot U) \right)} \right)} \right] \cdot \left(\frac{\gamma'_{ref}}{\gamma_l \cdot 0,902 + 1,078} \cdot \frac{\eta_l}{\eta_{ref}} \right)^{\sqrt{2}}$$

Equation (6) in Aulbach & Ziegler (2013)¹ applies for the plane case:

$$\frac{T}{H} = 0,32 \cdot A + (1,244 - 0,32 \cdot A) \cdot e^{\left(\frac{\frac{B}{H}}{U \cdot \left(0,541 + 0,395 \cdot \left(1 - e^{\left(1 - \frac{S}{H} \right)} \right)} \right)} \right)}$$

The result of the analysis is the necessary embedment depth.

¹ Aulbach, Benjamin; Ziegler, Martin:

Simplified design of excavation support and shafts for safety against hydraulic heave = *Einfache Bemessung von Baugruben und Schächten im Hinblick auf die Sicherheit gegen hydraulischen Grundbruch*. In: Geomechanics and Tunneling 6 (2013), H. 4, S. 362-374, ISSN 1865-7362.

7.30 Buoyancy

7.30.1 GGU-RETAIN: Buoyancy safety using global safety factors

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

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

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

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

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

7.30.2 GGU-RETAIN: 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

- $\gamma_{g,stab}$ = partial factor for stabilising permanent actions in the UPL (EC 7) limit state
- layer 1 ($i = 1$) is the uppermost layer

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

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

- μ_N = utilisation factor of layer N

Utilisation factors ≤ 1.0 mean that sufficient safety is given.

7.31 GGU-RETAIN: Analysis of sum H

When analysing the stability of soldier pile walls, the active earth pressure beneath the excavation base may be neglected (see EAB, R 15). However, it must be demonstrated that this neglected active pressure can be taken up with a sufficient margin of safety (generally 1.5) by the passive earth pressure. **GGU-RETAIN** performs the necessary investigation. It is possible that to comply with the EAB, R 15 an increase in embedment depth is necessary, which **GGU-RETAIN** will carry out automatically. If it is not possible to demonstrate wall stability, you are given advice on how to proceed according to the EAB.

7.32 Analysis of mobilised passive earth pressure

7.32.1 GGU-RETAIN: General note on mobilised passive earth pressure

The equilibrium of vertical forces must be verified in accordance with the EAB, R 9. **GGU-RETAIN** provides the necessary geostatic constants.

7.32.2 GGU-RETAIN: Analysis of mobilised passive earth pressure using global safety factors

In accordance with the EAB R 9 and Weißenbach (Geotechnical Engineering Handbook, Volume 3, 2001), two cases must be distinguished:

- **Case 1**
The vertical forces acting downwards are relatively small.

- **Case 2**

The vertical forces acting downwards are greater than the vertical components of passive earth pressure determined using a negative angle of wall friction.

In Case 1, Weißenbach (Geotechnical Engineering Handbook, Volume 3, 2001) gives the following relationship:

$$\eta_v = (P_v + E_{av} + C_v + G) / E_{pv}$$

- E_{av} = vertical component of active earth pressure
- C_v = vertical component of the counterforce C for toe fixation
- G = self-weight of wall
- P_v = permanent external loads (including vertical components of anchors)
- E_{pv} = vertical component of passive earth pressure

In accordance with the EAB, C_v is always determined as $1/3 \cdot \varphi$. The EAB gives a safety factor of 1.5.

In Case 2, Weißenbach (Geotechnical Engineering Handbook, Volume 3, 2001) gives the following relationship:

$$\eta_v = Q_g / (P_v + E_{av} + G)$$

- Q_g = vertical ultimate bearing capacity of the wall or soldier piles
 Q_g can be determined pursuant to DIN 4026 or DIN 4014.

Further useful information relating to verification of V can be found in Weißenbach (Geotechnical Engineering Handbook, Volume 3, 2001).

If the wall toe is bedded, E_{pv} is determined from the positive subgrade reactions and the angle of wall friction.

7.32.3 GGU-RETAIN: Analysis of mobilised passive earth pressure using partial safety factors

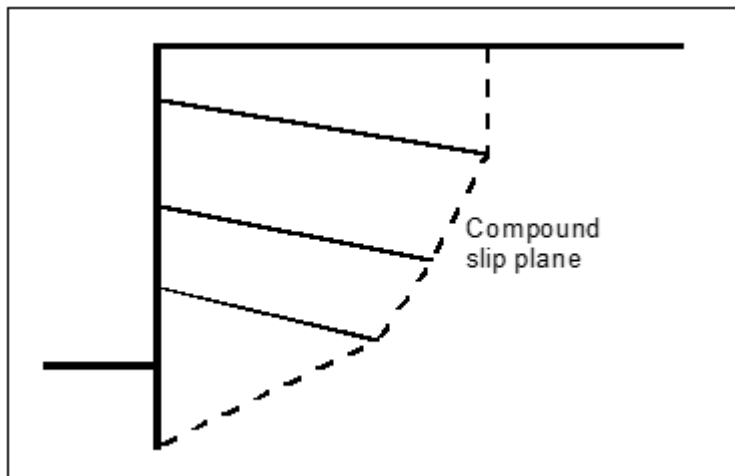
Detailed notes on this analysis concept are included in EAU 2012, Section 8.2.5.5, and in the Recommendations on Piling (2012).

7.33 GGU-RETAIN: Analysis of vertical capacity

Detailed notes on this analysis concept are included in EAU 2012, Section 8.2.5.5, and in the Recommendations on Piling (2012).

7.34 GGU-RETAIN: Verification of deep-seated stability

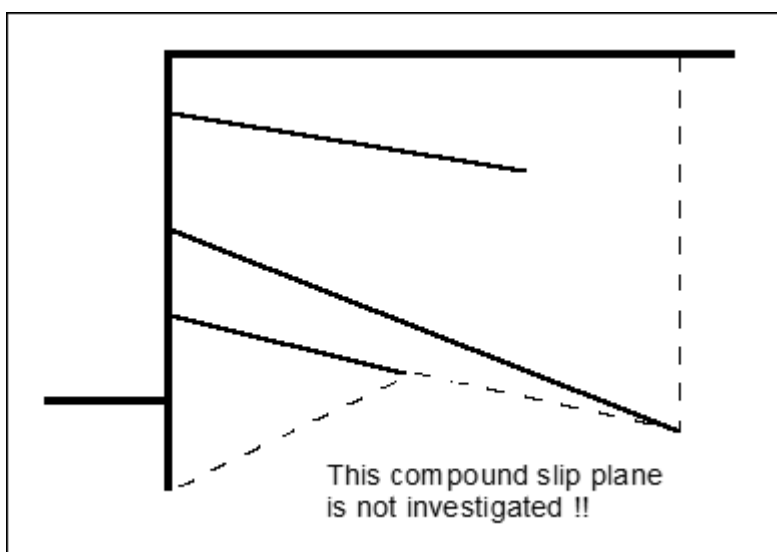
Verification of deep-seated stability is required for anchored cantilever walls. This primarily serves to determine the necessary anchor lengths. Verification uses the method described by Ranke/Ostermayer (*Bautechnik* 1968 (Construction Engineering), Issue 10). When **verifying deep-seated stability** 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.



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.



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 given if one of the following equations applies.

$A_{\text{poss},k}$ is determined from the force polygon with permanent loads only:

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

$A_{\text{poss},k}$ is determined from the force polygon with permanent and changeable loads:

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

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.

7.35 GGU-RETAIN: 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.

7.36 GGU-RETAIN: Construction phases

If you are investigating several advancing and retreating phases, **GGU-RETAIN** allows you to display the results in summarised form. For any selected phase it is possible to display the following conditions:

- envelope of moments,
- envelope of shear forces,
- envelope of normal forces,
- displacements,
- envelope of reinforcement.

You can skip through the individual phases using the mouse.

8 File menu

8.1 GGU-RETAIN: "New" menu item

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

The 'New data set' dialog box is a window for configuring a new project. It contains several sections with various options and input fields.

- Project identification:** A text input field for the project name.
- Standard:** Three radio buttons for selecting the standard:
 - ☒ Partial safety factors (EC 7) (with an 'Info EC 7' button)
 - ☐ Partial safety factors (DIN 1054:2005)
 - ☐ Global safety factor concept (DIN 1054 old)
- General:** A group of checkboxes and input fields:
 - ☐ Show excavation on right
 - Subgrade modulus unit: (dropdown)
 - ☒ Subgrade reaction via constr. modulus, EAB R 102 (with a '?' button)
 - ☐ Use absolute heights (Design:)
 - ☐ Differentiate active + passive soil properties (with a '?' button)
 - ☐ Use anchor/strut spacing
 - Anchor/strut spacing [m]: (with a '?' button)
- Wall inclination:** Wall inclination [°]: (with a '?' button)
- Steel design:** A group of checkboxes and radio buttons:
 - ☒ Using section list (with a '?' button)
 - ☐ Using several steel sections or inserted beams (with a '?' button)
 - ☒ Steel design to EC 3 (with a '?' button)
 - ☐ Buckling analysis limit criterion: $N_{Ed}/N_{cr} \leq 0.1$ (with a '?' button)
 - ☐ User-defined yield stress $f_{y,k}$ (steel pile) (with a '?' button)
- Concrete design:** ☒ Normal force characteristically (with a '?' button)
- Type of wall:** A grid of buttons for selecting the wall type:

Soldier pile wall	Sheet pile wall
Bored pile wall	Diaphragm wall
Bored soldier PW	MIP wall
Comb. sheet PW	

Below the grid are a 'Cancel' button and a checkbox for 'Soldier pile wall with tubular sections'.

You can enter a dataset description ("**Project identification**") of the problem going to process, which will then be used in the **General legend** (see menu item "**Graphics preferences/General legend**"). This can be particularly useful when you are working with the menu items "**Construction phases/Select files**" and "**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 next group box, the radio buttons are used to specify which safety concept to use for analysis and design. Additionally,

In the next group box, use the radio buttons to decide which safety concept should be used for your calculation and design.

Excavation visualisation to the right can be activated, as well as selecting kN/m^3 or MN/m^3 as the units for the modulus of subgrade reaction via a drop-down menu. If the wall base is to be calculated with elastic bedding, a subgrade reaction modulus curve must be specified. When the program starts, the "**Subgrade reaction via constrained modulus to EAB EB 102**" check box is activated. The subgrade reaction moduli are then determined using the constrained modulus entered in the "**Editor 1/Soils**" menu item. If you wish to enter the subgrade reaction modulus course yourself, deactivate the above check box and enter the appropriate course in the menu item "**Editor 2/Subgrade reaction moduli**".

If you select the "**Use absolute heights**" check box, you can enter all depths or heights in m AD (heights are positive upwards). If this check box is not selected, the top of the wall is assumed at 0.0 (height/depth) and all further entries, depth of soil layers, for example, are positive downwards. If, however, you want to work with absolute heights, enter the appropriate depths correspondingly altered. Thanks to **WYSIWYG** there is no danger of using incorrect data since all input is immediately visible on the screen.

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

If the "**Use anchor/strut spacing**" check box is activated the anchor or strut forces are output in [kN] and no longer in [kN/m]. Input in terms of EA and EI are also no longer entered per metre.

Below this, a wall inclination between -6° and $+6^\circ$ can be defined. Please read the information displayed after pressing the "?" button.

The "**Steel design:**" group box offers a list of supplied sections (soldier pile sections, sheet pile sections, etc.) for subsequent design. The section list is loaded automatically when the program starts. Sections from a variety of well-known manufacturers can be accessed here. Clicking the "?" button will display further information.

If you activate the "**Using several steel sections or inserted beams**" check box instead of the section list, you can allocate different sections when designing sheet pile walls, soldier pile walls and combined sheet pile walls in a retaining wall (see **Partial Corrosion Example**). For bored soldier pile walls, diaphragm walls and bored pile walls, an inserted I beam can be defined at the top of the wall, which you select from the soldier pile wall section list via the then active "**Editor 2/Inserted beam**" menu item.

Steel design should always follow EC 3.

If the normal force is a compressive force, concrete design using the normal force design value is not on conservative. If the "**Characteristic normal force**" check box in the "**Concrete design:**" group box is activated, design is performed using the characteristic normal force.

The buttons in the bottom section of the dialog box allow you to select the type of retaining wall you wish to analyse. Having selected a type of wall and entered all the data necessary for its analysis, you can subsequently return to this menu item and select another type of wall. **GGU-RETAIN** maintains the data already entered, which can be used to analyse the new type of wall.

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**" or "**Editor 1/Verifications/Partial factors**". Verifications and safety factors can be modified at any time using these menu items.

8.2 GGU-RETAIN: "Load" menu item

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

8.3 GGU-RETAIN: "Save" menu item

You can save data entered or edited during program use to a file, to have them available later, 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 the same visualisation as was present at the time of saving.

8.4 GGU-RETAIN: "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 ".vrb" 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, ".vrb" will be used automatically.

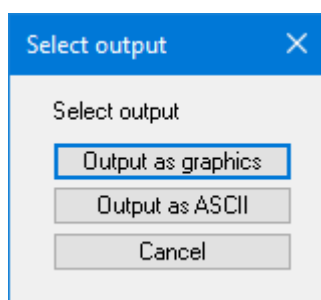
If the current system has been analysed at the time of saving, the analysis results are saved in the file. This is especially necessary if you want to use the results in this file for predeformations on other systems (see **"Theoretical principles: Predeformations"**), or if you want to subsequently generate summaries of certain construction phases (see **"Theoretical principles: Construction phases"**). 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.

8.5 "Print output table" menu item

8.5.1 GGU-RETAIN: Selecting the output format (Print output table)

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 program. To do this, select **"Output as graphics"** from the following options.



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

8.5.2 GGU-RETAIN: "Output as graphics" button (Print output table)

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.

Graphical table preferences

Page sizes

Page height [mm] 297.0

Page width [mm] 210.0

☒ Borders

Page margins [mm]

Left: 25.00 Right: 8.00

Top: 8.00 Bottom: 8.00

☒ Page margins

Output table margins

Upper margin [mm] 12.0

Lower margin [mm] 12.0

Left margin [mm] 5.0

Right margin [mm] 5.0

Font

Font size [mm] 2.5

Line spacing 1.2

Header

☒ With headers Edit

Footer

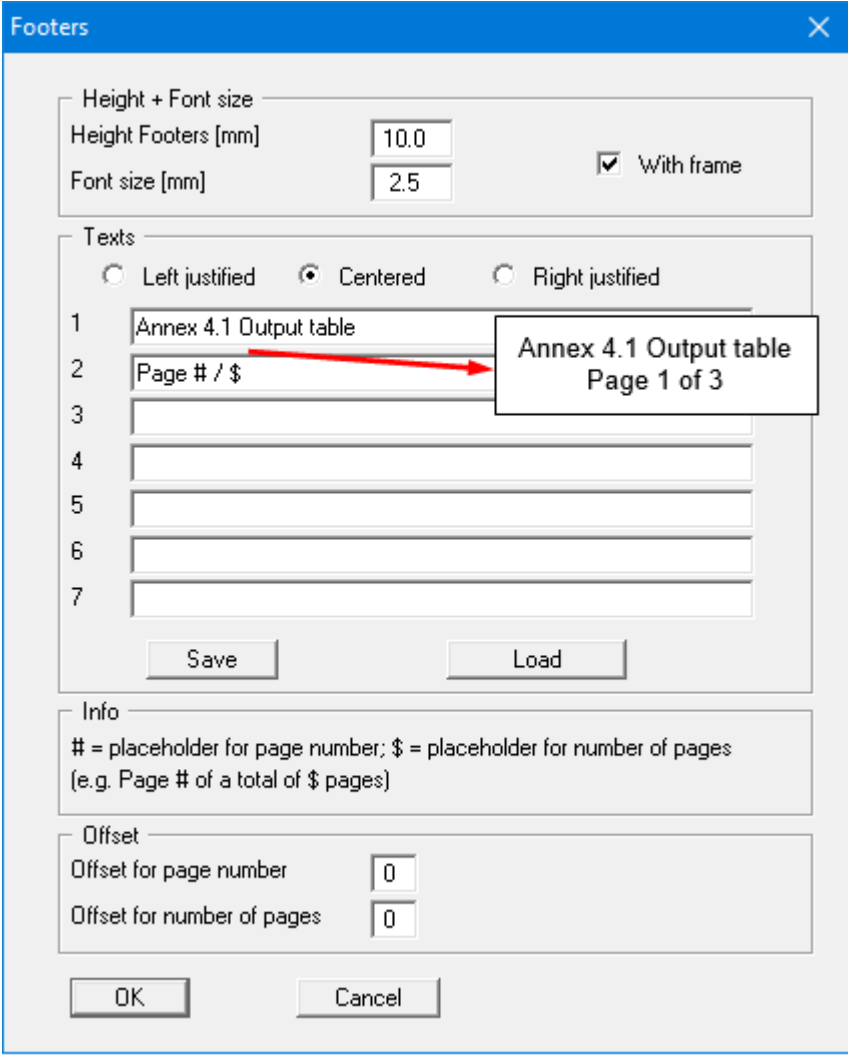
☒ With footers Edit

Save Load

OK Cancel

In the various group boxes of the dialog box, you can define preferences for the table output and layout. In some GGU programs a sketch of the system is integrated in the output table by activating an additional "**Incorporate graphics**" button.

If you need to add a header or footer (e.g. for page numbering), activate the appropriate check boxes "**With headers**" and/or "**With footers**" and click on the "**Edit**" button. You can then edit as required in a further dialog box. You can save your settings for the graphical output table presentation in a "**Protokoll.pin_ggu**" file at the program level so that they are loaded when the program starts. Using the "**Load**" button, the output table settings can also be subsequently loaded into an existing file, including that of another GGU program.



The 'Footers' dialog box is used to configure the footer information for the output. It includes sections for 'Height + Font size', 'Texts', 'Info', and 'Offset'. The 'Texts' section allows for multiple lines of footer text, with options for justification (Left, Center, Right). A red arrow points from the 'Page # / \$' text in line 2 to a preview box showing 'Annex 4.1 Output table' and 'Page 1 of 3'. The 'Info' section explains the placeholders: '#' for page number and '\$' for total number of pages. The 'Offset' section allows for offsets for the page number and total number of pages. Buttons for 'Save', 'Load', 'OK', and 'Cancel' are present.

Footers

Height + Font size
Height Footers [mm]
Font size [mm] ☒ With frame

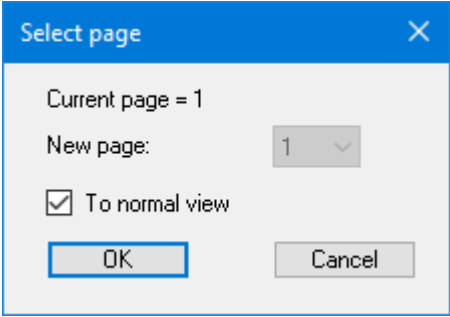
Texts
☐ Left justified ☒ Centered ☐ Right justified

1
2 Annex 4.1 Output table
Page 1 of 3
3
4
5
6
7

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

Offset
Offset for page number
Offset for number of pages

Automatic pagination can also be employed here if you work with the placeholders as described. After exiting the dialog boxes using "OK" you will see a further dialog box in which you can select the parameters to be used in the output table. After confirmation, the dialog box the output table is presented on the screen page by page. 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:



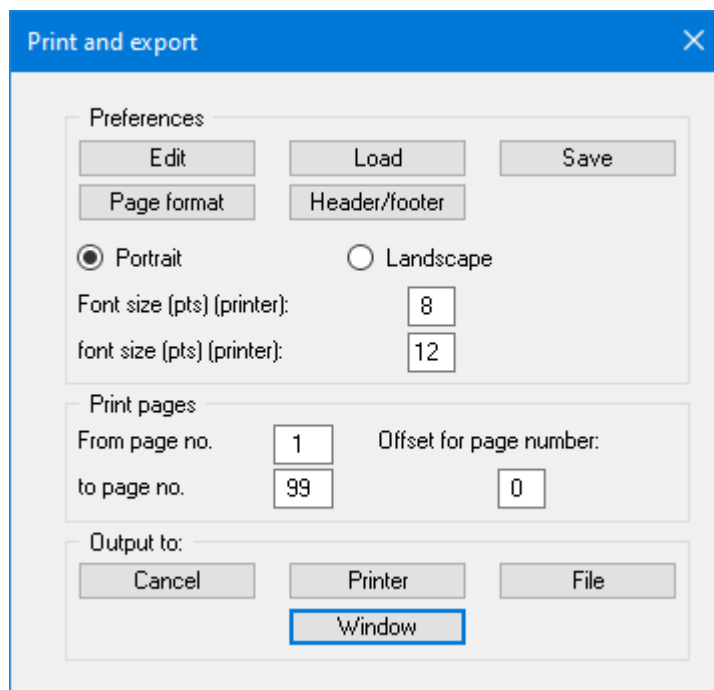
The 'Select page' dialog box is used to select the page to be displayed. It shows the current page (1) and allows the user to select a new page (1) from a dropdown menu. There is a checkbox for 'To normal view' which is checked. Buttons for 'OK' and 'Cancel' are present.

Select page

Current page = 1
New page:
☒ To normal view

8.5.3 GGU-RETAIN: "Output as ASCII" button (Print output table)

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



In the dialog box you can define output preferences.

- **"Output preferences"** group box

Using the **"Edit"** button the current output preferences can be changed or a different printer selected.

Using the **"Save"** button, all preferences from this dialog box can be saved to a file in order to have them available for a later session. If you select the program name with the extension **".drk"** as the file name and save this file at program level (default setting), this file will be loaded automatically the next time the program is started.

Using the **"Page format"** button you can define, amongst other things, the size of the left margin and the number of lines per page. The **"Header/footer"** button allows you to enter a header and footer text for each page. If the **"#"** symbol appears within the text, the current page number will be entered during printing (e.g. **"Page #"**). The text size is given in **"Pts"**. You can also change between **"Portrait"** and **"Landscape"** formats.

- **"Print pages"** group box

If you do not wish pagination to begin with **1** you can add an offset number to the check box. This offset will be added to the current page number. The output range is defined using **"From page no."** **"to page no."**.

- "Output to:" group box

Start output by clicking on "Printer" or "File". The file name can then be selected from or entered into the box. If you select the "Window" button the results are sent to a separate window. Further text editing options are available in this window, as well as loading, saving and printing.

8.6 GGU-RETAIN: "Export to GGU-STABILITY" menu item

The general stability can be simply verified by exporting the data 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.

You can export the retaining wall as a structural element.

8.7 GGU-RETAIN: "Output preferences" menu item

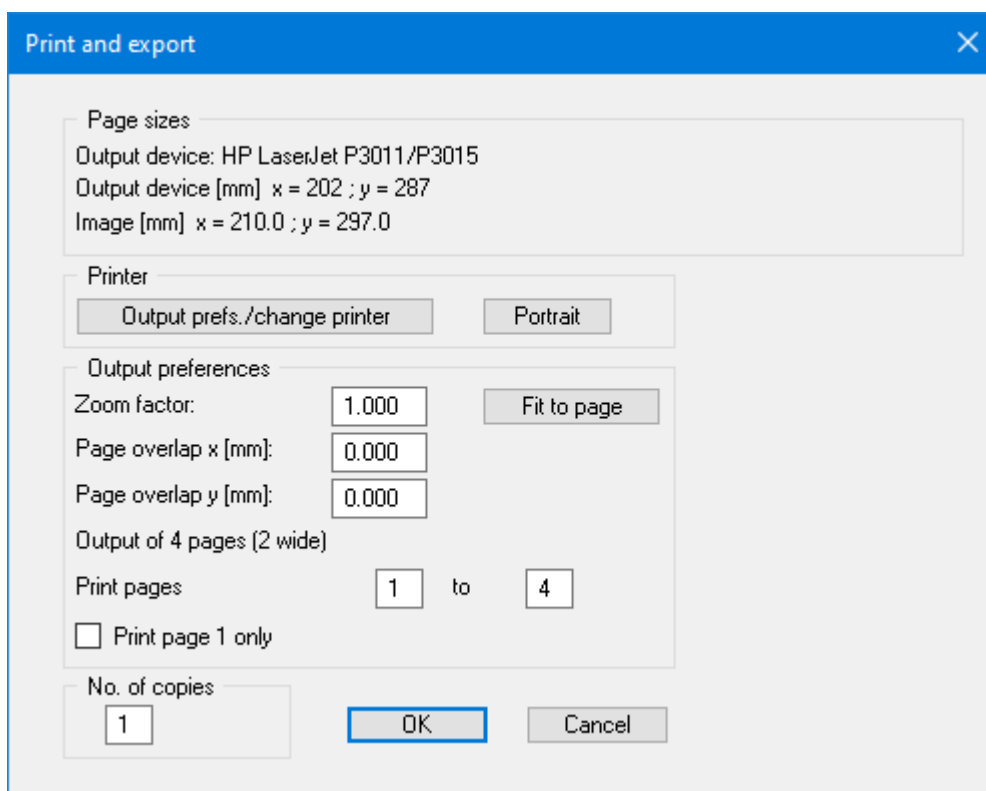
You can edit output preferences (e.g. swap between portrait and landscape) or change the printer in accordance with WINDOWS conventions.

8.8 GGU-RETAIN: "Print and export" menu item

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

- "Printer"

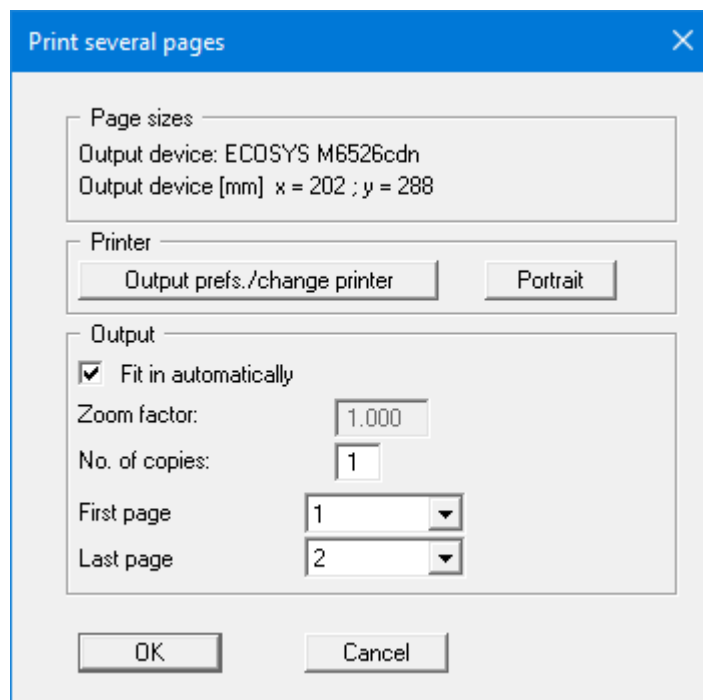
allows graphic output of the current screen contents (*graphical representation*) to the WINDOWS standard printer or to any other printer selected using the menu item "File/Output preferences". But you may also select a different printer in the following dialog box by pressing the "Output prefs./change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



You can select the pages of the report to be printed here. To obtain an output with a zoom factor of **1** (button "**Fit in automatically**" is deactivated), you must reduce the page format of your report page so that it lies within the printable area of the output device. To do this, use the dialog box in "**File/Print output table**" button "**Output as graphics**".

In some programs, you can transfer a representation of the system graphic to the output table if you activate the "**Print main graphic**" check box available there.

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

If you select the **"Copy/print area"** tool

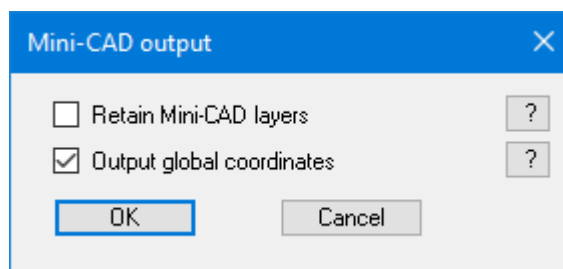


from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see **"Tips and tricks"**).

Using the **"Mini-CAD"** program module you can also import EMF files generated using other GGU applications into your graphics (see menu item **"Graphics preferences/Mini-CAD"**).

- **"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. After importing the file and pressing the [F9] function key (menu item **"Page size + margins/Auto-resize"**) the system coordinates and the scale are corrected according to the transferred world coordinates. -

"GGUMiniCAD"

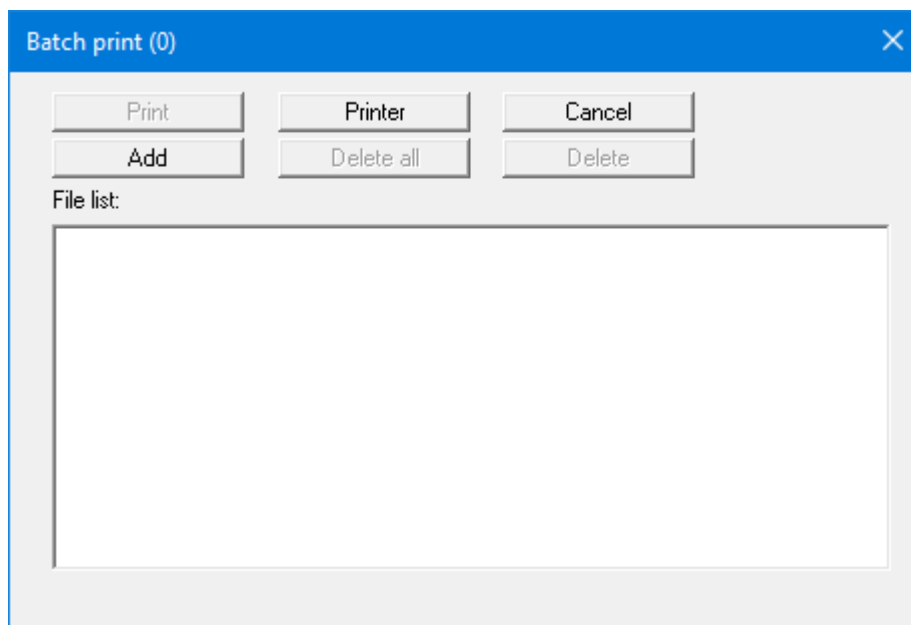
allows export of the graphics to a file in order to enable processing in the **GGUMiniCAD** program. -

"Cancel"

Printing is cancelled.

8.9 GGu-RETAIN: "Batch print" menu item

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



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

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

8.10 GGU-RETAIN: "Exit" menu item

After a confirmation prompt, you can quit the program.

8.11 GGU-RETAIN: "1, 2, 3, ..." menu items

If you have not stored a working directory for the program icon, the program jumps to the program directory after starting to load or save a file.

Under the menu items of the File menu, the most recently edited files are listed with the respective storage path ("**1, 2, 3, ...**"). Selecting one of these files loads the selected file. The next time the **"File/Load"** or **"File/Save"** menu items are called up in the same program session, the program automatically jumps to the directory of the last file loaded.

9 Editor 1 menu

9.1 GGU-RETAIN: "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 there).

9.2 GGU-RETAIN: "Excavation" menu item

A dialog box opens for defining the excavation and the selected retaining wall.

The 'Excavation' dialog box is shown with the following settings:

- Excavation**
 - Excavation base [m]: 3.00
 - Groundwater (right) [m]: 3.00
 - Groundwater (left) [m]: 3.00
 - Distributed load [kN/m²]: 10.00
 - Type of distributed load: Component above 10.0 kN/m² variable
 - ☒ Do not redistribute live loads
- Soldier pile wall**
 - Soldier pile width [m]: 0.500
 - Soldier pile spacing [m]: 2.00
- Passive side**
 - Distributed load [kN/m²]: 0.00

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

If the height of a previously defined system is subsequently set to absolute heights, a query follows after leaving the dialog box above asking for confirmation of whether soil strata and defined elements such as anchors, for example, should be adapted to the new wall top. 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. 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 following dialog box). "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 (see EAB, R 7).

In the group box below this in the dialog box you make the required entries for the retaining wall. This section of the box varies depending on the type of wall and the chosen safety factor concept. For instance, to analyse a bored pile wall you may see the following box:

Excavation

Excavation base [m] 3.00

Groundwater (right) [m] 3.00

Groundwater (left) [m] 3.00

Distributed load [kN/m²] 10.00

Type of distributed load: Component above 10.0 kN/m² variable

☒ Do not redistribute live loads

Bored pile wall

Bored pile diameter [m] 1.00

Bored pile spacing [m] 2.60

Number of unreinforced piles 2

☒ Unreinf. piles via qb,k+qs,k

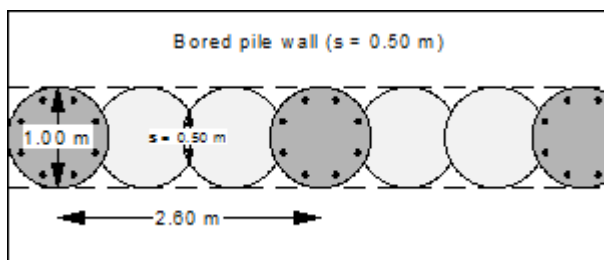
☐ Additional forces at base of beam

Passive side

Distributed load [kN/m²] 0.00

OK Cancel

You can define the diameter, the centres of the reinforced bored piles and the number of unreinforced piles (secant pile wall). The above adaptations lead to the following bored pile wall:



9.3 GGU-RETAIN: "Berms (active side)" menu item

You can define a maximum of 40 berms on the active side.

No.	Left [m]	Right [m]	delta h [m]	Surcharge [kN/m²]	Live load
0 berm(s) to edit					

Click on the "0 berm(s) to edit" button and enter the desired number of berms.

No.	Left [m]	Right [m]	delta h [m]	Surcharge [kN/m²]	Live load
1	1.000	2.000	0.500	0.00	☐

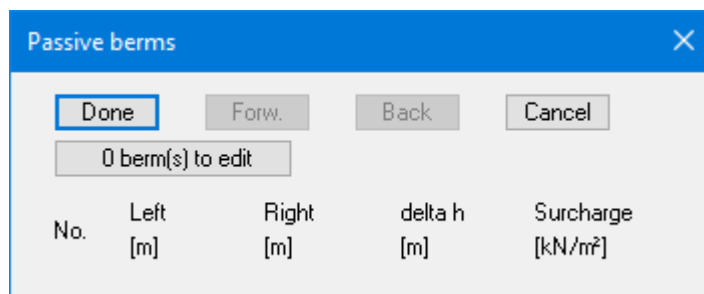
Enter the x-ordinates of the toe and head of the berm. With "**delta h**" you define the height of the berm; whereby negative values are also permitted. Finally, a "**Surcharge**" on the horizontal surface behind the head of the berm can be entered. This can be defined as a live load.

If berms already exist, you can delete existing berms by entering a "0".

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

9.4 GGU-RETAIN: "Berms (passive side)" menu item

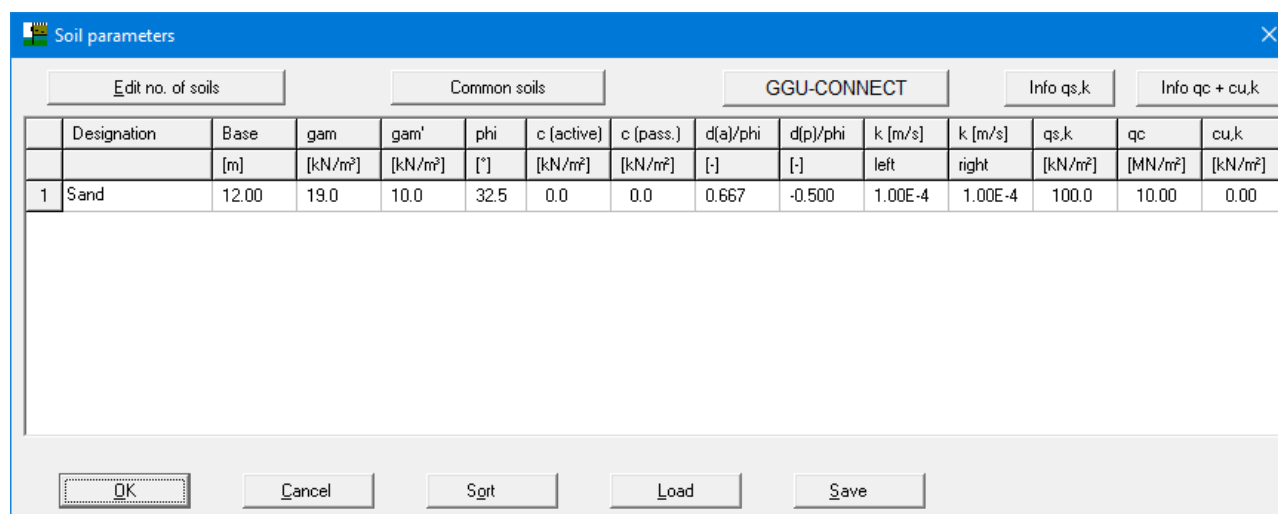
Berms on the passive side are defined in complete analogy to the berms on the active side, only the **"Live load"** check box is not available for this.



The "Passive berms" dialog box has a blue title bar with a close button. Below the title bar are four buttons: "Done" (highlighted with a blue border), "Forw.", "Back", and "Cancel". Below these buttons is a text field labeled "0 berm(s) to edit". At the bottom, there is a table with five columns: "No.", "Left [m]", "Right [m]", "delta h [m]", and "Surcharge [kN/m²]".

9.5 GGU-RETAIN: "Soils" menu item

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

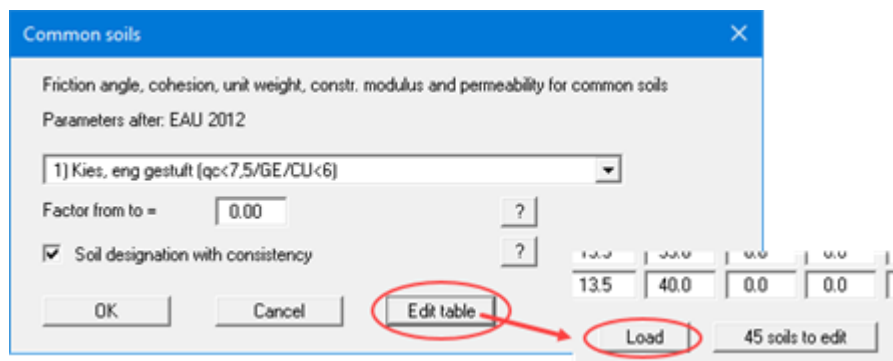


The "Soil parameters" dialog box has a blue title bar with a close button. Below the title bar are five buttons: "Edit no. of soils", "Common soils", "GGU-CONNECT", "Info qs,k", and "Info qc + cu,k". Below these buttons is a table with 14 columns: "Designation", "Base [m]", "gam [kN/m³]", "gam' [kN/m³]", "phi [°]", "c (active) [kN/m²]", "c (pass.) [kN/m²]", "d(a)/phi [-]", "d(p)/phi [-]", "k [m/s] left", "k [m/s] right", "qs,k [kN/m²]", "qc [MN/m²]", and "cu,k [kN/m²]". The first row of the table contains the following data: 1, Sand, 12.00, 19.0, 10.0, 32.5, 0.0, 0.0, 0.667, -0.500, 1.00E-4, 1.00E-4, 100.0, 10.00, 0.00. Below the table are five buttons: "OK", "Cancel", "Sort", "Load", and "Save".

If you would like to edit the number of soils, select the **"Edit number of soils"** or **"x soils to edit"** button and then enter the new number of soils. Using the **"Sort"** button, you can have the soils sorted according to depth. This sorting will be carried out automatically when leaving the dialog box, without the function being explicitly called-up. Erroneous input is thus impossible from the outset.

If several soils have been entered, and you would like to delete a soil which is not at the end of the list, you can assign this soil a large depth (e.g. 99.0). Then select the **"Sort"** button. The soil will now be at the end of the table and can be **deleted** by reducing the number of soils.

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 to open your modified database file when the program starts. You can also enter your own data ("**Edit table**" / "**x soils to edit**" button) and save it in the "**Soils.gng_ggu**" file. You can also use your adapted file in other GGU programs by means of the "**Common soils**" function if you copy the file into the appropriate GGU program folder.

All project data can be entered and managed in the **GGU-CONNECT** program. If you have already stored your drilling data as a soil strata model in your project, you can load the soil strata and properties as an xml file using the "**GGU-CONNECT**" button in the above dialog box. A brief overview of the procedure in **GGU-CONNECT** for exporting the data to the GGU analysis programs can be found here:

[GGU-CONNECT - Export contents for GGU analysis programs](#)

Layer depths (except when using the modulus of subgrade reaction) 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.

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

It is only necessary to enter permeabilities if hydrostatic pressures must be considered, and any hydraulic gradients, using flow conduits.

In addition, you can enter the soil parameter $q_{s,k}$ (= skin friction). However, this is only relevant if you want to analyse the pull-out resistance of anchors. Otherwise, it is not taken into consideration and does not appear in the soil properties legend. If the check box "**Verification with $q_{s,k}$** " in the "**Editor 2/Anchors**" dialog box is activated, the soil parameter $q_{s,k}$ is also shown in the soil properties legend.

To analyse the vertical capacity to EAU, EAB and EAP, enter the cone resistance q_c and the shear strength of the undrained soil c_u,k .

If you select analysis using the **p-y method**, you will see an adapted dialog box for entering the soil properties:

No.	Depth [m]	gamma [kN/m³]	gamma' [kN/m³]	phi [°]	c(akt) [kN/m²]	c(pas) [kN/m²]	d(akt)/phi [-]	d(pas)/phi [-]	qc MN/m²	cu,k kN/m²	eps 50% [-]	Type [-]	Designation
1	12.000	19.00	10.000	32.50	0.00	0.00	0.667	-0.667	10.00	0.00	0.0050	nicht bindig	Sand

Three types of soil are differentiated:

- non-cohesive
- cohesive soft
- cohesive stiff

An eps(50) value can also be defined for cohesive soils. More detailed explanations can be found by clicking the "?" button.

9.6 GGU-RETAIN: "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

General

☒ 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 [-]

Area loads

☒ Use active earth pressure for area loads

☐ Use at-rest earth pressure for area loads

☐ Use increased active earth pressure for area loads

Relationship: $(1.0 - \text{factor}) \cdot e(\text{active}) + \text{factor} \cdot e(\text{at-rest})$

Factor [-]

☒ Adapt wall friction for at-rest pressure and increased active earth pressure

The options for area loads can be specified separately.

The lower "**Adapt wall friction ...**" check box is the default setting. This setting is only required for analysis of vertical capacity.

9.7 GGU-RETAIN: "Active earth pressure" menu item

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

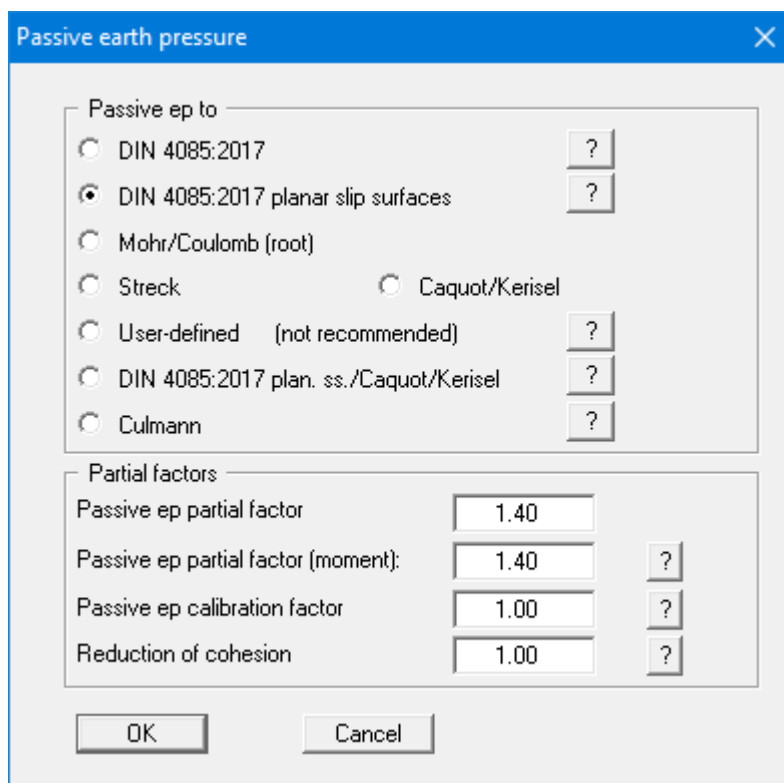
In the upper group box, you specify the type of active earth pressure calculation. If the "**Culmann**" check box is activated the active earth pressure is calculated after Culmann using a slice method with straight slip surfaces.

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

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

9.8 GGU-RETAIN: "Passive earth pressure" menu item

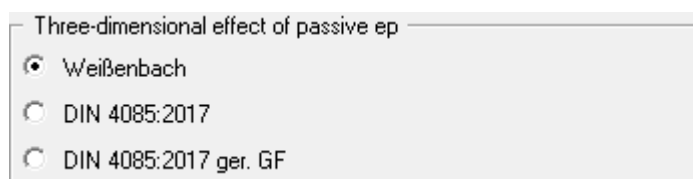
You can specify passive earth pressure preferences using this dialog box:



The dialog box titled "Passive earth pressure" contains two main sections. The first section, "Passive ep to", lists several calculation methods with radio buttons: DIN 4085:2017, DIN 4085:2017 planar slip surfaces (selected), Mohr/Coulomb (root), Streck, Caquot/Kerisel, User-defined (not recommended), DIN 4085:2017 plan. ss./Caquot/Kerisel, and Culmann. Each method has a "?" button next to it. The second section, "Partial factors", contains four input fields: "Passive ep partial factor" (1.40), "Passive ep partial factor (moment):" (1.40), "Passive ep calibration factor" (1.00), and "Reduction of cohesion" (1.00). Each input field has a "?" button next to it. At the bottom are "OK" and "Cancel" buttons.

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. The calculations use a slice method with straight slip surfaces.

An additional group box is displayed for soldier pile walls and bored soldier pile walls, in which the spatial effect of the passive earth pressure can be defined:



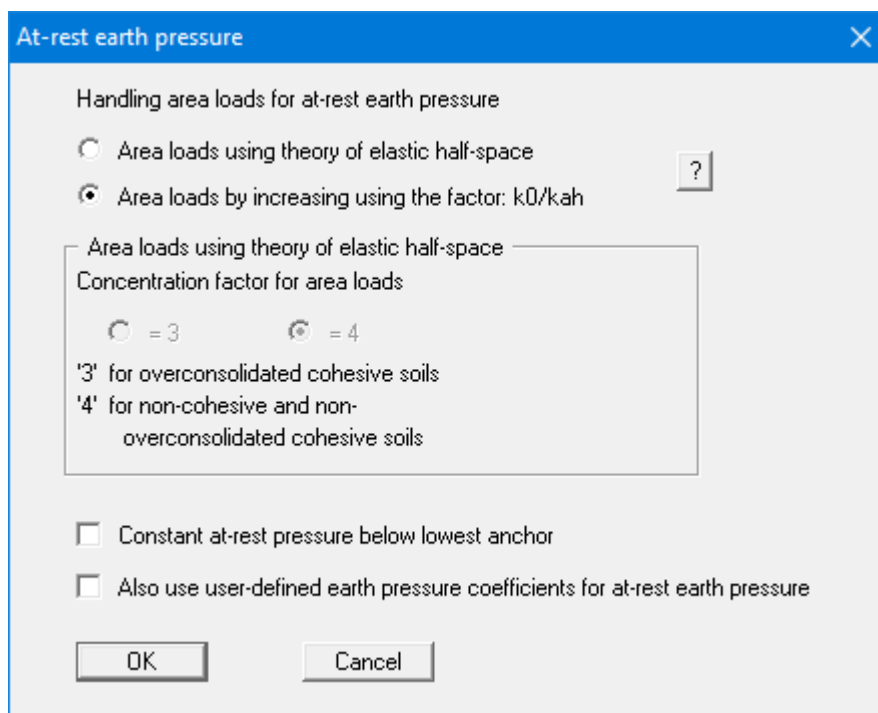
The group box titled "Three-dimensional effect of passive ep" contains three radio buttons: Weissenbach (selected), DIN 4085:2017, and DIN 4085:2017 ger. GF.

If you have selected partial safety factors, enter the partial factor for passive earth pressure in the dialog box and the calibration factor in accordance with the information in DIN 1054:2010/EC 7. The partial factor for passive earth pressure used to calculate the design moment can be reduced according to EAU 2012 under certain boundary conditions.

9.9 GGU-RETAIN: "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

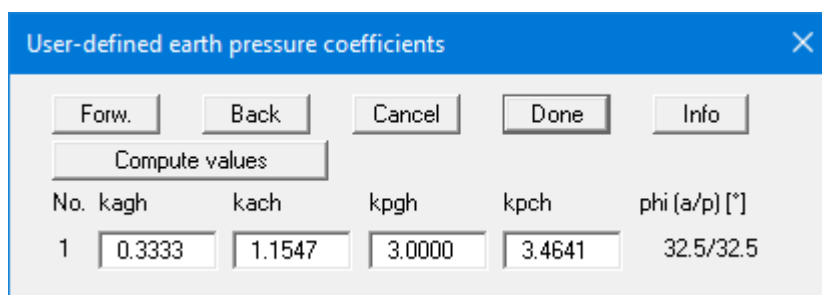
☐ Constant at-rest pressure below lowest anchor
☐ Also use user-defined earth pressure coefficients for at-rest earth pressure

OK Cancel

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

9.10 GGU-RETAIN: "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, the program will convert them for sloping ground using equation values for k_{ah0} and k_{ah} (see "**Theoretical principles: Berms**").



User-defined earth pressure coefficients

Forw. Back Cancel Done Info

Compute values

No.	k_{agh}	k_{ach}	k_{pgh}	k_{pch}	$\phi (a/p) [^\circ]$
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.

9.11 GGU-RETAIN: "Groundwater" menu item

With regard to the horizontal water pressure approach, please refer to the explanations in "Theoretical principles: Water".

Water pressure approach + hydraulic gradient

Horizontal water pressure approach:

☐ Classical ☒ With flow conduits

☐ No percolation around wall

☐ Toe potential to EAU 2004 R 115 ?

☐ Adopt flow below wall base

Flow below wall base [m] 5.000

Differential water pressure

☒ = set to '0.0' if directed to earth side

Consider hydraulic gradients i on:

☒ Active side ☒ Passive side

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

☒ Adhere to

OK Cancel

For homogeneous conditions, the "Toe potential to EAU 2004 R 115" check box can also be activated.

If the base of the sheet pile does not correspond to the base of the actual percolation (e.g. combined sheet pile wall with infill sheets not driven to the depth of the king pile), a depth may be defined for the base of the percolation below the toe after activating the check box "Adopt flow below wall base".

9.12 GGU-RETAIN: "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.

The screenshot shows the 'Seismic effects' dialog box with the following content:

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_{seis} [-]:

(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)

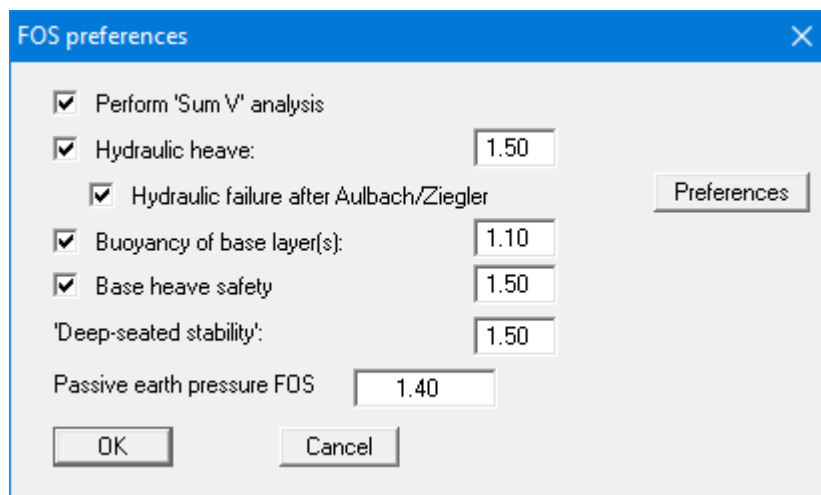
Info

$k_h = 0,1 \Rightarrow$ 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

9.13 GGU-RETAIN: "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.



FOS preferences

☒ Perform 'Sum V' analysis

☒ Hydraulic heave: 1.50

☒ Hydraulic failure after Aulbach/Ziegler

☒ Buoyancy of base layer(s): 1.10

☒ Base heave safety: 1.50

'Deep-seated stability': 1.50

Passive earth pressure FOS: 1.40

OK Cancel

Preferences

Then enter the safety factors for the individual verifications. The program's default values in the box are those required by the EAB. For soldier pile walls, the safety factors for "**Hydraulic heave**" and "**Buoyancy of base layer(s)**" are removed. Then, the verification of "**Sum H**" can be activated instead. If you do not want the program to perform one of the verifications, the corresponding check box must be deactivated.

9.14 GGU-RETAIN: "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 (here: Soldier pile wall selected).

Verifications/Partial factors preferences

Partial factors

Design situation: DIN EN 1997-1: DS-P

Permanent actions: 1.35

Permanent actions (water pressure): 1.35 ?

Permanent actions (at-rest pressure): 1.20

Variable actions: 1.50

Passive ep: 1.40

Passive ep (moment): 1.40 ?

☐ Moment resulting from $\gamma(E_{p,red})$ with full length ?

Passive ep adjustment factor: 1.00 ?

Pull-out resistance (piles): 1.10 ?

☒ Always design struts for DS-P ?

☐ Increase anchor force (ep) Factor: 1.15

☐ Always design anchors for DS-P ?

'Sum V' analysis

☒ Perform 'Sum V' analysis

☐ Simplified analysis for soldier pile walls

$C_v = C_h \cdot \tan(\text{factor} \cdot \phi)$ Factor [-]: 0.333 ?

☒ Perform 'Vertical capacity' analysis (EAU, EAP and EAB)

☐ Use own empirical data ?

☒ 2019 annual report ☒ EAB 2021

Analysis of heave safety

☒ Perform analysis $\gamma(\text{bearing capacity})$: 1.40

Default values

To DIN EN 1997-1 To ÖNORM EN 1997-1

OK Cancel

The partial factor for the permanent action because of water pressure can be reduced according to EAU 2012 under certain boundary conditions. The same applies to the passive earth pressure partial factor used for calculating the design moment.

In the dialog box you can deactivate verifications that you do not want to carry out. If you have selected **"Soldier pile wall"** or **"Bored soldier PW"** as the retaining wall, a further area appears for verifying the sum H:

'Sum H' analysis

☒ Perform analysis ☒ Bh via computed support ?

☒ $Bh' = Bh - 0.5 \cdot C_h$?

☒ $E_{ph,d}$ with adjustment factor ?

☒ $E_{ph,d}$ with reduction of cohesion

In the "**Default values**" group box you can use the "**To DIN 1054:2010**" button to transfer the partial safety factors for the various load cases of DIN 1054:2010 or EC 7. The selected design situation is automatically entered at the top of the dialog box and can, if desired, be displayed in the **General legend**. However, you can also use your own designations here.

In the partial safety concept according to EC 7, the designations of the load cases have been changed:

- 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.

9.15 GGU-RETAIN: "Deep-seated stability/Heave of anchor soil" menu item

Use this menu item to define preferences for analysis of deep-seated stability.

Deep-seated stability/heave of anchor soil preferences

Deep-seated stability

Analysis for increased earth pressure/at-rest pressure

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

☒ Use partial factors DS-P/T ?

Wall friction angle

☐ Enter wall friction angle (equiv. wall) manually

Wall friction angle / phi [-]: 0.000 ?

Earth pressure on equivalent wall

☒ Area loads rectangular on equivalent wall

Surch. earth pressures on equiv. wall

Eh,g [kN/m]: 0.00

Eh,q [kN/m]: 0.00

Ev,g [kN/m]: 0.00

Ev,q [kN/m]: 0.00

Intersection with wall

☐ Intersection with wall 'manually'

Depth of intersection [m]: 5.000

Heave of anchor soil

☒ Perform analysis

☐ Earth pressures with mean partial safety factors

☐ Use your own adjustment factor ?

Passive earth pressure adjustment factor [-]: 1.00

OK Cancel

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 like the method described in Section 7.3.4 of the Piling Handbook 1977.

9.16 GGU-RETAIN: "Buoyancy + Hydraulic heave" menu item

This menu item opens a dialog box in which you can activate 'traditional' hydraulic heave analysis and analysis of the buoyancy of the excavation base. In addition, the hydraulic heave safety can be analysed using the Aulbach/Ziegler method (Aulbach, Benjamin; Ziegler, Martin: Simplified design of excavation support and shafts for safety against hydraulic heave = *Einfache Bemessung von Baugruben und Schächten im Hinblick auf die Sicherheit gegen hydraulischen Grundbruch*. In: Geomechanics and Tunnelling 6 (2013), No. 4, p. 362-374, ISSN 1865-7362.).

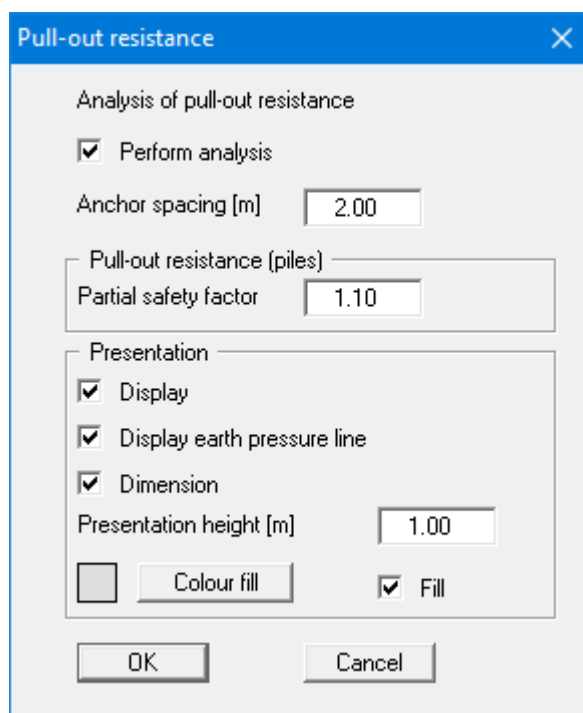
If you have activated "**Hydraulic failure after Aulbach/Ziegler**", you can enter the required system data via the "**Preferences**" button.

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.

9.17 GGU-RETAIN: "Pull-out resistance" menu item

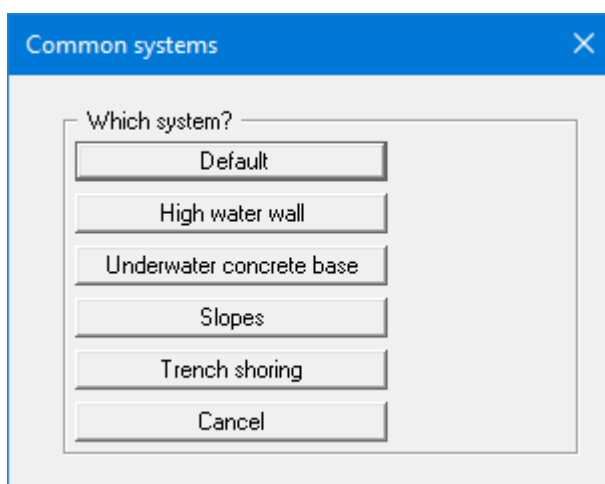
If anchors with activated "Verification with $q_{s,k}$ " check box are employed (see menu item "**Editor 2/Anchors**"), the pull-out resistance analysis can be activated via this menu item.



Preferences for the following graphical visualisation can be specified in the lower group box of the dialog box.

9.18 GGU-RETAIN: "Common systems" menu item

You can quickly design a system for the shoring wall systems shown in the dialog box. Without having to work through the various menu items one after the other, you can enter the essential elements such as excavation geometry, groundwater, anchors, soil properties etc. in just one dialog box.



For example, if you want to calculate a default system, select the corresponding button and the following dialog box will appear:

Default system

☐ Mirror system after generation

Excavation base [m]: 3.00

System base [m]: 12.00

GW (active) [m]: 3.00

GW (passive) [m]: 3.00

☐ Generate anchor

Height [m]: 0.75

Length [m]: 8.00

Inclination [°]: 10.00

Anchor spacing [m]: 2.00

Distributed load [kN/m²]: 10.00

Soil properties

phi (top)	32.5	phi (bottom)	27.5
c (top)	0.5	c bottom	5.0
gam(f) (top)	19.0	gam(f) (bottom)	20.0
Design. (top)	S (qc>10)	Design. (bottom)	SU

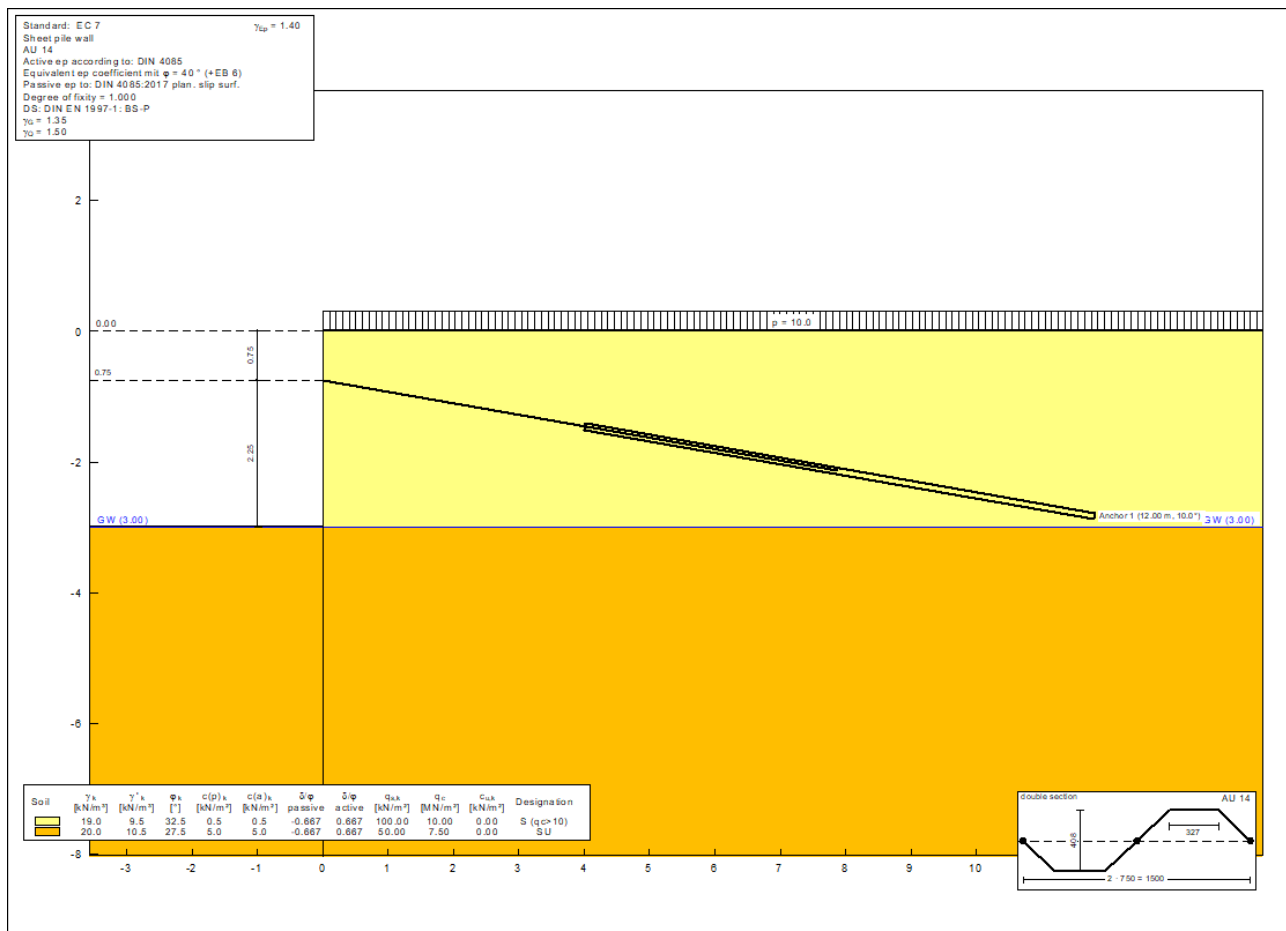
☐ Calculate with flow conduits

☐ System is embedded

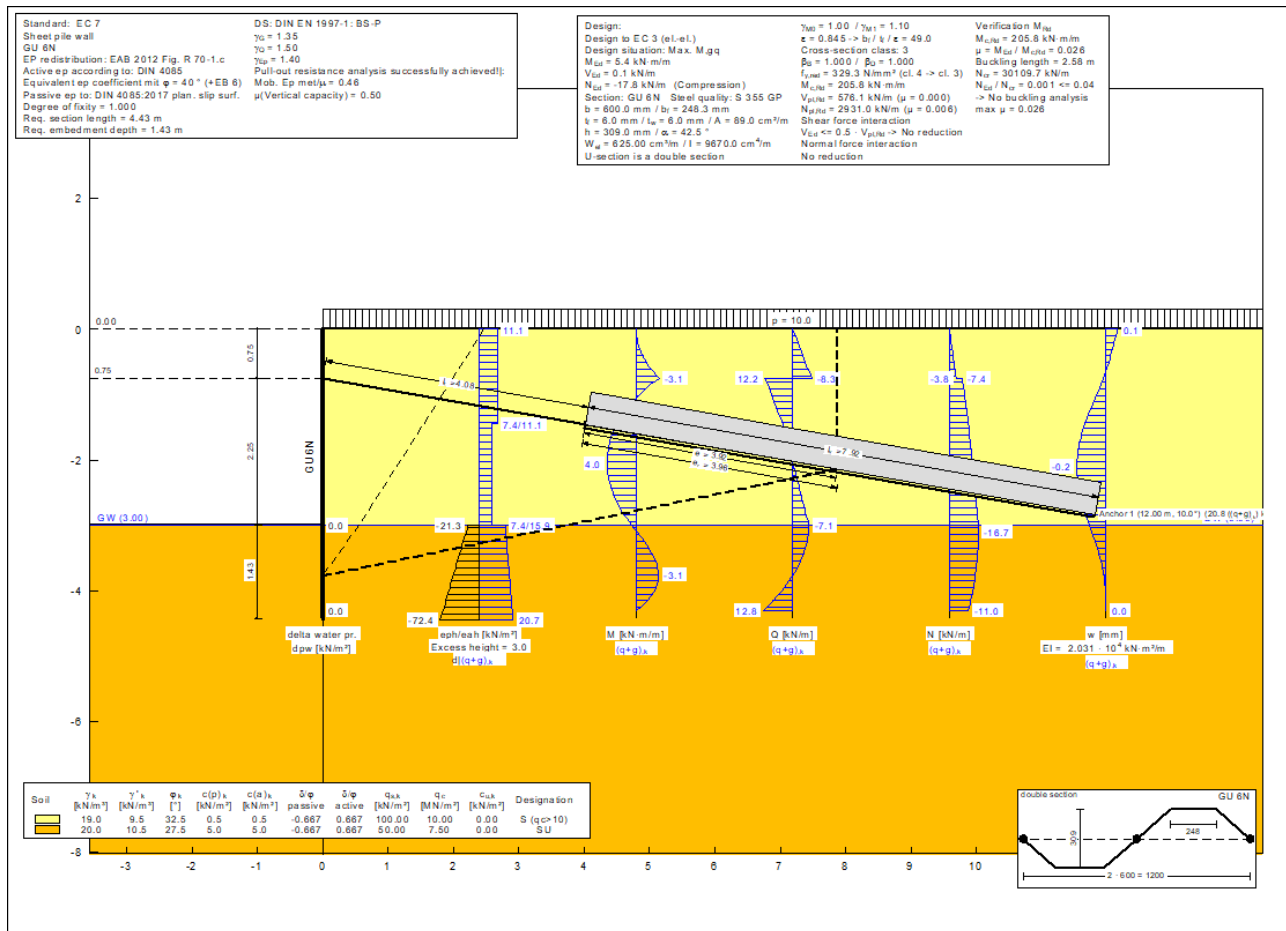
Which type of wall: Sheet pile wall

OK Cancel

After confirming the entries, the following system is displayed.



You can now start your calculation as usual via " **System/Analyse**" or [F5] and receive the following result, for example:



10 Editor 2 menu

10.1 GGU-RETAIN: "Lateral pressures" menu item

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

No.	top [m]	bottom [m]	e.k (top) [kN/m²]	e.k (bottom) [kN/m²]	Type
1	1.000	2.000	10.00	20.00	Permanent

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.

The lateral pressures can be represented by permanent or variable loads, water pressures or passive earth pressures. The characteristic values are entered and are automatically multiplied or divided by the appropriate partial safety factors.

10.2 GGU-RETAIN: "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)	☐

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 "**Theoretical principles: Area loads**"). To define an area load as a live load, activate the corresponding "**Live**" check box on the right.

Use the "Loads with limited plan dimensions" button, to define such loads (see also "Theoretical principles: 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).

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]:
 $p(h)$ [kN/m²]:
 $x(\text{left})$ [m]:
 $x(\text{right})$ [m]:
 b [m]:
 Depth [m]:
 No. of subsections:
 Type of load:
☐ Delete existing loads
 Analyse compliant with:

10.3 GGU-RETAIN: "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:

Done Forw. Back Cancel Sort

1 bounded surcharge(s) to edit ☒ Do not redistribute live loads

No.	p [kN/m ²]	x [m]	Depth [m]	Live
1	1.00	0.000	0.000	☐

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.

10.4 GGU-RETAIN: "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
0 double-bounded surcharge(s) to edit					

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 in "**Theoretical principles: Double-bounded surcharges (active site)**") may be adopted at its full value for the bounded surcharge with the negative sign has not yet been conclusively clarified. This can be influenced using the control parameter:

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

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

10.5 GGU-RETAIN: "Action boundary conditions" menu item

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

No.	Depth [m]	M,g [kN·m/m]	M,q [kN·m/m]	H,g [kN/m]	H,q [kN/m]	V,g [kN/m]	V,q [kN/m]
1	0.00	0.00	0.00	-15.00	0.00	0.00	0.00

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 adopting the **partial safety factor concept** the forces are subdivided into permanent and changeable forces.

10.6 GGU-RETAIN: "Displacement boundary conditions" menu item

You can introduce displacement boundary conditions anywhere along the wall.

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

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

10.7 GGU-RETAIN: "Anchors" menu item

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

No.	Depth [m]	Incl. [°]	Length [m]	EA [kN/m]	H DM [m]	L GR [m]	Verification with qs,k	FL [m]	TL [m]	D [m]
1	0.50	20.0	9.00	2.100E+7	0.00	5.00	☐	6.50	11.51	0.100

It is necessary to enter the following information:

- Depth [m] = position of anchor head
- Inclination [°] of anchor
- Length [m] of anchor = starting point of deep-seated stability
- 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.

By activating the "**Verification with qs,k**" check box instead of "**H DM**" and "**L GR**" the following data have to be entered:

- FL [m] = free length (top)
- TL [m] = total injection length
- D [m] = diameter

Then, the skin friction values qs,k defined in the menu item "**Editor 1/Soils**" are used for calculation.

10.8 GGU-RETAIN: "Struts" menu item

Struts are defined in this dialog box.

No.	Depth [m]	Incl. [°]	Length [m]	EA [kN/m]	EI [kN·m²/m]	V load [kN/m²/m]	Hinge Left	Hinge right	No. of rods
1	0.50	0.0	9.00	2.100E+7	2.100E+7	0.00	☒ Yes	☒ 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
- EI [kN·m²/m] = bending stiffness of the strut
- V Load [kN/m²/m] = vertical load on the strut
- Hinge (right)
Here you specify whether the strut is hinged or rigidly fixed to the wall.
- Hinge (left)
Here you specify the support conditions of the strut *on the "opposite side"*.

If neither hinge option is selected, the following happens:

The strut is then modelled with only half length!! On the left-hand edge of the "half" strut complete fixation is assumed. The support can be vertically displaced.

- 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 **"Theoretical principles: Structural system"**). The maximum number of rods that can be specified is 20.

10.9 GGU-RETAIN: "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.

10.10 GGU-RETAIN: "Groundwater potentials" menu item

You can define additional potentials anywhere along the wall (see **"Theoretical principles: Water pressure"**).

No.	Depth [m]	Value [m]	left/right
1	6.000	8.000	☒ Right
2	6.000	8.000	☐ Right

In the above example, potentials of 8.0 m have been entered for 6.0 m below the top of the wall on either side. Additional potentials are only taken into consideration if you selected the water pressure approach using flow conduits.

Instead of potentials in [m] the data may also be entered as water pressures in [kN/m²] after clicking **"Switch to: Enter water pressure"**.

10.11 GGU-RETAIN: "Subgrade reaction moduli" menu item

If the analysis is to be performed for an elastically bedded wall toe, it is necessary to enter a modulus of subgrade reaction. In this dialog box you can enter the profile of the subgrade reaction moduli.

Subgrade reaction moduli

Done Forw. Back Cancel Sort Subgrade modulus unit: kN/m³

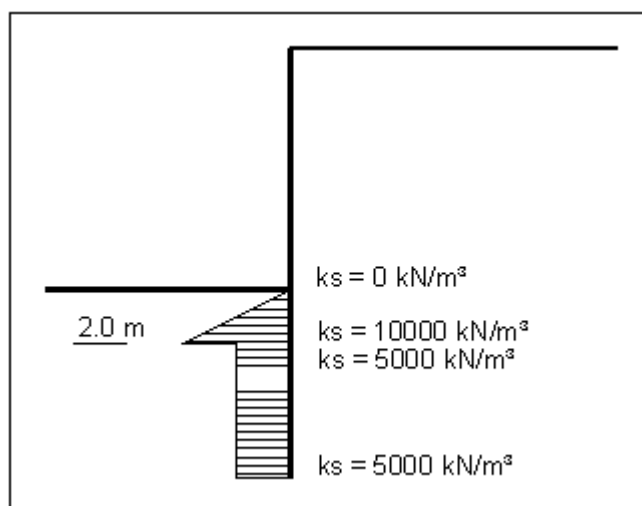
2 subgr. modulus(i) to edit Lock to layer boundaries Info "μ(ks)"

Values valid from excavation base!

No.	to depth [m]	ks top [MN/m ²]	ks bottom [MN/m ²]	μ(ks) [-]
1	2.000	0.000	10.000	0.0000
2	8.000	5.000	5.000	0.0000

The number of subgrade reaction moduli can be adapted to your system using the "**x subgr. modulus(i) to edit**" button. It must be noted, however, that in this case the depth data is with reference to the excavation base and are positive downwards, regardless of the initial setting of the "**Use absolute heights**" check box. You can also have the program take care of the depth input by activating the "**Lock to layer boundaries**" check box. Now, you only need to enter the values for the modulus of subgrade reaction.

The following modulus of subgrade reaction profile was modelled using the values shown in the dialog box:



The tangential bedding is given in the "**μ(ks)**" field as a multiple of the horizontal bedding. However, the factor is generally of little importance, since the wall toe is considered to be vertically non-displaceable, meaning that deformations in a longitudinal direction, and any associated tangential subgrade reaction moduli, are small.

10.12 GGU-RETAIN: "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.

The screenshot shows a dialog box titled "Compaction earth pressure" with a close button (X) in the top right corner. The dialog contains several sections with checkboxes and input fields:

- Adopt compaction earth pressure:** A checked checkbox.
- Surface loading:**
 - ☐ Distributed load as surface load (with a help icon ?)
 - Compaction plane [m]: A text input field containing "0.00" (with a help icon ?).
- Compaction:**
 - ☒ Intense compaction
 - ☐ Light compaction (with a help icon ?)
- Wand:**
 - ☐ Yielding wall
 - ☒ Non-yielding wall
 - Width of backfill space [m]: A text input field containing "1.00".
- Soil properties:**
 - ☒ Use uppermost soil layer properties
 - ☐ Use user-defined soil properties
 - Friction angle $\phi_{i,k}$ [°]: A text input field containing "32.50".
 - $\gamma_{m,k}$ [kN/m³]: A text input field containing "19.00".

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

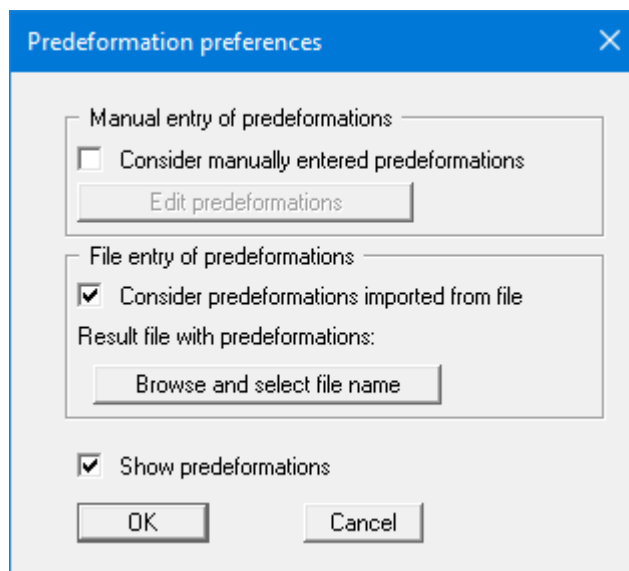
Two files containing analyses taken from *Geotechnische Nachweise nach EC 7 und DIN 1054* (Geotechnical analyses to EC7 and DIN 1054), 3rd edition 2012, by Martin Ziegler, can be found in the program's example folder. The analyses refer to pages 70 and 72 of the literature references.

10.13 GGU-RETAIN: "Predeformation information" menu item

You will see information on predeformations.

10.14 GGU-RETAIN: "Predeformation preferences" menu item

If predeformations from previous construction phases must be taken into consideration as boundary conditions (see also "**Theoretical principles: Predeformation**"), the necessary settings are made here.

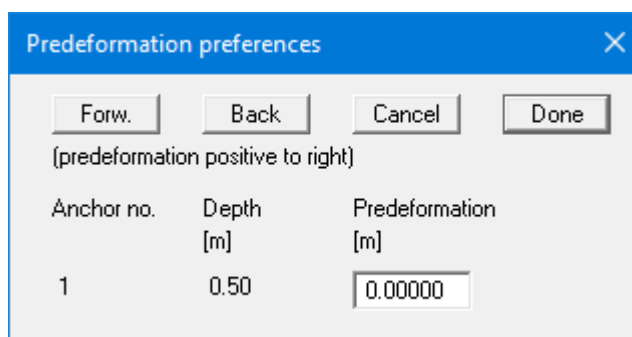


In the bottom section of the dialog box, you can choose whether or not to have the loaded predeformation data shown on screen. If you choose to have it shown, the system graphics show the system's bending line with predeformations.

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 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.

10.15 "Soldier pile"/"Sections"/"Bored pile wall"/"Diaphragm wall"/ "Bored soldier pile wall"/"Girder"/"Section data"/"Inserted beams" menu items

10.15.1 General note to soldier piles and sections

This menu item contains various designations and dialog boxes, depending on the type of wall selected in "**File/New**" or "**Editor 1/Analysis options**" and on whether you use the section list or several steel sections or inserted beams.

10.15.2 GGU-RETAIN: "Soldier piles" menu item

If you selected the "**Soldier pile wall**" and "**Using section list**" check boxes in the "**File/New**" or "**Editor 1/Analysis options**" menu items, you will see the following dialog box with the currently loaded list of soldier piles. For this view the "**Steel design to EC 3**" check box in the "**Editor 1/Analysis options**" menu item must be activated.

Soldier piles

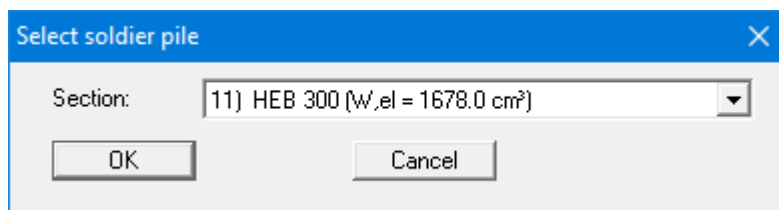
Forw. Back Cancel Done Load Save Sort Delete Delete doubles Simulate corrosion Weld on plates

Go to no.: 1 86 Soldier piles to edit Notation info Select section Create old record Create waling

No.	Designation	h [mm]	b [mm]	t _f [mm]	t _w [mm]	r [mm]	W _{el} [cm ²]	W _{pl} [cm ²]	A [cm ²]	I [cm ⁴]	I section	Available steel qualities		
												S 235	S 355	S 460
1	HEB 100	100.0	100.0	10.0	6.0	12.0	89.9	104.0	26.0	450.0	✓ Yes	✓	✓	✓
2	HEB 120	120.0	120.0	11.0	6.5	12.0	144.0	165.0	34.0	864.0	✓ Yes	✓	✓	✓
3	HEB 140	140.0	140.0	12.0	7.0	12.0	216.0	245.0	43.0	1509.0	✓ Yes	✓	✓	✓
4	HEB 160	160.0	160.0	13.0	8.0	15.0	311.0	354.0	54.3	2492.0	✓ Yes	✓	✓	✓
5	HEB 180	180.0	180.0	14.0	8.5	15.0	426.0	481.0	65.3	3831.0	✓ Yes	✓	✓	✓
6	HEB 200	200.0	200.0	15.0	9.0	18.0	570.0	643.0	78.1	5696.0	✓ Yes	✓	✓	✓
7	HEB 220	220.0	220.0	16.0	9.5	18.0	736.0	827.0	91.0	8091.0	✓ Yes	✓	✓	✓
8	HEB 240	240.0	240.0	17.0	10.0	21.0	938.0	1053.0	106.0	11259.0	✓ Yes	✓	✓	✓
9	HEB 260	260.0	260.0	17.5	10.0	24.0	1148.0	1283.0	118.0	14919.0	✓ Yes	✓	✓	✓
10	HEB 280	280.0	280.0	18.0	10.5	24.0	1376.0	1534.0	131.0	19270.0	✓ Yes	✓	✓	✓
11	HEB 300	300.0	300.0	19.0	11.0	27.0	1678.0	1869.0	149.0	25166.0	✓ Yes	✓	✓	✓
12	HEB 320	320.0	300.0	20.5	11.5	27.0	1926.0	2149.0	161.0	30824.0	✓ Yes	✓	✓	✓
13	HEB 340	340.0	300.0	21.5	12.0	27.0	2156.0	2408.0	171.0	36656.0	✓ Yes	✓	✓	✓
14	HEB 360	360.0	300.0	22.5	12.5	27.0	2400.0	2683.0	181.0	43193.0	✓ Yes	✓	✓	✓
15	HEB 400	400.0	300.0	24.0	13.5	27.0	2884.0	3232.0	198.0	57680.0	✓ Yes	✓	✓	✓
16	HEB 450	450.0	300.0	26.0	14.0	27.0	3551.0	3982.0	218.0	79888.0	✓ Yes	✓	✓	✓
17	HEB 500	500.0	300.0	28.0	14.5	27.0	4287.0	4815.0	239.0	107176.0	✓ Yes	✓	✓	✓
18	HEB 550	550.0	300.0	29.0	15.0	27.0	4971.0	5591.0	254.0	136691.0	✓ Yes	✓	✓	✓
19	HEB 600	600.0	300.0	30.0	15.5	27.0	5701.0	6425.0	270.0	171041.0	✓ Yes	✓	✓	✓
20	HEB 650	650.0	300.0	31.0	16.0	27.0	6480.0	7320.0	286.0	210616.0	✓ Yes	✓	✓	✓

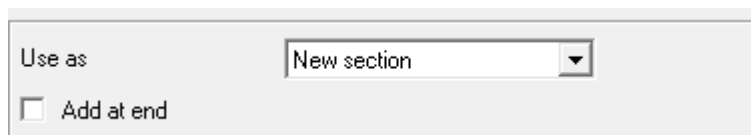
The following options are available:

- You can navigate through the list using **"Forw."** and **"Back"**. **"Go to no."** allows you to jump to the soldier pile specified.
- **"Cancel", "Done"**
Exit the dialog box either saving or rejecting your modifications using these buttons.
- **"x Soldier piles to edit", "Notation info"**
Using this button you may expand or reduce the list of soldier piles. New soldier piles are added to the end of the list. A description of the abbreviations used to enter the section data is available via the **"Notation info"** button.
- **"Load", "Save", "Sort", "Delete", "Delete doubles"**
A different soldier pile list can be opened by pressing the **"Load"** button. It is then possible to append the new list to an already open soldier pile list. After appending soldier piles, it may be expedient to delete any double sections in the list by pressing the **"Delete doubles"** button. The soldier piles can then be sorted either by moment of inertia or by name by pressing the **"Sort"** button. The modified soldier pile list may then be saved to the program folder as **".tbw_ggu"** file for subsequent analyses by pressing the **"Save"** button. Any number of sections in a sequence in the list can be deleted using the **"Delete"** button.
- **"Select section"**
A dialog box opens in which you select the required soldier pile for design.



- **"Simulate corrosion"**

You can define the corrosion for any soldier pile, which is then used to calculate the new section data. The new section data can be allocated to the selected soldier pile, or to a different, existing soldier pile. The data can also be allocated to a new soldier pile, which is added to the existing list (also see **"Worked example 2: Sheet pile wall with corrosion"**).



- **"Weld on plates"**

If you need to simulate reinforcement using weld-on plates, click the previous **"Simulate corrosion"** button and enter a negative corrosion.

- **"Create old record"**

Using this button you can save a record containing the old section tables based on the global safety factors (".tbw", ".spw").

- **"Create waling"**

Using this button you can generate a waling data set from the soldier pile section list and save it to a ".grt" file. This file can be loaded in the **"Editor 2/Waling design"** menu item after clicking the **"Edit waling"** button.

If Steel design to EC 3 is not activated, the old dialog box opens. A check box in front of the number of each soldier pile can be activated. This is used to select the respective soldier pile. If you then press the **"Selected section as design section"** button, this section is used as the design section. The section parameters ("h" = height; "b" = width; "A" = area; "I" = moment of inertia) can be edited. The program uses I and h to determine the section modulus **W** during the subsequent design phase. The variable **S** is the first moment of area of the section and **s** is the web thickness. The value of "**S/s**" is required for verification of shear stress.

10.15.3 GGU-RETAIN: "Sections" menu item

Upon selecting **"Sheet pile wall"** and **"Using section list"** you see a dialog box which appears almost exactly the same as that for soldier piles. Here, too, the **"Steel design to EC 3"** check box must be activated. The sheet pile wall sections can be stored in a ".spw_ggu" file.

If you do **not** design to EC 3, it is important to note that for sheet pile wall sections the area "A" must be entered in [cm²/m] and the moment of inertia "I" in [cm⁴/m]. Also note that the section values given in the table assume that a shear resistant connection exists between the supplied double sections. The variable **S** is the first moment of area of the section and **s** is the web thickness. The "**alp**" variable is the opening angle of the section. The value of "**S·sin(alp)/s**" is required for verification of shear stress.

If "**Combined sheet pile wall**" is selected, the sections dialog box is expanded slightly:

No.	Designation	h [mm]	b [mm]	b.f [mm]	t.f [mm]	t.w [mm]	alpha [°]	W.el [cm ³ /m]	W.pl [cm ³ /m]	A [cm ² /m]	I [cm ⁴ /m]	Z section	Yes
1	AU 14	408.0	750.0	327.2	10.0	8.3	47.8	1405.0	1663.0	132.0	28680.0	☐	☐
2	AU 16	411.0	750.0	327.2	11.5	9.3	47.8	1600.0	1891.0	147.0	32850.0	☐	☐
3	AU 18	441.0	750.0	365.4	10.5	9.1	54.7	1780.0	2082.0	150.0	39300.0	☐	☐
4	AU 20	444.0	750.0	365.4	12.0	10.0	54.7	2000.0	2339.0	165.0	44440.0	☐	☐
5	AU 23	447.0	750.0	406.0	13.0	9.5	59.6	2270.0	2600.0	173.0	50700.0	☐	☐
6	AU 25	450.0	750.0	406.0	14.5	10.2	59.6	2500.0	2866.0	188.0	56240.0	☐	☐
7	AZ 12-700	314.0	700.0	356.4	8.5	8.5	42.8	1205.0	1415.0	123.0	18880.0	☒	☐
8	AZ 12-770	344.0	770.0	351.0	8.5	8.5	39.5	1245.0	1480.0	120.0	21430.0	☒	☐
9	AZ 13-700	315.0	700.0	356.4	9.5	9.5	42.8	1305.0	1540.0	135.0	20540.0	☒	☐
10	AZ 13-700-10/10	316.0	700.0	356.4	10.0	10.0	42.8	1355.0	1600.0	140.0	21370.0	☒	☐
11	AZ 13-770	344.0	770.0	351.0	9.0	9.0	39.5	1300.0	1546.0	126.0	22360.0	☒	☐
12	AZ 14-700	316.0	700.0	356.4	10.5	10.5	42.8	1405.0	1665.0	146.0	22190.0	☒	☐
13	AZ 14-770	345.0	770.0	351.0	9.5	9.5	39.5	1355.0	1611.0	132.0	23300.0	☒	☐
14	AZ 14-770-10/10	345.0	770.0	351.0	10.0	10.0	39.5	1405.0	1677.0	137.0	24240.0	☒	☐
15	AZ 17	379.0	630.0	356.0	8.5	8.5	55.4	1665.0	1944.0	138.0	31580.0	☒	☐
16	AZ 17-700	420.0	700.0	352.8	8.5	8.5	51.2	1730.0	2027.0	133.0	36230.0	☒	☐
17	AZ 18	380.0	630.0	356.0	9.5	9.5	55.4	1800.0	2104.0	150.0	34200.0	☒	☐
18	AZ 18-10/10	381.0	630.0	356.0	10.0	10.0	55.4	1870.0	2189.0	157.0	35540.0	☒	☐
19	AZ 18-700	420.0	700.0	352.8	9.0	9.0	51.2	1800.0	2116.0	139.0	37800.0	☒	☐
20	AZ 18-800	449.0	800.0	435.8	8.5	8.5	51.8	1840.0	2135.0	129.0	41320.0	☒	☐

You can have new sections generated by the program using the "**Generate tubular section**" button. The new sections created here can be saved in a ".kom_ggu" file.

10.15.4 GGU-RETAIN: "Bored pile wall"/"Diaphragm wall"/"Bored soldier pile wall" menu items

If you specified a "**Bored pile wall**" or a "**Bored soldier pile wall**" and "**Using section list**", a dialog box opens for entering the "**Bored pile diameter**".

If you select a "**Diaphragm wall**" and "**Using section list**" you must enter the "**Diaphragm wall thickness**" in a dialog box.

10.15.5 GGU-RETAIN: "Girder" menu item

If you selected the "**CMG wall**" and "**Using section list**" check boxes in the "**File/New**" or "**Editor 1/Analysis options**" menu items, you will see the dialog box with the currently loaded list of soldier piles (see "**Editor 2/Soldier piles**").

10.15.6 GGU-RETAIN: "Section data" menu item

For sheet pile walls, soldier pile walls and combined sheet pile walls, you can define several different sections or soldier piles for the wall. This allows partial corrosion of a sheet pile wall to be modelled, for example (see **"Worked example 2: Sheet pile wall with corrosion"**). In the **"File/New"** or **"Editor 1/Analysis options"** menu items, activate the **"Using several steel sections or inserted beams"** check box. In the subsequent dialog box, you can select different individual sections for the various depths.

No.	Depth [m]	Designation	Steel quality
1	3.000	AU 23(4/0)	S 240 GP
2	16.000	AU 23	S 240 GP

10.15.7 GGU-RETAIN: "Inserted beams" menu item

You can define a beam inserted at the top of the wall for a bored soldier pile wall, a diaphragm wall or a bored pile wall. In the **"File/New"** or **"Editor 1/Analysis options"** menu items, activate the **"Using several steel sections or inserted beams"** check box. In the subsequent dialog box, select the beam to insert from the soldier pile wall section list and define the depth and inserted beam spacing.

Depth [m]	Designation	Steel quality
1.000	HEB 300	S 235

Inserted beam centres [m]: 2.00

10.16 GGU-RETAIN: "Young's modulus/Specific weight"/"Specific weight" menu items

This menu item opens two different dialog boxes depending on whether you are working with a section list or with user-defined section data:

- **"E modulus/Specific weight"**

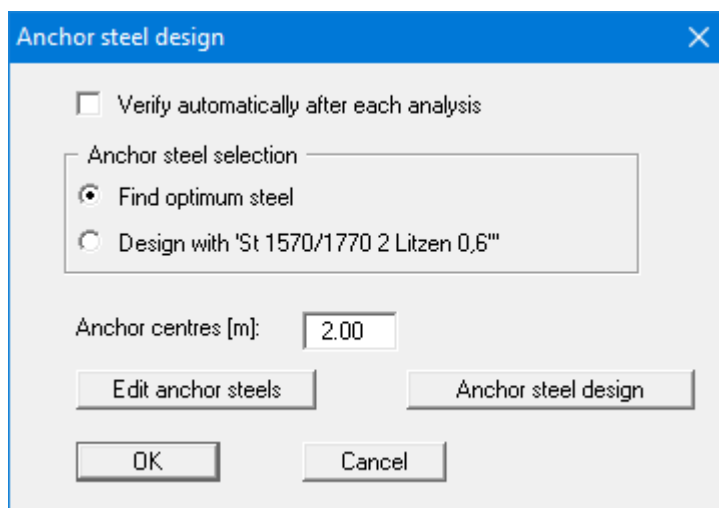
When working with the section list, you must enter Young's modulus and specific weight of the wall. From the former the bending line is calculated, from the latter the self-weight of the wall.

- "Specific weight"

Using user-defined section data Young's modulus is entered in the corresponding dialog box, so that here it is only necessary to specify the specific weight of the wall.

10.17 GGU-RETAIN: "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**".

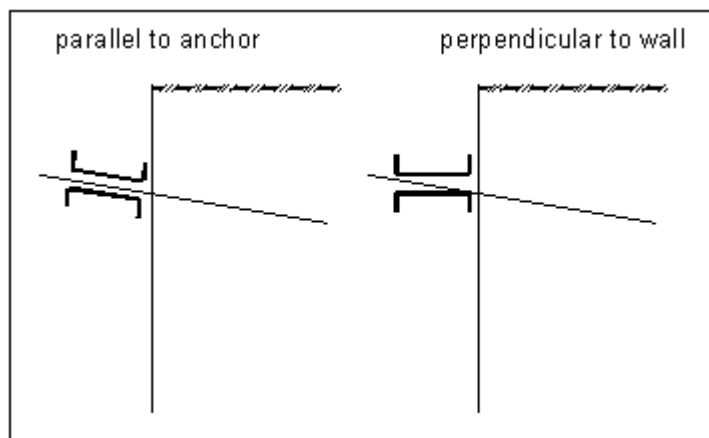
The "**Anchor steel design**" button only appears if the system is already analysed!

10.18 GGU-RETAIN: "Waling design" menu item

Using this menu item, you can specify preferences and then carry out the design for an analysed system by means of the "**Waling design**" button. The bending and shear verifications of the waling are then displayed on the screen.

If you activate the "**Verify automatically after each analysis**" check box, you will be automatically presented with a query for waling design when analysis is complete.

You can choose to design with a specified waling or to search for the optimum waling from a list of existing waling. When using partial factors, you must enter the "**Yield point**" and "**gamma(steel)**", and the allowable stresses for the global factors. Moreover, you must also enter the "**Anchor centres**" and select the type of "**Moment loading**" on which the design is based. Finally, you can specify whether the waling runs parallel to the anchor or perpendicular to the wall.



The "**Edit waling**" button opens a dialog box with a list of walings showing their names, section moduli and the values of "**S/s**" (first moment of area and web thickness) required for shear design. Here, you can select a waling and use it as the design waling using the button "**Selected waling as design waling**". New waling can be entered by clicking "**x waling(s) to edit**" and specifying a new number. With the buttons "**Load**" and "**Save**" you can access files with waling lists.

The button "**Waling design**" only appears if your system has already been calculated!

11 System menu

11.1 GGU-RETAIN: "Info" menu item

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

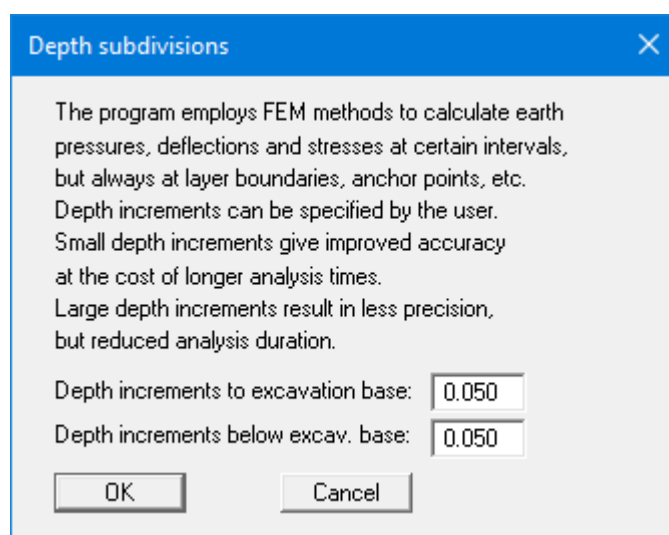
11.2 GGU-RETAIN: "Special preferences" menu item

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.

11.3 GGU-RETAIN: "Depth subdivisions" menu item

The program uses the finite element method, which requires the system to be divided into a number of finite elements (rods, see **"Theoretical principles: Structural system"**). You can specify the size of these depth increments for the region above and below the excavation base.



11.4 GGU-RETAIN: "Length surcharge" menu item

Here you can specify the way in which length surcharge is calculated for full or partial wall fixity (see **"Theoretical principles: Length surcharge"**).

Length surcharge for fixity [X]

Determine length surcharge dx for partial or full fixity

Length surcharge with % value

$dt = \text{length surcharge [\%]} \cdot (\text{depth} + \text{embedment})$
 Length surcharge [%] generally 20%

☐ Use length surcharge with % value

Length surcharge [%]:

Length surcharge via equation

$dt = Ch / eph(\phi, \text{deltap})$
 deltap/ϕ generally 0.333 $C = \text{automatic equiv. force}$

☒ Calculate length surcharge via equation

$\text{deltap}/\phi [-]$:

☒ Earth pressure coefficients after Streck

☒ Determine ϕ at level of theoretical toe

☒ Minimum value nach EAU 2012 8.2.9 Adopt

☒ Consider active berms

11.5 "Analyse" menu item

11.5.1 GGU-RETAIN: Start dialog box (Analyse)

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 appears with three sections, which are explained separately in the following sections. 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. At the bottom right of the dialog box a button can be seen labelled with the currently selected design section. If you select this button you can view and edit the current values for the design section.

Analyse sheet pile wall

Embedment depth via:

Toe is bedded
Section length will be determined

☒ Vertical support at wall toe ?

☐ Reduce subgrade with $\gamma(E_p)$?

☐ Analysis of $Bh,d \leq E_{ph,d}$ to the wall toe ?

☐ Subgrade to R 102 ?

At-rest e_p level: 0.00

Select method

Type of redistribution

☒ Do not redistribute

☐ EAB 1988

☐ EAB 2012

☐ Rectangular

☐ Birectangular Preferences

☐ Triangular Preferences

☐ Trapezoidal Preferences

☐ Quadrilateral Preferences

☐ User-defined Preferences

☐ EAU 2012 Preferences

Section: AU 14

Special preferences

☐ Redistribute e_p to toe

Buckling analysis DIN EN 1993-1-1

☐ 2nd order theory

Pre-curvature: 1 / 150 ?

☐ Double value in parabolic regions ?

☐ Pre-curvature to ground side

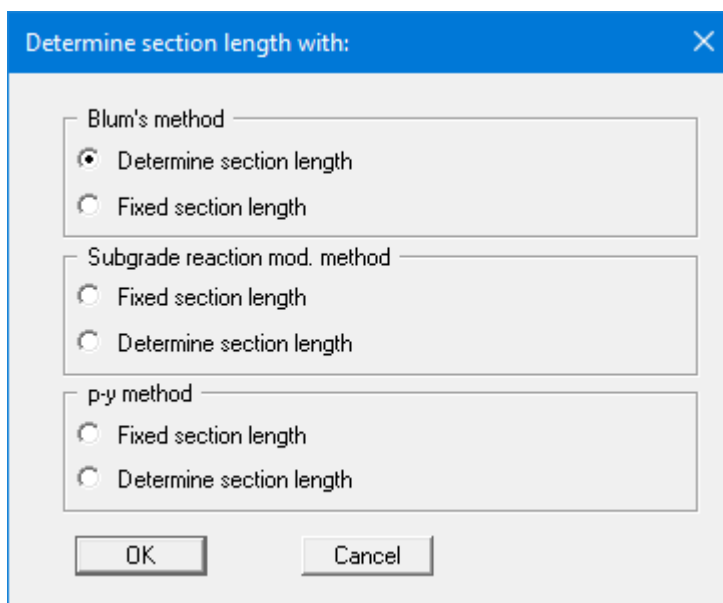
OK Cancel

When all the options are selected as you require, you can start the analysis by clicking "OK". Initially, GGU-RETAIN carries out a comprehensive plausibility check of the data you have entered and informs you of any inconsistencies. The system is then analysed. Information relating to the current stage of the analysis is displayed in the title bar. The analysis process can be terminated at any time (perhaps because you want to alter one of the parameters) by clicking the right mouse button.

Once the analysis is complete the **design** can follow by clicking "Yes" in the prompt. The design phase is more closely described in menu item "System/Design defaults". If "No" is now clicked or following completion of design, the results are presented in message 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"). If an intermediate state is involved, it may be useful to enter further explanations for the data record in addition to the file name in "Project identification" in the dialog box of the menu item "Editor 1/Analysis options" or "File/New".

11.5.2 GGU-RETAIN: "Embedment depth via:" group box (Analyse)

In the "**Embedment depth via**" group box of the start dialog box of the "**System/Analyse**" menu item you can define the method of calculation for the embedment depth. If you prefer to use another than the default method to determine the embedment depth, use the "**Select method**" button. The following dialog box opens:



If you select one of the options using fixed section lengths "**Fixed section length**" appears in the start dialog box instead of the degree of fixity; you must enter the value, you require.

You can choose between Blum's method or the subgrade reaction modulus method for the calculation. For soldier pile walls and bored soldier pile walls you also have the option of calculating according to the p-y method.

Where a **bedded toe** is used, the subgrade reaction can be reduced using the partial factor $\gamma(E_p)$ by activating the check box. The reduction can be made using characteristic values to DIN 1054, GL. (47) and (48). If the "**Reduce subgrade with $\gamma(E_p)$** " check box is not activated, the subgrade reaction can be taken into consideration in accordance with the EAB, R 102.

Embedment depth via:

Toe is bedded

Section length will be determined

☒ Vertical support at wall toe ?

☐ Reduce subgrade with gamma(Ep) ?

☐ Analysis of $Bh,d \leq Eph,d$ to the wall toe ?

☐ Subgrade to R 102 ?

At-rest ep level: 0.00

☒ Subgrade on continuous wall ?

Select method

Using soldier pile walls or bored soldier pile walls the "**Subgrade on continuous wall**" check box appears additionally. If the check box is deactivated, the subgrade will be adopted for the girder width.

11.5.3 GGU-RETAIN: "Special preferences" group box (Analyse)

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

Retaining walls subject to buckling hazards can be analysed using "**2nd order theory**". A system pre-curvature, which can be adopted at 1/150, must be defined for buckling analysis to DIN EN 1993-5 or DIN EN 1993-1-1. The direction of pre-curvature must also be defined (to the ground side or the atmosphere side). Whether a pre-curvature towards the ground side or the atmosphere side provides the more unfavourable design values is system-dependent: it is therefore absolutely vital that both directions are investigated in a buckling analysis (see "**Theoretical principles: 2nd order theory**").

11.5.4 GGU-RETAIN: Type of redistribution (Analyse)

The following options are offered in the "**Type of redistribution**" group box in the start dialog box of the "**System/Analyse**" menu item:

- "**Do not redistribute**"

The analysis is performed using classical earth pressure redistribution.

- "**EAB 1988**" and "**EAB 2012 / 2021**"

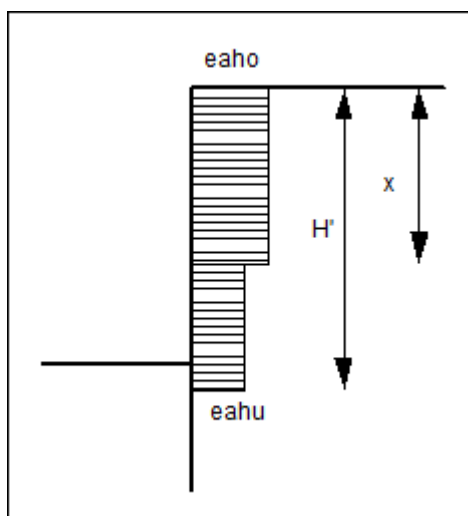
Redistribution diagrams are given as a function of anchor positioning in EAB 1988 and EAB 2012 or 2021. The program selects the appropriate redistribution from these diagrams. If no agreement is found an appropriate error message is shown.

- "**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.

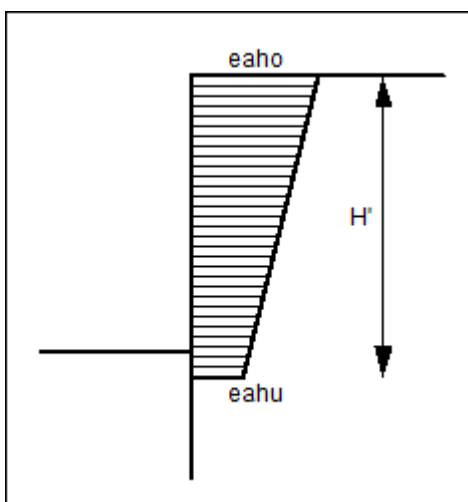


- **"Triangular"**

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

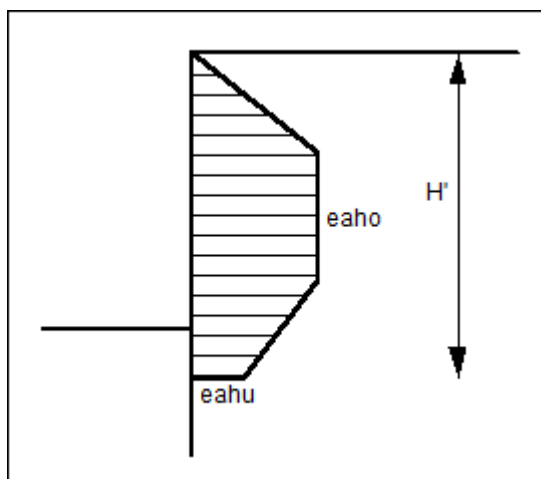
- **"Trapezoidal"**

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



- **"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 $eaho/eahu$.



- "User-defined"

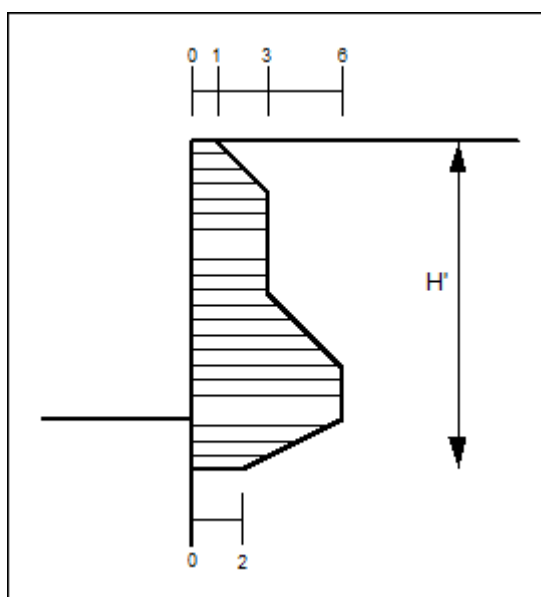
If all the earth pressure redistributions offered so far do not meet your requirements, you can also carry out any earth pressure redistribution by defining a polygon course using the "**Preferences**" button. You can define several depths between the top of the wall and the base of the excavation and assign specific, freely definable earth pressure coordinates to these depths. This creates a polygonal composite surface. The calculated earth pressure is redistributed to precisely this area during subsequent analysis. For the example in the following dialog box, the following image is shown:

User-defined ep redistribution

Done Forw. Back Cancel

6 point(s) to edit Sort

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



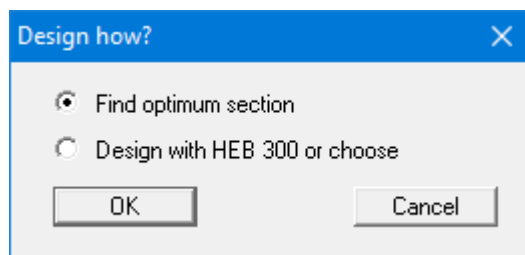
- "EAU 2012"

The earth pressure is redistributed in accordance with EAU 2012. After clicking on the "**Preferences**" button you can differentiate between an excavated or a backfilled wall.

11.6 GGU-RETAIN: "Design defaults" menu item

Once the current system has been analysed you can progress directly to the retaining wall design (see menu item "**System/Analyse**"). In addition, using this menu item, you can also carry out subsequent design for a different section to that used for the analysis.

The program reacts differently depending on the type of retaining wall (soldier pile wall, sheet pile wall or in-situ concrete wall). For **soldier pile walls**, for example, you will see the following dialog box:



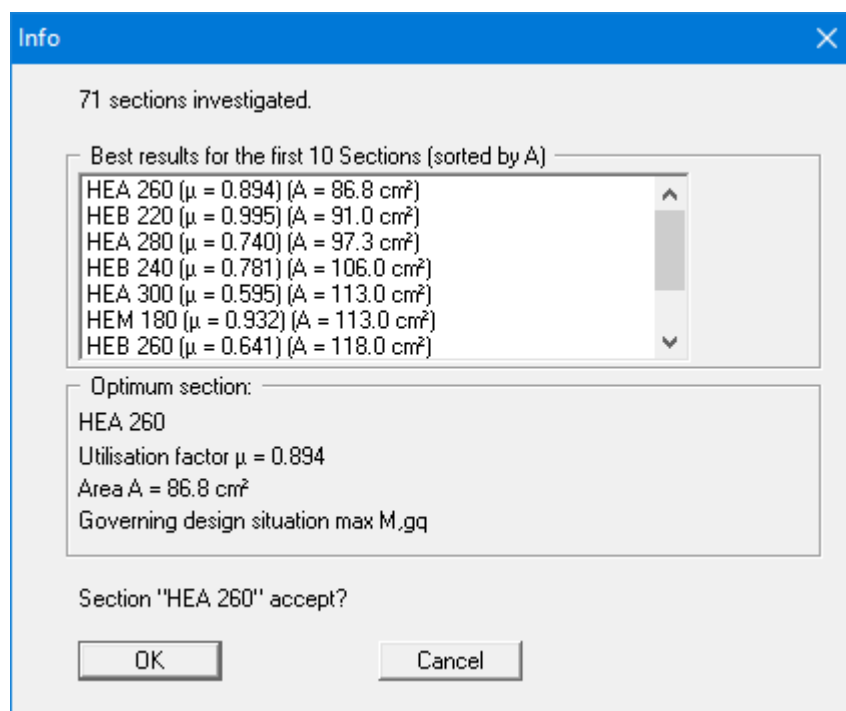
If you would prefer to have the program search for the optimum section, select the upper radio button and the following dialog box opens:

 A dialog box titled "Find optimum section" with a close button (X) in the top right corner. It contains several sections:

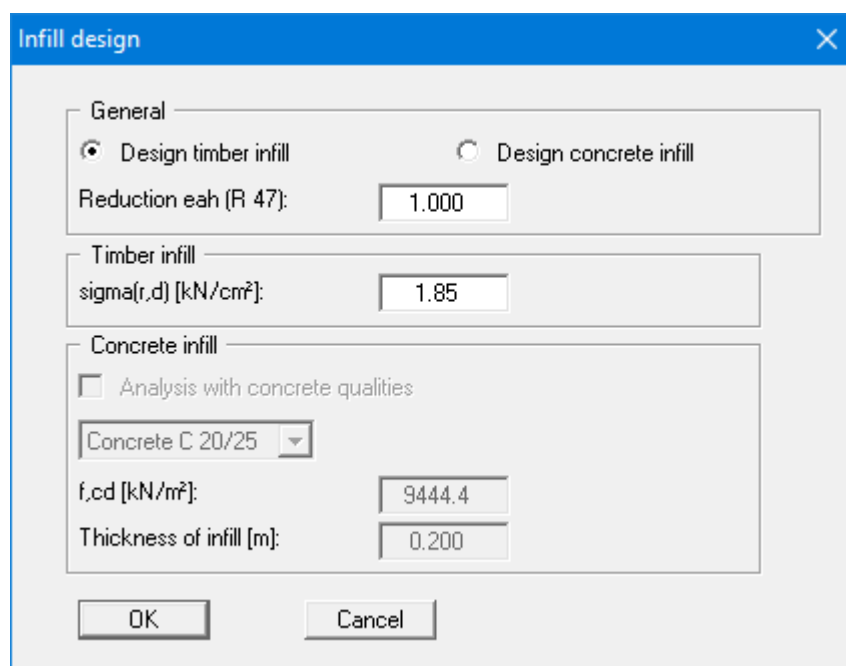
- Section selection**: "Optimum section is a" with two radio buttons: "I beam" (selected) and "Double U beam". Below are two checked checkboxes: "Reduce yield stress for Class 4" and "Search criterion: 'Find smallest section area'".
- Steel selection**: "Steel quality:" with a dropdown menu showing "S 235".
- Actions**: A table with design data.

Design situation	Max. M	Max. Q	Max. N
Moment M _{Ed} [kN·m]:	193.3	0.0	95.6
Shear force V _{Ed} [kN]:	-1.2	127.4	-82.3
Normal force N _{Ed} [kN]:	-82.8	-22.5	-121.2
Buckling length L [m]:	6.9	6.9	6.9
- Partial factors**: Two input fields: "gam,M0 [-]:" with value "1.00" and "gam,M1 [-]:" with value "1.10".
- At the bottom are two checked checkboxes: "Design cross-sections Class 1 or 2 plastically" and "Perform buckling analysis".
- At the very bottom are "OK" and "Cancel" buttons.

The search result is displayed in another dialog box. If you want to adopt the optimum section proposed by the program, click "OK".



Following this the analysis results are displayed in a message box. If you selected the proposed optimum soldier pile, you should adapt the soldier pile width to suit the section in your system by pressing the "Use" button. Subsequent reanalysis is recommended. The infill walling for soldier pile walls can then be designed.



The program then searches for the maximum active earth pressure ordinate, *max eah*, determines the maximum moment for the infill from

$$(\text{max eah}) \cdot (\text{soldier pile centres})^2 / 8$$

and calculates the required thickness of the infill elements using the allowable stress. Alternatively, to timber infill, concrete infill can also be designed.

When design is complete you are informed of the verification results in an information box and are presented with the principal data on which the design was based. After confirming with "**OK**", the state variables and the system are presented on the screen.

11.7 GGU-RETAIN: "Graph positioning preferences" menu item

If you are not happy with the automatic graph arrangement, you can use this menu item to create an arbitrary arrangement. The dialog box of the **GGU-RETAIN** program is shown here as an example. Other graphs may be listed in other GGU programs.

Graph name	x [m]	Width [m]
Water pressure, ..	0.00	2.00
Earth pressure	5.00	2.00
Moment	10.00	2.00
Shear force	15.00	2.00
Normal force	20.00	2.00
Displacement	25.00	2.00
Subgrade react. mod.	30.00	2.00
Lateral pressures	-5.00	2.00

For a customised graph arrangement, activate the "**Manual graph positioning**" check box. The graphs are then displayed centred at the sheet position "**x**" 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.

11.8 GGU-RETAIN: "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 flow tube

☐ Flow tube diagram

Earth pressure and state variable hatching

☒ Fixed spacing Spacing [mm] =

Presentation of action effects, etc.

☒ Moment ☒ Shear force ☒ Normal force

☒ Displacement ☒ SR modulus

☒ Deep-seated stability ☒ Lateral pressures

☒ Shear flow $q(z)$ ☒ Acc. shear force $F(z)$

Display (earth pressure):

☐ d ☒ (g+q).k ☐ g.k ☐ q.k

Display (state variables):

☐ d ☐ g.d ☒ (g+q).k ☐ g.k ☐ q.k ☐ w.k

Display (displacement):

☐ d ☐ g.d ☒ (g+q).k ☐ g.k ☐ q.k ☐ w.k

☐ Same height for all Presentation height (loads) [m]:

Load 'hatching':

Hatch spacing [mm]:

Excess passive/active earth pressure [-]:

State variable for strut:

On the whole, the dialog box is self-explanatory. When "**ep + pw**" is selected, the sum of earth pressure and water pressure (pw) is displayed in the diagram for earth pressure. If a system has been analysed using 2nd order theory with buckling analysis to DIN EN 1993-1-1 activated, either the "**Displacement**" or the "**Pre-curvature**" can be selected for display in the result graphics in the dialog box shown above. Any adopted pre-curvature will always be given in the output table.

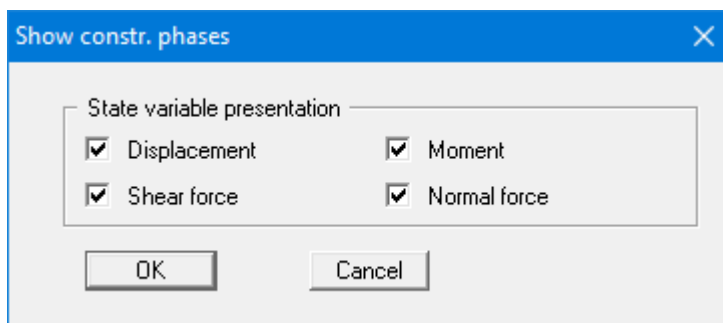
With version 12.34, the additional internal force derivatives shear flow $q(z)$ and accumulated shear force $F(z)$ have been added. These are automatically filled in by the calculation and displayed **together with the shear force line!** If no shear force is selected, the results are superimposed on the moment line.

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.

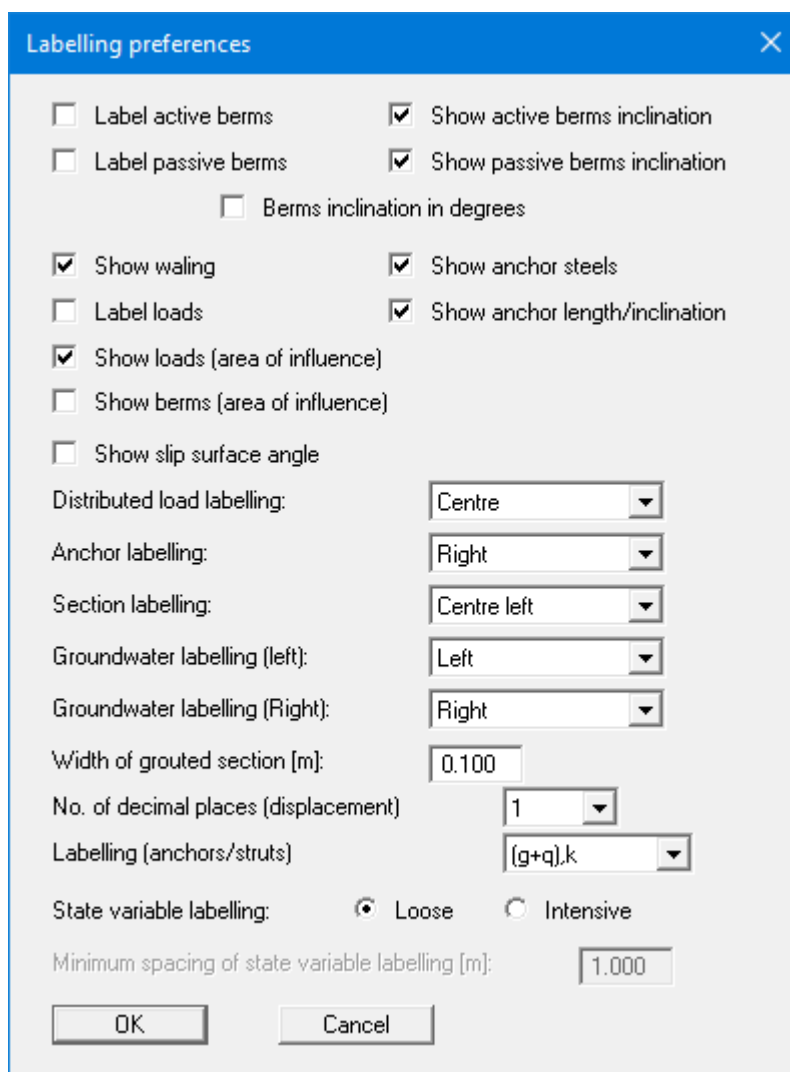
Leave the dialog box by pressing "**Show system**". If the system has already been analysed, you can leave the box by pressing "**Show results**" and then view the result graphics on the screen.

When you are working with a particular construction phase (menu item "**Construction phases/Display summary**") the following dialog box appears:



11.9 GGU-RETAIN: "Labelling preferences" menu item

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



The 'Labelling preferences' dialog box contains the following settings:

- ☐ Label active berms
- ☒ Show active berms inclination
- ☐ Label passive berms
- ☒ Show passive berms inclination
- ☐ Berms inclination in degrees
- ☒ Show waling
- ☒ Show anchor steels
- ☐ Label loads
- ☒ Show anchor length/inclination
- ☒ Show loads (area of influence)
- ☐ Show berms (area of influence)
- ☐ Show slip surface angle
- Distributed load labelling: Centre
- Anchor labelling: Right
- Section labelling: Centre left
- Groundwater labelling (left): Left
- Groundwater labelling (Right): Right
- Width of grouted section [m]: 0.100
- No. of decimal places (displacement): 1
- Labelling (anchors/struts): (g+q).k
- State variable labelling: ☒ Loose ☐ Intensive
- Minimum spacing of state variable labelling [m]: 1.000

Buttons: OK, Cancel

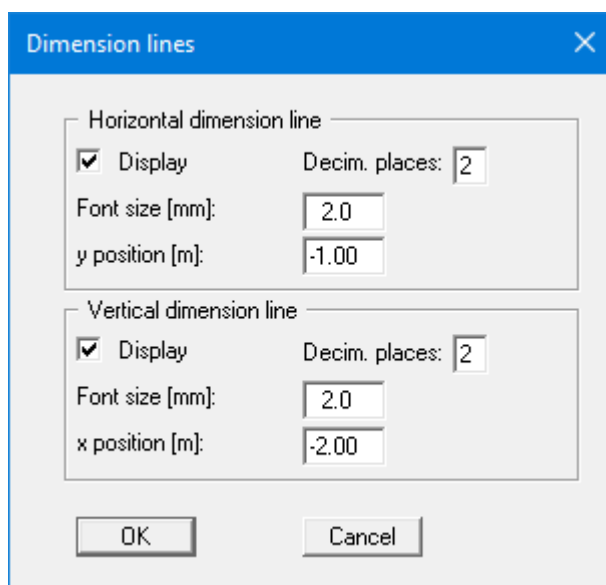
In the dialog box, you activate the required check boxes and select the preferences for alignment or font sizes. In addition, groundwater labelling and the width of the graphical visualisation of any grouted section can be edited.

11.10 GGU-RETAIN: "Graph grid preferences" menu item

You can define graph grids for moments, shear forces and normal forces.

11.11 GGU-RETAIN: "Dimension lines" menu item

You can include a vertical and/or horizontal dimension line in the graphic to illustrate the system dimensions. If the dimension lines are displayed on the screen, you can also open the following dialog box directly by double-clicking the left mouse button over a dimension line.



The distance to the wall or pile 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 wall or pile. All values are in metres in the scale selected (see the menu item "**Page size + margins/Manual resize (editor)**").

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.

11.12 GGU-RETAIN: "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.

11.13 GGU-RETAIN: "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.

12 Evaluation menu

12.1 GGU-RETAIN: General note on evaluation

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.

12.2 GGU-RETAIN: "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.

12.3 GGU-RETAIN: "Main output summary" menu item

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

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

12.4 GGU-RETAIN: "Maximum reaction summary" menu item

You will see the maximum values for all state variables displayed in a message box.

12.5 GGU-RETAIN: "Anchor and strut summary" menu item

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

12.6 GGU-RETAIN: "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 = 4.20 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	0.50	8.00	22.3	122.1	0.183	22.3	122.1	0.183

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 0.183 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 = 4.05

☒ 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**.

12.7 GGU-RETAIN: "Sum V FOS summary" menu item

Here you obtain all data relating to analysis of sum V displayed in a message box (see also "**Theoretical principles: Analysis of mobilised passive earth pressure**").

12.8 GGU-RETAIN: "Sum H FOS summary" menu item

When working on a soldier pile wall, this is where you obtain all data relating to analysis of sum H displayed in a message box (see also "**Theoretical principles: Analysis of sum H**").

12.9 GGU-RETAIN: "Hydraulic heave FOS summary" menu item

When working with sheet pile walls (in-situ concrete walls) you obtain all data relating to analysis of hydraulic heave safety displayed in a message box (see also "**Theoretical principles: Hydraulic heave**").

12.10 GGU-RETAIN: "Buoyancy FOS summary" menu item

When working on sheet pile walls (in-situ concrete walls), you obtain all data relating to analysis of the buoyancy safety displayed in a message box also "**Theoretical principles: Hydraulic heave**").

12.11 GGU-RETAIN: "Heave of anchor soil" menu item

If a value unequal to '0' is entered for the height of the dead man when entering the anchors in the dialog box associated with the "**Editor 2/Anchors**" menu item, the program performs verification of anchor soil heave similar to the method described in Section 7.3.4 of the Piling Manual.

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**").

12.12 GGU-RETAIN: "Heave FOS summary" menu item

When analysing the system using global safety factors the base heave safety factor is determined automatically (see also "**Theoretical principles: Base heave safety**"). If this cannot be verified, you will see a corresponding warning even before analysis is complete. Otherwise, you can query the safety factor using this menu item.

12.13 GGU-RETAIN: "Verification of pull-out resistance" menu item

If analysis was performed using anchors with activated "**Verification with $q_{s,k}$** " check box and verification is activated in the "**Editor 1/Pull-out resistance**" menu item, this menu item displays the analysis results in a message box.

12.14 GGU-RETAIN: "Earth support analysis" menu item

Using this menu item, all results of earth support analysis will be displayed in a message box.

12.15 GGU-RETAIN: "Grouting points analysis" menu item

When you click on the menu item "Verification of grouting points", the following dialog box opens:

Section: PU 18

Existing number of grouting points / m [-] ?

Rd [kN/pressing point] ?

2 shear ranges

Shear_range	from	to	M1	M2
[-]	[m]	[m]	[kN·m/m]	[kN·m/m]
1	442.20	436.42	0.00	-374.46
2	436.42	432.50	-374.46	0.00

OK Cancel

The relevant standard setting for R_d is 75 kN per grouting point. This can be found in the National Annex to DIN EN 1993-5/6.4. Higher grouting point resistances must always be verified.

If no calculation has been performed yet, no shear ranges are specified.

Once a calculation has been performed, you can see the shear ranges displayed below. Shear ranges are located between the zero crossings of the shear force line and are considered as such by the program.

Grouting points analysis [X]

Section: PU 18
 Existing number of grouting points/m [-] = 4.50
 R_d [kN/pressing point] = 75.00
 I_y [cm⁴/m] = 38650.00
 S_y [cm³/m] = 1067.00
 b_y [cm] = 120.00
 LQ = distance of the shear force transition points
 Required number req. $n = \eta \cdot M / (R_d \cdot LQ)$
 $\eta = S_y / I_y \cdot b_y$
 $M = M_2 - M_1$
 2 shear ranges

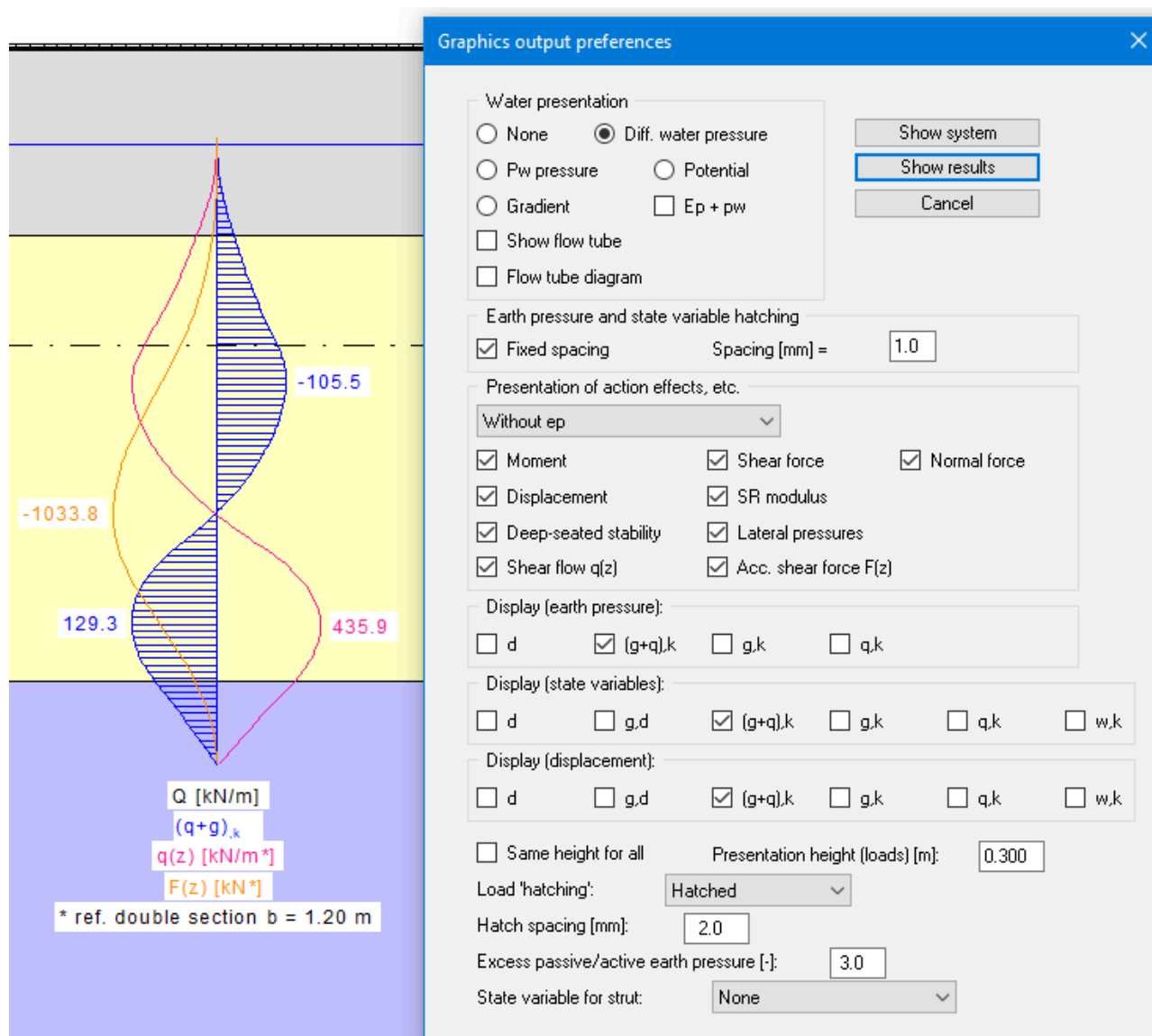
Shear_range	LQ	M	η	req_n	Utilisation_factor
[-]	[m]	[kN·m/m]	[-]	[-]	[-]
1	5.78	374.46	3.31	2.86	0.64
2	3.92	374.46	3.31	4.22	0.94

Maximum utilisation factor = 0.94

OK Cancel Recalculate

The verification in the program is always on the safe side, as the selected number from Figure 1 is always compared with the selected number of the respective shear range. For each shear range, the integral of the shear force (-> the moment) is used and averaged over the entire shear range with height LQ and on the related width of a double sheet pile wall. It is recommended to stagger the points if shear areas have significantly different required numbers of grouting points.

With version 12.34, the accumulated shear force $F(z)$ and the shear flow $q(z)$ were implemented in the program for each reference width:



This makes it easier to check the plausibility of the results. The smallest possible value for the number of grouting points should not fall below the value of $F(z, \max) / R_d$ if the verification of the grouting points is not fulfilled.

12.16 GGU-RETAIN: "Welding lengths analysis" menu item

When working on sheet pile walls, you can verify the welding lengths using this menu item. After selecting the steel and entering the weld thickness, the results are displayed in an inbox.

13 Construction phases menu

13.1 GGU-RETAIN: General notes on construction phases

When the system you are working on consists of several construction phases, each phase must initially be calculated separately, and the results saved in corresponding files. Subsequently, from this menu, you can combine the different file contents. The following graphics can be created:

- envelope of moments;
- envelope of shear forces;
- envelope of normal forces;
- displacements;
- envelope of reinforcement.

The resulting graphic is displayed in the current system. The "**Construction phases**" menu will only work with the results from the files compiled using this menu.

For diaphragm walls, bored pile walls and bored soldier pile walls, please note:

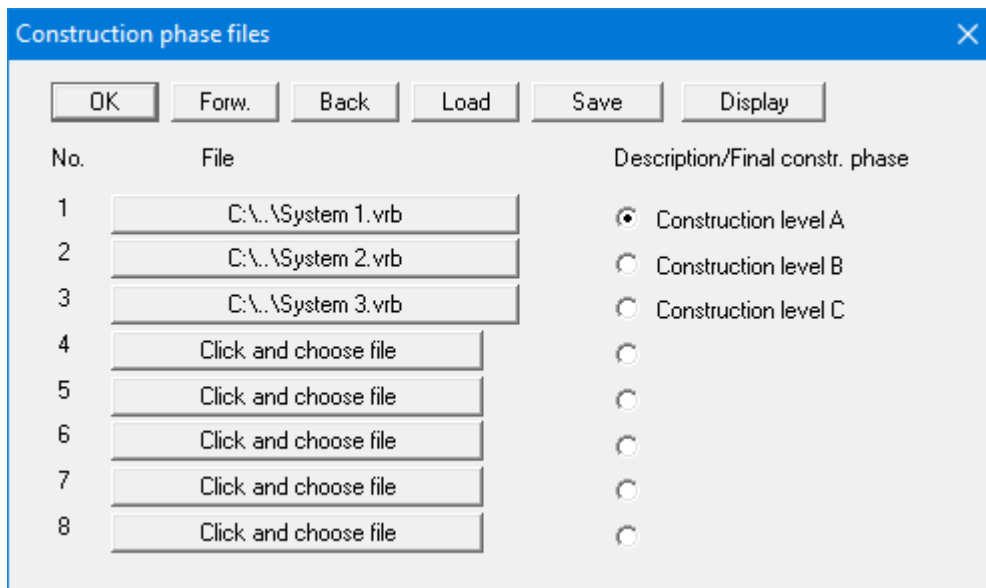
Construction phase files created using program versions older than **GGU-RETAIN 10.20** must first be opened and then saved again in the new program versions from 10.20 on, to achieve reinforcement envelope visualisation.

13.2 GGU-RETAIN: "Info" menu item (Construction phases)

You will see information on the options provided by the "**Construction phases**" menu. In addition, you can also activate displacement visualisation as cumulative values.

13.3 GGU-RETAIN: "Select files" menu item

Using this dialog box, you can compile a list of files to be summarised. A total of 15 files can be loaded into the list. The files must be saved together with the analysis result data, otherwise an error message appears.



The current system results are not displayed automatically. They must first be saved as a result file and then be loaded into the file list. The currently loaded system serves only to represent the excavation, which is displayed in the graphics in addition to the summarised results. You can move through the list using **"Forw."** and **"Back"**. Using the **"Save"** button the selected files can be saved in order to be loaded into the application at a later session. If you save the current system, the file names will also be saved in the dataset. Using the **"Display"** button, the summarised files will be displayed on the screen.

After clicking one of the **"Click and choose file"** buttons, the desired file can be selected. The button is then labelled with the file name. The project identification is shown after this (in the example above **"Construction level A"**, **"Construction level B"**, **"Construction level C"**). If you need to remove a file from the list, click on the button with the file name and select the **"Cancel"** button from the file selector box. The file sequence should correspond to the construction phase sequence. The radio button behind the file name serves to indicate up to which file the summary should be displayed.

There are two arrow icons



in the program menu toolbar. These allow to easily switch between the individual construction phases, facilitating an animation-like visualisation of the bending line in the individual construction phases. Of course, the arrow icons only work if files have been selected and you have left the dialog box using the **"Display"** button.

13.4 GGU-RETAIN: "Display summary" menu item

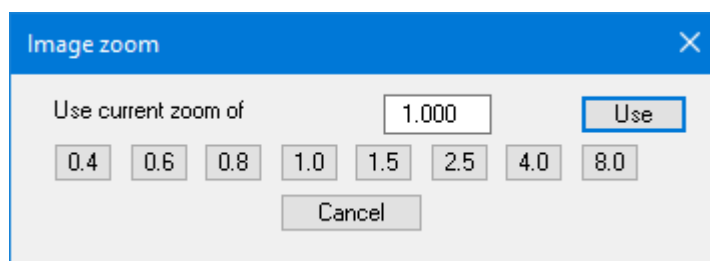
If the system graphics or the results of the current analysis are actually displayed on the screen, this option will switch to a visualisation of the construction phases.

14 Graphics preferences menu

14.1 GGU-RETAIN: "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.

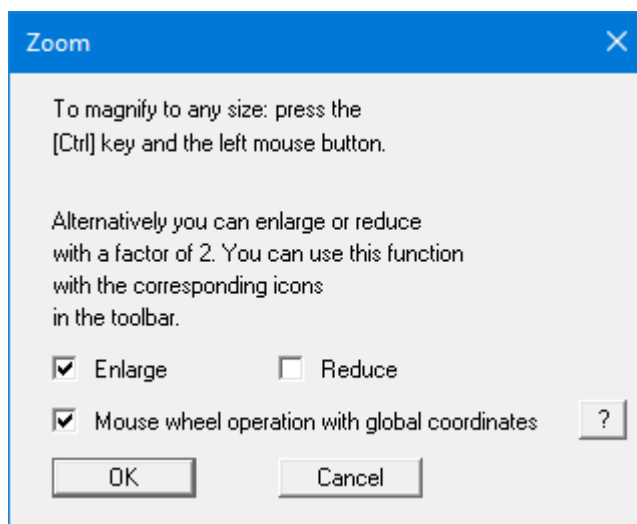


A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "**Use**" to exit the box the current factor is accepted. By clicking on the "0.4", "0.6", etc. buttons, the selected factor is used directly, and the dialog box closed.

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.

14.2 GGU-RETAIN: "Zoom info" menu item

An info box informs you about the activation and possibilities of the zoom functions.



A quick way to enlarge a specific section of the screen is to open a magnifying glass window by holding down the **[Ctrl]** key and the mouse button.

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"**.

14.3 GGU-RETAIN: "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.

14.4 GGU-RETAIN: "Pen colour and width" menu item

In order to enhance the clarity of the graphics you can preset the pen settings for various graphic elements in the GGU programs.

You can change the pen widths for the elements listed in the dialog box and adjust the pen and/or fill colours by clicking on the button with the element name. In some GGU programs you can also define a line colour for selected elements.

For the graphic output of colours on **monochrome printers** (e.g. laser printers), colours are replaced by an equivalent shade of grey. In the case of very light colours, the corresponding graphic elements are then barely visible on the printer. In such cases, it makes sense to change the colour setting to darker colours.

14.5 GGU-RETAIN: "Mini-CAD toolbar" and "Header toolbar" menu items

Using this menu item you can add free text, lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.) to the main program graphics. PDF files can also be imported as images. A pop-up menu opens, the icons and functions used are described in more detail in the [Mini-CAD manual](#).

The differences between the Mini-CAD and Header CAD are as follows:

- Drawing objects created with **Mini-CAD** refer to the coordinate system (generally in [m]) in which the drawing is created and are displayed accordingly. You should therefore select this menu item if you wish to enter additional information on the system. These objects can be saved in a **“.mcd”** file using the



icon in the pop-up menu and loaded as such in other GGU programs using the



icon in the pop-up menu.

- Drawing objects created with the **Header CAD** refer to the sheet format (in [mm]). They therefore always remain at the same sheet position, regardless of the coordinate system of the measuring points. You should select this menu item if you want to enter general information on the drawing (e.g. company logo, report number, system number, stamp). If you save this so-called header data in a **“.kpf”** file using the icon



in the pop-up menu, you can reload this header data for a completely different system (with different system coordinates) and also in other GGU programs using the icon



in the pop-up menu. The saved header data can then be found in the same position. This considerably simplifies the creation of general sheet information.

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

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

Not all of the icons described below are available in all GGU programs.



"Next page"/"Previous page"

You can use these icons to scroll forwards and backwards between individual pages. In many GGU programs, you can use this in the **output table representation**.



"Select page" or "Diagram/Output table"

If you are in the **output table representation**, you can use this symbol to jump to a specific page or switch back to the normal display, i.e. your graphic display.



"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**").

**"Colour on/off" or "Colour/hatching"**

If you want to remove the colour from the system display, e.g. to create a black and white printout, you can do this using this on/off switch.

In several GGU programs, 4 colour and hatching settings are possible via this icon, which you can click through in sequence. A colour filled representation is the default setting, the next click shows the hatching, then coloured and hatched. The fourth click shows a representation with neither colour nor hatching. The next click starts again.

**"Move object"**

Use this icon to position legends and diagrams at the desired position on the output sheet.

**"Undo move object"**

By clicking this icon the last performed movement of graphical elements made using the [F11] function key or the menu item **"Output preferences/Move objects"** can be undone.

**"Restore move object"**

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

**"Construction phase back"/"Construction phase forward"**

If you are in the display of construction phases, you can use these icons to scroll back and forth between the individual construction phases.

14.7 GGU-RETAIN: "Soil properties legend" menu item

A legend containing the soil properties is displayed with the graphics. If the **"Show legend"** check box is activated, you can edit the representation in the dialog box for this menu item.

You can define and edit the position of the legend using the values "**x**" and "**y**". You can control the size of the legend via the "**Font size**". You can also define a background colour in some legends.

The fastest way to modify the position or size of the legend is to press the [**F11**] function key and then to either pull the legend to the new position or to move the sides of the legend to their new size with the left mouse button pressed.

In the case of a calculation with bedding according to the **p-y method**, an additional check box is available here to highlight the shear values:

☒ Shear values (p-y method) marked red

By activating the "**With permeabilities**" and "**With depths**" check boxes the corresponding data of the individual soil layers will be shown in the soil properties legend. The designations entered for the individual ground layers are displayed in the system graphics adjacent to the layer depths if the "**Soil designations in main graphics**" check box is activated.

If "**System coloured**" is activated as presentation format in the combi-box, the soils will be displayed coloured both in the soil properties legend and in the system graphics. You can also define either hatching or colour fill and hatching for the different soil types in the combi-box. If you select "**System without all**", the soils are merely numbered. The various preferences can also be accessed via the "**Colour/hatching**" icon in the menu items toolbar.

In the "**Soil colours + soil hatching**" group box, you can make the desired adjustments according to your individual requirements.

If you have activated the "**Differentiate active + passive soil properties**" check box in the "**Editor 1/Analysis options**" menu item, the "**Soil colours (passive side)**" check box is also displayed:

- **"Soil colours"/"Soil colours (passive side)"**

You will see a dialog box in which you can make your desired settings. Here you can assign a new colour to each soil layer after clicking on the button with the desired number or reassign the colours via **"Soil colours/Reorganise"** button. You can save your colour settings to a file under **"Soil colours/Save"** and reuse them for other systems using the **"Soil colours/Load"** button. In the lower area, you can transfer the colour settings to the Windows colour box as user-defined colours, for example, or vice versa. Click on the **"Info"** button for further explanations.

- **"Hatching"**

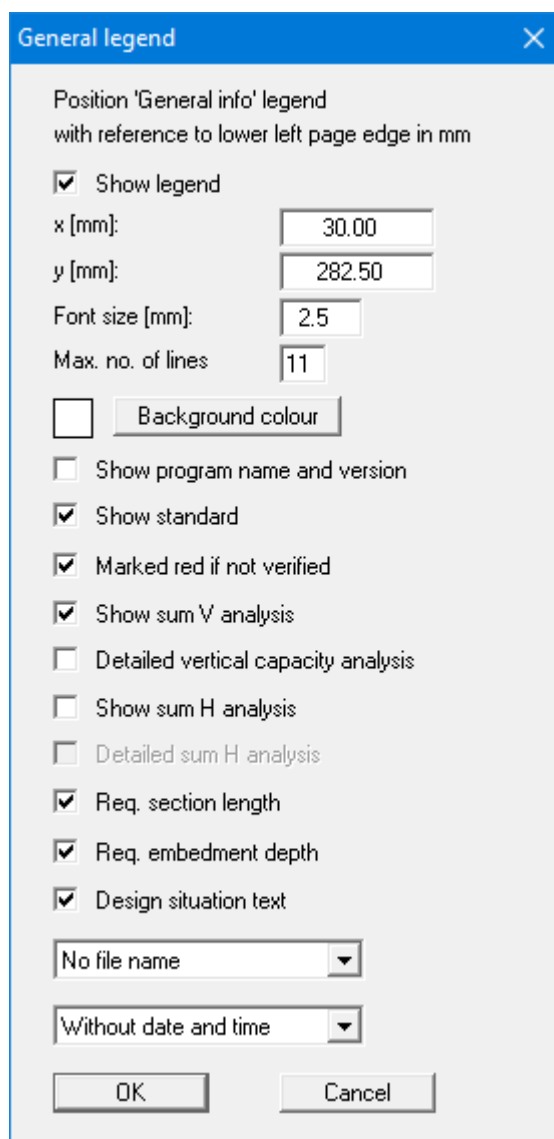
You will see a dialog box in which you can define different hatchings for each soil.

- **"Legend hatching factor"**

A narrower hatching can be achieved in the small boxes of the legend than in the system presentation. To do this, select an entry < 1.00 .

14.8 GGU-RETAIN: "General legend" menu item

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



General legend

Position 'General info' legend
with reference to lower left page edge in mm

☒ Show legend

x [mm]: 30.00

y [mm]: 282.50

Font size [mm]: 2.5

Max. no. of lines 11

☐ Background colour

☐ Show program name and version

☒ Show standard

☒ Marked red if not verified

☒ Show sum V analysis

☐ Detailed vertical capacity analysis

☐ Show sum H analysis

☐ Detailed sum H analysis

☒ Req. section length

☒ Req. embedment depth

☒ Design situation text

No file name

Without date and time

OK Cancel

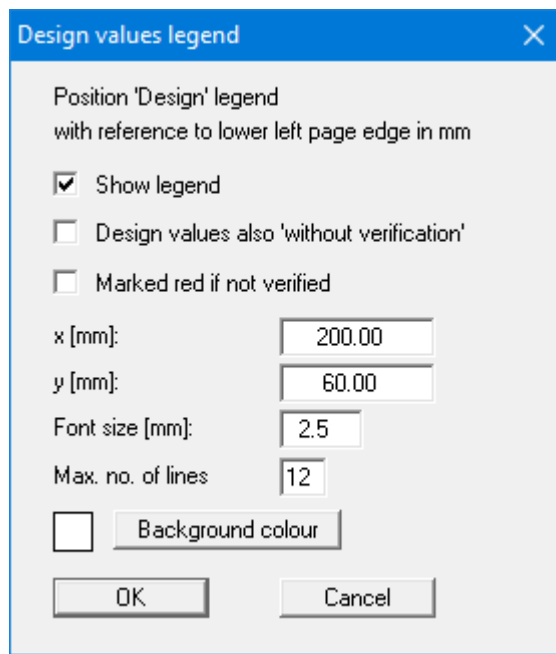
You can define and edit the position of the legend using the values "x" and "y". You can control the size of the legend via the "Font size" and "Max. no. of lines"; if necessary, the legend is displayed in several columns. You can also define a background colour in some legends.

The fastest way to modify the position or size of the legend is to press the [F11] function key and then to either pull the legend to the new position or to move the sides of the legend to their new size 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 design situation (load case) as well as on the current file (name, path, time info). Additionally the display of further information as e.g. "Show sum V analysis" can be activated depending on the selected retaining wall. Any project identification entered in the "File/New" or "Editor 1/Analysis options" dialog box will be shown automatically in the general legend.

14.9 GGU-RETAIN: "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**" and "**y**". You can control the size of the legend via the "**Font size**" and "**Max. no. of lines**"; if necessary, the legend is displayed in several columns. You can also define a background colour in some legends.

The fastest way to modify the position or size of the legend is to press the [**F11**] function key and then to either pull the legend to the new position or to move the sides of the legend to their new size with the left mouse button pressed.

14.10 GGU-RETAIN: "Subgrade modulus legend" menu item

A legend is displayed on the screen containing the subgrade reaction moduli entered for the sections along the retaining wall defined for the subgrade reaction profile. 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**" and "**y**". You can control the size of the legend via the "**Font size**". You can also define a background colour in some legends.

The fastest way to modify the position or size of the legend is to press the [**F11**] function key and then to either pull the legend to the new position or to move the sides of the legend to their new size with the left mouse button pressed.

14.11 GGU-RETAIN: "p-y curves legend" menu item

If you work with the p-y method, a legend of the p-y curves can be displayed. Using this menu item you can alter the type of presentation or turn off the legend completely.

Function p = f(y)

☒ Show function p = f(y)

Heading: p = f(y)

x value [mm]: 320.00 Width [mm]: 80.00

y value [mm]: 85.00 Height [mm]: 60.00

Font size (heading) [mm]: 2.0

Font size (axes) [mm]: 2.0

☐ Automatic axes max. y [m]: 0.500

Pen width [mm]: 0.50

Show which soil: 1

☐ Use soil colours

p-y curves on: Stratum centre

☐ Background colour

OK Cancel

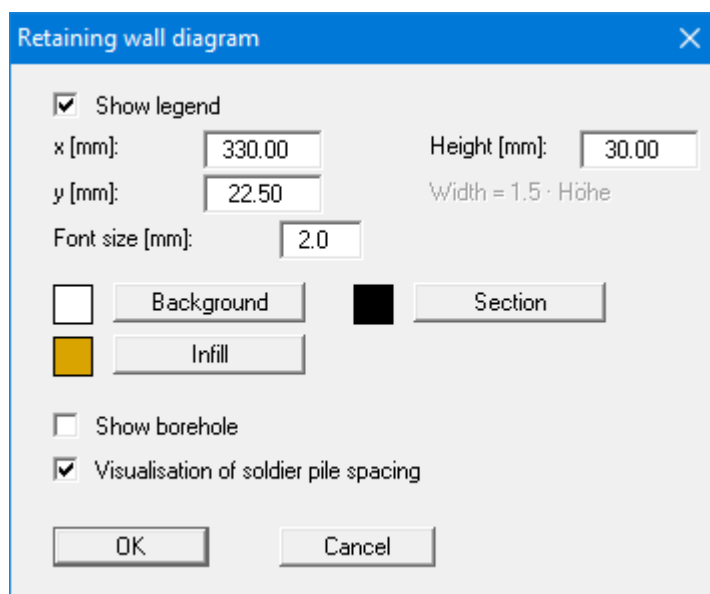
You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Width**" and "**Height**". In some legends, you can also adjust the font size of the label and/or specify a background colour.

The fastest way to modify the position or size of the legend is to press the [**F11**] function key and then to either pull the legend to the new position or to move the sides of the legend to their new size with the left mouse button pressed.

You can adapt the display of the function $p = f(y)$ to meet your requirements based on the selected soil, alignment of the p-y curves or line colours.

14.12 GGU-RETAIN: "Retaining wall diagram" menu item

A legend containing a small sketch of the retaining wall employed is displayed on the screen. Using this menu item you can alter the type of presentation or turn off the legend completely. The following dialog box opens for a **soldier pile wall**, for example.



If you edit the font sizes you may have to alter the size of the legend to suit, otherwise the dimensions in the legend may no longer be visible.

14.13 GGU-RETAIN: "Move objects" menu item

If you select this entry, you can then use the mouse to move the various objects and legends. Move the mouse over the object of your choice. If you are over a movable object, the mouse pointer will take on the shape of a cross. Now press the left mouse button and drag the object to the desired position while holding down the button.

After going to this menu item **only one object** at a time can be moved using the mouse or its size be altered.

If you want to edit several objects, you can also activate the function more quickly by pressing the **[F11]** key or the symbol



. An info box will then no longer appear.

In many GGU programs you can also alter the size of an object using this menu item or the **[F11]** function key. If you are positioned over the frame of a resizable object after activating the function, the mouse takes on the shape of a double arrow. Hold down the left mouse button and drag the frame until the object has reached the desired size. Drag a corner to maintain the aspect ratio of the sides. If you drag on one side, the object becomes taller or wider.

Using the **[Back]** key or by clicking on the icon



you can undo the last change to the position or size of an object.

In programs in which you can position the diagrams yourself (menu item "**System/Graph positioning preferences**"), the result diagrams can also be moved using this function if you have switched to "**Manual graph positioning**".

14.14 GGU-RETAIN: "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-RETAIN.alg**" as file name, and save the file on the same level as the program, the data will be automatically loaded the next time the program is started and need not be entered again.

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

14.15 GGU-RETAIN: "Load graphics preferences" menu item

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

15 Page size + margins menu

15.1 GGU-RETAIN: "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.

15.2 GGU-RETAIN: "Manual resize (mouse)" menu item

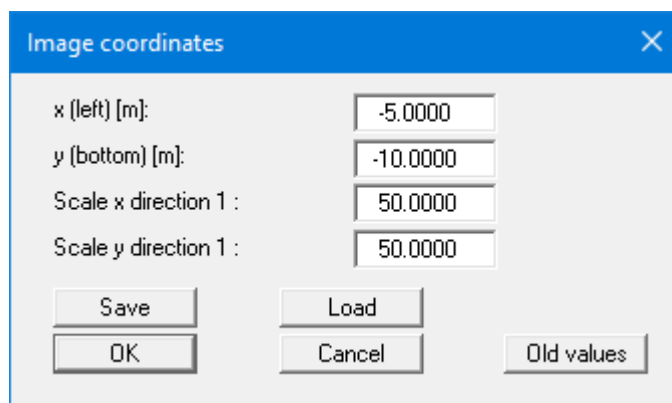
You can transfer the coordinates of a section of your previous graphic display as new image coordinates by holding down the **[Ctrl]** and **[Shift]** keys and marking the desired area with the left mouse button. The scales of the x-direction and y-direction are adjusted accordingly.

In the default setting, the **"Proportional section"** check box is activated. This retains the previous proportions (scale x-direction/scale y-direction).

If you activate the **"Redefine origin"** check box, you can adjust the image coordinates without changing the scale. In this way, you can quickly achieve the desired visualisation without having to try out the x/y coordinates if the scale is set appropriately.

15.3 GGU-RETAIN: "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.



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.

15.4 GGU-RETAIN: "Zoom" menu item

You can linearly enlarge or reduce the image coordinates.

15.5 GGU-RETAIN: "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.

15.6 GGU-RETAIN: "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 deactivating 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.

15.7 GGu-RETAIN: "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 icon



in the toolbar.

15.8 GGU-RETAIN: "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 icon



in the toolbar.

15.9 GGU-RETAIN: "Preferences" menu item

You can activate or deactivate the undo functions.

16 Info menu

16.1 GGU-RETAIN: "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 the GGU program.

16.2 GGU-RETAIN: "GGU on the web" menu item

Using this menu item, you can access the [GGU Software website](#).

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

16.3 GGU-RETAIN: "GGU support" menu item

This menu takes you directly to the [GGU Software Support Portal](#).

Here you can enter your support request directly and upload your edited GGU program file.

16.4 GGU-RETAIN: "Maxima" menu item

Here you can check the defaults for maximum values of the program.

16.5 GGU-RETAIN: "Active wall friction angle" menu item

A message box appears with information relating to active angle of wall friction according to the EAB.

16.6 GGU-RETAIN: "Compare earth pressure coefficients" menu item

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

16.7 GGU-RETAIN: "Vertical capacity" menu item

You can compare $q_{b,k}$ and $q_{s,k}$ for given q_c and $c_{u,k}$ values to EAU 2012 and Recommendations on Piling 2012.

16.8 GGU-RETAIN: "Active earth pressure (constrained slip surface)" menu item

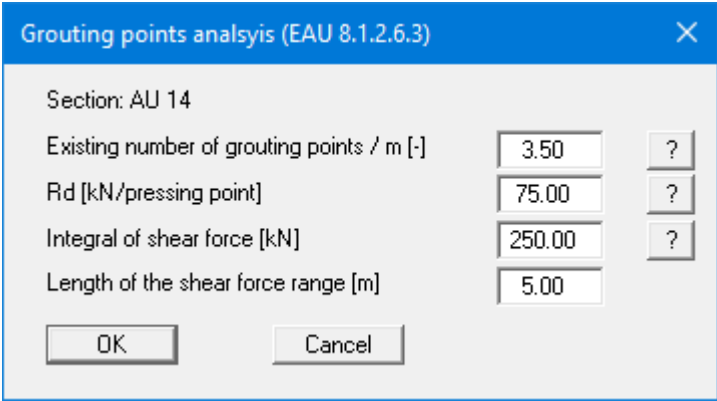
The active earth pressure is determined for a constrained slip surface.

16.9 GGU-RETAIN: "Steel design to DIN EN 1993" menu item

Design values can be entered by hand irrespective of any analysis results. They are then used to perform steel design to EN 1993/EC 3.

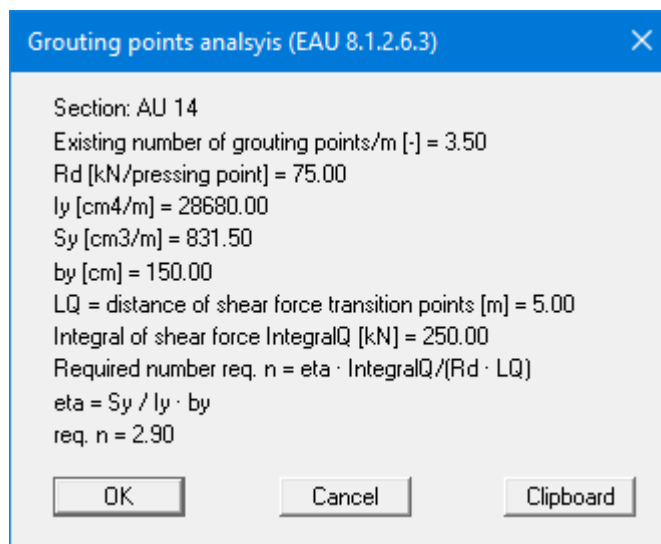
16.10 GGU-RETAIN: "Grouting points analysis (U sections)" menu item

Using this menu item, you can have the grouting points verified for a selected section. You can use the following dialog box to examine various settings for the selected section.



Grouting points analysis (EAU 8.1.2.6.3)		
Section: AU 14		
Existing number of grouting points / m [-]	3.50	?
Rd [kN/pressing point]	75.00	?
Integral of shear force [kN]	250.00	?
Length of the shear force range [m]	5.00	
OK Cancel		

You will then receive the following verification, for example, which you can transfer to your report via the Windows clipboard ("**Clipboard**" button).



Grouting points analysis (EAU 8.1.2.6.3)

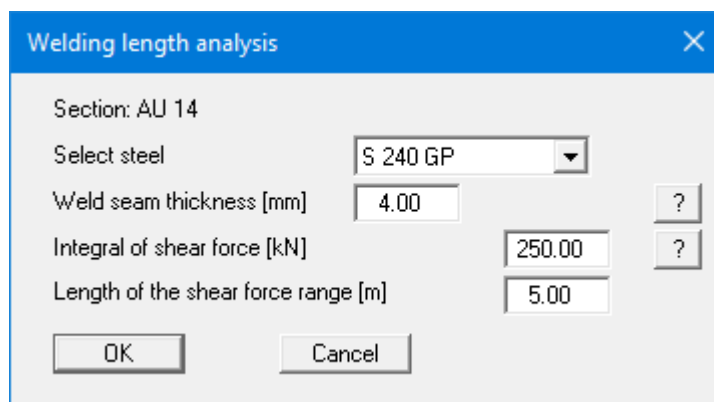
Section: AU 14
Existing number of grouting points/m [-] = 3.50
Rd [kN/pressing point] = 75.00
Iy [cm⁴/m] = 28680.00
Sy [cm³/m] = 831.50
by [cm] = 150.00
LQ = distance of shear force transition points [m] = 5.00
Integral of shear force IntegralQ [kN] = 250.00
Required number req. $n = \eta \cdot \text{IntegralQ} / (Rd \cdot LQ)$
 $\eta = Sy / Iy \cdot by$
req. $n = 2.90$

OK Cancel Clipboard

After clicking on "OK", you can verify the grouting points for another section or for the same section with different specifications. Click "Cancel" to exit the menu item.

16.11 GGU-RETAIN: "Welding lengths analysis (U sections)" menu item

Using this menu item, you can have the welding length verified for a selected section. You can use the following dialog box to examine various settings for the selected section.



Welding length analysis

Section: AU 14

Select steel: S 240 GP

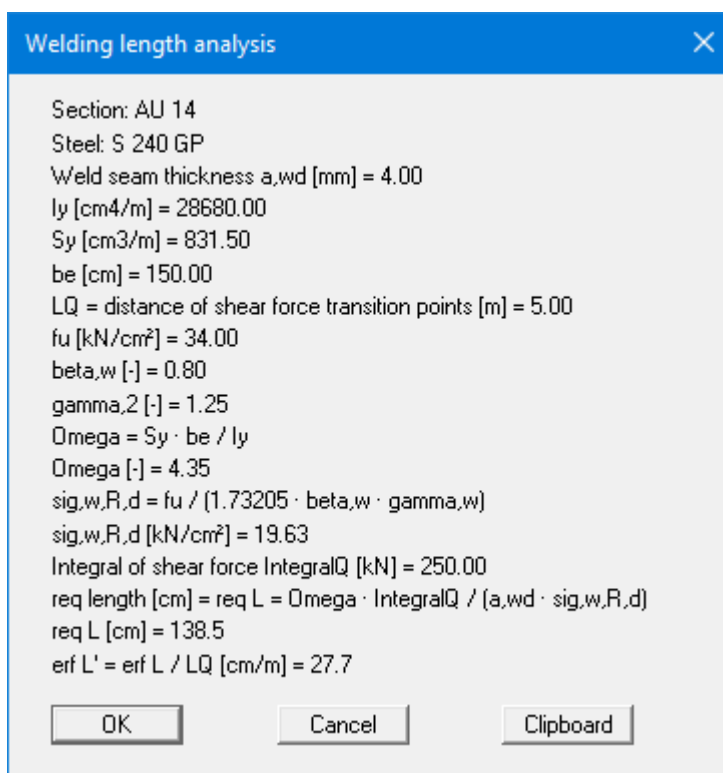
Weld seam thickness [mm]: 4.00

Integral of shear force [kN]: 250.00

Length of the shear force range [m]: 5.00

OK Cancel

You will then receive the following verification, for example, which you can transfer to your report via the Windows clipboard ("Clipboard" button).



After clicking on "OK", you can verify the welding length for another section or for the same section with different specifications. Click "Cancel" to exit the menu item.

16.12 GGU-RETAIN: "Help" menu item

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

16.13 GGU-RETAIN: "What's new?" menu item

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

16.14 GGU-RETAIN: "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.