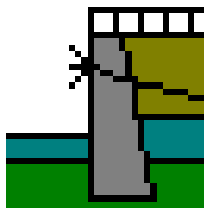


Analysis of underpinning and footings
to DIN 4123, DIN 4017, DIN 4019 and EAB

GGU-UNDERPIN

VERSION 9



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

January 2022
Prof. Dr. Johann Buß
Civilserve GmbH, Steinfeld

Contents:

1	Preface.....	7
2	Capabilities	8
3	Licence protection	10
4	Language selection	10
5	Starting the program	11
6	First steps using a worked example.....	12
6.1	System description	12
6.2	Step 1: Select analysis options	13
6.3	Step 2: Define the excavation.....	14
6.4	Step 3: Define load on the underpinning body	15
6.5	Step 4: Define geometry of the underpinning body.....	16
6.6	Step 5: Define specification for concrete	17
6.7	Step 6: Define berm.....	18
6.8	Step 7: Define soil	18
6.9	Step 8: Define type of earth pressure	19
6.10	Step 9: Analyse system.....	20
6.11	Step 10: Evaluate and visualise the results.....	21
7	Theoretical principles	22
7.1	Underpinning body.....	22
7.2	Soil properties	23
7.3	Active earth pressure.....	23
7.4	At-rest earth pressure.....	23
7.5	Increased active earth pressure	23
7.6	Passive earth pressure.....	24
7.7	Water pressure.....	24
7.7.1	Classical water pressure approach	24
7.7.2	Water pressure approach using flow conduits	25
7.7.3	Recommendation for water pressure approach	26
7.7.4	Possible conceptual error using flow conduits.....	26
7.7.5	What is a potential?.....	27
7.8	Berms	29
7.9	Area loads.....	30
7.10	Loads with limited plan dimensions	31
7.11	Bounded surcharges (active side).....	32
7.12	Double-bounded surcharges (active side).....	33
7.13	Bounded surcharges (passive side).....	34
7.14	Double-bounded surcharges (passive side)	34
7.15	Structural system	35
7.16	Base inclination	36
7.17	Design	37
7.18	Bedding conditions of underpinning body base	37
7.19	Action and displacement boundary conditions.....	37
7.20	Predeformations	38

7.21 Pretensioning.....	39
7.22 Earth pressure redistribution	39
7.23 Bearing capacity safety	40
7.23.1 Analysis principles.....	40
7.23.2 Determining the mean governing soil properties	41
7.24 Settlements	42
7.25 Sliding safety	42
7.26 General stability	42
7.27 Hydraulic heave.....	43
7.27.1 Hydraulic heave safety using global safety factors	43
7.27.2 Utilisation factor (hydraulic heave) using partial safety factors	43
7.28 Buoyancy.....	44
7.28.1 Buoyancy safety using global safety factors	44
7.28.2 Utilisation factor (buoyancy) using partial safety factors	44
7.29 Verification of deep-seated stability	45
7.30 Verification of unreinforced concrete.....	47
7.30.1 Analysis to DIN 1045 (old).....	47
7.30.2 Analysis to EC 2/DIN 1045-1	48
7.30.3 Analysis to DIN 4093	49
7.30.3.1 Simplified analysis concept	49
7.30.3.2 Analysis concept using principal stresses	50
7.30.3.3 Slenderness.....	52
7.31 Construction phases.....	53
8 Description of menu items.....	54
8.1 File menu.....	54
8.1.1 "New" menu item.....	54
8.1.2 "Load" menu item	55
8.1.3 "Save" menu item	55
8.1.4 "Save as" menu item	56
8.1.5 "Print output table" menu item.....	56
8.1.5.1 Selecting the output format	56
8.1.5.2 Button "Output as graphics"	57
8.1.5.3 Button "Output as ASCII"	59
8.1.6 "Export" menu item	60
8.1.7 "Output preferences" menu item.....	60
8.1.8 "Print and export" menu item.....	60
8.1.9 "Batch print" menu item	63
8.1.10 "Exit" menu item	63
8.1.11 "1, 2, 3, 4" menu items.....	63
8.2 Editor 1 menu	64
8.2.1 "Analysis options" menu item.....	64
8.2.2 "Excavation" menu item	64
8.2.3 "Body (general)" menu item	65
8.2.4 "Body (geometry)" menu item	65
8.2.5 "Concrete" menu item.....	65
8.2.6 "Berms (active side)" menu item	66
8.2.7 "Berms (passive side)" menu item	66

8.2.8	"Soils" menu item	67
8.2.9	"Type of earth pressure" menu item.....	68
8.2.10	"Active earth pressure" menu item	69
8.2.11	"Passive earth pressure" menu item	70
8.2.12	"At-rest earth pressure" menu item	71
8.2.13	"User-defined earth pressure coefficients" menu item.....	72
8.2.14	"Groundwater" menu item	73
8.2.15	"Verifications/Safety factors" menu item	74
8.2.16	"Verifications/Partial factors" menu item	75
8.2.17	"Buoyancy + Hydraulic heave" menu item.....	76
8.2.18	"Seismic effects" menu item	77
8.3	Editor 2 menu	78
8.3.1	"Lateral pressures" menu item	78
8.3.2	"Area loads" menu item	79
8.3.3	"Bounded surcharges" menu item.....	80
8.3.4	"Double-bounded surcharges" menu item	81
8.3.5	"Action boundary conditions" menu item.....	82
8.3.6	"Displacement boundary conditions" menu item.....	82
8.3.7	"Anchors" menu item.....	83
8.3.8	"Struts" menu item.....	84
8.3.9	"Pretensioning" menu item	85
8.3.10	"Groundwater potentials" menu item.....	85
8.3.11	"Settlements" menu item.....	86
8.3.12	"Deep-seated stability" menu item.....	87
8.3.13	"Predeformation info" menu item	88
8.3.14	"Predeformation preferences" menu item	88
8.3.15	"Anchor steel design" menu item.....	90
8.4	System menu	91
8.4.1	"Info" menu item	91
8.4.2	"Special preferences" menu item	91
8.4.3	"Depth subdivisions" menu item.....	91
8.4.4	"Analyse" menu item	92
8.4.4.1	Start dialog box	92
8.4.4.2	"Type of toe pivot" group box.....	92
8.4.4.3	"Special preferences" group box	93
8.4.4.4	"Type of redistribution" group box	93
8.4.5	"Graph positioning preferences" menu item	96
8.4.6	"Graphics output preferences" menu item	97
8.4.7	"Labelling preferences" menu item.....	99
8.4.8	"Body pens + colours" menu item	100
8.4.9	"Dimension lines" menu item	101
8.4.10	"Display system" menu item.....	101
8.4.11	"Display results" menu item	101
8.5	Evaluation menu.....	102
8.5.1	General notes	102
8.5.2	"Earth pressure redistribution" menu item.....	102
8.5.3	"Main output summary" menu item.....	102

8.5.4	"Anchor and strut summary" menu item.....	102
8.5.5	"Bearing capacity" menu item	102
8.5.6	"Settlements" menu item.....	102
8.5.7	"Sliding safety" menu item	102
8.5.8	"Deep-seated stability summary" menu item.....	103
8.5.9	"Hydraulic heave FOS summary" menu item	104
8.5.10	"Buoyancy FOS summary" menu item	104
8.5.11	"EQU analysis" menu item	104
8.6	Construction phases menu.....	104
8.6.1	General notes	104
8.6.2	"Info" menu item	104
8.6.3	"Select files" menu item.....	105
8.6.4	"Display summary" menu item.....	105
8.7	Graphics preferences menu	106
8.7.1	"Refresh and zoom" menu item	106
8.7.2	"Zoom info" menu item	106
8.7.3	"Legend font selection" menu item.....	106
8.7.4	"Pen colour and width" menu item	107
8.7.5	"Mini-CAD toolbar" and "Header toolbar" menu items	107
8.7.6	"Toolbar preferences" menu item	107
8.7.7	"Soil properties legend" menu item	109
8.7.8	"General legend" menu item.....	110
8.7.9	"Design legend" menu item	111
8.7.10	"Settlement legend" menu item.....	112
8.7.11	"Move objects" menu item.....	112
8.7.12	"Save graphics preferences" menu item.....	112
8.7.13	"Load graphics preferences" menu item	112
8.8	Page size + margins menu	113
8.8.1	"Auto-resize" menu item.....	113
8.8.2	"Manual resize (mouse)" menu item.....	113
8.8.3	"Manual resize (editor)" menu item.....	113
8.8.4	"Zoom" menu item.....	113
8.8.5	"Font size selection" menu item.....	113
8.8.6	"Page size and margins" menu item.....	114
8.8.7	"Undo" menu item	115
8.8.8	"Restore" menu item	115
8.8.9	"Preferences" menu item.....	115
8.9	Info menu	115
8.9.1	"Copyright" menu item.....	115
8.9.2	"Maxima" menu item.....	115
8.9.3	"Active wall friction angle" menu item	115
8.9.4	"Compare earth pressure coefficients" menu item	115
8.9.5	"Help" menu item	116
8.9.6	"GGU on the web" menu item	116
8.9.7	"GGU support" menu item.....	116
8.9.8	"What's new?" menu item	116
8.9.9	"Language preferences" menu item	116

9 Tips and tricks.....	117
9.1 "?" buttons	117
9.2 Keyboard and mouse	118
9.2.1 Function keys	119
9.3 "Copy/print area" icon.....	120
10 Index.....	121

List of figures:

<i>Figure 1 Illustration of worked example 1</i>	<i>12</i>
<i>Figure 2 Underpinning body.....</i>	<i>22</i>
<i>Figure 3 Classical water pressure approach</i>	<i>24</i>
<i>Figure 4 Water pressure approach using flow conduits</i>	<i>25</i>
<i>Figure 5 Possible conceptual error using flow conduits.....</i>	<i>26</i>
<i>Figure 6 Definition of potential</i>	<i>28</i>
<i>Figure 7 Berm on the active side.....</i>	<i>29</i>
<i>Figure 8 Area load.....</i>	<i>30</i>
<i>Figure 9 At-rest earth pressure from area loads.....</i>	<i>31</i>
<i>Figure 10 Bounded surcharge (active side).....</i>	<i>32</i>
<i>Figure 11 Two bounded surcharges.....</i>	<i>33</i>
<i>Figure 12 Double-bounded surcharge</i>	<i>33</i>
<i>Figure 13 Bounded surcharge (passive side).....</i>	<i>34</i>
<i>Figure 14 Structural system of a wall without anchors or struts</i>	<i>35</i>
<i>Figure 15 Base inclination.....</i>	<i>36</i>
<i>Figure 16 Logarithmic spiral.....</i>	<i>41</i>
<i>Figure 17 Compound "deep slip planes"</i>	<i>45</i>
<i>Figure 18 Compound "deep slip plane", which is not investigated.....</i>	<i>46</i>
<i>Figure 19 n-value and width for concrete verification.....</i>	<i>47</i>
<i>Figure 20 DIN 4093:2012-08: Figure 1</i>	<i>50</i>
<i>Figure 21 DIN 4093:2012-08: Stresses in cross section.....</i>	<i>51</i>
<i>Figure 22 DIN 4093:2012-08: Utilisation factor.....</i>	<i>52</i>
<i>Figure 23 Birectangular earth pressure redistribution.....</i>	<i>93</i>
<i>Figure 24 Earth pressure redistribution in a trapezoidal</i>	<i>94</i>
<i>Figure 25 Earth pressure redistribution in a quadrilateral</i>	<i>94</i>
<i>Figure 26 User defined earth pressure redistribution.....</i>	<i>95</i>

1 Preface

GGU-UNDERPIN allows the analysis and design of underpinning walls and, in special cases, footings.

The program is essentially based on the Recommendations of the Working Group on Excavations (*German: EAB*). Virtually all the cases considered by the Working Group are included. 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 factors* to DIN 1054 (old) and the *partial safety factors* to EC 7 may be taken into consideration.

The application is designed to allow simple data input or modification. The input is immediately shown on the screen, giving you optimum control over what you are doing. Reading of the manual can mostly be dispensed with, because

"?" buttons 

dealing with almost all geotechnical and program-specific problems are available in the dialog boxes. You are presented with the necessary information by clicking the "?" button (see also Section 9.1).

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

The application was developed in 1995, in part using previously existing modules. It has been tested on numerous examples found in the literature and in practice. It is now used by a large number of civil engineering firms, construction companies and university institutes, and has successfully completed well over 1000 structural tests. Nevertheless, liability for completeness and correctness of the program and the manual, and for any consequential damage resulting from incompleteness or incorrectness, cannot be accepted.

2 Capabilities

GGU-UNDERPIN has the following characteristics and maximum capabilities:

- Almost any type of underpinning body may be defined.
- Base inclination of the underpinning body
- Up to 50 soil layers
- Up to 40 berms on the active earth pressure side
- Up to 40 berms on the passive earth pressure side
- Analysis with active and increased active earth pressure and with at-rest earth pressure
- Coefficients of active earth pressure to DIN 4085
- Coefficients of passive earth pressure to DIN 4085, Streck, Caquot/Kerisel
- Classical water pressure approach for impermeable 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 the passive earth pressure sides
- Analysis of safety factors against deep-seated failure with optimisation of anchor lengths
- Analysis of bearing capacity safety to DIN 4017 in addition to verification of footings in slopes and footings with inclined bases
- Analysis of settlements to DIN 4019
- Analysis of sliding safety factor
- Analysis of safety against hydraulic heave
- Analysis of safety against buoyancy
- Verification of unreinforced concrete to EC 2/DIN 1045 or DIN 4093:2012-08
- Analysis of static equilibrium in the EQU limit state
- 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 wall by means of a two-dimensional rod construction module based on finite-element methods. In contrast to many other applications, the influence of inclined anchors or struts and their interactions are thus directly considered in the analysis approach. Analysis can even be performed using 2nd order theory, making the generally onerous buckling length investigation of struts and wall unnecessary. In addition, it is possible to consider loads on the struts (e.g. for auxiliary bridges additionally acting as struts) during analysis.
- Specification of up to 5 displacement boundary conditions (rotation, displacement in x or y directions) at any location

- Specification of up to 5 action boundary conditions (moment, shear force and normal force) at any location
- 20 anchor and strut sets. In defining the anchors and struts, axial stiffness and bending stiffness can be specified, so that, for example, passive *soft* anchors can also be incorporated.
- Consideration of pretensioning of anchors and struts possible
- Consideration of seismic effects via additional horizontal loads and/or altered earth pressure coefficients
- Definition of up to 20 additional potentials for the analysis of flow conduits to the left and right of the wall for complex groundwater conditions
- Automatic computation of earth pressure redistribution in accordance with EAB recommendations
- Other earth pressure redistribution options:
 - No redistribution
 - Rectangle
 - 2 Rectangles
 - Triangle (maximum can be optionally placed at top, middle or bottom)
 - Trapezoid
 - Quadrilateral with maximum at anchor locations or any other location.
 - User-defined polygon
- Footing pressure can be redistributed either to the wall toe or to the transition point.
- Passive earth pressure can be superimposed (*global safety factor concept*) or placed in front of the wall.
- Following computation of the system the earth pressure, pore water pressure, moment, shear, normal force and bending line are displayed on the screen. The visualisation can be varied within wide limits. For example, the distributions of the potential and gradient, etc., can also be displayed.
- Display of the pressure line, the 1st and 2nd kernel width and the centroid axis
- At anchor points, predeformations that are already present can be defined as boundary conditions.
- Previously analysed records can be combined, in order to investigate the cumulated deformations for advancing and retreating conditions. In addition, action effect profiles can be represented as an envelope.
- Legends can be displayed on screen indicating soil characteristics, general computation data, data needed for settlement analysis and the main design data. Thus, virtually all the raw data necessary for the computation and the results are shown on screen.
- **GGU-UNDERPIN**'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).

- By clicking the "**Copy/print area**" icon on the toolbar you can copy any part of the graphics to the clipboard or save it as an EMF file (Enhanced Metafile Format). Using the "**Mini-CAD toolbar**" or "**Header toolbar**" modules, you can insert EMF files into your graphics. Thus, the results of a slope failure analysis or of a grading analysis, for example, can easily be imported into the current graphics.

3 Licence protection

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

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

4 Language selection

GGU-UNDERPIN is a bilingual program. The program always starts with the language setting applicable when it was last ended.

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

5 Starting the program

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

- File
- Info

By going to the **"File"** menu, a previously analysed system can be loaded by means of the **"Load"** menu item, or a new one created using **"New"**. After clicking the **"New"** menu item a dialog box opens for specifying general preferences for your new system (see Section 8.1.1). You then see nine menus in the menu bar:

- File
- Editor 1
- Editor 2
- System
- Evaluation
- Construction stages
- Graphics preferences
- Page size + margins
- Info

After clicking one of these menus, the so-called menu items roll down, allowing you access to all program functions.

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

If you would like to refresh the screen contents, press either **[F2]** or **[Esc]**. Additionally, the **[Esc]** key sets the screen presentation back to your current zoom, which has the default value 1.0, corresponding to an A3 format sheet.

6 First steps using a worked example

6.1 System description

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

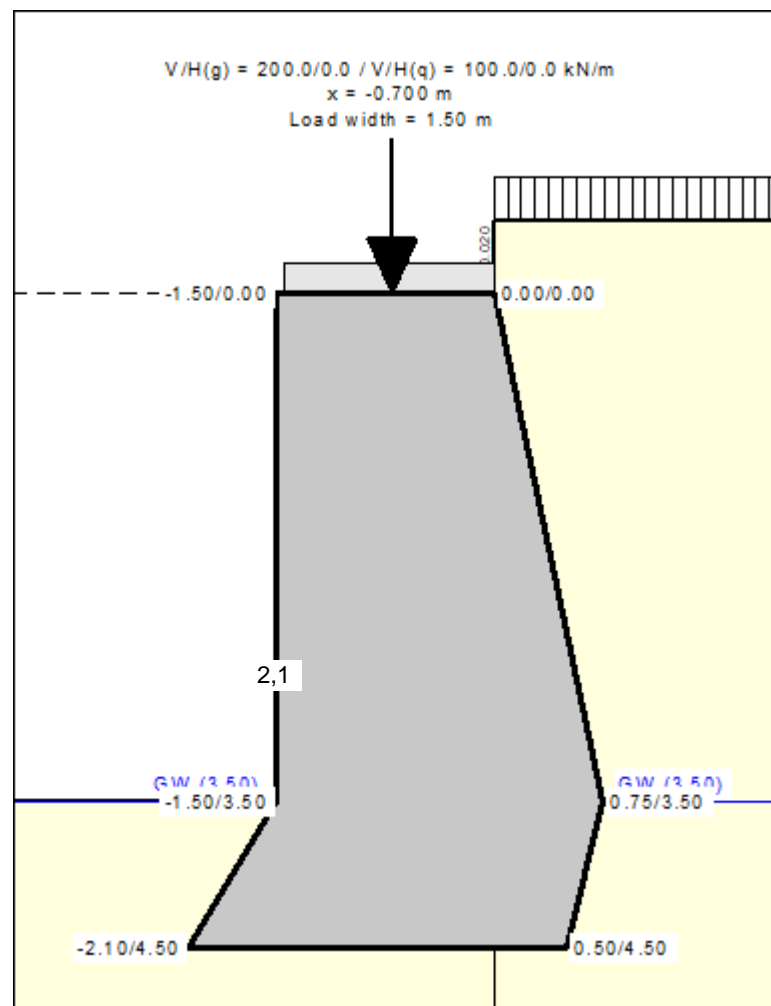


Figure 1 Illustration of worked example 1

The groundwater level lies on excavation base. At the top of the wall a permanent vertical load of 200 kN/m and a changeable vertical load of 100 kN/m are present. The load is transferred with a transmission width of 1.50 m. The unit weight of the concrete is = 23 kN/m³. Behind the top of the wall the ground surface is elevated by 0.5 m. A live load of 5 kN/m² acts on the ground surface.

6.2 Step 1: Select analysis options

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

The "New dataset" dialog box contains the following settings:

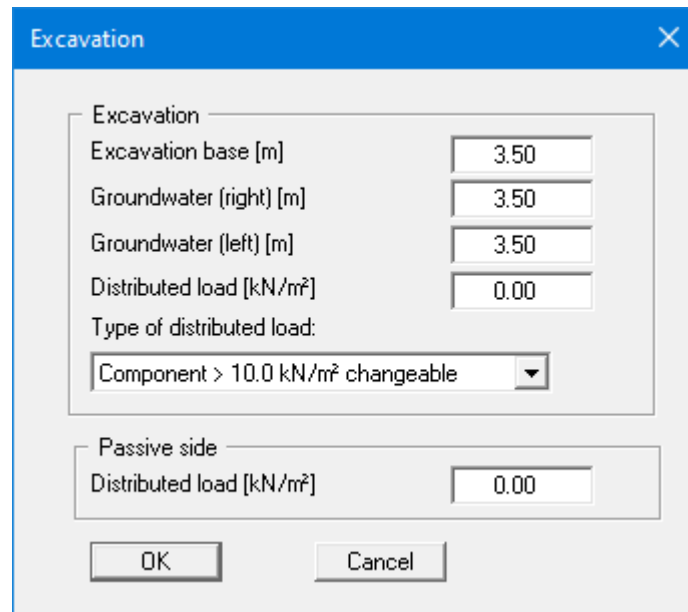
- Project identification:** Example 1
- Standard:** ☒ Partial safety factor concept (EC 7) (Info EC 7)
☐ Partial safety factors (DIN 1054:2005)
☐ Global safety factors (DIN 1054 old)
- Analysis of unreinforced concrete:** ☐ To DIN 4093:2012-08
☒ DIN 4093:2012-08 using principal stresses
☒ To EC 2/DIN 1045
Shear analysis to DIN 4093 simplified
- General:**
☐ Show excavation on right
☐ Do not investigate overturning
☒ Use absolute heights (Design: m)
☐ Differentiate active + passive soil properties (with regard to: phi, gamma + gamma')
- Equivalent wall:**
☐ Equivalent wall compliant with EAB 6th Edition
☐ Use wall friction for bounded surcharges
Wall friction angle $\delta, k/\phi =$ 0.50

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

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

6.3 Step 2: Define the excavation

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



The image shows a software dialog box titled "Excavation" with a close button (X) in the top right corner. The dialog box contains two main sections. The first section, labeled "Excavation", includes five input fields: "Excavation base [m]" with a value of 3.50, "Groundwater (right) [m]" with a value of 3.50, "Groundwater (left) [m]" with a value of 3.50, "Distributed load [kN/m²]" with a value of 0.00, and a dropdown menu for "Type of distributed load:" currently set to "Component > 10.0 kN/m² changeable". The second section, labeled "Passive side", includes one input field: "Distributed load [kN/m²]" with a value of 0.00. At the bottom of the dialog box are two buttons: "OK" and "Cancel".

Field	Value
Excavation base [m]	3.50
Groundwater (right) [m]	3.50
Groundwater (left) [m]	3.50
Distributed load [kN/m²]	0.00
Type of distributed load:	Component > 10.0 kN/m² changeable
Passive side Distributed load [kN/m²]	0.00

The live load of 5 kN/m² for the example is entered later in "**Editor 1/Berms (active side)**".

6.4 Step 3: Define load on the underpinning body

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

Underpinning body

Loads + geometry

	Permanent(g)	Changeable(q)
Vertical load [kN/m]:	200.0	100.0
Horizontal load [kN/m]:	0.0	0.0
x [m]:	-0.700	
Load transmission width [m]:	1.500	
Footing length [m]: (for bearing capacity + settlement)	100.00	
Base inclination [°]:	0.00	

OK Cancel

Diagram: A cross-section of a wall with a vertical load +V and a horizontal load +H. The load +V is applied at a distance x from the top right edge of the wall.

Define the "Vertical load" and the "Horizontal load" as permanent loads at the top of the wall. Define the location of the vertical load from the top right edge of the underpinning wall using "x(load)". The "Load transmission width" is required for verification of internal stability to EC 2 (see "Theoretical principles/Verification of unreinforced concrete", Section 7.30). Define the appropriate value, here.

By entering the "Footing length" you define the length of the underpinning body. This value, however, is only used in the bearing capacity and settlement analyses. Additionally, you can define any "Base inclination" of the underpinning body in the above dialog box (see "Theoretical principles/Base inclination", Section 7.16).

For the example enter the values shown in the above dialog box.

6.5 Step 4: Define geometry of the underpinning body

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

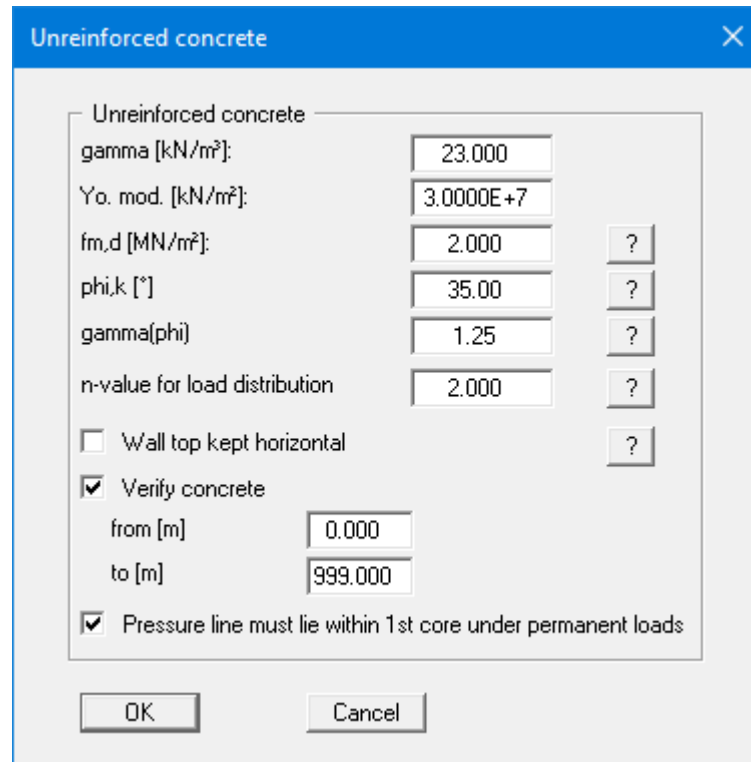
No	Depth [m]	x(left) [m]	x(right) [m]
Top of wall = 0.000		-1.500	0.000
2	3.500	-1.500	0.750
3	4.500	-2.100	0.500

Change the number of points to **3** and enter the values shown above. The depth values are relative to the top of the wall elevation. For every depth two coordinates (left and right edge of the wall) need to be given (see "**Theoretical principles/Underpinning body**", Section 7.1).

After closing the box press [**F9**] for optimised graphics display.

6.6 Step 5: Define specification for concrete

Select the menu item "**Editor 1/Concrete**". Enter the values given in the following dialog box. "**Yo. mod.**" designates the Young's modulus of the concrete. This value is required for the structural analysis of the wall.



The dialog box is titled "Unreinforced concrete" and contains the following fields and options:

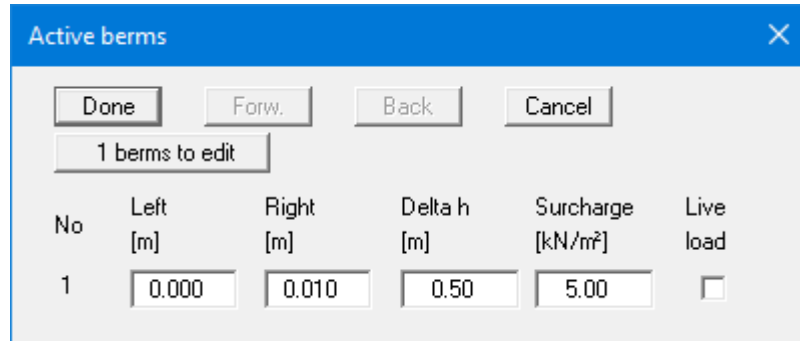
Parameter	Value	Help
gamma [kN/m ²]:	23.000	
Yo. mod. [kN/m ²]:	3.0000E+7	
f _{m,d} [MN/m ²]:	2.000	?
phi _k [°]	35.00	?
gamma(phi)	1.25	?
n-value for load distribution	2.000	?
<input type="checkbox"/> Wall top kept horizontal		?
<input checked="" type="checkbox"/> Verify concrete		
from [m]	0.000	
to [m]	999.000	
<input checked="" type="checkbox"/> Pressure line must lie within 1st core under permanent loads		

Buttons: OK, Cancel

If the "**Verify concrete**" check box is activated, the unreinforced concrete is verified, if necessary, by means of a defined partial section. Values for "**f_{m,d}**" (design value of concrete compressive strength) and "**n-value for load distribution**" are required here (see explanation in Section 7.30).

6.7 Step 6: Define berm

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



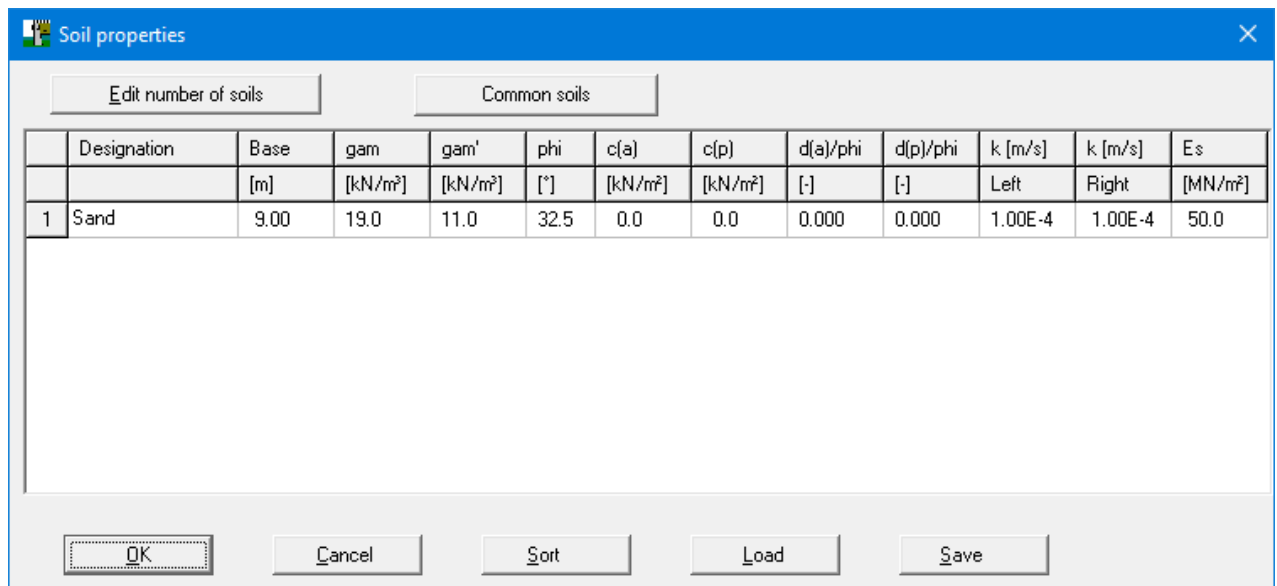
The "Active berms" dialog box has a blue title bar with a close button. It contains four buttons at the top: "Done", "Forw.", "Back", and "Cancel". Below these is a button labeled "1 berms to edit". The main area is a table with the following columns: "No", "Left [m]", "Right [m]", "Delta h [m]", "Surcharge [kN/m²]", and "Live load". The first row contains the values: "1", "0.000", "0.010", "0.50", "5.00", and an unchecked checkbox.

No	Left [m]	Right [m]	Delta h [m]	Surcharge [kN/m²]	Live load
1	0.000	0.010	0.50	5.00	<input type="checkbox"/>

Enter the values shown above and click "Done".

6.8 Step 7: Define soil

Go to "Editor 1" menu and select "Soils".



The "Soil properties" dialog box has a blue title bar with a close button. It contains two buttons at the top: "Edit number of soils" and "Common soils". Below these is a table with the following columns: "Designation", "Base [m]", "gam [kN/m²]", "gam' [kN/m²]", "phi [°]", "c(a) [kN/m²]", "c(p) [kN/m²]", "d(a)/phi [-]", "d(p)/phi [-]", "k [m/s] Left", "k [m/s] Right", and "Es [MN/m²]". The first row contains the values: "1", "Sand", "9.00", "19.0", "11.0", "32.5", "0.0", "0.0", "0.000", "0.000", "1.00E-4", "1.00E-4", and "50.0". Below the table is a large empty text area. At the bottom are five buttons: "OK", "Cancel", "Sort", "Load", and "Save".

	Designation	Base [m]	gam [kN/m²]	gam' [kN/m²]	phi [°]	c(a) [kN/m²]	c(p) [kN/m²]	d(a)/phi [-]	d(p)/phi [-]	k [m/s] Left	k [m/s] Right	Es [MN/m²]
1	Sand	9.00	19.0	11.0	32.5	0.0	0.0	0.000	0.000	1.00E-4	1.00E-4	50.0

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

6.9 Step 8: Define type of earth pressure

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

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 [-] 0.50

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 [-] 0.50

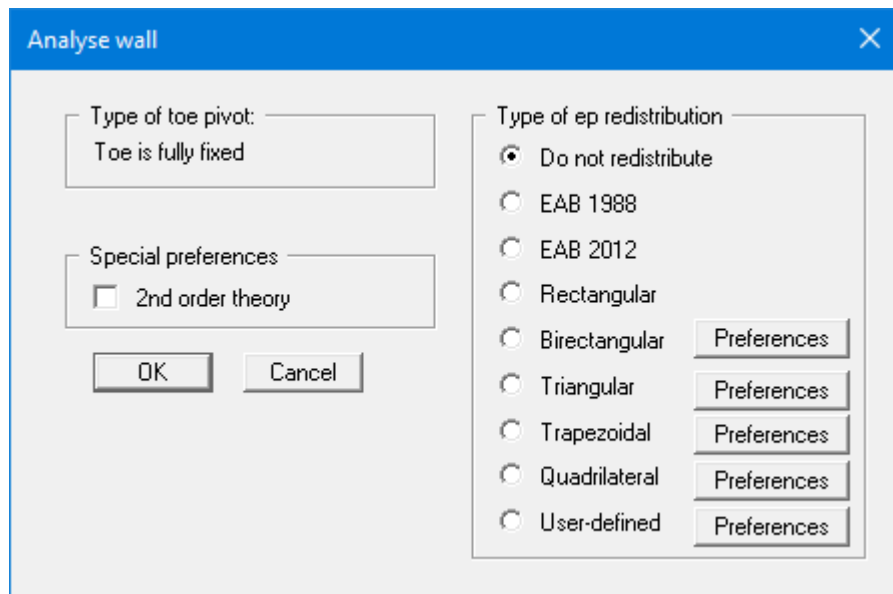
OK Cancel

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

This completes data input.

6.10 Step 9: Analyse system

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



After clicking the "OK" button the analysis begins. The program first performs an intensive plausibility check of the data input.

Once the analysis is complete the system is displayed on the screen, together with the results. Besides the action effects the following are shown in the body:

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

6.11 Step 10: Evaluate and visualise the results

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

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

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

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

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

7 Theoretical principles

7.1 Underpinning body

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

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

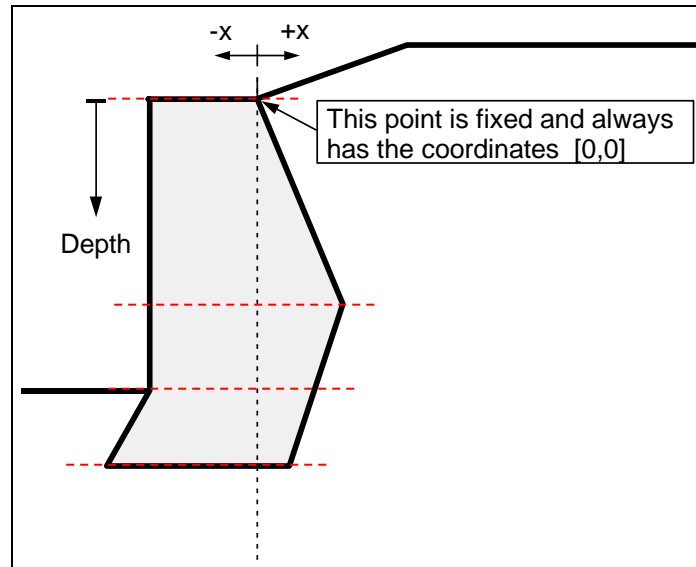


Figure 2 Underpinning body

To define the body of figure 2, you must enter the depth and corresponding two x-values in four horizontal sections. In addition, you can specify a base inclination (see Section 7.16).

7.2 Soil properties

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

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

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

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

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

7.3 Active earth pressure

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

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

7.4 At-rest earth pressure

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

7.5 Increased active earth pressure

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

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

$$0.0 \leq f \leq 1.0$$

7.6 Passive earth pressure

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

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

Mohr/Coulomb (root)

This approach is obsolete and should no longer be used!

In addition, calculation of the passive earth pressure after Culmann is possible. 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.

7.7 Water pressure

7.7.1 Classical water pressure approach

For 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:

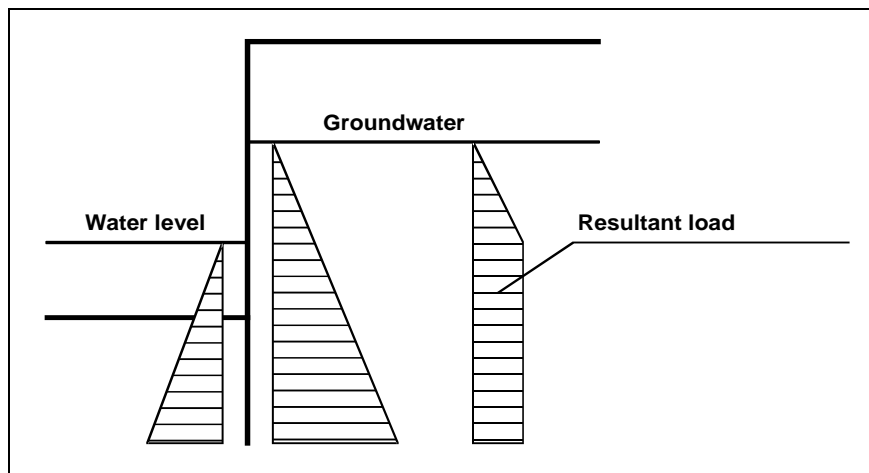


Figure 3 Classical water pressure approach

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 wall.

7.7.2 Water pressure approach using flow conduits

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

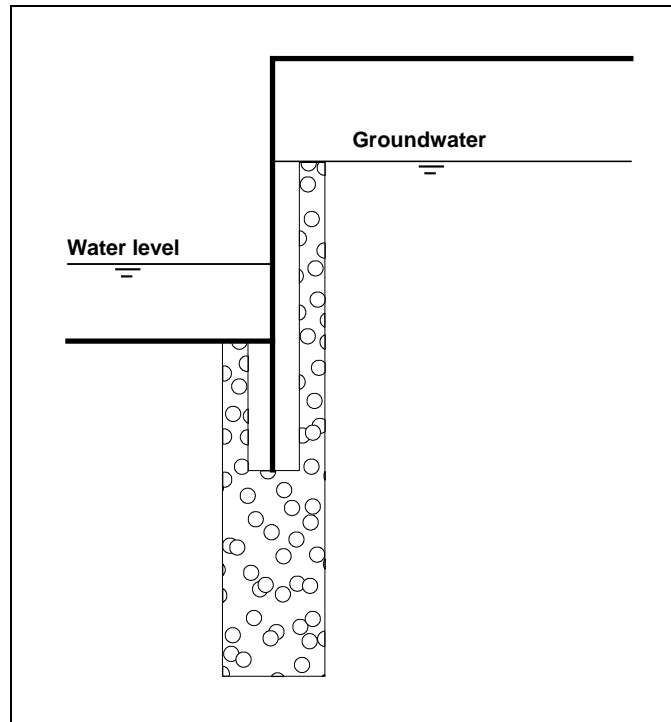


Figure 4 Water pressure approach using flow conduits

The spacing of the two upper flow conduits is the result of the largest wall thickness.

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

7.7.3 Recommendation for water pressure approach

The classical water pressure approach is valid when the toe of the wall is embedded in a layer of **low permeability**. Otherwise the difference in water pressure at the underpinning body base (see Figure 3 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 underpinning body base, the water pressure approach using flow conduits should always be preferred. If doubts remain, you will have to carry out a two-dimensional groundwater analysis according to the EAU. The potentials determined at the wall can be incorporated into flow conduits.

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

7.7.4 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:

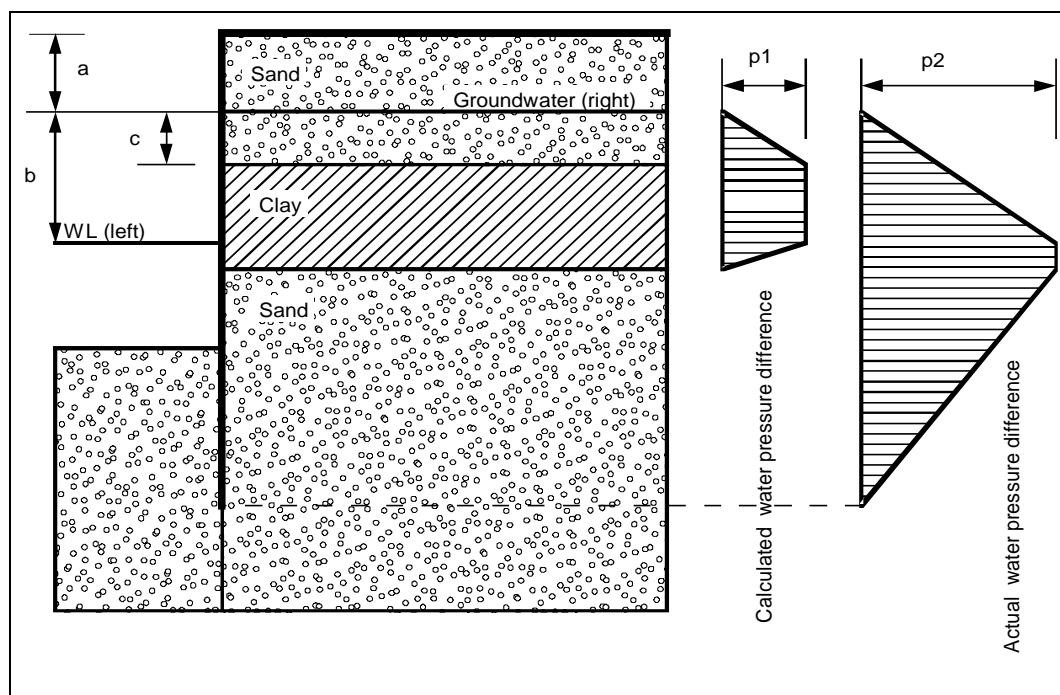


Figure 5 Possible conceptual error using flow conduits

$$p1 = \gamma_w \cdot c$$

$$p2 = \gamma_w \cdot b$$

The permeabilities of the clay and sand layers are $1 \cdot 10^{-8}$ m/s and $1 \cdot 10^{-4}$ m/s, respectively. Although **GGU-UNDERPIN** conforms completely to potential theory, the calculated difference in water pressure, $p1$ [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 $p1$ shown in Figure 5, 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, so as to achieve the desired distribution of water pressure differentials, $p2$, as shown in Figure 5. Although **GGU-UNDERPIN** 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 **GGU-UNDERPIN** graphically displays the difference in water pressure between the left and right sides of the wall.

7.7.5 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/m² at a particular point, then water pressure, u (in metres) = p/γ_w (γ_w = unit weight of water $\cong 10.0$ kN/m³).

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

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

An example:

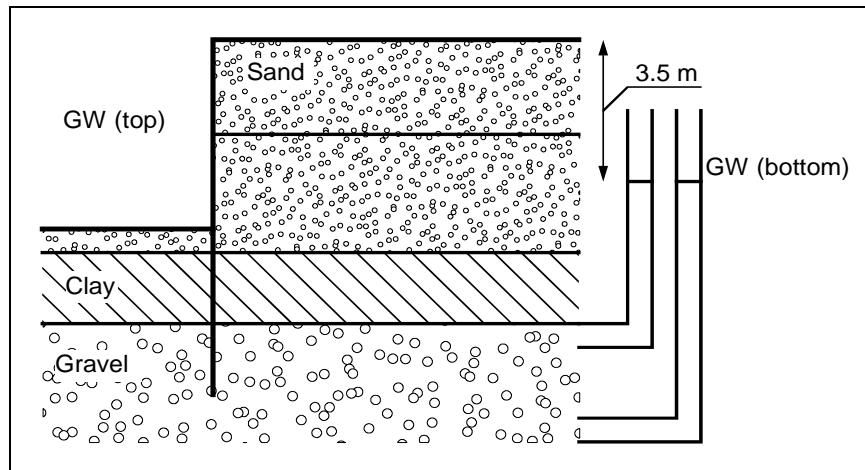


Figure 6 Definition of potential

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

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

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

For the gradient (i) we have:

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

Δh = potential difference between two points

ΔL = distance between the two points

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

7.8 Berms

GGU-UNDERPIN can handle up to 10 berms on both the active and the passive sides of the wall. The berms may include surcharges. The effect on earth pressure is taken into consideration according to the Piling Handbook (Krupp Hoesch Stahl).

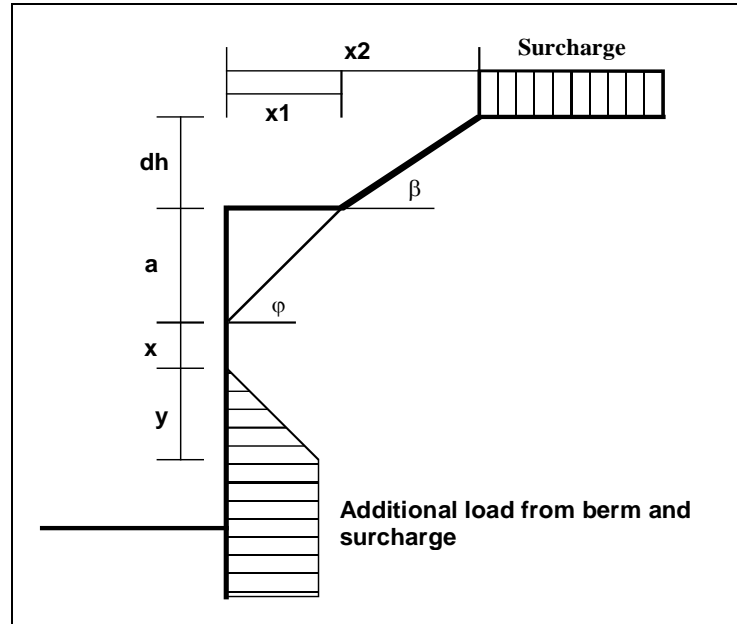


Figure 7 Berm on the active side

The following relationships apply for the parameters x and y :

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

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

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

γ = unit weight of soil in the berm area

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

7.9 Area loads

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

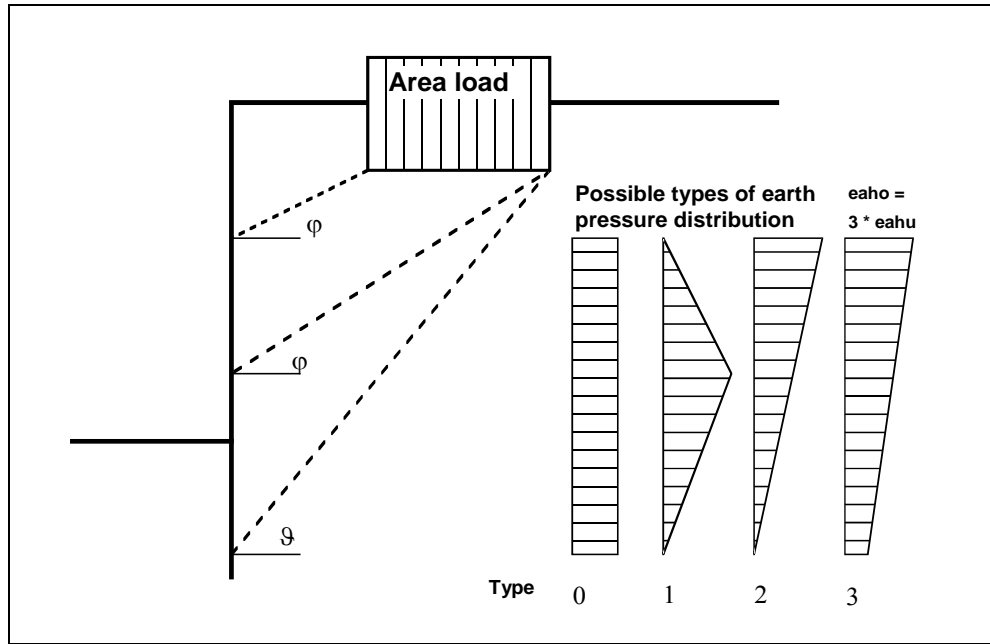


Figure 8 Area load

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

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

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

For at-rest earth pressure, the area loads are calculated by increasing by the factor k_0/k_{ah} in compliance with DIN 4085 Section 6.4.3.

Alternatively, it is possible to determine the demands placed on a wall from area loads using the theory of elastic half-space. The two load concentration factors '3' and '4' can be taken into consideration (see also Figure 9).

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

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

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

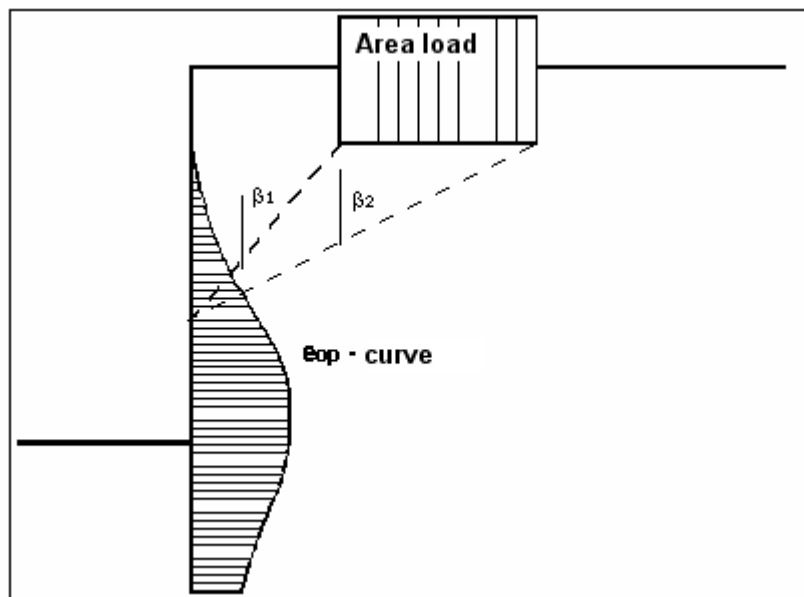


Figure 9 At-rest earth pressure from area loads

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

7.10 Loads with limited plan dimensions

Loads with limited plan dimensions can be reduced compliant with "Piling Handbook" Figure 4.20 (page 64) or DIN 4085:2017-08 (page 17), see also Section 8.3.2.

7.11 Bounded surcharges (active side)

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

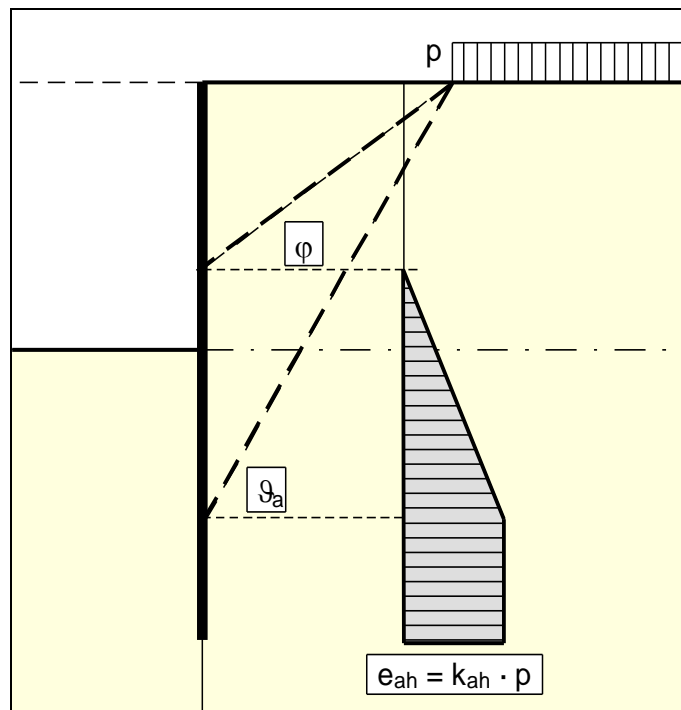


Figure 10 Bounded surcharge (active side)

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

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

7.12 Double-bounded surcharges (active side)

If two bounded surcharges are defined as follows:

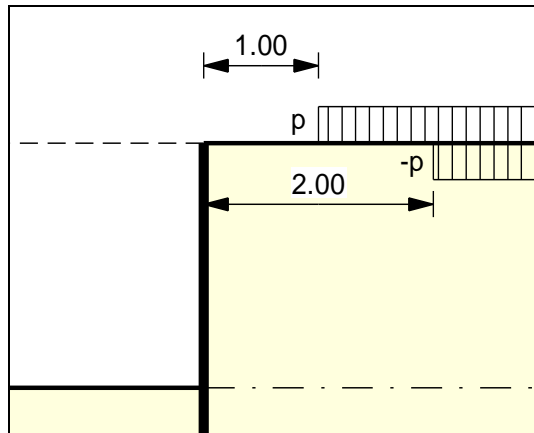


Figure 11 Two bounded surcharges

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

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

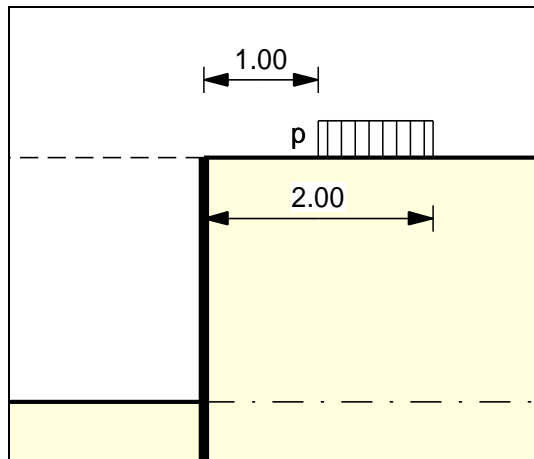


Figure 12 Double-bounded surcharge

The same result is achieved as for the definition provided by Figure 11 but with improved graphics.

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

7.13 Bounded surcharges (passive side)

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

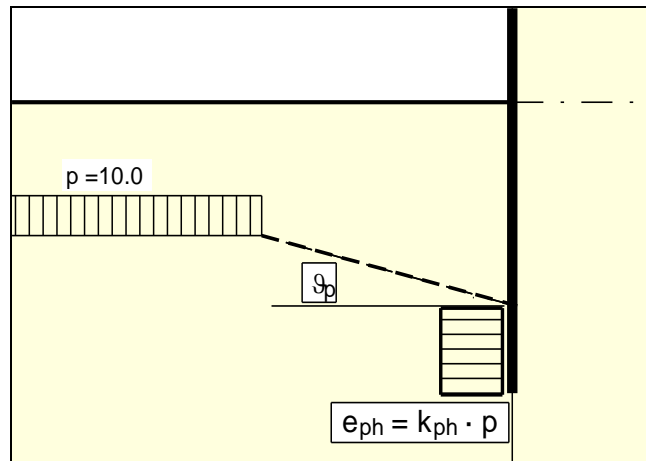


Figure 13 Bounded surcharge (passive side)

7.14 Double-bounded surcharges (passive side)

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

7.15 Structural system

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

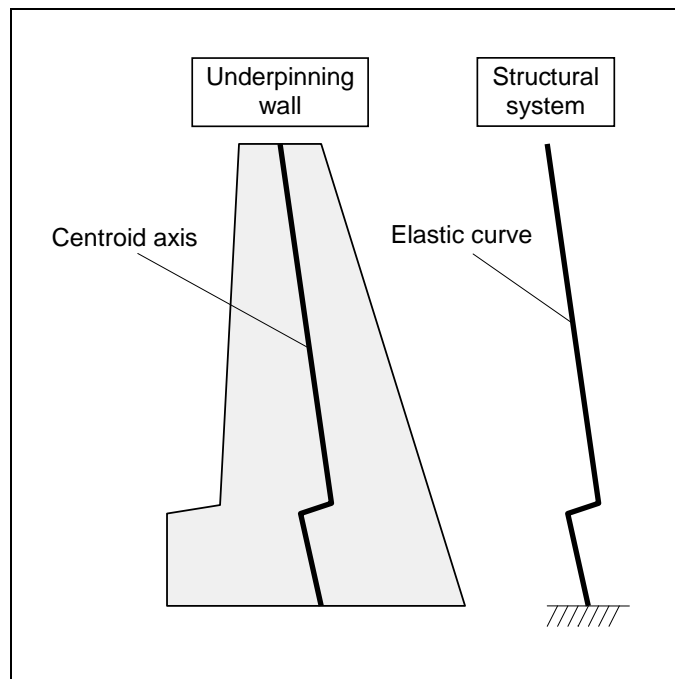


Figure 14 Structural system of a wall without anchors or struts

The stiffness EI of the underpinning wall is acquired from $E \cdot d^3/12$, where d is the horizontal thickness of the wall in the section under consideration. The stiffness of each element is assumed as constant, where the thickness is the thickness at the centre of the section. The remaining values of any anchors or struts can be specified before beginning the analysis.

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

The theoretical principles of the rod construction module can be traced back to an article by Dudeck/Ahrens (*Betonkalender* 1976 (Concrete Calendar), Volume 2). Basically, this is a finite element method based on the displacement method. The division of the wall and the struts into rod sections (finite elements) can be controlled by input (see "**System/Depth subdivisions**" menu item in Section 8.4.3).

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

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

7.16 Base inclination

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

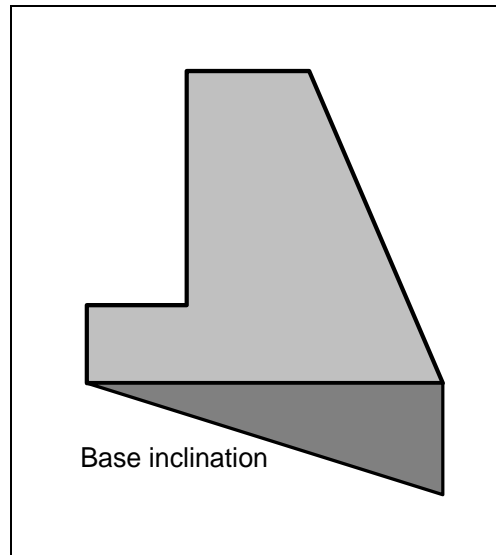


Figure 15 Base inclination

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

In order to disqualify contradictory results from the outset, the base inclination is **not** taken into consideration when determining the action effects. Only when verifying the **safety against sliding** are the active and passive earth pressures at the left and right of the wedge cumulated and this component allocated to the horizontal force when verifying the safety against sliding. When verifying the **bearing capacity safety** (see Section 7.23.1), base inclination is taken into consideration. The additional horizontal force component is **not** incorporated in the analysis, leading to a more conservative result.

7.17 Design

When you begin an analysis, the program requires the stiffnesses of the underpinning wall and any anchors and struts present. The stiffness EI of the wall is acquired from $E \cdot d^3/12$, where d is the thickness of the wall in the section under consideration. The stiffness of each element is assumed as constant, where the thickness is the thickness at the centre of the section. The remaining values of any anchors or struts can be specified before beginning the analysis.

Once the structural analysis is complete the program performs the following verifications for all rods (finite elements):

- calculation of bearing capacity safety to DIN 4017 and additional verification of footing within slopes and for footing with base inclination, where necessary;
- calculation of settlements to DIN 4019;
- calculation of sliding safety;
- calculation of hydraulic heave safety;
- calculation of buoyancy safety;
- verification of the unreinforced concrete to DIN 1045.

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

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

7.18 Bedding conditions of underpinning body base

For systems without anchors or struts the underpinning body base is fully fixed.

For systems using anchors and/or struts, the underpinning body base can either be defined as fixed, freely pivoted or with an embedded base.

7.19 Action and displacement boundary conditions

GGU-UNDERPIN automatically sets the boundary conditions described in Section 7.15 ("**Theoretical principles/Structural system**"). It is also possible to enter additional boundary conditions for any point on the wall. All six state variables can be defined as boundary conditions:

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

7.20 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, 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, **GGU-UNDERPIN** 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, **GGU-UNDERPIN** takes care of the rest. You also have the option of automatically adopting predeformations from a previous analysis.

Designing an underpinning body involves a multitude of different adjustments, making it all too easy to lose one's bearings. **GGU-UNDERPIN** 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 begin analysis;
- accept warning and do not begin analysis.

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

GGU-UNDERPIN 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 a number of files simultaneously is incompatible with this concept. Therefore, **GGU-UNDERPIN** does not create a single dataset when dealing with a number of 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, **GGU-UNDERPIN** 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.

7.21 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.

7.22 Earth pressure redistribution

Classical earth pressure redistribution is not appropriate for braced or tied-back walls. Earth pressure calculated according to classical theory must be redistributed. **GGU-UNDERPIN** includes a large number of pre-programmed redistribution figures. In addition, by defining a polygon, the user can create any redistribution figure. The figures available are given below:

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

For *normal* underpinning walls, earth pressure redistribution according to the EAB is probably the simplest way of finding the correct redistribution figure. If you select this option, you only need to wait for the analysis to be completed. **GGU-UNDERPIN** selects the most appropriate from the available EAB figures (R 71 for in-situ concrete walls). If the program is unable to find a suitable EAB figure because of *abnormal* anchor or strut positions, analysis is terminated, and a warning message is issued. You must then give some thought to selecting or defining your own redistribution figure.

GGU-UNDERPIN gives you the option of including or excluding area loads in redistribution. Water pressure is never redistributed.

7.23 Bearing capacity safety

7.23.1 Analysis principles

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

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

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

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

Bearing capacity coefficients N_c N_d and N_b

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

The following values are adopted:

- N_{c0} N_{d0} N_{b0} according to DIN 4017
- v_c v_d v_b according to DIN 4017 (shape coefficients)

When entering the system data input of a length is principally of importance for the settlement analysis. The program consistently uses the values for a and b , including for calculation of the shape coefficients v_d and v_b for strip footings, as this delivers somewhat more favourable values.

$$v_d = 1 + 0.2 \cdot b/a \quad (\text{instead of } 1.0 \text{ for strip footings})$$
$$v_b = 1 + b/a \cdot \sin(\varphi) \quad (\text{instead of } 1.0 \text{ for strip footings})$$

- i_c i_d i_b according to DIN 4017 (load inclination coefficients)
- λ_c λ_d λ_b according to DIN 4017 (ground inclination coefficient)
- ξ_c ξ_d ξ_b according to DIN 4017 (base inclination coefficients)

- T_c T_d = depth coefficients according to the Danish standard (not permitted to DIN 4017)
 $T_c = T_d = 1 + 0.35 \cdot GS / b \leq 2.0$
 GS = footing base
 b = footing width

7.23.2 Determining the mean governing soil properties

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

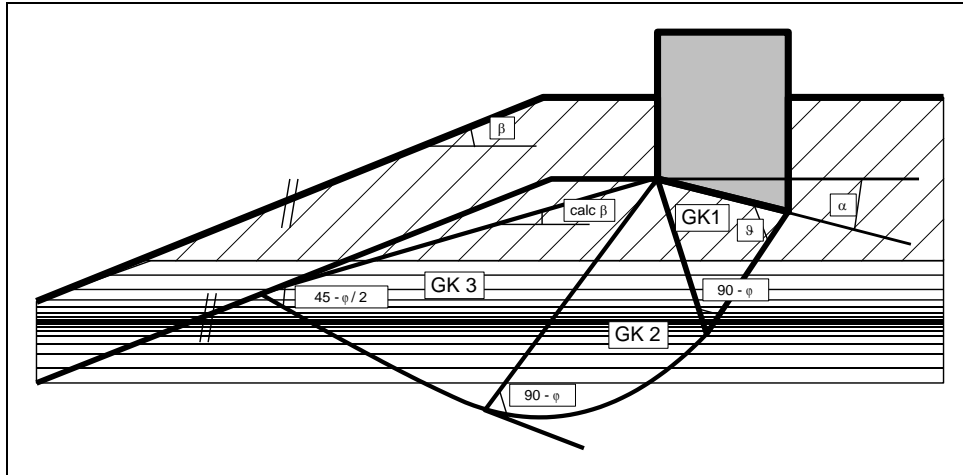


Figure 16 Logarithmic spiral

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

$$\varphi = \Sigma \tan \varphi_i \cdot l_i / \Sigma l_i$$

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

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

l_i = length within individual layer

A_i = area of individual layer

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

7.24 Settlements

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

The limiting depth can be defined in three different ways:

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

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

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

7.25 Sliding safety

The sliding safety is calculated using

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

V_k = characteristic vertical force

H_d = design value of horizontal force

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

7.26 General stability

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

7.27 Hydraulic heave

7.27.1 Hydraulic heave safety using global safety factors

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

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

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

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

7.27.2 Utilisation factor (hydraulic heave) using partial safety factors

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

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

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

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

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

- μ_N = utilisation factor of layer N

Utilisation factors ≤ 1.0 mean that sufficient safety is given.

7.28 Buoyancy

7.28.1 Buoyancy safety using global safety factors

The buoyancy safety for each soil layer within the excavation is determined via a comparison of the soil weights to the water pressures at the respective layer bases. The self-weight 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.

7.28.2 Utilisation factor (buoyancy) using partial safety factors

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

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

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

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

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

- μ_N = utilisation factor of layer N

Utilisation factors ≤ 1.0 mean that sufficient safety is given.

7.29 Verification of deep-seated stability

According to the EAB, R 44, the deep-seated stability of tied-back excavations must be verified. This primarily serves to determine the necessary anchor lengths. Verification uses the method described by Ranke/Ostermayer (*Bautechnik* 1968 (Construction Engineering), Issue 10). When **verifying deep-seated stability** each anchor is first investigated (including the influence of the remaining anchors on the slip plane). Compound slip planes, which are determined by connecting the end points of the anchors involved, are then analysed.

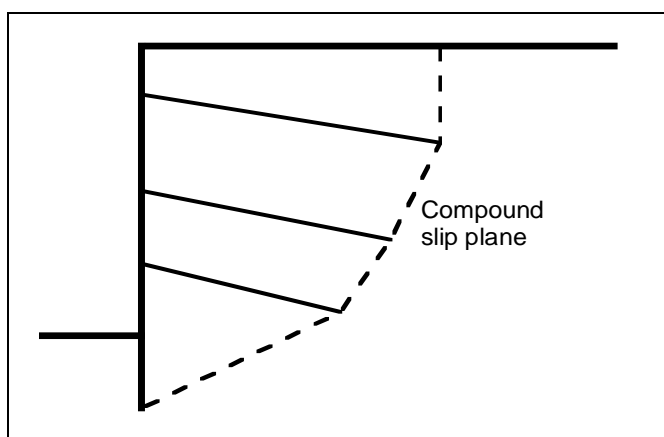


Figure 17 Compound "deep slip planes"

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

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

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

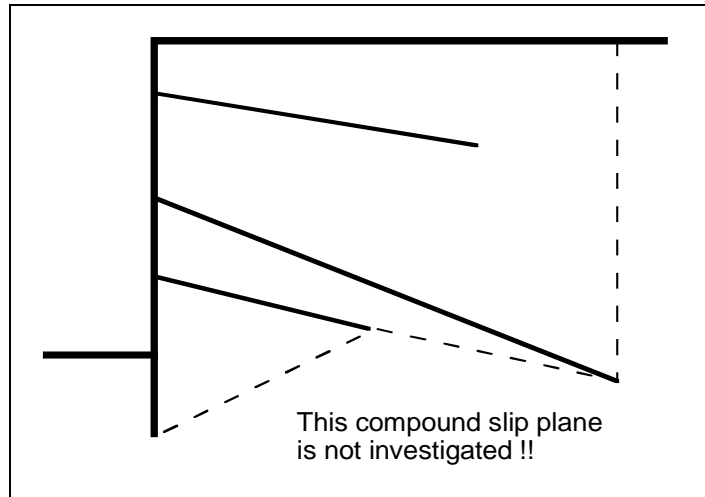


Figure 18 Compound "deep slip plane", which is not investigated

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

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

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

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

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

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

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

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

7.30 Verification of unreinforced concrete

7.30.1 Analysis to DIN 1045 (old)

To verify the unreinforced concrete the following data is required:

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

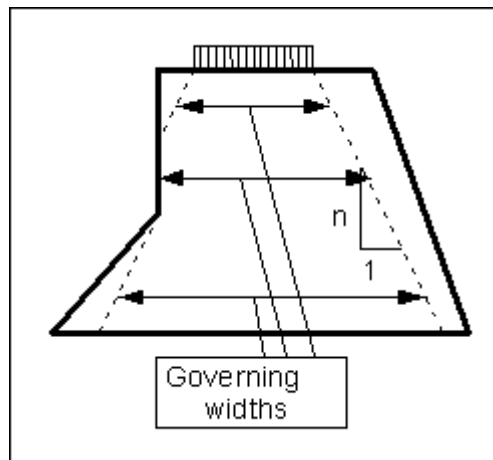


Figure 19 n-value and width for concrete verification

Verification of the unreinforced concrete is performed using the following relationship to DIN 1045:

$$\text{allow. } N = \kappa \cdot 1 / 2,1 \cdot b \cdot \text{beta}(r) \cdot (1,0 - 2 \cdot e) \text{ [kN/m]}$$

b = governing width of concrete

e = eccentricity with regard to governing width

κ = reduction for slenderness of structural element

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

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

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

λ = slenderness (for rectangular sections = $s_k / b \cdot \sqrt{12}$)

s_k = buckling length

The buckling length s_k in systems without anchors or struts results approximately from twice the separation between the top of the wall and half of the embedment depth. For walls that are horizontally restrained near the top by other engineering means (e.g. adjoining floor slabs), the single separation may be adopted. The horizontal restraint can be specified using the "**Wall top kept horizontal**" check box. In system which includes anchors or struts, s_k is acquired from twice (once) the separation between the top of the wall and the upper row of anchors (struts), or the separation between the individual anchors (struts). The larger of these values is adopted. The slenderness λ is recalculated for every section using the respective governing concrete width.

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

$$g = \text{work. } N / \text{allow. } N$$

Verification of unreinforced concrete can also be performed using a partial section.

7.30.2 Analysis to EC 2/DIN 1045-1

The following verification is required to *EC 2/DIN 1045-1*:

$$N_{R,d} = - (b \cdot h \cdot f_{m,d} \cdot \varphi)$$

where

$$\varphi = 1.14 (1 - 2e_{tot}/h) - 0.02 l_0/h$$

and

$$0 \leq \varphi \leq 1 - 2e_{tot}/h$$

Where:

$N_{R,d}$ = design value of the acceptable longitudinal force;

b = width of section;

h = thickness of section;

$f_{m,d}$ = design value of concrete compressive strength

φ = coefficient for considering the second order theory effects on the bearing capacity of compression members of unreinforced concrete in immovably braced structures;

e_{tot} = total eccentricity = $e_0 + e_a + e_\varphi$

e_0 = load eccentricity according to first order theory, where necessary taking the actions from adjoining floors into consideration (e.g. bending moments as a result of fixity, transferred from a slab to the wall), as well as the horizontal actions of wind e_a , and the additional unintentional eccentricity as a result of geometric imperfections. If more precise data is not available, e_a may be assumed at $= 0.5 l_0/200$.

e_φ = eccentricity resulting from creep; generally, the e_φ component can be disregarded.

7.30.3 Analysis to DIN 4093

7.30.3.1 Simplified analysis concept

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

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

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

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

W = section modulus

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

A = cross-sectional area

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

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

b = width of cross-section

e = eccentricity

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

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

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

$Q_{E,d}$ = design value of the shear force

A_v = cross-sectional area under shear loading

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

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

α = coefficient for considering a long-term effect = 0.85

γ_m = partial safety factor for the compressive strength

(= 1.5 for DS-P and DS-T

= 1.3 for DS-A)

7.30.3.2 Analysis concept using principal stresses

In the analysis using principal stresses the design values of the load may not exceed the stress states defined as allowable as shown in Figure 1 taken from DIN 4093.

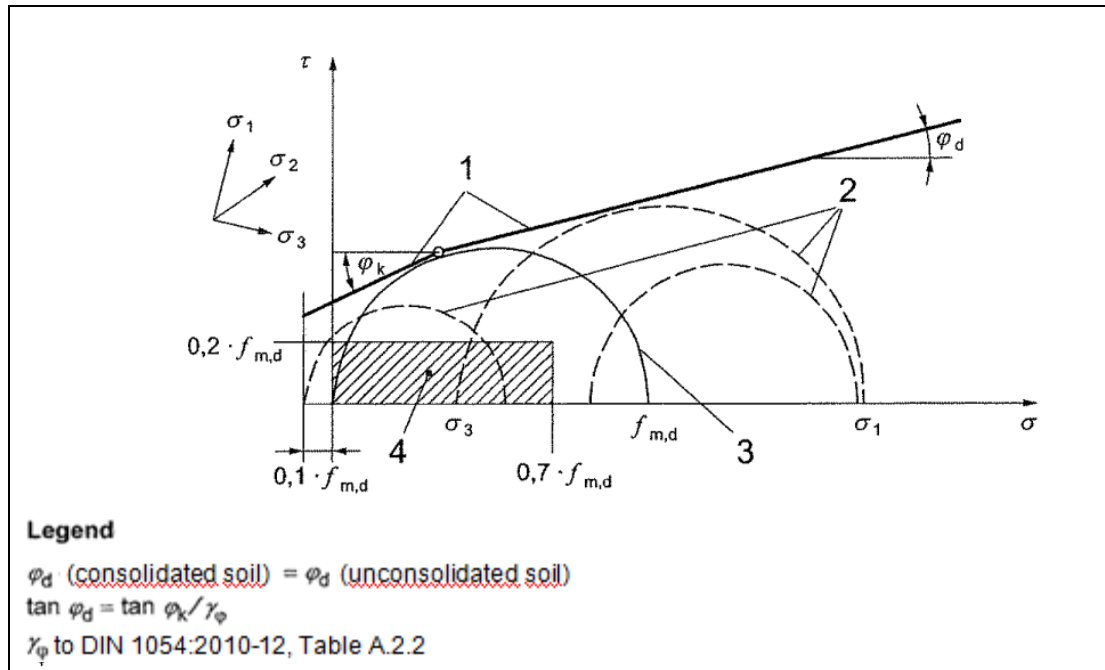


Figure 20 DIN 4093:2012-08: Figure 1

The normal stress σ_y is distributed linearly across the cross section. The shear stress τ has a parabolic graph. The stress σ_x parallel to the cross section is approximately 0.0.

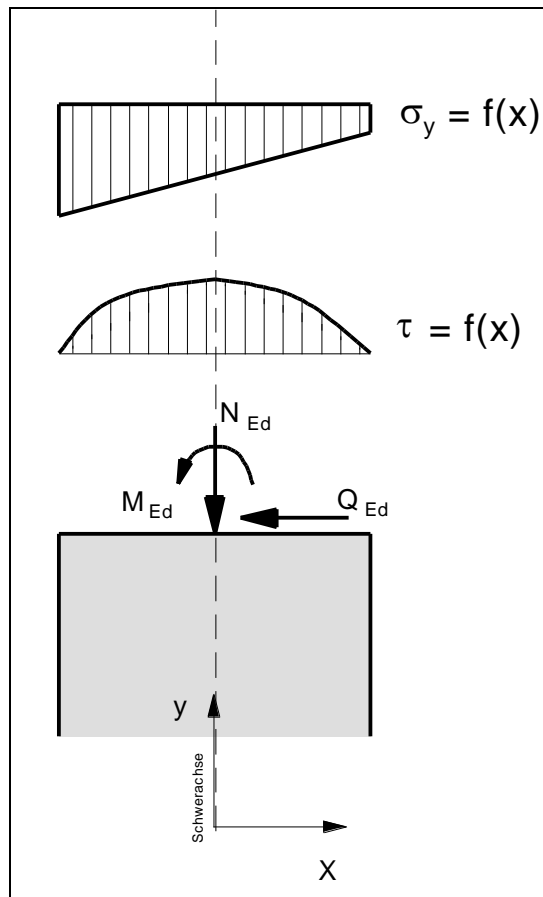


Figure 21 DIN 4093:2012-08: Stresses in cross section

Using the variables σ_y , σ_x and τ the principal stresses σ_1 and σ_3 can be calculated for any point on the cross section

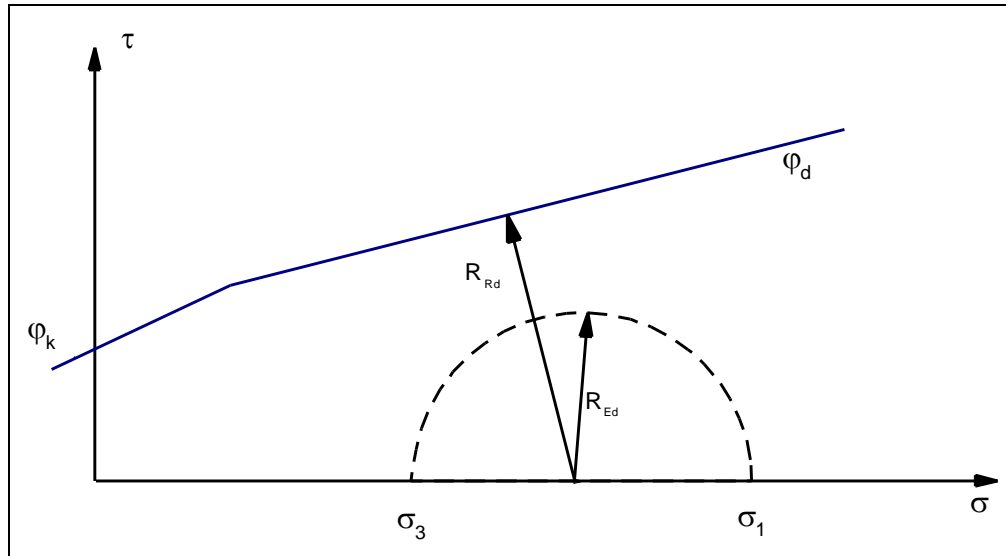


Figure 22 DIN 4093:2012-08: Utilisation factor

Figure 22 shows Mohr's circle for a major principal stress. The utilisation factor μ can be calculated using the radius R_{Ed} of this circle and the distance R_{Rd} from the centre of the circle to the boundary line.

$$\mu = R_{Ed}/R_{Rd}$$

7.30.3.3 Slenderness

DIN 4093:2012-08 stipulates that the slenderness $\lambda \leq 15$, if no buckling safety analysis is performed. EN 1993-1-1:2010-12 states that buckling safety analysis may be dispensed with if the following condition is adhered to:

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

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

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

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

If the conditions:

- a) slenderness $\lambda \leq 15$ and
- b) $N_{Ed}/N_{cr} \leq 0.10$

are not adhered to, an analysis of the unreinforced concrete compliant with DIN 1045-1 may be performed instead. The relationships in DIN 1045-1 include a buckling safety analysis.

7.31 Construction phases

If you are investigating a number of advancing and retreating phases, **GGU-UNDERPIN** allows you to display a 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;
- sum of deformations.

You can skip through the individual phases using the mouse.

8 Description of menu items

8.1 File menu

8.1.1 "New" menu item

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

New dataset

Project identification

Standard:

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

☐ Partial safety factors (DIN 1054:2005)

☐ Global safety factors (DIN 1054 old)

Analysis of unreinforced concrete

☐ To DIN 4093:2012-08 ?

☒ DIN 4093:2012-08 using principal stresses ?

☒ To EC 2/DIN 1045 ?

Shear analysis to DIN 4093 simplified ?

General

☐ Show excavation on right

☐ Do not investigate overturning

☐ Use absolute heights Design: mHNN

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

Equivalent wall

☐ Equivalent wall compliant with EAB 6th Edition ?

☐ Use wall friction for bounded surcharges

Wall friction angle delta,k/phi = 0.50 ?

OK Cancel

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

The default analysis of the unreinforced concrete is performed compliant with EC 2/DIN 1045-1 (see Section 7.30.2). By default the shear analysis is performed compliant with DIN 4093 using the simplified equation $\tau_{E,d} = 1.5 \cdot Q_{E,d}/A_v \leq 0.2 \cdot f_{m,d}$. However, you may also calculate without a shear analysis or, when applying EC 2, perform shear analysis using principal stresses compliant with DIN 4093.

Alternatively, you can analyse the unreinforced concrete based completely on DIN 4093:2012-08 (see Section 7.30.3). This then also includes the shear analysis using principal stresses. Depending on the system, analysis without principal stresses may return more favourable values. You may therefore also deactivate this analysis.

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

If you select the **"Use absolute heights"** check box, you can enter all depths and heights in **m AD** (heights are positive upwards). If this check box is not selected, the top of the wall is assumed to have a height of **0.0** and all further entries, depth of soil layers, for example, are positive downwards.

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

In the lower group box, you can select the new regulations concerning the equivalent wall compliant with EAB 6th Edition. The new regulations are described in "Bautechnik Issue 8 (2020)".

If the safety concept has been altered using the **"File/New"** dialog box, the dialog box for specifying either the safety factors or the partial factors always opens after leaving the dialog box by clicking **"OK"**. These dialog boxes can also be accessed via the menu items **"Editor 1/Verifications/ Safety factors"** (see Section 8.2.15) or **"Editor 1/Verifications/Partial factors"** (see Section 8.2.16). Verifications and safety factors can be modified at any time using the above menu items.

8.1.2 "Load" menu item

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

8.1.3 "Save" menu item

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

8.1.4 "Save as" menu item

You can save data entered during program use to an existing file or to a new file, i.e. using a new file name. For reasons of clarity, it makes sense to use ".ufg" 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, ".ufg" will be used automatically.

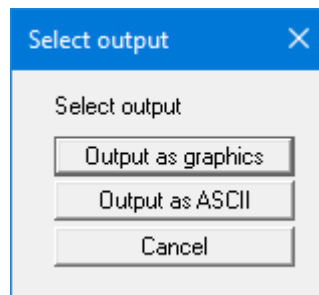
If the current system has been analysed at the time of saving, the analysis results are saved in the file. This is especially necessary if you want to use the results in this file for *predeformations* on other systems (see Section 7.20), or if you want to subsequently generate summaries of certain *construction phases* (see Section 7.31). If your current system represents an intermediate construction stage, it may be expedient to enter a further description of the file in "**Project identification**", in the "**File/New**" or "**Editor 1/Analysis options**" dialog box (see Section 8.1.1).

8.1.5 "Print output table" menu item

8.1.5.1 *Selecting the output format*

You can have a table printed containing the current analysis results. The results can be sent to the printer or to a file (e.g. for further editing in a word processor). The output contains all information on the current state of analysis, including the system data.

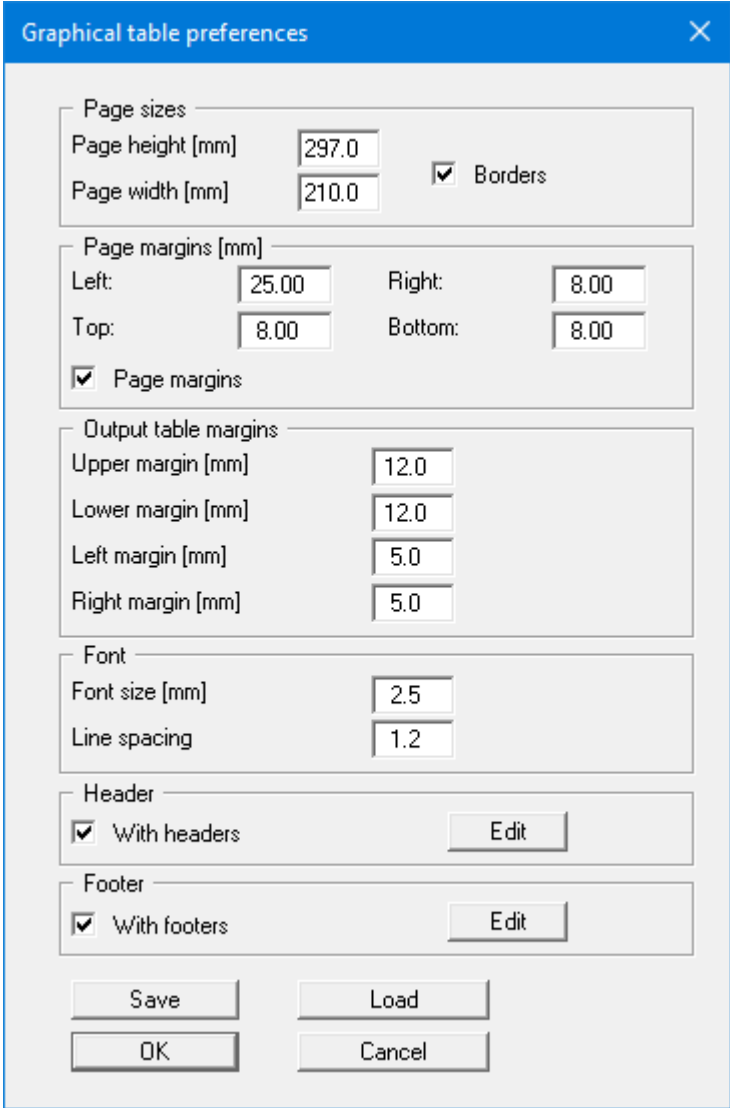
You have the option of designing and printing the output table as an annex to your report within the **GGU-UNDERPIN** application. To do this, select "**Output as graphics**" from the following options.



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

8.1.5.2 Button "Output as graphics"

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



The dialog box titled "Graphical table preferences" contains several sections for configuring the output of graphical tables. The "Page sizes" section includes input fields for "Page height [mm]" (297.0) and "Page width [mm]" (210.0), along with a checked "Borders" checkbox. The "Page margins [mm]" section has input fields for "Left" (25.00), "Right" (8.00), "Top" (8.00), and "Bottom" (8.00), with a checked "Page margins" checkbox. The "Output table margins" section includes input fields for "Upper margin [mm]" (12.0), "Lower margin [mm]" (12.0), "Left margin [mm]" (5.0), and "Right margin [mm]" (5.0). The "Font" section has input fields for "Font size [mm]" (2.5) and "Line spacing" (1.2). The "Header" section has a checked "With headers" checkbox and an "Edit" button. The "Footer" section has a checked "With footers" checkbox and an "Edit" button. At the bottom, there are buttons for "Save", "Load", "OK", and "Cancel".

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

Footers [X]

Height + Font size

Height Footers [mm]

Font size [mm]

☒ With frame

Texts

☐ Left justified ☒ Centered ☐ Right justified

1

2

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

Annex 4.1 Output table
Page 1 of 3

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

Select page [X]

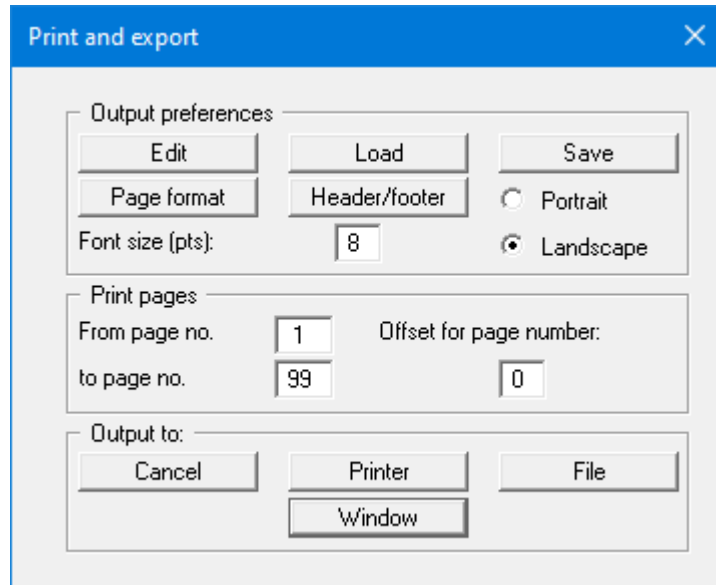
Current page = 5

New page:

☒ To normal view

8.1.5.3 Button "Output as ASCII"

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



In the dialog box you can define output preferences.

- **"Output preferences"** group box
Using the **"Edit"** button the current output preferences can be changed or a different printer selected. Using the **"Save"** button, all preferences from this dialog box can be saved to a file in order to have them available for a later session. If you select **"GGU-UNDERPIN.drk"** as file name and save the file in the program folder (default), the file will be automatically loaded the next time you start the program.

Using the **"Page format"** button you can define, amongst other things, the size of the left margin and the number of lines per page. The **"Header/footer"** button allows you to enter a header and footer text for each page. If the **"#"** symbol appears within the text, the current page number will be entered during printing (e.g. **"Page #"**). The text size is given in **"Pts"**. You can also change between **"Portrait"** and **"Landscape"** formats.
- **"Print pages"** group box
If you do not wish pagination to begin with **"1"** you can add an offset number to the check box. This offset will be added to the current page number. The output range is defined using **"From page no."** **"to page no."**.
- **"Output to:"** group box
Start output by clicking on **"Printer"** or **"File"**. The file name can then be selected from or entered into the box. If you select the **"Window"** button the results are sent to a separate window. Further text editing options are available in this window, as well as loading, saving and printing.

8.1.6 "Export" menu item

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

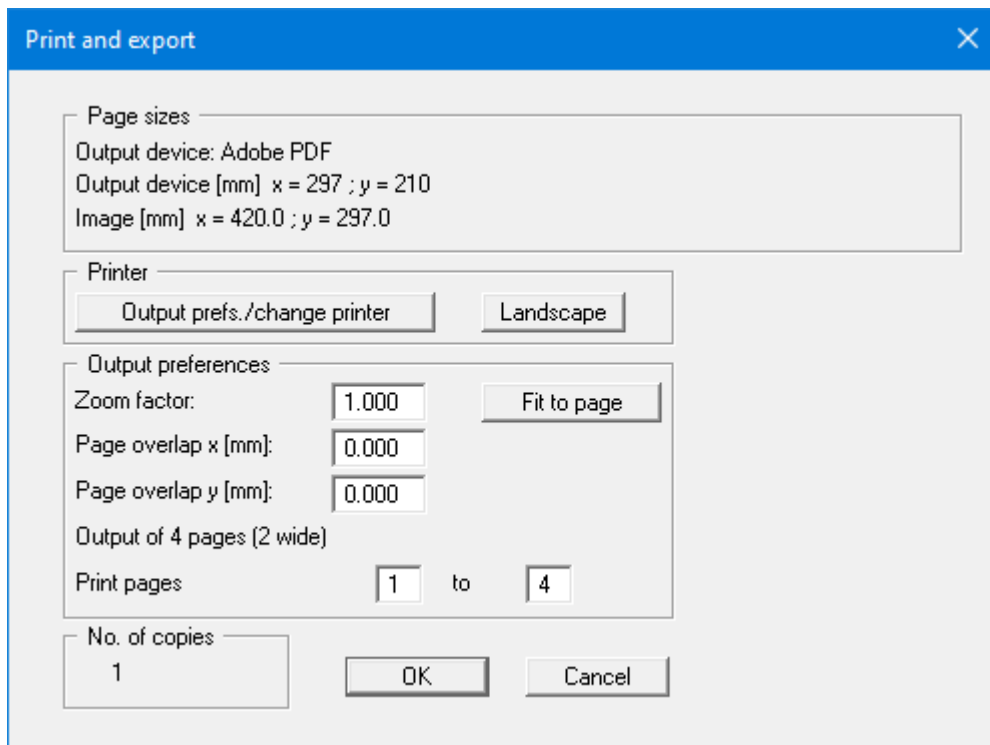
8.1.7 "Output preferences" menu item

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

8.1.8 "Print and export" menu item

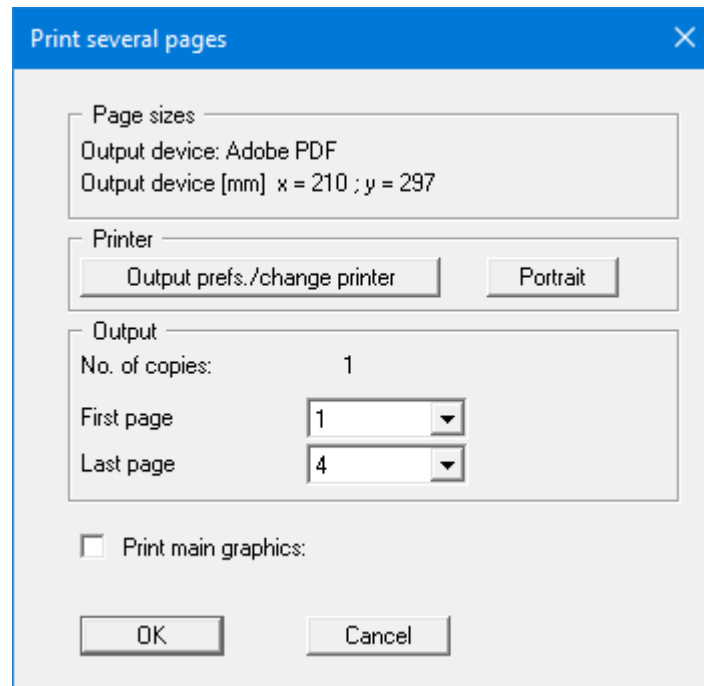
You can select your output format in a dialog box. You have the following options:

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




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

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



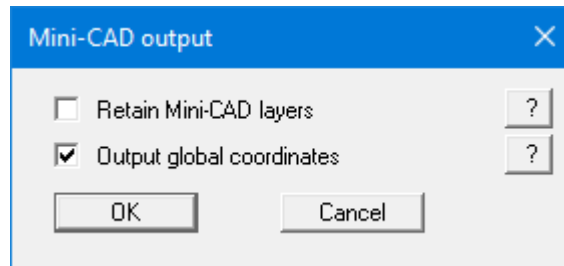
Here, you can select the table pages to be printed. In order to achieve output with a zoom factor of 1 (button "**Fit in automatically**" is deactivated), you must adjust the page format to suit the size format of the output device. To do this, use the dialog box in "File/Print output table" button "Output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

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

If you select the "Copy/print area" tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see "**Tips and tricks**", Section 9.3).

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

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



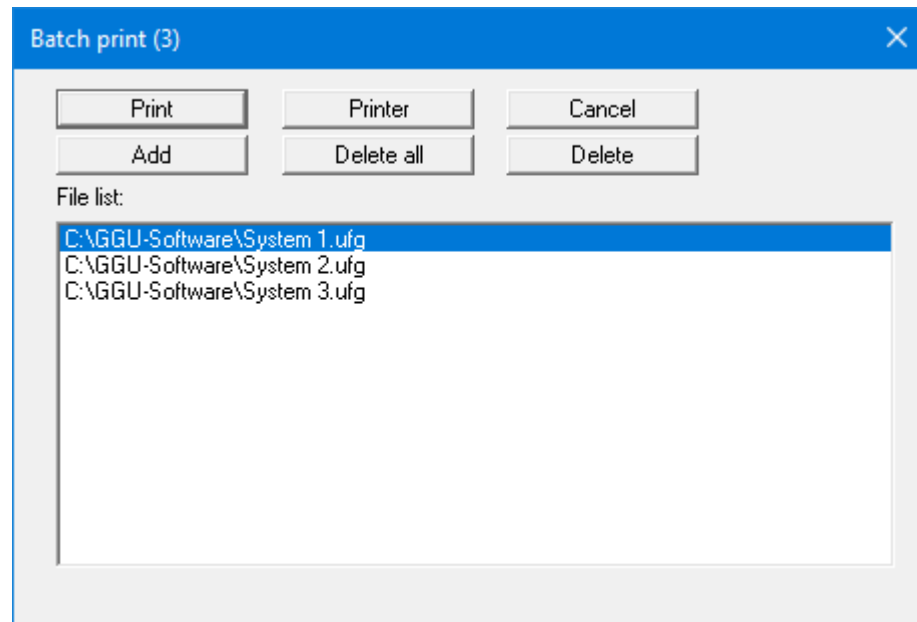
If the **"Retain Mini-CAD layers"** check box is activated, the layer allocations for any existing **Mini-CAD** elements are saved. Otherwise, all **Mini-CAD** elements are saved on Layer 1 and are also inserted into Layer 1 in other **GGU** programs via the **"Load"** function in the **Mini-CAD** pop-up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

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

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

8.1.9 "Batch print" menu item

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



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

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

8.1.10 "Exit" menu item

After a confirmation prompt, you can quit the program.

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

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

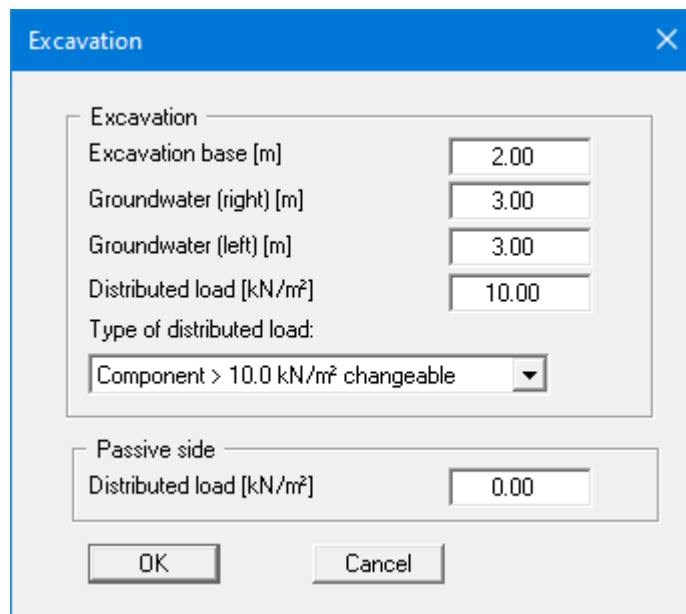
8.2 Editor 1 menu

8.2.1 "Analysis options" menu item

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

8.2.2 "Excavation" menu item

A dialog box opens for defining the excavation:



The image shows a software dialog box titled "Excavation" with a blue header bar and a close button (X) in the top right corner. The dialog is divided into two main sections. The upper section, titled "Excavation", contains five input fields: "Excavation base [m]" with a value of 2.00, "Groundwater (right) [m]" with a value of 3.00, "Groundwater (left) [m]" with a value of 3.00, "Distributed load [kN/m²]" with a value of 10.00, and a dropdown menu for "Type of distributed load:" currently set to "Component > 10.0 kN/m² changeable". The lower section, titled "Passive side", contains one input field: "Distributed load [kN/m²]" with a value of 0.00. At the bottom of the dialog are two buttons: "OK" and "Cancel".

In the upper group box, enter the depth of the excavation level and the groundwater levels. If you want to use the program to analyse a footing in which it is not necessary to consider support for a terrace, enter **0.01** for the excavation base.

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

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

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

If you are working with the *partial safety factors* used in DIN 1054 (new), 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.

8.2.3 "Body (general)" menu item

Enter the loads and geometry of any footings. A more detailed description can be found in **"First steps using worked examples"** in Section 6.4.

When using the *partial safety factor concept*, the loads are entered separately as **"Permanent (g)"** and **"Changeable (q)"** loads (see Section 6.4).

8.2.4 "Body (geometry)" menu item

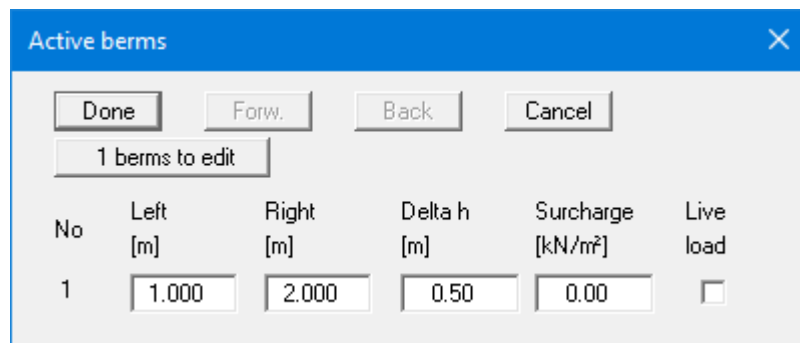
Enter the geometry of the underpinning body. A more detailed description can be found in **"First steps using worked examples"**, Section 6.5.

8.2.5 "Concrete" menu item

Enter the properties for the concrete and for verification of the unreinforced concrete. A more detailed description can be found in **"First steps using worked examples"** (Section 6.6) and in the **"Theoretical principles"** (Section 7.30).

8.2.6 "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
1	1.000	2.000	0.50	0.00	<input type="checkbox"/>

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

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

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

8.2.7 "Berms (passive side)" menu item

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

8.2.8 "Soils" menu item

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

	Designation	Base	gam	gam'	phi	c(a)	c(p)	d(a)/phi	d(p)/phi	k [m/s]	k [m/s]	Es
		[m]	[kN/m ²]	[kN/m ²]	[°]	[kN/m ²]	[kN/m ²]	[-]	[-]	Left	Right	[MN/m ²]
1	Sand	6.50	19.0	10.0	32.5	0.0	0.0	0.000	0.000	1.00E-4	1.00E-4	10.0

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

Layer depths are always with reference to the top of the wall, or are absolute heights (m AD), if this was selected in the initial dialog box of the "**File/New**" menu item. It is only necessary to enter permeabilities if hydrostatic pressures must be considered, and any hydraulic gradients, using flow conduits (see Section 7.7.2).

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

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

You can also use this function to eliminate a soil from the table.

Simply assign the soil to be eliminated a greater layer depth and then click the "**Sort**" button. The corresponding soil is now the last soil in the table and can be *deleted* by reducing the number of soils.

8.2.9 "Type of earth pressure" menu item

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

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 [-]

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

8.2.10 "Active earth pressure" menu item

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

Active earth pressure

Active earth pressure after:

☒ DIN 4085

☐ Culmann ?

☐ User-defined (not recommended) ?

Equivalent ep coefficient

☒ Use equivalent ep coefficient

Equivalent kah [-] 0.200

☒ Equivalent ep coefficient with phi = 40°

☒ Minimum earth pressure and distributed load compliant with EAB ? EB 6

☒ Suppress negative active earth pressures

☒ Additional ordinates at 0.5 m spacing

Surcharge factor for active earth pressure

Surcharge factor [-] 1.0000 ?

OK Cancel

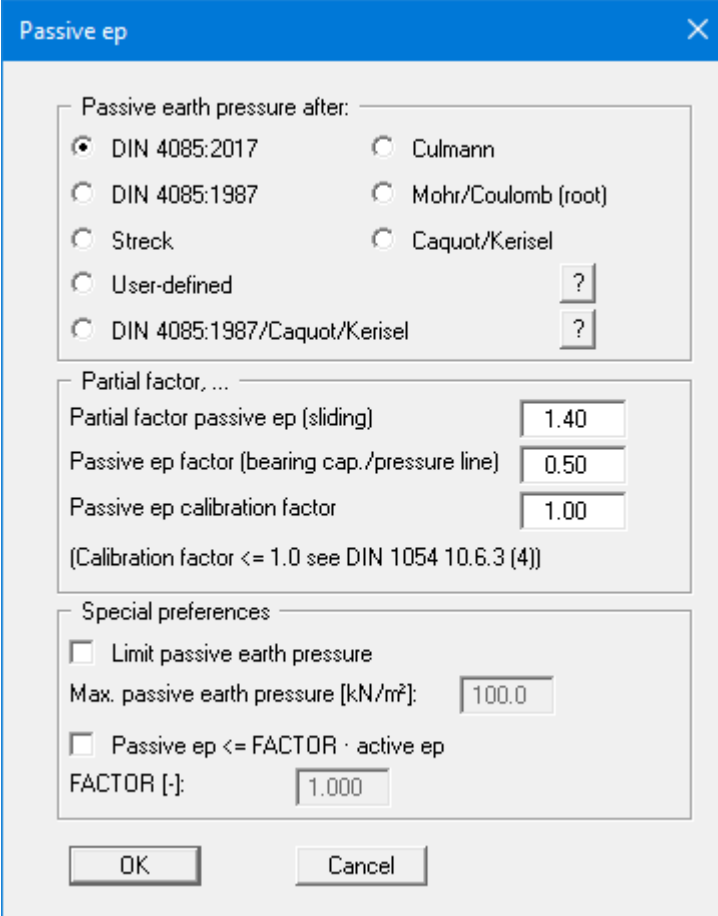
In the upper group box, you specify the type of active earth pressure calculation. If the "**Culmann**" check box is activated the active earth pressure is calculated after Culmann using a slice method with straight slip surfaces.

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

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

8.2.11 "Passive earth pressure" menu item

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

The image shows a software dialog box titled "Passive ep" with a blue header bar and a close button (X) in the top right corner. The dialog is divided into three main sections. The first section, "Passive earth pressure after:", contains six radio button options: "DIN 4085:2017" (selected), "DIN 4085:1987", "Streck", "User-defined", "DIN 4085:1987/Caquot/Kerisel", "Culmann", "Mohr/Coulomb (root)", and "Caquot/Kerisel". The last three options have a question mark icon to their right. The second section, "Partial factor, ...", contains three input fields: "Partial factor passive ep (sliding)" with a value of 1.40, "Passive ep factor (bearing cap./pressure line)" with a value of 0.50, and "Passive ep calibration factor" with a value of 1.00. Below these is a note: "(Calibration factor <= 1.0 see DIN 1054 10.6.3 (4))". The third section, "Special preferences", contains two checkboxes: "Limit passive earth pressure" (unchecked) and "Passive ep <= FACTOR · active ep" (unchecked). The first checkbox has an input field for "Max. passive earth pressure [kN/m²]" with a value of 100.0. The second checkbox has an input field for "FACTOR [-]" with a value of 1.000. At the bottom of the dialog 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.

According to R 14 and R 19 of the EAB, the location of the earth pressure resultant for a free earth support can, under certain conditions, be adopted higher than for classical earth pressure theory. If you analyse using the *global safety factors*, specify the necessary settings in the central section of the dialog box. The value of the safety factor for the passive earth pressure can also be specified here.

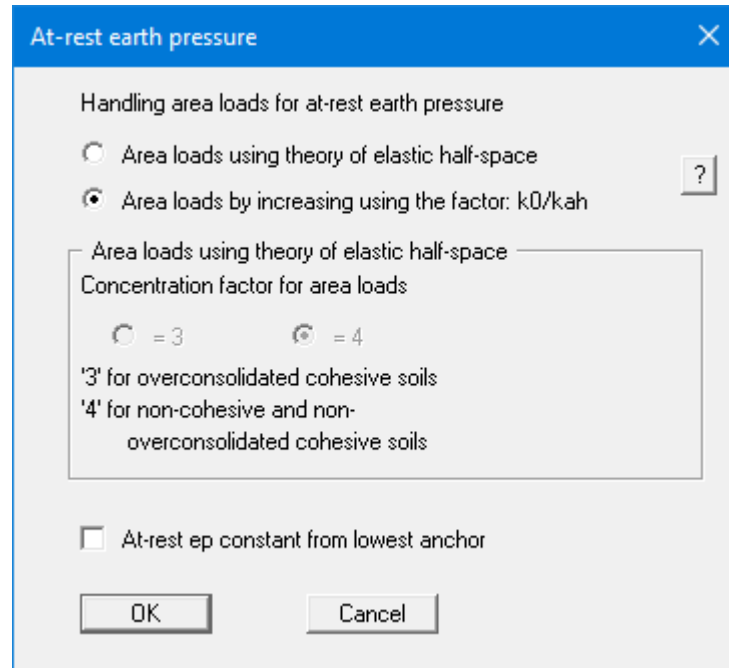
If you have selected *partial safety factors*, enter the partial factor for passive earth pressure in the dialog box and the calibration factor in accordance with the information in DIN 1054:2005/EC 7.

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

8.2.12 "At-rest earth pressure" menu item

Using the at-rest earth pressure function, the area loads are calculated by increasing by the factor k_0/k_{ah} in compliance with DIN 4085 Section 6.4.3.

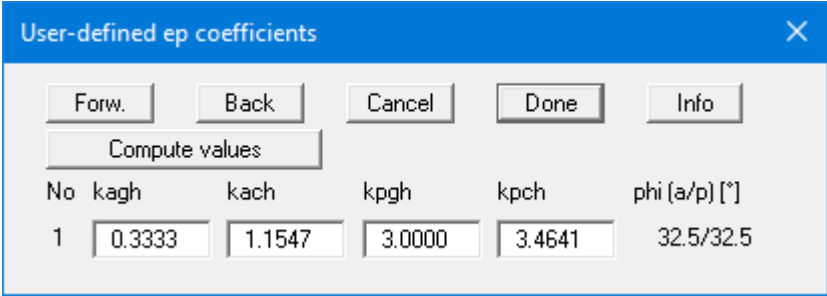
Alternatively, it is possible to determine the demands placed on the wall by area loads according to the theory of elastic half-space. The required concentration factor is specified in the following dialog box, if the upper check box is activated.



According to the EAB, R 23 (Paragraph 2), for underpinning 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.

8.2.13 "User-defined earth pressure coefficients" menu item

If you want to work with user-defined earth pressure coefficients instead of with those computed by the program you can enter them here. Enter the coefficients for horizontal ground. If necessary, **GGU-UNDERPIN** will convert them for sloping ground using equation values for k_{ah0} und $k_{ah\beta}$ (see "**Theoretical principles/Berms**", Section 7.8).



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

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

The earth pressure coefficients entered here are only adopted if the "**User-defined**" check box is activated in "**Editor 1/Active earth pressure**" and/or in "**Editor 1/Passive earth pressure**" menu items (see Sections 8.2.10 and 8.2.11).

8.2.14 "Groundwater" menu item

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

The dialog box is titled "Water pressure approach + hydraulic gradient" and contains the following settings:

- Horizontal water pressure approach:**
 - ☒ Classical
 - ☐ With flow conduits
- Hydraulic gradients i to be considered on the:**
 - ☐ Active side
 - ☒ Passive side
 - ☒ Incl. bearing capacity
- According to EAB, R 63, the hydraulic gradient on the passive side must be considered. It may be ignored on the active side.

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

☒ Adhere to

Buttons: OK, Cancel

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

8.2.15 "Verifications/Safety factors" menu item

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

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

- Hydraulic heave:** FOS = 1.50, ☒ Verify
- Buoyancy:** FOS = 1.10, ☒ Verify
- 'Deep-seated stability':** FOS = 1.50
- Sliding safety:** FOS = 1.50, ☒ Verify, ☒ Verify to DIN 1054, ☐ Sliding safety without passive ep (Info), ☐ Friction angle in base plane [°] = 30.00
- Bearing capacity:** FOS = 2.00, ☒ Verify, ☐ Bearing capacity with depth factor (Info), ☒ Check and correct 5° condition, Decrement [°]: 0.10 (Info), ☒ Top of bearing cap. failure body parallel to slope

Buttons: OK, Cancel

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

8.2.16 "Verifications/Partial factors" menu item

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

Verifications/Partial factors preferences

Actions/Resistance

Design situation: DIN 1054: DS-P

Permanent actions: 1.35

Permanent actions (at-rest ep): 1.20

Changeable actions: 1.50

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

Passive earth pressure (sliding): 1.40

Passive ep factor (bearing cap./pressure line): 0.50 ?

Passive ep calibration factor: 1.00 ?

Bearing capacity

☒ Verify ☒ To new DIN 4017

gamma(bearing capacity): 1.40

☐ Bearing capacity with depth factor ?

☒ Check and correct 5° condition

Decrement [°]: 0.10 ?

☒ Top of bearing cap. failure body parallel to slope

Sliding

☒ Verify

gamma(sliding): 1.10

☐ Friction angle in base plane [°] = 30.00 ?

Serviceability verification

☐ Sliding safety without passive ep ?

☒ Serviceability verification ?

EQU analysis

☒ Verify

gamma(G,dst): 1.10

gamma(G,stb): 0.90

gamma(Q,dst): 1.50

Default values

To DIN 1054:2010 To ÖNORM EN 1997-1

OK Cancel

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

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

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

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

8.2.17 "Buoyancy + Hydraulic heave" menu item

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

Partial safety factors

☒ Verify 'hydraulic heave'

☒ Verify 'base buoyancy'

Weight

Favourable permanent actions:

DS-P: 0.95/DS-T: 0.95/DS-A: 0.95/DS-E: 1.00

Buoyancy (pwp)

Unfavourable permanent actions:

DS-P: 1.05/DS-T: 1.05/DS-A: 1.00/DS-E: 1.00

Hydraulic heave

Flow force:

Favourable subsoil: DS-P: 1.45/DS-T: 1.45/DS-A: 1.25/DS-E: 1.00

Unfavourable subsoil: DS-P: 1.90/DS-T: 1.90/DS-A: 1.45/DS-E: 1.00

Default values

If you press the "**Info**" button, you will see further information on the subsoil situation (favourable/unfavourable).

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

8.2.18 "Seismic effects" menu item

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

Seismic effects [X]

Consideration of seismics

- ☒ Use DIN EN 1998-5 Dec. 2010 [?]
- ☐ Horizontal loading from rigid structure [?]
- ☒ Horizontal loading from $k_h \cdot$ 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) [-]: [Info]

(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]

8.3 Editor 2 menu

8.3.1 "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 on the active side into consideration, define them here.

No	top [m]	bottom [m]	e (top) [kN/m²]	e (bottom) [kN/m²]	Live
1	1.000	2.000	10.00	20.00	<input type="checkbox"/>

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

8.3.2 "Area 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]	Type	Live
1	10.00	0.00	0.000	1.000	0.000	Triangle (max. top)	<input type="checkbox"/>

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

Use the "**Loads with limited plan dimensions**" button, to define such loads (see also Section 7.10):

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

Whereby: $p(\text{new}) = p \cdot b / (2 \cdot a + b)$
 b = width parallel to wall
 Piling Handbook 1977: a = distance of front of surcharge from the wall
 DIN 4085:2017-08: a = distance of rear of surcharge from the wall

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

p(v) [kN/m²]: 100.000
 p(h) [kN/m²]: 0.000
 x(left) [m]: 1.000
 x(right) [m]: 5.000
 b [m]: 0.500
 Depth [m]: 1.000
 No. of subsections: 4
 Type of load: Rectangle
☐ Delete current loads
 Analyse compliant with: DIN 4085:2017-08

8.3.3 "Bounded surcharges" menu item

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

No	p [kN/m²]	x(left) [m]	Depth [m]	Live
1	0.00	0.000	0.000	<input type="checkbox"/>

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

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

8.3.4 "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
----	-----------------	----------------	-----------------	--------------	------

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

Double-bounded surcharges consist of 2 bounded surcharges with opposing signs. Whether or not the linear component of the resulting earth pressure (see Figure 10 in Section 7.10) may be adopted at its full value for the bounded surcharge with the negative sign has not yet been conclusively clarified. This can be influenced using the control parameter:

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

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

8.3.5 "Action boundary conditions" menu item

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

Point loads

Done Forw. Back Cancel

1 point loads to edit

Moment positive clockwise
Horizontal force positive to right
Vertical force positive downwards

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

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

8.3.6 "Displacement boundary conditions" menu item

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

Displacement boundary conditions

Forw. Back Cancel Done

1 displacement BCs to edit

wx positive to the right
wy positive downwards
phi positive clockwise

No	Depth [m]	Size [m or radian]	Rotation phi
1	2.00	0.00000	Rotation phi

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

8.3.7 "Anchors" menu item

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

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

It is necessary to enter the following information:

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

8.3.8 "Struts" menu item

Struts are defined in this dialog box.

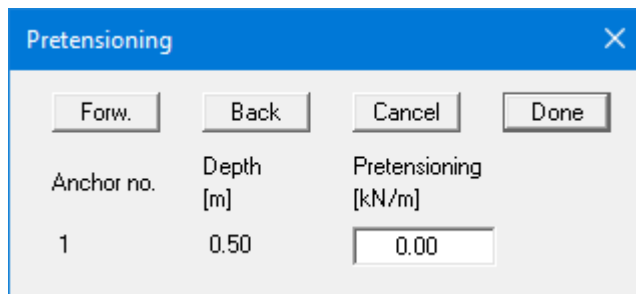
No.	Depth [m]	Inclin. [°]	Length [m]	EA [kN/m]	EI [kN·m²/m]	V load [kN/m²/m]	Hinge	No. of rods
1	0.50	5.0	8.00	2.100E+7	2.100E+7	0.00	<input checked="" type="checkbox"/> Yes	10

It is necessary to enter the following information:

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

8.3.9 "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

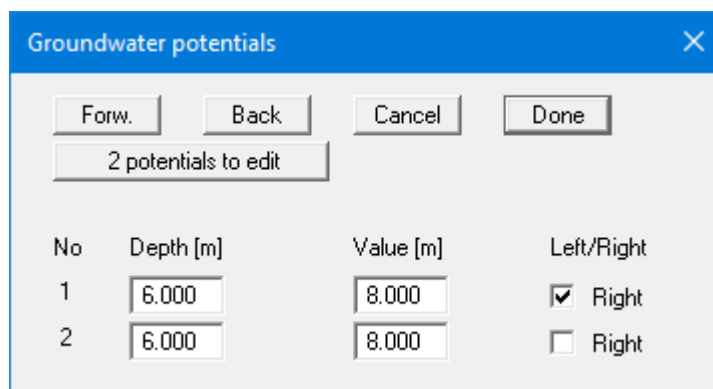
For anchors, pretensioning must be entered as positive. Faulty input is pointed out before analysis begins. From a structural perspective, the program inserts an action boundary condition equivalent in size and direction to the pretensioning force into the system at the appropriate depth.

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.

8.3.10 "Groundwater potentials" menu item

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



No	Depth [m]	Value [m]	Left/Right
1	6.000	8.000	<input checked="" type="checkbox"/> Right
2	6.000	8.000	<input type="checkbox"/> 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* (see Section 7.7.2).

8.3.11 "Settlements" menu item

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

Settlements

☒ Calculate settlements

Limiting depth

☐ Limiting depth with a fixed value of t
t [m b. FB]: 5.000

☐ Limiting depth with $x \cdot b'$
x: 2.000

☒ Limiting depth with p%
p: 20.0

☒ Settlements as a result of permanent loads ?

Preconsolidation [kN/m²]: 0.000 ?

Increments [m]: 0.100 ?

OK Cancel

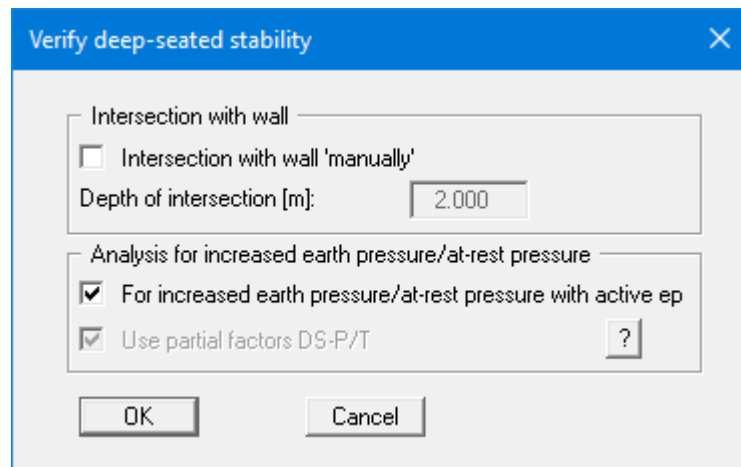
According to DIN 4019, settlement analysis can be terminated at a depth (limiting depth) in which the stress from soil self-weight and ground surcharges (overburden stress) is greater than 20% of the stresses caused by the body. The stress analysis for determining the limiting depth is performed for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

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

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

8.3.12 "Deep-seated stability" menu item

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



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

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

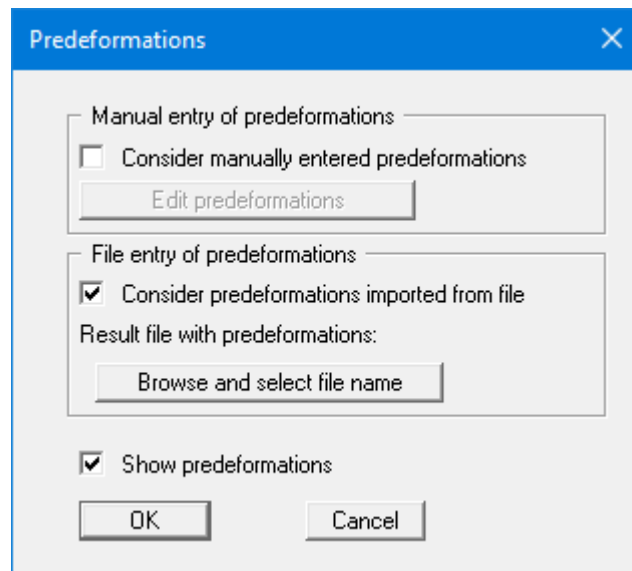
The partial factors for the actions to DS-P/T are around 5% greater than those to DS-T. Simplified, but conservatively, the anchor forces are multiplied by the factor '1.05'. R 44 does not always return higher utilisation factors.

8.3.13 "Predeformation info" menu item

You will see information on predeformations.

8.3.14 "Predeformation preferences" menu item

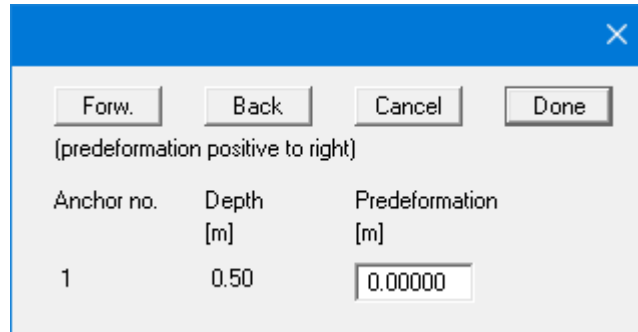
If predeformations from previous construction phases must be taken into consideration as boundary conditions (see Section 7.20), the necessary settings are made here.



Predeformations can be defined in two different ways:

- **Manual entry of predeformations**

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



Anchor no.	Depth [m]	Predeformation [m]
1	0.50	0.00000

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

- **File entry of predeformations**

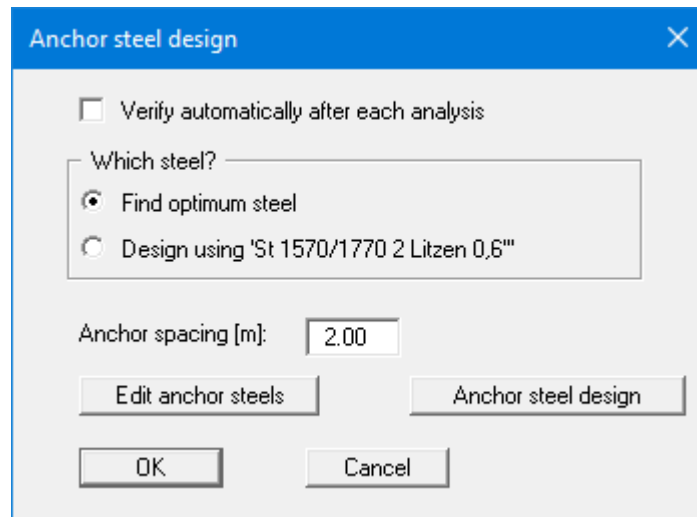
Alternatively, predeformation data can be automatically imported from an existing **GGU-UNDERPIN** file. Select "**Consider predeformations imported from file**" and "**Browse and select file name**". A dialog box will open for selecting the appropriate file, which must have been saved with the results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

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

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

8.3.15 "Anchor steel design" menu item

Using this menu item, you can specify preferences for designing the anchor steel and then 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 steels to edit**".

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

8.4 System menu

8.4.1 "Info" menu item

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

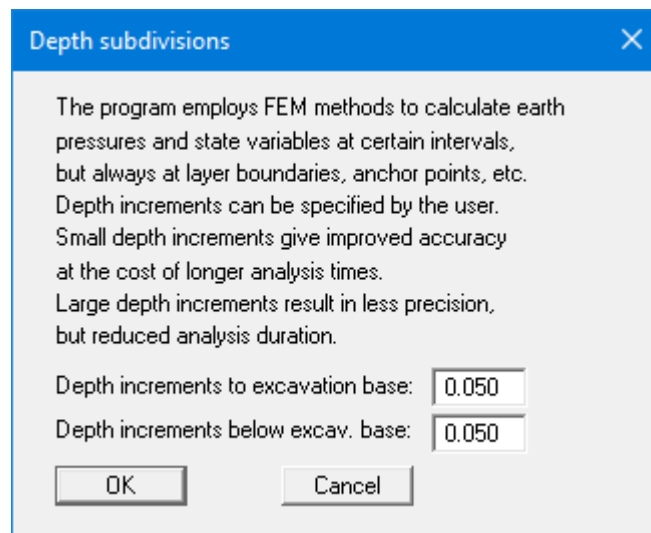
8.4.2 "Special preferences" menu item

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

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

8.4.3 "Depth subdivisions" menu item

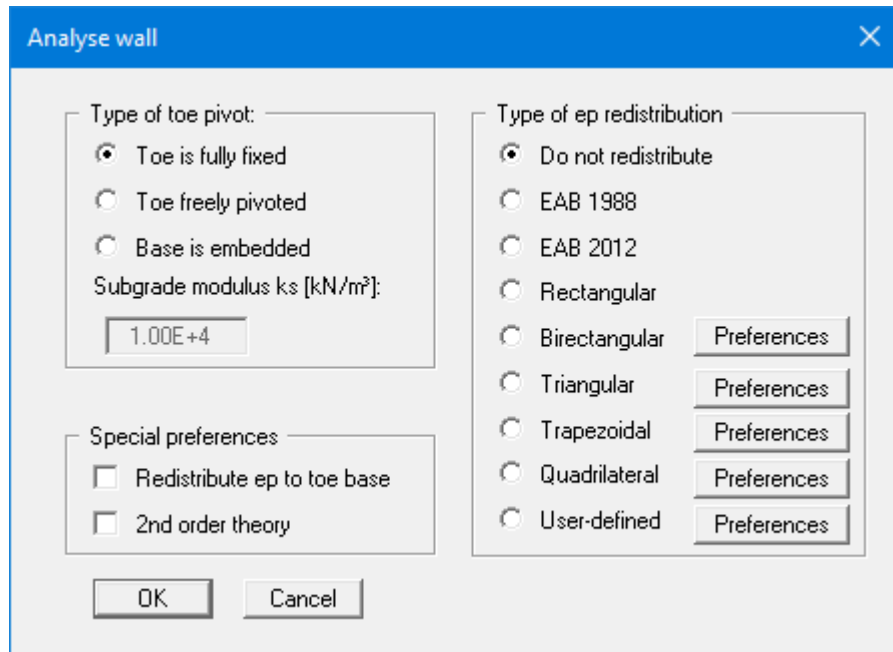
GGU-UNDERPIN uses the finite element method, which requires the system to be divided into a number of finite elements (rods) (see Section 7.15). You can specify the size of these depth increments for the region above and below the excavation base.



8.4.4 "Analyse" menu item

8.4.4.1 Start dialog box

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



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

Once analysis is complete, the results are presented in message boxes and then visualised graphically on the screen. The system data can be saved in a file, together with the results (menu item **"File/Save as"**, Section 8.1.4). If your system represents an intermediate construction stage, it may be expedient to enter a further description of the file in the **"Project identification"** field, found in the **"Editor 1/Analysis options"** or **"File/New"** dialog boxes (see Section 8.1.1).

8.4.4.2 "Type of toe pivot" group box

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

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

8.4.4.3 "Special preferences" group box

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

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

8.4.4.4 "Type of redistribution" group box

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

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

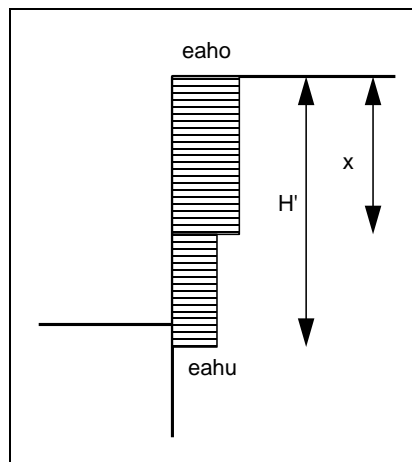


Figure 23 Birectangular earth pressure redistribution

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

- **"Trapezoidal"**

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

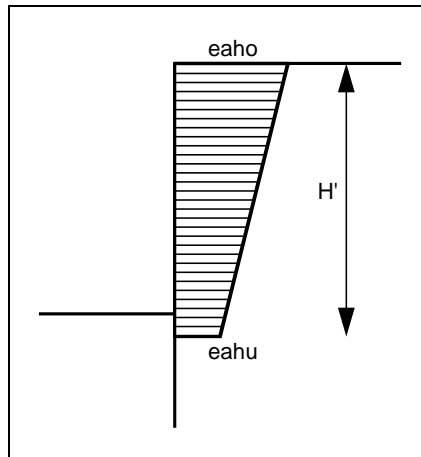


Figure 24 Earth pressure redistribution in a trapezoidal

- **"Quadrilateral"**

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

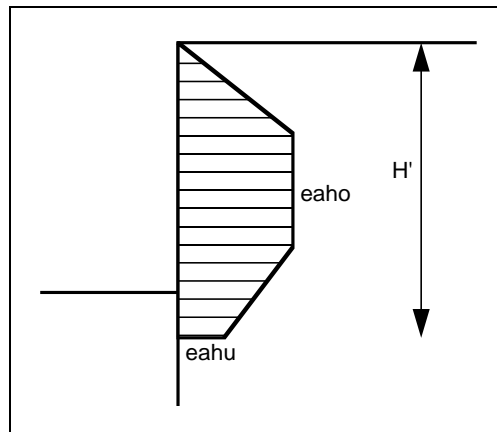


Figure 25 Earth pressure redistribution in a quadrilateral

- **"User-defined"**

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

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

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

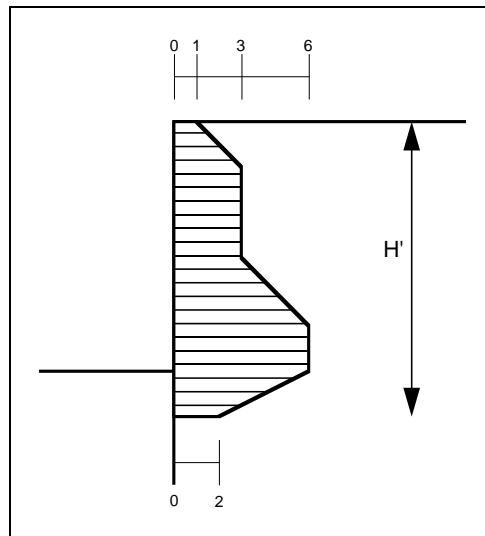
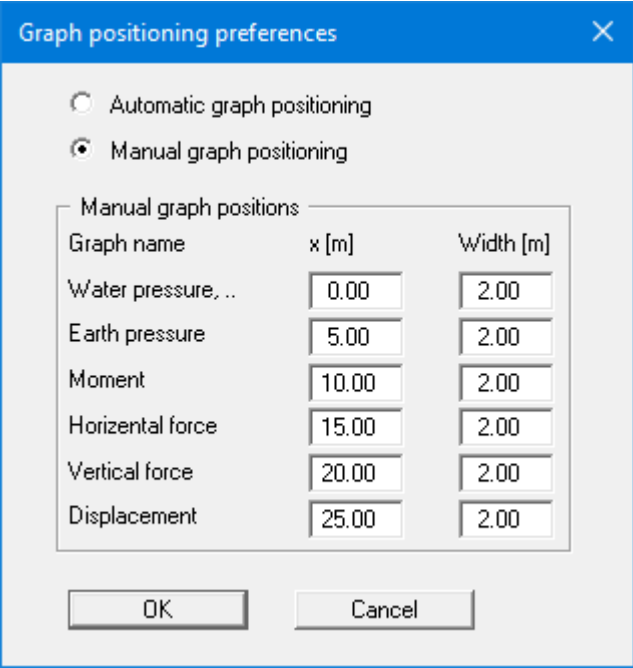


Figure 26 User defined earth pressure redistribution

8.4.5 "Graph positioning preferences" menu item

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



The dialog box titled "Graph positioning preferences" has a blue header bar with a close button (X) in the top right corner. It contains two radio buttons: "Automatic graph positioning" (unselected) and "Manual graph positioning" (selected). Below the radio buttons is a section titled "Manual graph positions" which contains a table with three columns: "Graph name", "x [m]", and "Width [m]". The table lists six graph types with their corresponding x and width values. At the bottom of the dialog are "OK" and "Cancel" buttons.

Graph name	x [m]	Width [m]
Water pressure, ...	0.00	2.00
Earth pressure	5.00	2.00
Moment	10.00	2.00
Horizontal force	15.00	2.00
Vertical force	20.00	2.00
Displacement	25.00	2.00

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

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

8.4.6 "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

☐ Pore water pressure ☐ Potential

☐ Gradient ☐ EP + pw

Show system

Show results

Cancel

Earth pressure and state variable hatching

☒ Fixed spacing Spacing [mm] = 2.0

Presentation of action effects, etc.

☒ Earth pressure ☒ Moment ☒ Horizontal force

☐ Displacement ☒ Vertical force

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

☐ Settlements ☐ Logarithmic spiral

Display (earth pressure):

☐ d ☒ (g+q).k ☐ g.k ☐ q.k

Display (state variables):

☐ d ☐ g.d ☒ (g+q).k ☐ g.k ☐ q.k

Display (displacement):

☐ d ☐ g.d ☒ (g+q).k ☐ g.k ☐ q.k

☒ With footing rotation

Load presentation

Presentation height [m]: 0.300 Load 'hatching': With hatching

☐ Same height for all

Hatch separation [mm]: 2.0

Passive/active earth pressure oversizing [-]: 1.0

Strut variables: Moment

☒ Only display water pressure if water pressure <> 0.0

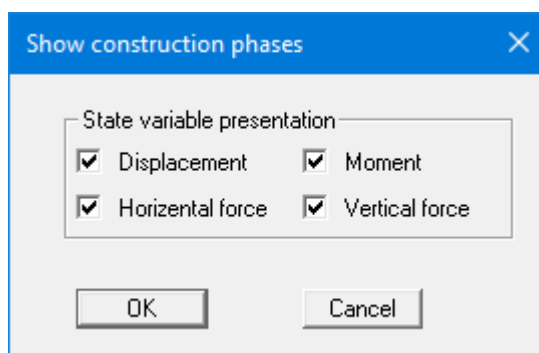
On the whole, the dialog box is self-explanatory. For example, if the "**Settlements**" check box is activated, the stresses below the footing for the characteristic point are visualised. When "**EP + pw**" is selected, the sum of earth pressure and water pressure (pw) is displayed in the diagram for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

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

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

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

When you are working with a particular construction phase the following dialog box appears:



8.4.7 "Labelling preferences" menu item

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

The 'Labelling' dialog box contains the following settings:

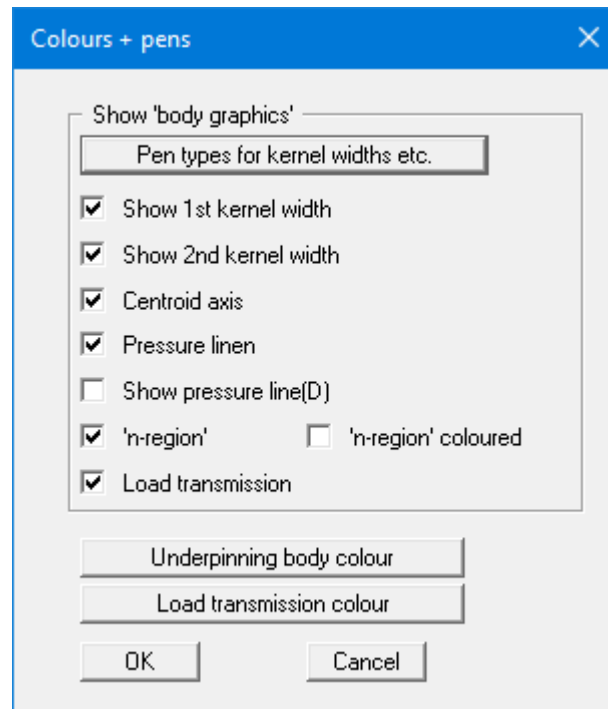
- ☐ Label active berms
- ☒ Show slope of active berms
- ☐ Label passive berms
- ☒ Show slope of passive berms
- ☐ Label loads
- ☐ Label bodies
- ☒ Show loads (area of influence)
- ☐ Label logarithmic spiral
- ☒ Label stress distribution
- ☒ Show anchor steels
- ☒ Show anchor lengths/inclinations
- Labelling (Anchor / Struts):
- Decimal places (displacement):
- Distributed load labelling: ☐ Left ☒ Central ☐ Right
- Groundwater labelling (left):
- Groundwater labelling (right):
- Anchor labelling:
- Font size [mm]:
- Force arrow factor [-]:
- Width of grouted section [m]:

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 sections can be edited.

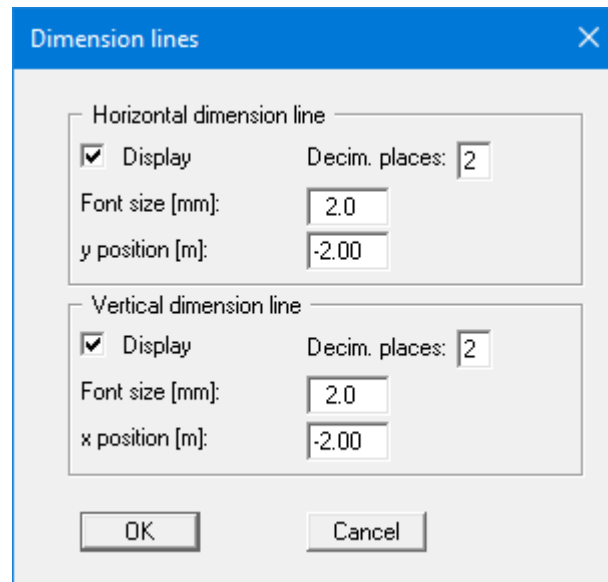
8.4.8 "Body pens + colours" menu item

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



8.4.9 "Dimension lines" menu item

You can define a vertical and/or horizontal dimension line for the graphics in order to emphasise and clarify the system dimensions.



The screenshot shows a dialog box titled "Dimension lines" with a close button (X) in the top right corner. It contains two main sections: "Horizontal dimension line" and "Vertical dimension line". Each section has a "Display" checkbox which is checked, a "Decim. places" field set to 2, a "Font size [mm]" field set to 2.0, and a position field. For the horizontal line, the position is "y position [m]" set to -2.00. For the vertical line, the position is "x position [m]" set to -2.00. At the bottom of the dialog are "OK" and "Cancel" buttons.

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

The fastest way to modify the position of the dimension line is to press the [F11] function key and then to pull the legend to the new position with the left mouse button pressed.

8.4.10 "Display system" menu item

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

8.4.11 "Display results" menu item

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

8.5 Evaluation menu

8.5.1 General notes

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

8.5.2 "Earth pressure redistribution" menu item

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

8.5.3 "Main output summary" menu item

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

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

8.5.4 "Anchor and strut summary" menu item

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

8.5.5 "Bearing capacity" menu item

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

8.5.6 "Settlements" menu item

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

8.5.7 "Sliding safety" menu item

The calculated sliding safety is displayed in a message box.

8.5.8 "Deep-seated stability summary" menu item

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

Deep-seated stability FOS

OK Forw. Back

Start point of slip plane in wall area = 3.00 m

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

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

μ_{ue} = Utilisation factor ≤ 1.0

No	Depth [m]	Length [m]	$A_{h,g+q,d}$ [kN/m]	Poss. $A_{h,g+q,d}$ [kN/m]	$\mu_{ue}(g+q)$ [-]	$A_{h,g,d}$ [kN/m]	Poss. $A_{h,g,d}$ [kN/m]	$\mu_{ue,g}$ [-]
1	0.50	8.00	13.6	105.7	0.128	13.6	105.7	0.128

Optimise

In addition to the information regarding the anchors, the critical safety factor or the utilisation factor for deep-seated stability for each anchor is given. In the example shown here, the anchor has a safety factor which is higher than the required value of 1.5. Select the "**Optimise**" button to optimise the length of the anchor.

Optimise anchor lengths

Anchor length increment [m]: 0.050

Minimum anchor length [m]: 4.000

Maximum anchor length [m]: 35.000

Current anchor length = 8.00

☒ Search completely

Optimisation can be cancelled during calculating using the right mouse button.

OK Cancel

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

8.5.9 "Hydraulic heave FOS summary" menu item

You obtain all data relating to verification of hydraulic heave safety (see Section 7.27).

8.5.10 "Buoyancy FOS summary" menu item

You obtain all data relating to verification of buoyancy safety (see Section 7.27.1).

8.5.11 "EQU analysis" menu item

All available information on the EQU analysis (limit state of loss of static equilibrium) is displayed in a message box.

8.6 *Construction phases menu*

8.6.1 General notes

When the system you are working on consists of a number of construction phases, each phase must initially be calculated separately, and the results saved in corresponding files (see Section 8.1.4). Subsequently, from this menu, you can combine the different file contents. The following graphics can be created:

- envelope of moments,
- envelope of horizontal forces,
- envelope of vertical forces,
- sum of displacements as a new bending line.

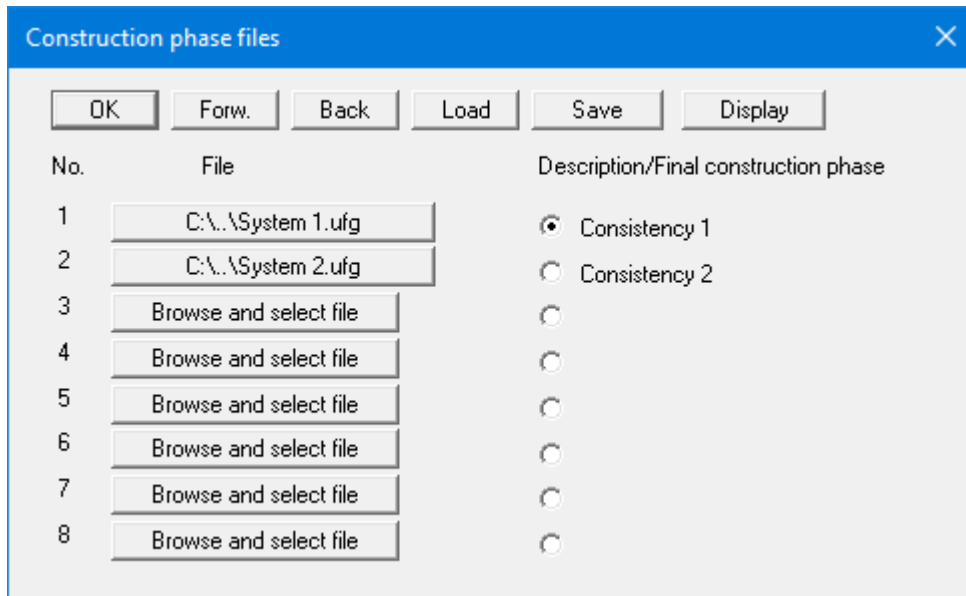
The resulting graphic is displayed in the current system. The "**Construction phases**" menu will only work with the results from the files compiled using this menu.

8.6.2 "Info" menu item

You will see information on the options provided by the "**Construction phases**" menu.



8.6.3 "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 redisplay 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 "**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 "**Phase 1**", "**Phase 2**"). 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 toolbar (see Section 8.7.6). 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.

8.6.4 "Display summary" menu item

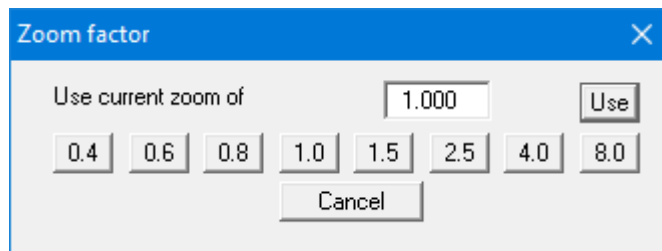
If the system graphics or the results of the current analysis are actually displayed on the screen, this option will switch to a visualisation of the construction phases.

8.7 Graphics preferences menu

8.7.1 "Refresh and zoom" menu item

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

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



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

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

8.7.2 "Zoom info" menu item

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

8.7.3 "Legend font selection" menu item

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

8.7.4 "Pen colour and width" menu item

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

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

8.7.5 "Mini-CAD toolbar" and "Header toolbar" menu items

Using these two menu items you can add free text to the graphics and add lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, etc.). PDF files can also be imported as images. The same pop-up menu opens for both menu items, the icons and functions used are described in more detail in the **Mini-CAD** manual saved in the 'C:\Program Files (x86)\GGU-Software\Manuals' folder during installation. The differences between the Mini-CAD and Header CAD are as follows:

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

8.7.6 "Toolbar preferences" menu item

After starting the program, a horizontal toolbar for menu items appears below the program menu bar. If you would rather work with a popup window with several columns, you can specify your preferences using this menu item. The smart icons can also be switched off.

At the bottom of the program window, you find a status bar with further information. You can also activate or switch off the status bar here. The preferences will be saved in the "**GGU-UNDERPIN.alg**" file (see menu item "**Graphics preferences/Save graphics preferences**") and will be active at the next time the program is started.

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



"Next page"/"Previous page"

Using this icon, you can navigate between the individual pages in the *tabular representation*.



"Select page"

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



"Zoom out"

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



"Zoom (-)"/"Zoom (+)"

With the zoom functions you can zoom in or out of parts of the image, by clicking the left mouse button.



"Copy/print area"

Use this tool to copy only parts of the graphics in order to paste them, e.g. to a report. You will see information on this function and can then mark an area, which is copied to the clipboard or can be saved in a file. Alternatively you can send the marked area directly to your printer (see "**Tips and tricks**", Section 9.3).



"Colour on/off"

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



"Undo"

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



"Restore "

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

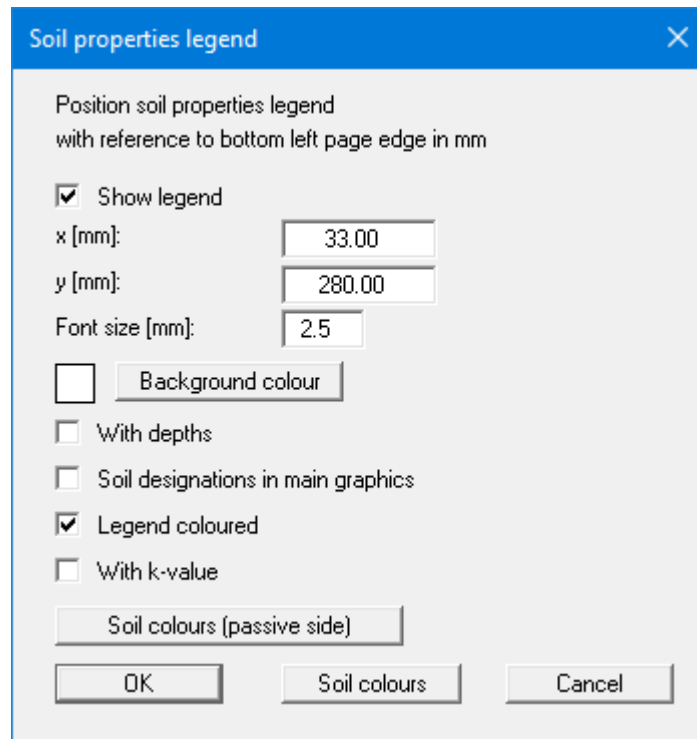


"Back one phase"/"Forw. one phase"

If you have activated a summarised result of various construction phases, you can move from phase to phase using the arrow buttons.

8.7.7 "Soil properties legend" menu item

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



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

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

- **"Background colour"**
Using this button you can define the background colour for the legend.
- **"With depths"**
The depths of the individual soil layers will be shown in the soil properties legend.
- **"Soil designations in main graphics"**
The designations entered for the individual ground layers are displayed in the system graphics adjacent to the layer depths.
- **"Legend coloured"**
The soil colours are displayed in the legend. Otherwise, they will be numbered.
- **"With k-value"**
The k-values of the individual soil layers will be shown in the soil properties legend.

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

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

8.7.8 "General legend" menu item

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

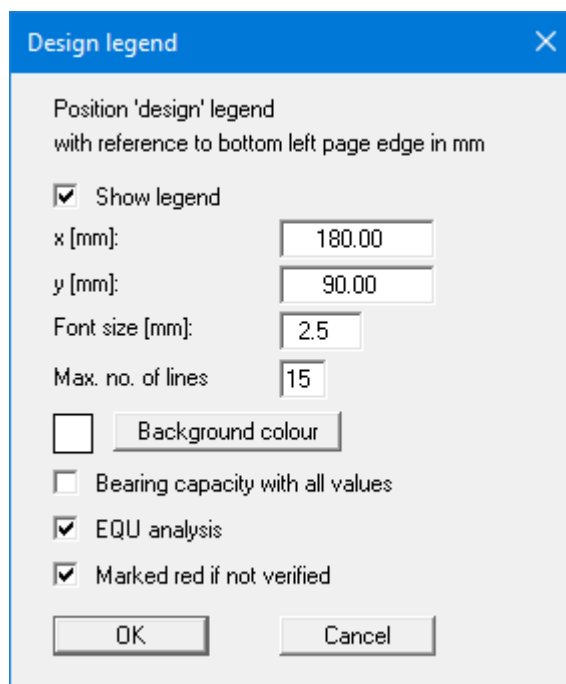
You can define and edit the position of the legend using the values **"x value"** and **"y value"**. You control the size of the legend using **"Font size"** and **"Max. no. of lines"**; where necessary, several columns are used. Additionally, you can define a background colour for the legend.

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

In the **General legend** you can, if wished, display information on the program (name and version), on the adopted standard and on the current file (name, path, time info). Additionally, the display of further information as e.g. hydraulic heave can be activated. Any project identification entered in the "**File/New**" or "**Editor 1/Analysis options**" dialog box will also be shown in the **General legend**.

8.7.9 "Design legend" menu item

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



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

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

The number of values displayed in the graphical design legend can be influenced using the lower check boxes. These values are automatically contained in the output table that can be opened using the "**File/Print output table**" menu item (see Section 8.1.5).

8.7.10 "Settlement legend" menu item

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

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

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

8.7.11 "Move objects" menu item

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

If **"Manual graph positioning"** has been selected in the **"System/Graph positioning preferences"** menu item (see Section 8.4.5), the result graphs can also be repositioned using this function.

8.7.12 "Save graphics preferences" menu item

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

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

8.7.13 "Load graphics preferences" menu item

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

8.8 Page size + margins menu

8.8.1 "Auto-resize" menu item

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

8.8.2 "Manual resize (mouse)" menu item

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

- You can use the coordinates of a section of the visualisation as the new image coordinates by marking the desired area with the mouse, pressing the left mouse button and holding the [Ctrl] and [Shift] keys. The scales of the x- and y-axes are adjusted accordingly. If the previous proportions (scale x-direction/scale y-direction) need to be retained, the "**Proportional section**" check box must be activated.
- By activating the "**Redefine origin**" button you can change the image coordinates without altering the scale. In this way, you can quickly achieve the desired presentation with a suitably defined scale without annoying trial and error with the x/y coordinates. The previous scale preferences are not affected by this.

8.8.3 "Manual resize (editor)" menu item

You can alter the current image coordinates by direct numerical input in a dialog box. This allows precise scale input. The coordinates refer to the *drawing area*. This can be defined in the "**Page size + margins/Page size and margins**" menu item by means of the plot margins (see Section 8.8.6).

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

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

8.8.4 "Zoom" menu item

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

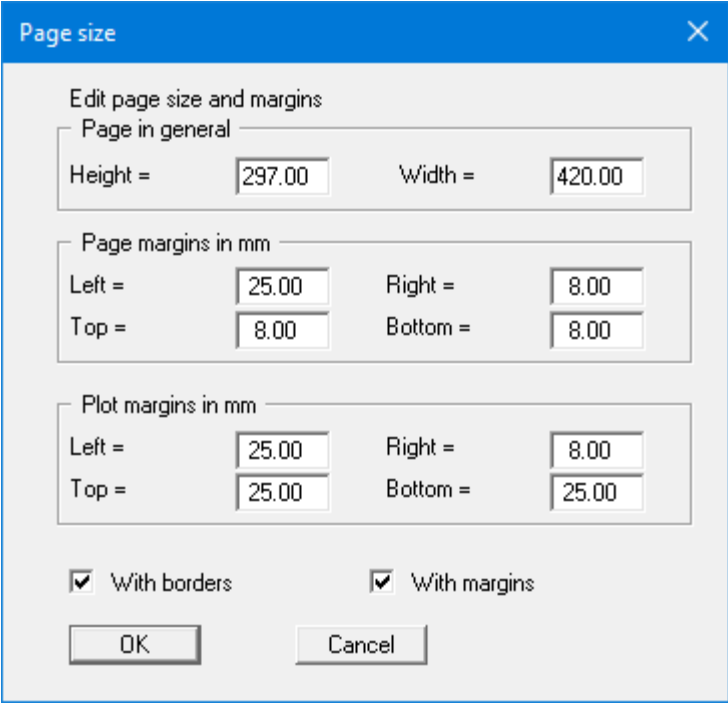
8.8.5 "Font size selection" menu item

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

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

8.8.6 "Page size and margins" menu item

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



The "Page size" dialog box is used to configure page dimensions and margins. It features three main sections for editing:

- Page in general:** Allows setting the overall page size. The default height is 297.00 and the default width is 420.00.
- Page margins in mm:** Allows setting the distance from the page edges to the content area. The default values are Left: 25.00, Right: 8.00, Top: 8.00, and Bottom: 8.00.
- Plot margins in mm:** Allows setting the distance from the page edges to the actual drawing area. The default values are Left: 25.00, Right: 8.00, Top: 25.00, and Bottom: 25.00.

At the bottom of the dialog, there are two checked checkboxes: "With borders" and "With margins". The "OK" and "Cancel" buttons are located at the bottom center.

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

8.8.7 "Undo" menu item

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

8.8.8 "Restore" menu item

When this menu item is selected the last change made in a dialog box or the last change in the position of objects, which you undid using the menu item "**Page size + margins/Undo**" will be restored. This function can also be reached by using the key combination [Ctrl] + [Back] or the appropriate tool in the toolbar (see Section 8.7.6).

8.8.9 "Preferences" menu item

You can activate or deactivate the undo functions.

8.9 Info menu

8.9.1 "Copyright" menu item

You will see a copyright message and information on the program version number.

The "**System**" button shows information on your computer configuration and the folders used by GGU-UNDERPIN.

8.9.2 "Maxima" menu item

Here you can check the defaults for maximum values.

8.9.3 "Active wall friction angle" menu item

An info box appears with information relating to active angle of wall friction according to the EAB.

8.9.4 "Compare earth pressure coefficients" menu item

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

8.9.5 "Help" menu item

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

8.9.6 "GGU on the web" menu item

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

Get information on updates and modifications on a regular basis from your program module page. You can also subscribe to email notifications, which provide information on all modifications on a monthly basis.

8.9.7 "GGU support" menu item

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

8.9.8 "What's new?" menu item

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

8.9.9 "Language preferences" menu item


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

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

9 Tips and tricks

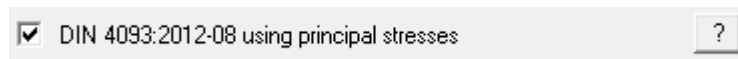
9.1 "?" buttons

Reading of the manual can mostly be dispensed with, because

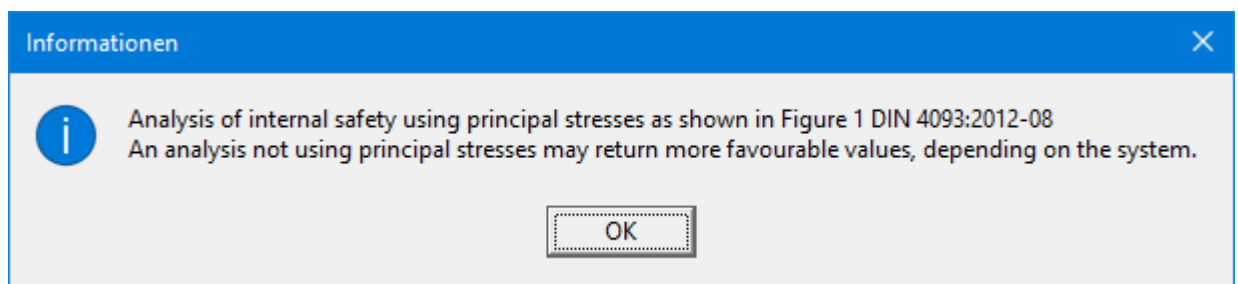
"?" buttons 

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

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

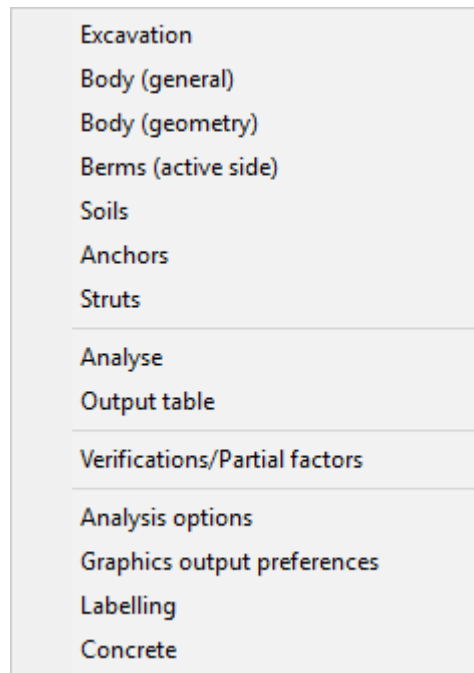


If you click on the question mark, you will see the following message box:



9.2 Keyboard and mouse

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



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

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

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

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

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

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

Change screen coordinates:


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

9.2.1 Function keys

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

- [Esc] refreshes the screen contents and sets the screen back to the given format. This is useful if, for example, you have used the zoom function to display parts of the screen and would like to quickly return to a complete overview.
- [F1] opens the manual file.
- [F2] refreshes the screen without altering the current magnification.
- [F5] opens the menu item "System/Analyse".
- [F6] opens the menu item "Evaluation/Main output summary".
- [F7] opens the menu item "Evaluation/Bearing capacity".
- [F8] opens the menu item "Evaluation/Settlements".
- [F9] opens the menu item "Page size + margins/Auto-resize".
- [F11] activates the menu item "Graphics preferences/Move objects".

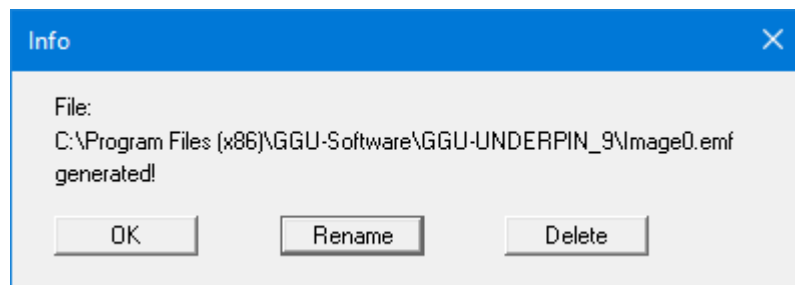
9.3 "Copy/print area" icon

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

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

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

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



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

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

10 Index

2

2nd order theory, select for analysis..... 93

A

Absolute heights, activate use..... 55

Action boundary conditions,

define..... 82

equivalent for pretensioning 39

Active earth pressure,

activate calculation after Culmann 69

analysis principles 23

calculation after Culmann 23

edit preferences 69

select for analysis 68

Anchor steel design 90

Anchor wall 83

Anchors,

adopt predeform. as lowering of supports 38

consider pretensioning..... 39

define..... 83

define pretensioning 85

display design values in a message box..... 102

optimise length..... 103

Angle of wall friction,

enter/adopt from soil database..... 67

Area loads,

consider in analysis 30

define..... 79

include into earth pressure redistribution 93

select ep analysis method 71

select type of earth pressure 68

At-rest earth pressure,

coefficient to DIN 4085..... 23

keep constant..... 71

select for analysis 68

B

Base inclination,

coefficients in bearing capacity equation 40

consider in analysis 36

define..... 15

Bearing capacity coefficients,

in bearing capacity equation..... 40

Bearing capacity failure plane, determine..... 41

Bearing capacity safety,

analysis principles 40

consider base inclination 36

display analysis results in a message box ... 102

Bending line, display with predeformations 89

Berms,

activate labelling 99

define on active side..... 66

define on passive side..... 66

Bounded surcharges,

consider in analysis 32, 34

define..... 80

Buckling length, determination..... 48

Buoyancy safety,

analysis using global safety factors 44

analysis using partial safety factors 44

display analysis results in a message box ... 104

C

Caquot/Kerisel,

passive ep analysis 24

select for passive ep analysis..... 70

Centroid axis,

activate/edit display 100

consider acting forces 35

Characteristic point,

display settlements 102

limiting depth determination 86

Cholesky 35

Clipboard 61

Cohesion,

coefficient determination methods 23

enter/adopt from soil database..... 67

Colour,

define for soils..... 110

switch on/off..... 108

Colour/hatching, define for loads 98

Colour/pens,

define for graphical elements 107

define for underpinning body 100

Company letterhead, add via Mini-CAD 107

Concrete verification,

analysis to DIN 1045..... 47

analysis to DIN 4093..... 49

analysis to EC2/DIN 1045-1 48

select analysis method 55

Concrete,

activate verification 17

enter values for structural analysis 17

Constrained modulus,

for settlement analysis 42

Construction phases,

activate display..... 105

animated presentation..... 105

consider predeformations 38

possible state variables presentation..... 104

select files..... 105

Context menu, open..... 118

Coordinates,

alter via editor 113

alter with mouse 113

optimise/reset 113

save/load..... 113

Copy/print area 61, 108, 120

Culmann,

activate active ep calculation..... 69

activate passive ep calculation..... 70

active ep calculation 23

passive ep analysis 24

Cutting borders, switch on/off 114

D

Danish standard	41
Database,	
for soil properties of common soils	23, 67
Dataset description,	
display	111
enter.....	54
Deep-seated stability,	
define intersection with wall	87
safety/utilisation factor for anchors	103
verification principles.....	45
Delete, soil layers	67
Depth coefficients,	
definition to Danish standard	41
Design situation,	
activate display in legend	111
enter text for legend.....	75
Design strength,	
define for concrete.....	17
Design, display results in legend	111
Dimension lines,	
define.....	101
move with mouse	112
DIN 1045	47
DIN 4017	40
DIN 4019	42
DIN 4085,	
consider in ep analysis.....	23
passive ep analysis	24
select for active ep analysis	69
select for passive ep analysis.....	70
DIN 4093, select for concrete verification.....	55
Displacement boundary conditions, define	82
Displacement method	35
Displacement, define for underpinning body	82
Distributed load, define as live load	65
Double-bounded surcharges,	
consider in analysis	33, 34
define.....	81
Drawing area, define.....	114
Duddeck/Ahrens	35
DXF file,	
export	61
import.....	7

E

Earth pressure coefficients,	
be calculated.....	115
calculate	72
user-defined.....	72
Earth pressure redistribution,	
display in a message box	102
to EAB	39, 93
user-defined.....	95
Earth pressure,	
edit preferences for active	69
edit preferences for passive	70
select for analysis	68
Earth quake, as design situation to EC 7.....	76
EC 7, load cases description	76
Eccentricity, unintentional	47
Editor window, output table	59
Elastic curve, for inclined body	36
Elevation head, for potential definition.....	27

Embedment depth, footing.....	40
EMF format	61
EQU analysis,	
display results in a message box.....	104
Equation solution to Cholesky	35
Equivalent earth pressure coefficient,	
activate use	69
Equivalent wall, regulations compliant with EAB 6th Edition.....	55
Excavation,	
change alignment.....	55
define base.....	64

F

File,	
display name in legend.....	111
load/save.....	55
Finite element module,	
for structural system.....	35
for water pressure determination	25
Flow conduits approach, possible error	26
Flow force,	
characteristic	43
for hydraulic heave safety	43
influence on earth pressure.....	73
Font selection.....	106
Font size,	
design legend.....	111
dimension lines.....	101
edit for graphical elements	113
general legend	110
settlement legend.....	112
soil properties legend	109
system/result graphics	99
Footer, output table	57
Footing,	
define excavation base.....	64
define length.....	15
Force boundary conditions,	
equivalent for pretensioning	39
Foundation gap, explanation.....	20
Friction angle, enter/adopt from soil database ..	67
Function keys.....	119

G

General page information,	
add via Mini-CAD.....	107
General stability,	
verification using file export	42, 60
GGU-CAD file, export	61
GGUMiniCAD file, export	62
GGU-STABILITY file, export	60
Global coordinates,	
use for Mini-CAD output	62
Global safety factors/verifications define	74
Graph positions,	
alter via editor	96
alter with mouse	112
Graphics, add via Mini-CAD.....	107
Ground inclination coefficients,	
in bearing capacity equation.....	40
Groundwater,	
activate labelling	99

consider storeys/confined aquifers	25
define colour	107
enter additional potentials	85
enter level	64
Grouted section,	
enter length for graphics	83
enter width for graphics	99

H

Hatching, define for loads	98
Header CAD, application explanations	107
Header, output table	57
Hinge, define for struts	84
Horizontal force, calculation	36
Hydraulic gradient,	
calculation	25, 28
determine for analysis	73
Hydraulic heave safety,	
analysis using global safety factors	43
analysis using partial safety factors	43
display analysis results in a message box ...	104

I

Increased active earth pressure,	
coefficient	23
select for analysis	68

K

Kernel widths,	
activate/edit display	100
explanation	20

L

Labelling, system/result graphics	99
Language preferences	10, 116
Lateral pressures, define	78
Layout, output table	57
Legends, move with mouse	112
Licence protection	10
Limiting depth,	
defining methods	42
select determination	86
Live load, consider distributed load as	65
Load cases,	
adopt to DIN 1054-2010/EC 7	76, 77
adopt to ÖNORM EN 1997-1	76, 77
Load concentration factor,	
consider	31
define	71
Load distribution, for concrete verification	47
Load inclination coefficients,	
in bearing capacity equation	40
Load transition point,	
using redistribution	94
Load transmission,	
activate/edit display	100
define width	15
Loads with limited plan dimensions,	
define	79
reduce	31, 79
Loads, edit graphical output preferences	98

Logarithmic spiral,	
display below wall	97
in bearing capacity failure plane	41

M

Manual, open as PDF file	116
Maximum values, display for system	115
Mean friction angle	41
Mean soil properties	41
Metafile, export	61
Mini-CAD,	
application explanations	107
export file	62
output with global coordinates	62
Moment load, consider on rod axis	35
Mouse click functions	118
Mouse wheel functions	119

N

Navigation, output table	108
n-region,	
activate/edit display	100
explanation	20
n-value, define for load distribution	17

O

Objects, move with mouse	112
ÖNORM EN 1997-1,	
select partial safety factors	76, 77
Output preferences	59, 60
Output table,	
edit output as ASCII	59
edit output as graphics	57
navigation	58
save/load settings for graphical display	57
select output format	56
switch to system graphics	58, 108
Overburden stress,	
consider in settlement analysis	86
Overturning analysis, deactivate	55

P

Page,	
copy/print section	108, 120
define format	114
switch margins on/off	114
Pagination, automatic	58, 59
Partial safety factors,	
define	55, 77
select from DIN 1054-2010/EC 7	76
select from ÖNORM EN 1997-1	76, 77
Passive earth pressure,	
activate calculation after Culmann	70
analysis methods for coefficients	24
calculation after Culmann	24
limit	70
specify calculation	70
PDF file, import via Mini-CAD	7, 107
Pen preferences,	
edit for graphical elements	107
edit for underpinning body	100

Permeabilities,		
consider in analysis	23	
display in legend	109	
enter/adopt from soil database	67	
using flow conduits	25	
Piling Handbook,		
consider loads with limited dimensions	31	
effect on earth pressure	29	
Plausibility checks,		
show during analysis	92	
view results	91	
Plot margins, define	114	
Potential, definition	27	
Preconsolidation,		
consider in settlement analysis	42, 86	
Predeformations,		
consider in analysis	38	
define	89	
Presentation height, define for loads	98	
Pressure line,		
activate/edit display	100	
explanation	20	
Pretensioning,		
consider in analysis	39	
define for anchors/struts	85	
Print,		
graphics	60	
output table	61	
section	61, 108, 120	
several files	63	
Program,		
display name in legend	111	
save/load preferences	112	
show improvements	116	
show information	115	
Project data, add via Mini-CAD	107	
Project identification,		
display	111	
enter	54	
R		
Redistribution figures,		
available	39	
select	93	
Result graphics,		
display	101	
edit preferences	97	
label	99	
move with mouse	112	
position	96	
Results, display for selected depth	102, 118	
Rod construction module	35	
Rods,		
define for struts	84	
define for system	91	
effect on analysis	35	
Rotation, define for underpinning body	82	
S		
Safety concept, select	55	
Safety factors, define	55, 74	
Scale,		
alter via factor input	113	
alter with mouse	113	
define manually	113	
determine automatically	113	
Scroll the screen	118	
Seismic effects,		
as design situation to EC 7	76	
consider loads	77	
Settlement analysis,		
display results in a message box	102	
display results in legend	112	
edit preferences	86	
limiting depth definition	42	
subtraction preconsolidation loading	42	
Settlements, display share in legend	112	
Shape coefficients, for strip footings	40	
Slenderness, structural element	47	
Sliding safety,		
analysis principles	42	
consider base inclination	36	
display analysis results in a message box	102	
Slip planes, compound	45	
Slope stability application	60	
Smart icons, for menu items	107	
Soil colours/numbers, activate display	109	
Soil designations,		
activate presentation in system graphics	109	
define	67	
Soil layers,		
adapt to absolute heights	64	
define/delete	67	
maximum number	23	
Soil properties,		
activate separate input act./pass. values	55	
display in legend	109	
enter/adopt from soil database	67	
Standard, display in legend	111	
Status bar main program, activate	107	
Stiffness of underpinning wall,		
consider in analysis	35, 37	
Streck,		
passive ep analysis	24	
select for passive ep analysis	70	
Stress analysis, for limiting depth	86	
Struts,		
consider pretensioning	39	
define	84	
define pretensioning	85	
display design values in a message box	102	
System,		
analyse	92	
display	101	
display properties in legend	110	
division into rods	91	
edit graphics output preferences	97	
show information	91, 115	
structural	35	
T		
Theory of elastic half-space	31	
Toolbar, edit for menu items	107	
Transition point,		
using redistribution	94	
Translation, activate	116	
True-type font	106	

U

Underpinning body,	
define colour/pens	100
define geometry	16, 22
define loads	15
define rotation/displacement	82
possible bedding conditions	37
Unit weight, enter/adopt from soil database	67
User-defined ep coefficients, enter	72
Utilisation factor,	
buoyancy	45
deep-seated stability	103
hydraulic heave	43
unreinforced concrete	48

V

Verification of unreinforced concrete,	
select method	55
to DIN 1045	47
to DIN 4093	49
to EC2/DIN 1045-1	48
Verifications,	
activate to DIN 1054 old	74
activate to DIN 1054-2010/EC 7	75, 76
Version number,	
display in a message box	115

display in legend	111
Vertical force, calculation	36

W

Wall toe, select bedding conditions	92
Wall top, kept horizontal,	
activate	17
consider in analysis	48
Water pressure approach,	
classical	24
recommendation	26
select for analysis	73
using flow conduits	25
Water pressure,	
resulting difference	27
user-defined distribution	25
What you see is what you get	106

Y

Young's modulus,	
define for concrete	17

Z

Zoom factor, define for full-screen display	106
Zoom function, activate	106, 108, 118