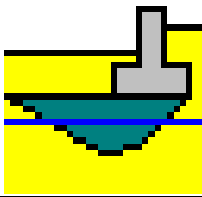


Analysis of footings to DIN 4017, DIN 4019,
DIN 1054 and EC 7

GGU-FOOTING

VERSION 10



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

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

Contents:

1	Preface.....	5
2	Licence protection	6
3	Language selection	6
4	Starting the program	7
5	Tips and tricks.....	8
5.1	"?" and "Info" buttons	8
5.2	Function keys	8
5.3	Keyboard and mouse	9
5.4	"Copy/print area" icon.....	10
6	Short description and worked examples	11
6.1	Output sheet design	11
6.1.1	Generate result graphics.....	11
6.1.2	Move graphic elements.....	13
6.1.3	Edit and turn off graphic elements	13
6.1.4	Edit page size and margins	15
6.2	Example 1: Multiple footings.....	16
6.2.1	Enter analysis options (Example 1)	16
6.2.2	Enter system parameters (Example 1)	17
6.2.3	Define footings (Example 1).....	20
6.2.4	Define soils (Example 1)	21
6.2.5	Select the limiting depth (Example 1).....	22
6.2.6	Analyse system (Example 1)	23
6.3	Example 2: Single rectangular footing	24
6.3.1	Enter analysis options (Example 2)	24
6.3.2	Enter system parameters (Example 2)	25
6.3.3	Define footing (Example 2)	26
6.3.4	Define soils (Example 2)	27
6.3.5	Analyse system (Example 2)	27
6.3.6	Represent results in the plan view of footing (Example 2)	28
6.3.7	Consider depth coefficients (Example 2).....	30
6.3.8	Represent results in the pad footing legend (Example 2).....	31
6.4	Example 3: Single circular footing (EC 7)	32
6.4.1	Enter analysis options (Example 3)	32
6.4.2	Enter system parameters (Example 3)	33
6.4.3	Define footing (Example 3)	34
6.4.4	Define soils (Example 3)	35
6.4.5	Analyse system (Example 3)	35
6.4.6	Represent results in the plan view of footing (Example 2)	36
7	Theoretical principles	38
7.1	Analysis of bearing capacity safety	38
7.2	Theoretical principles circle/annulus.....	40
7.3	Determining the mean governing soil properties	41
7.4	Differences between DIN 1054 (old) and DIN 1054:2005/EC 7	42
7.5	Settlement analysis	46

8	Description of menu items.....	47
8.1	File menu.....	47
8.1.1	"New" menu item.....	47
8.1.2	"Load" menu item.....	49
8.1.3	"Save" menu item.....	49
8.1.4	"Save as" menu item.....	49
8.1.5	"Print output table" menu item.....	50
8.1.5.1	Output as graphics.....	50
8.1.5.2	Output as ASCII.....	52
8.1.6	"Export to GGU-STABILITY" menu item.....	53
8.1.7	"Export to GGU-SLAB" menu item.....	54
8.1.8	"Output preferences" menu item.....	55
8.1.9	"Print and export" menu item.....	55
8.1.10	"Batch print" menu item.....	57
8.1.11	"Exit" menu item.....	57
8.1.12	"1, 2, 3, 4" menu items.....	57
8.2	Edit menu.....	58
8.2.1	"Analysis options" menu item.....	58
8.2.2	"Project identification" menu item.....	58
8.2.3	"Ground level" menu item.....	59
8.2.4	"System parameters" menu item.....	60
8.2.5	"Footing details" menu item.....	61
8.2.6	"Soils" menu item.....	62
8.2.7	"Limiting depth" menu item.....	62
8.2.8	"Line loads" menu item.....	63
8.2.9	"Berms" menu item.....	64
8.2.10	"Partial safety factors" menu item.....	65
8.3	System menu.....	66
8.3.1	"Analyse" menu item.....	66
8.3.2	"Optimise footing width" menu item.....	66
8.4	Output preferences menu.....	67
8.4.1	"Footing analysis diagram" menu item.....	67
8.4.2	"Footing plan" menu item.....	69
8.4.3	"Soil properties" menu item.....	70
8.4.4	"System presentation" menu item.....	71
8.4.5	"General legend" menu item.....	72
8.4.6	"Output table" menu item.....	73
8.4.7	"Pad footing" menu item.....	74
8.4.8	"Stress influence diagram" menu item.....	75
8.4.9	"Line loads" menu item.....	76
8.4.10	"Reset all graphics to defaults" menu item.....	76
8.4.11	"Page size and margins" menu item.....	77
8.4.12	"Move objects" menu item.....	77
8.5	Graphics preferences menu.....	78
8.5.1	"Refresh and zoom" menu item.....	78
8.5.2	"Fill colours" menu item.....	79
8.5.3	"Zoom info" menu item.....	79

8.5.4	"Legend font selection" menu item.....	79
8.5.5	"Margins and borders" menu item	79
8.5.6	"Pen colour and width" menu item	80
8.5.7	"Mini-CAD toolbar" menu item	80
8.5.8	"Toolbar preferences" menu item	81
8.5.9	"Load graphics preferences" menu item	82
8.5.10	"Save graphics preferences" menu item.....	82
8.6	Info menu	83
8.6.1	"Copyright" menu item.....	83
8.6.2	"Help" menu item	83
8.6.3	"GGU on the web" menu item	83
8.6.4	"GGU support" menu item.....	83
8.6.5	"What's new?" menu item.....	83
8.6.6	"Test vibrodisplacement compaction" menu item	84
8.6.7	"Language preferences" menu item	84
9	Index.....	85

List of figures:

<i>Figure 1</i>	<i>Result screen</i>	<i>12</i>
<i>Figure 2</i>	<i>Equivalent area for circle (Leibniz University, Hannover).....</i>	<i>40</i>
<i>Figure 3</i>	<i>Logarithmic spiral.....</i>	<i>41</i>
<i>Figure 4</i>	<i>Line load.....</i>	<i>63</i>
<i>Figure 5</i>	<i>Berms.....</i>	<i>64</i>

1 Preface

The **GGU-FOOTING** program allows the analysis of bearing capacity according to DIN 4017 and of settlement according to DIN 4019. Both the *global safety factors* according to DIN 1054 (old) and the *partial safety factors* according to DIN 1054:2005 or EC 7 may be taken into consideration. In addition to the standard DIN 4017 methods, the methods after Terzaghi, Meyerhoff, Hansen and Vesic that are well known from many literature references may also be adopted. Furthermore, soil improvement with vibrodisplacement compaction according to Priebe can be considered.

The program provides two different types of analysis:

- **Mode 1: "Multiple footings"**
Multiple footings of a single type (pad or strip footing), which only vary in their width, are analysed. In this mode it is possible to generate footing analysis diagrams from which the bearing resistance design value (EC7), the acceptable bearing pressure (DIN 1054:2005) or the allowable footing pressure (DIN 1054 old) can be read off as a function of the footing width, together with the corresponding settlement.
- **Mode 2: "Pad footing"**
A pad footing is analysed. It is possible to select between either a *rectangular*, a *circular or annular footing*. Analysis of the torsional stiffness, which can be determined using this mode for circular or annular footings, is often required for wind energy converters.

Further calculations for sloped systems with **GGU-STABILITY** and larger slabs with **GGU-SLAB** can be performed using the corresponding export functions.

The program system allows comfortable data input. Reading of the manual can mostly be dispensed with, because

"?" buttons  and "Info" buttons 

dealing with almost all geotechnical and program-specific problems are available in the dialog boxes. You are presented with the necessary information by clicking the "?" or "Info" buttons (see also Section 5.1).

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

The program is used almost daily during GGU engineering work. Further to this, the program has been thoroughly tested on numerous examples from the literature and in engineering practice. No faults have been found. Nevertheless, liability for completeness and correctness of the program and the manual, and for any damage resulting from incompleteness or incorrectness, cannot be accepted.

2 Licence protection

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

To be able to use the GGU licences in a CmContainer, a runtime environment, the *CodeMeter Runtime Kit*, must be installed on your computer via driver software. To be simplified, we will refer to your computer with installed CodeMeter Runtime Kit and CmContainer as the *CodeMeter licence server*.

We provide 3 alternative CmContainer types, that can be used on your CodeMeter licence server:

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

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

3 Language selection

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

The language preferences can be changed at any time in the "**Info**" menu, using the menu item "**Spracheinstellung**" (for German) or "**Language preferences**" (for English) (see Section 8.6.6).

4 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). After leaving the box you then see six menus in the menu bar:

- File
- Edit
- System
- Output preferences
- Graphics preferences
- Info

After clicking one of these menus, the 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-FOOTING** screen is not refreshed after every alteration.

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

5 Tips and tricks

5.1 "?" and "Info" buttons

Reading of the manual can mostly be dispensed with, because

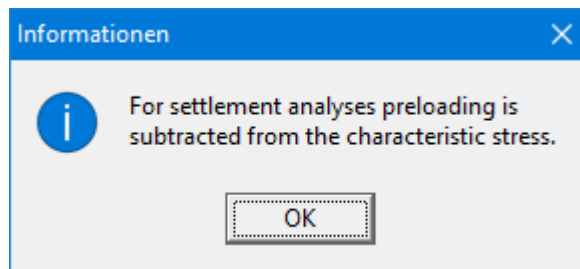
"?" buttons  and "Info" buttons 

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

For example, the following check box can be found in the "Edit/System parameters" dialog box:



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



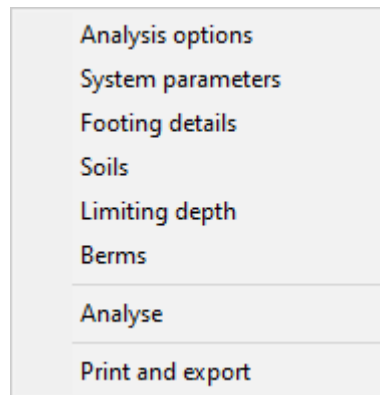
5.2 Function keys

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

- [Esc] refreshes the screen contents and sets the screen back to the given format. This is useful if, for example, you have used the zoom function to display parts of the screen and would like to quickly return to a complete overview.
- [F1] opens the manual file.
- [F2] refreshes the screen without altering the current magnification.
- [F3] opens the menu item "Edit/System parameters".
- [F5] opens the menu item "System/Analyse".
- [F6] opens the menu item "System/Optimise footing width".
- F11] activates the menu item "Output preferences/Move objects".

5.3 Keyboard and mouse

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



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


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.

Furthermore, you can use the mouse wheel to zoom in or out or to move the screen display:

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

From a zoomed representation you always return to the full screen with **[Esc]**.

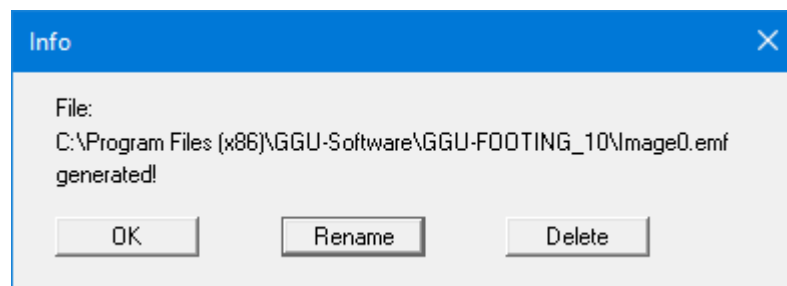
5.4 "Copy/print area" icon

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

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

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

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



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

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

6 Short description and worked examples

6.1 Output sheet design

6.1.1 Generate result graphics

After starting the program, the logo is at first displayed. Select the menu item "**File/New**". Then, the following dialog box is shown.

New

Equation after: DIN 4017:2006

☐ Bearing cap. failure with English designations

Standard:

☒ Partial safety factor concept (EC 7) ?

☐ Partial safety factor concept (DIN 1054:2005)

☐ Global safety factor concept (DIN 1054 old)

Type of analysis:

☒ Multiple footings Analysis and output with: sigma(R,d) ?

☐ Rectangular footing ☐ Circular/annular footing

☐ Do not investigate overturning

Ref. parameter (bearing cap.):

☒ Load ☐ Shear coefficients

Vibrodisplacement compaction

☐ Use ?

☐ Edit shear values too ?

☐ Use reduction due to load spreading ?

☐ Use absolute heights

Designation for abs. height: mNHN

☐ Use kappa correction coefficients (DIN 4019)

☒ Punching analysis to DIN 4017:2006 (recommended)

Project identification:

My first attempt

OK Cancel

Leave the dialog box clicking on the "**DS-P**" button and then select the "**System/Analyse**" menu item. First, an info box concerning the punching verification appears. Then the analysis results with all graphics for the values given by the program are shown on the screen.

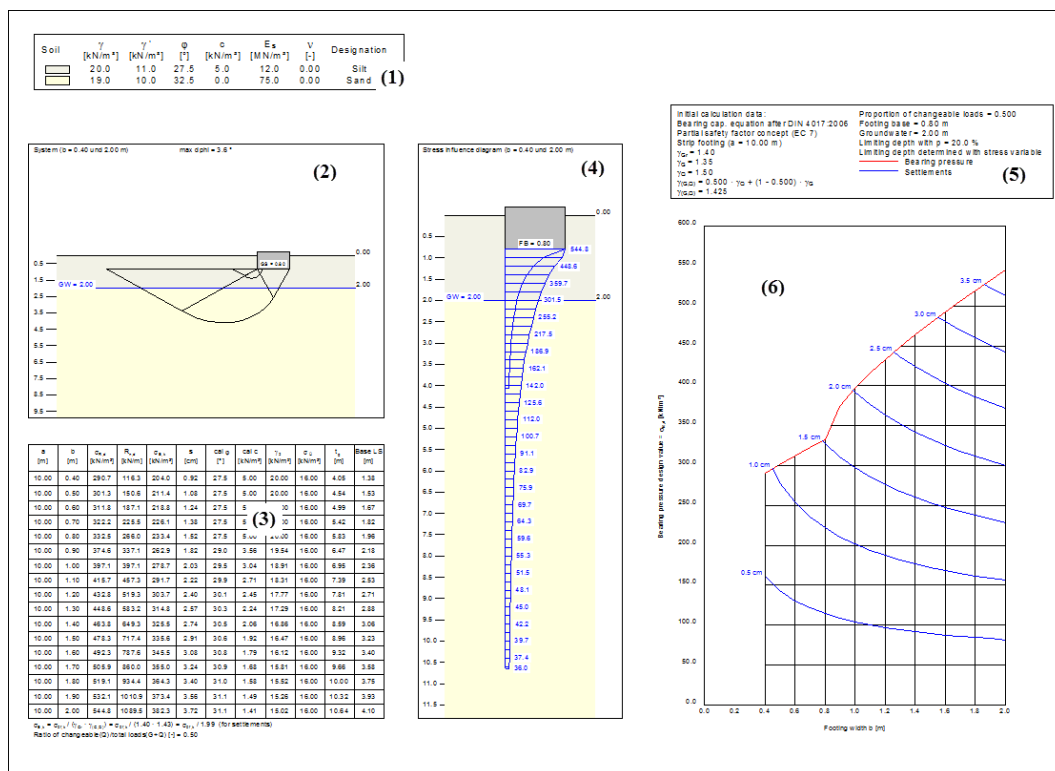


Figure 1 Result screen

The graphic consists of a total of seven elements (also see "**Output preferences**" menu).

- Legend with **soil properties (1)**,
- Legend with **line loads** (if present),
- Graphical representation of the **system (2)** (soil structure, footing and associated logarithmic spiral),
- **Output table (3)** with analysis results for the selected footing types,
- **Stress influence diagram (4)** with the soil structure, footing and associated stress distribution,
- Legend with **analysis fundamentals (5)** such as bearing capacity safety factor, groundwater level, etc.,
- **Footing analysis diagram (6)** with analysis results for bearing resistance design value or allowable soil pressure and settlements.

6.1.2 Move graphic elements

All seven elements can be arranged at will. Select the "**Move objects**" menu item from the "**Output preferences**" menu. Confirm the info box with "**OK**". Move the mouse over the object to be moved, press the left mouse button and keep it pressed whilst moving the object. After releasing the mouse button, the object is shown at its new position.

The size of some objects can also be altered. To alter the size of an object, move the mouse to the top or right-hand edge of the object in question and move the edge with the left mouse button pressed. If you are within the boundaries of a graphic element, the mouse pointer changes to a cross with double arrows pointing *left-right* and *up-down*. If the mouse pointer is at the top or right-hand edge of a resizable element, the mouse pointer changes to a simple double-arrow, pointing either *left-right* or *up-down*. To carry out further alterations, the menu item must be reactivated after each movement.

You can activate this program function without the info box by pressing the [F11] function key.

6.1.3 Edit and turn off graphic elements

You can completely remove graphic elements or alter their size and position. Select one of the following menu items from the "**Output preferences**" menu:

- "**Footing analysis diagram**"/"**Footing plan**",
- "**Soil properties**",
- "**System presentation**",
- "**General legend**",
- "**Output table**"/"**Pad footing**",
- "**Stress influence diagram**",
- "**Line loads**".

This can be achieved even more easily by double-clicking in the appropriate element.

For example, you will see the following dialog box by selecting the "**Output preferences/System presentation**" menu item:

System presentation

☒ Show system

x [mm]: 32.50 Width [mm]: 130.00

y [mm]: 140.00 Height [mm]: 100.00

Show logarithmic spiral

☐ None

☐ For smallest footing width

☐ For largest footing width

☒ For smallest + largest footing width

☐ For all footing widths

Display soil names

☒ None

☐ Left

☐ Right

☐ Central

☐ Label logarithmic spiral

☒ With groundwater

☒ Display bearing cap. failure plane to left

Berm labeling

Load presentation

Presentation height (m): 0.400

☐ Fill colours

☒ Same height for all

Load 'hatching': Hatched

OK Cancel

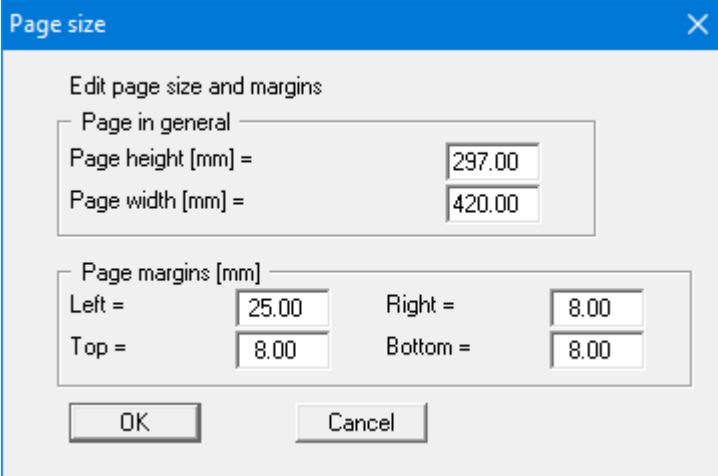
You can switch off the representation of the system using the "**Show system**" check box. Just deactivate the check box. If you leave the dialog box by means of "**OK**", the system presentation will no longer be displayed.

The size and position of the system presentation can be defined in the dialog box. You can also decide whether certain elements and labelling should be displayed in the diagram.

The dialog boxes for other graphic elements are similarly structured.

6.1.4 Edit page size and margins

The program uses A3 format as the default drawing size. However, the size can be edited to suit your needs. Go to the "**Output preferences/Page size and margins**" menu item and enter your personal page format.



The screenshot shows a dialog box titled "Page size" with a close button (X) in the top right corner. The dialog contains two main sections: "Page in general" and "Page margins [mm]".

Page in general

Page height [mm] =	297.00
Page width [mm] =	420.00

Page margins [mm]

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

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

Page margins defines the position of a frame as a distance to the cutting border. This frame encloses the subsequent diagram.

6.2 Example 1: Multiple footings

6.2.1 Enter analysis options (Example 1)

Select the menu item "File/New". When the program is started the check boxes "Partial safety factor concept (EC 7)" and "Multiple footings" are already activated. Alternatively, the standard and the analysis mode can be selected by going to the menu item "Edit/Analysis options".

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

- Equation after:** DIN 4017:2006
- Bearing cap. failure with English designations:** ☐
- Standard:**
 - ☒ Partial safety factor concept (EC 7)
 - ☐ Partial safety factor concept (DIN 1054:2005)
 - ☐ Global safety factor concept (DIN 1054 old)
- Type of analysis:**
 - ☒ Multiple footings
 - ☐ Rectangular footing
 - ☐ Circular/annular footing
 - ☐ Do not investigate overturning
- Analysis and output with:** sigma(R,d)
- Ref. parameter (bearing cap.):**
 - ☒ Load
 - ☐ Shear coefficients
- Vibrodisplacement compaction:**
 - ☐ Use
 - ☐ Edit shear values too
 - ☐ Use reduction due to load spreading
- Use absolute heights:**
 - ☐ Use absolute heights
 - Designation for abs. height: mNHN
- Use kappa correction coefficients (DIN 4019):** ☐
- Punching analysis to DIN 4017:2006 (recommended):** ☒
- Project identification:** 'Multiple footings' worked example

After confirming with "OK" you can select the load case. Choose "DS-P". The partial safety factors are used in the system data.

You can enter a project identification for the current problem, which is subsequently shown in the **General legend**. A project identification is entered when selecting the type of system in the above dialog box or via the **"Edit/Project identification"** menu item.

6.2.2 Enter system parameters (Example 1)

The appropriate partial safety factors have already been entered into the **"Edit/System parameters"** menu item. The quantities described below can be entered or edited. Use the default values for the example.

- **"Partial FOS (bearing capacity/permanent actions/changeable actions):"/"Bearing capacity FOS [-]:"**
safety factor η according to DIN 4017
- **"Groundwater table [m]:"**
measured from ground level in m
- **"Footing base [m]:"**
measured from ground level in m
- **"Slope inclination [°]:"**
positive values only (= falling slope)

The older version of DIN 4017 did not envisage taking slope inclination into consideration. If you are not investigating slopes in accordance with DIN 4017:2006 (the appropriate check box is deactivated, see below), you should therefore examine the approach for your problem used in "**Theoretical principles**" as a matter of urgency. Input of slope inclination here does not represent a substitute for slope stability analysis according to DIN 4084!!!. The program assumes that the slope is stable.

- **"Berm width [m]:"**
measured in m beside the footing

The current issue of DIN 4017 does not envisage consideration of berms. You should therefore examine the "**Theoretical principles**" for an approach to your problem.

- **"Slope to DIN 4017:2006"**
Previous versions of DIN 4017 did not provide for consideration of slopes. We therefore installed our own model for taking slopes into consideration.
- **"Base inclination [°]:"**
inclination of the footing base in [°]
- **"Preloading [kN/m²]:"**
For settlement analyses, a preloading in kN/m² can be subtracted from the current footing 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.
- **"Surcharge (bearing capacity) [kN/m²]:"**
The bearing capacity surcharge defines a stress on the ground, which generates a stress on the failure body in addition to the embedment depth.
- **"Surcharge (limiting depth) [kN/m²]:"**
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 is greater than 20 % of the stresses caused by the footing. You can enter the ground surcharge for this limiting depth calculation, here. Input is only of importance if you calculate the limiting depth as a function of the surcharge stress (see also Section 6.2.5).
- **"H/V [-]:"**
Ratio of the horizontal force H to the vertical force V
In the bearing capacity equation in DIN 4017, a horizontal force can be taken into consideration. In **GGU-FOOTING** this force can be defined as a ratio to the vertical force.
- **"Calculate using depth coefficients"**
In the bearing capacity equations in DIN 4017, the effects of the shear strength of the soil above the footing base are neglected. According to Section 7.1 ("**Theoretical principles/Analysis of bearing capacity failure**"), this influence can be considered by applying the depth coefficients d_c and d_d . Four different procedures are available, which can be selected via the dropdown menu. The depth coefficient equations can be found in the above-mentioned Section 6.1 of this manual.
- **"Friction angle for bearing capacity failure"**
Three of the four depth coefficient equations use a friction angle:
Check box deactivated ==> mean friction angle at footing base will be adopted.
Check box activated ==> friction angle for bearing capacity analysis will be adopted.
- **"Depth coefficients for slopes too"**
If the slope inclination $\neq 0$, the parameters d_c and d_d are set to "**1.0**" if the check box is deactivated. If the check box is activated, the influence of the shear strength of the soil above the footing base is also taken into consideration in a slope.

- **"Verify bearing capacity for short side only"**
The program verifies the bearing capacity safety for the short and the long sides of the footing. In rare, exceptional cases (see the included file "**DIN 4017 new Example 6 a flexible.gdg**") it may be expedient to verify the short side only. You should then activate this check box.
- **"Limit soil pressure"** group box
For predefined footing types, the program calculates the allowable soil pressure as a function of the footing width. If footing pressures are determined, which are too high for your taste, in particular for large footing widths, possibly due to excessively high settlement or for simple safety reasons (the structural engineer always computes with the allowable values provided by you), you can limit the allowable footing pressure to a defined level. Activate the **"Apply"** check box and enter the required maximum value.
- **"5° condition"** group box
According to DIN 4017 the soil properties can be averaged out for stratified ground. The condition for this is that the averaged friction angle demonstrates a maximum deviation of 5° to the true friction angles. The program checks this condition. If it is not adhered to, a warning signal is given after the analysis. Alternatively, you can have the program reduce all friction angles in stages, which lie above the average value. The size of the reduction is specified with **"Decrement [°]"**. Small decrement values mean long computation times with high precision. A reduction only takes place if the **"Check and correct"** check box is activated.
- **"Ratio changeable/total loads [-]:"** group box
The failure stress is first divided by the bearing capacity partial safety factor. This gives the bearing capacity stress. A further reduction by the partial safety factors for actions is necessary for calculation of an allowable footing pressure or acceptable bearing pressure. In load case 1, e.g., a partial safety factor of $\gamma_m(G) = 1.35$ for permanent loads and one of $\gamma_m(Q) = 1.50$ for changeable loads is demanded. From a ratio of 0.40, for example, a ***total partial factor*** results from:
$$\text{Ratio} \cdot 1.50 + (1.0 - \text{ratio}) \cdot 1.35 = 1.410$$
- **"Punching analysis"** group box
Punching analysis can be activated here. DIN 4017:2006 describes a new punching analysis method. To use this new analysis method the **"Punching analysis using new method (recommended)"** check box in the 'File/New' dialog box is activated by default when the program starts. The punching analysis equations given in DIN 4017:2006 differentiate between flexible and rigid footings. Accordingly, you will activate or deactivate the **"Flexible footing"** check box.

6.2.3 Define footings (Example 1)

In the "Edit/Footing details" menu item you define the size and shape of the footing.

The screenshot shows a software dialog box titled "Footing shape". It is divided into two main sections. The first section, "Define footing width b", contains three input fields: "Start width [m]:" with the value "0.400", "End width [m]:" with the value "2.000", and "Width increment [m]:" with the value "0.100". The second section, "Type of footing (a = footing length)", contains three radio button options. The first option, "Strip footing with a [m] =", is selected and has a value of "10.000". The second option, "Pad footing with a/b =", has a value of "1.000". The third option, "Pad footing with a [m] =", has a value of "2.500". At the bottom of the dialog are "OK" and "Cancel" buttons.

The "Start width", the "End width" and the "Delta width" are entered in this box. You thus define the width range to be investigated. Thus, using the example given above, the widths 0.4, 0.5, 0.6, ... to 2.0 m will be investigated.

You must also enter whether a strip or a pad footing is to be analysed. The example above would analyse a strip footing. When analysing a strip footing, enter the strip length applicable to all widths. This strip length is required for the subsequent settlement analysis.

To analyse a pad footing you define whether a constant length/width ratio (a/b) or a constant footing length (a) is analysed. If you have activated a pad footing with constant length a , you may find, for an appropriate start width, that $a < b$. In this case a warning message is issued to inform you that $a = b$.

6.2.4 Define soils (Example 1)

In the "**Edit/Soils**" menu item, enter the values for the soils present as shown in the following dialog box.

	Designation	Base	gam	gam'	phi	c	Es	nu
		[m]	[kN/m³]	[kN/m³]	[°]	[kN/m²]	[MN/m²]	[-]
1	Silt	2.00	20.0	11.0	27.5	5.0	12.0	0.00
2	Sand	6.00	19.0	10.0	32.5	0.0	75.0	0.00

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

The layer depths (base) are with reference to ground level and are positive downwards, as are all other inputs. But if you have activated the "**Use absolute heights**" check box in the dialog box in the "**File/New**" menu item, layer depths are positive upwards. In addition, you must enter a value in the "**Edit/Ground level**" dialog box. In this case you can enter the layer depths and other height-referenced values e.g. in m AD.

Furthermore:

- gam = unit weight of soil above groundwater
- gam' = buoyant unit weight
- phi = soil friction angle
- c = cohesion of soil
- Es = soil constrained modulus
- nu = Poisson's ratio of soil

If the constrained modulus is used for analysis, nu equals 0.0 for all soils. The value of nu displayed in the soil properties legend can then be switched off (see Section 8.4.3).

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

If several soils have been entered, and you would like to delete a soil which is not at the end of the list, you can assign this soil a **large** depth (e.g. 99.0). Then select the **"Sort"** button. The soil will now be at the end of the table. With the **"Edit number of soils"** button you can reduce the number by 1. The soil has now been deleted from the list, without a lot of typing.

6.2.5 Select the limiting depth (Example 1)

In the **"Edit/Limiting depth"** menu item, the type of limiting depth calculation can be defined in three different ways.

Limiting depth

Limiting depth

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

☐ Limiting depth = $x \cdot b'$
x: 1.500

☒ Limiting depth of p %
p: 20.0

Stress influence diagram
Increment [m] (for drawing): 0.200

OK Cancel

If the limiting depth lies below the depth of the lowest soil layer, the values of this layer are used in the settlement analysis.

For settlement analyses, a preloading in kN/m² can be subtracted from the current footing pressure. Settlement analysis will then be performed with the reduced values. The overburden stress is also reduced by the preloading value when calculating the limiting depth.

6.2.6 Analyse system (Example 1)

After editing one of the previous dialog boxes, the system is first represented without the analysis results. If you want to have the footing reanalysed with the new values go to the "**System**" menu and select "**Analyse**". Alternatively, press the [F5] function key or click on the *calculator* in the tool bar.

First, an info box concerning the punching verification appears. Then, the analysis results are presented to you in the various drawing elements (table and graphics). The current screen contents can be sent to the printer at any stage of project processing. Select the "**Print and export**" menu item from the "**File**" menu (see Section 8.1.9).

6.3 Example 2: Single rectangular footing

6.3.1 Enter analysis options (Example 2)

Select the menu item "**File/New**". Activate the "**Rectangular footing**" switch. Alternatively, you can alter standard and analysis mode in the "**Edit/Analysis options**" dialog box.

The 'New' dialog box is used to configure analysis parameters. It includes the following sections:

- Equation after:** A dropdown menu set to 'DIN 4017:2006'.
- ☐ Bearing cap. failure with English designations
- Standard:** Radio buttons for:
 - ☒ Partial safety factor concept (EC 7) [?]
 - ☐ Partial safety factor concept (DIN 1054:2005)
 - ☐ Global safety factor concept (DIN 1054 old)
- Type of analysis:** Radio buttons for:
 - ☐ Multiple footings
 - ☒ Rectangular footing
 - ☐ Circular/annular footing
- Analysis and output with:** A dropdown menu set to 'sigma(R,d)' [?]
- ☐ Do not investigate overturning
- Ref. parameter (bearing cap.):** Radio buttons for:
 - ☒ Load
 - ☐ Shear coefficients
- Vibrodisplacement compaction:**
 - ☐ Use
 - ☐ Edit shear values too [?]
 - ☐ Use reduction due to load spreading [?]
- ☐ Use absolute heights
- Designation for abs. height:** A text box containing 'mNHN'
- ☐ Use kappa correction coefficients (DIN 4019)
- ☒ Punching analysis to DIN 4017:2006 (recommended)
- Project identification:** A text box containing 'Rectangular footing' worked example

Buttons: OK, Cancel

Confirm with "**OK**". After this you can select the load case. Choose "**DS-P**". The partial safety factors given are used in the system data. You can enter a project identification for the current problem, which is subsequently shown in the *General legend*.

6.3.2 Enter system parameters (Example 2)

The appropriate partial safety factors have already been entered into the "**Edit/System parameters**" menu item. Enter '1.00' in the "**Footing base**" input box.

System parameters

Partial FOS (bearing capacity): 1.40

Partial FOS (permanent actions): 1.35

Partial FOS (changeable actions): 1.50

Groundwater table [m]: 2.00

Footing base [m]: 1.00

Slope inclination [°]: 0.00

Berm width [m]: 0.00

☒ Slope to DIN 4017:2006

Base inclination [°]: 0.00

Preloading [kN/m²]: 0.00 ?

Surcharge (bearing cap.) [kN/m²]: 0.00 ?

Surcharge (limiting depth) [kN/m²]: 0.00 ?

Depth coefficients

☐ Calculate using depth coefficients ?

Depth coeff. after: GCOC (ES) ?

☐ Friction angle for bearing capacity failure ?

☐ Depth coefficients for slopes too ?

☒ Additional bearing cap. analysis with V(permanent), M(total) and H(total)

☐ Calculate settlements below permanent loads

☐ Verify bearing capacity for short side only ?

5° condition

☒ Check and correct ?

Decrement [°]: 0.10

Punching analysis

☒ Carry out punching analysis ?

☒ Flexible footing

EQU analysis

☒ Verify

gamma(G,dst): 1.10

gamma(G,stb): 0.90

gamma(Q,dst): 1.50

Done Cancel

The dialog box largely corresponds to that of the "**Multiple footings**" mode. The input values are described in Section 6.2.2. In addition, when analysing pad footing (*rectangle* or *circle/annulus*) with the *partial safety factor concept*, the following two check boxes can be activated:

- "**Additional bearing cap. analysis with V(permanent), M(total) and H(total)**"
Bearing capacity is analysed with the total loads = permanent + changeable loads. However, if the load combination V(permanent), M(total) and H(total) is possible because the loads can occur independently, this load combination may provide unfavourable results. It is therefore possible to investigate this load combination separately.
- "**Calculate settlements below permanent loads**"
If this check box is activated, only the settlements below permanent loads will be calculated. Otherwise, settlements below total loads will be calculated (= permanent + changeable loads).

6.3.3 Define footing (Example 2)

Select the menu item "**Edit/Footing details**".

Rectangular footing

Dimensions

Length a [m]: 4.000 Width b [m]: 2.000

Equivalent width from circle/annulus

Stress, characteristic

Forces act at the footing base level

	Permanent	Changeable
Vertical force [kN]:	1250.00	0.00
Horizontal force (x) [kN]:	125.00	0.00
Horizontal force (y) [kN]:	0.00	0.00
Moment (x) [kN·m]:	500.00	0.00
Moment (y) [kN·m]:	500.00	0.00

Self weight

☐ Automatic self weight

Gamma concrete [kN/m³]: 25.00

Slip resistance

☐ Verify slip resistance

Partial FOS gamma(R,h) [-]: 1.10

☐ Sliding resistance with user-defined angle

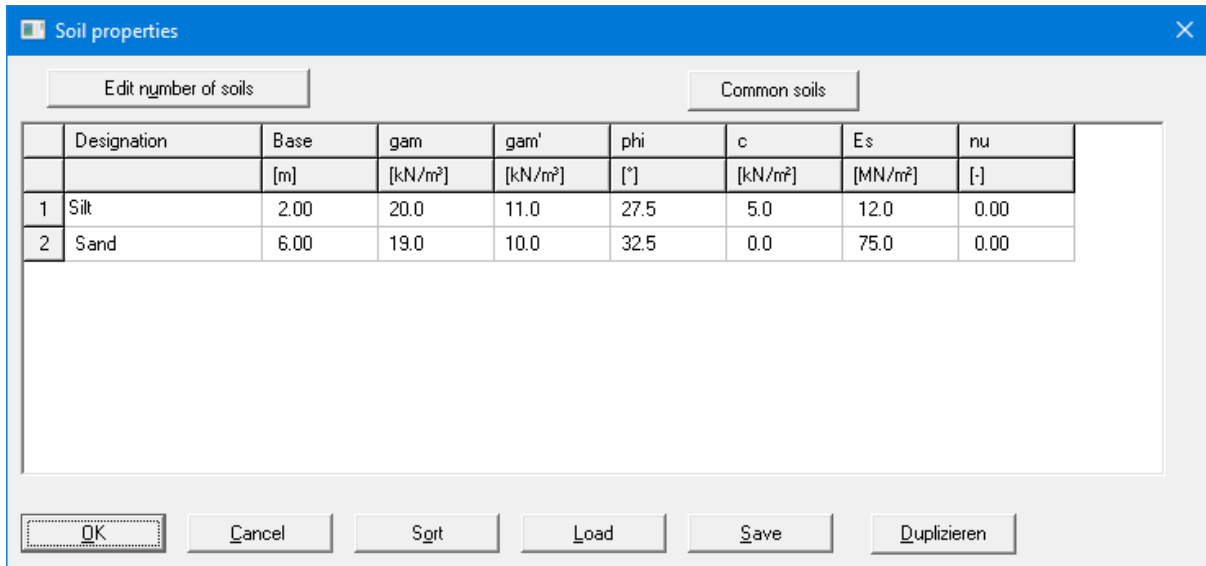
User-defined angle [°]: 30.00

OK Cancel

According to the new *partial safety factor concept*, you can define differing stresses for permanent and changeable loads.

6.3.4 Define soils (Example 2)

If you have previously analysed the "**Multiple footings**" worked example, you can leave the soil properties unchanged. Otherwise, enter the values for the soils present in the menu item "**Edit-it/Soils**", as shown in the following dialog box.



The dialog box titled "Soil properties" contains two tabs: "Edit number of soils" and "Common soils". The "Edit number of soils" tab is active, showing a table with 9 columns: Designation, Base, gam, gam', phi, c, Es, and nu. The table has two rows of data: Row 1 (Silt) and Row 2 (Sand). Below the table are buttons for OK, Cancel, Sort, Load, Save, and Duplizieren.

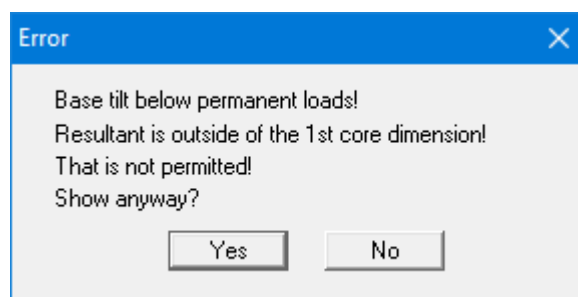
	Designation	Base	gam	gam'	phi	c	Es	nu
		[m]	[kN/m²]	[kN/m²]	[°]	[kN/m²]	[MN/m²]	[-]
1	Silt	2.00	20.0	11.0	27.5	5.0	12.0	0.00
2	Sand	6.00	19.0	10.0	32.5	0.0	75.0	0.00

The input values are more closely described in Section 6.2.4.

6.3.5 Analyse system (Example 2)

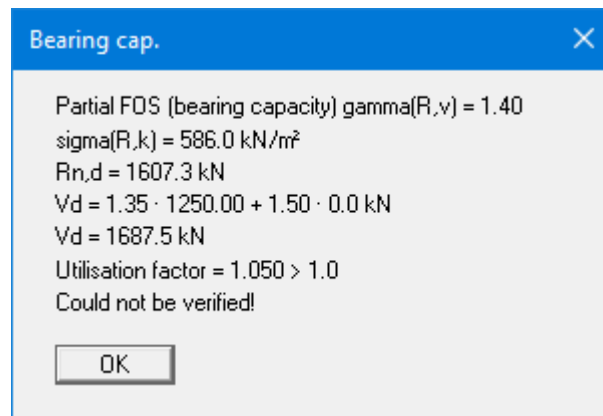
After editing one of the previous dialog boxes, the system is first represented without the analysis results. If you want to have the footing reanalysed with the new values, go to the "**System**" menu and select "**Analyse**". Alternatively, press the [F5] function key or click on the *calculator* in the tool bar.

After a short time, the following message appears.



An error dialog box with a blue header and a close button. The text inside reads: "Base tilt below permanent loads! Resultant is outside of the 1st core dimension! That is not permitted! Show anyway?". At the bottom are "Yes" and "No" buttons.

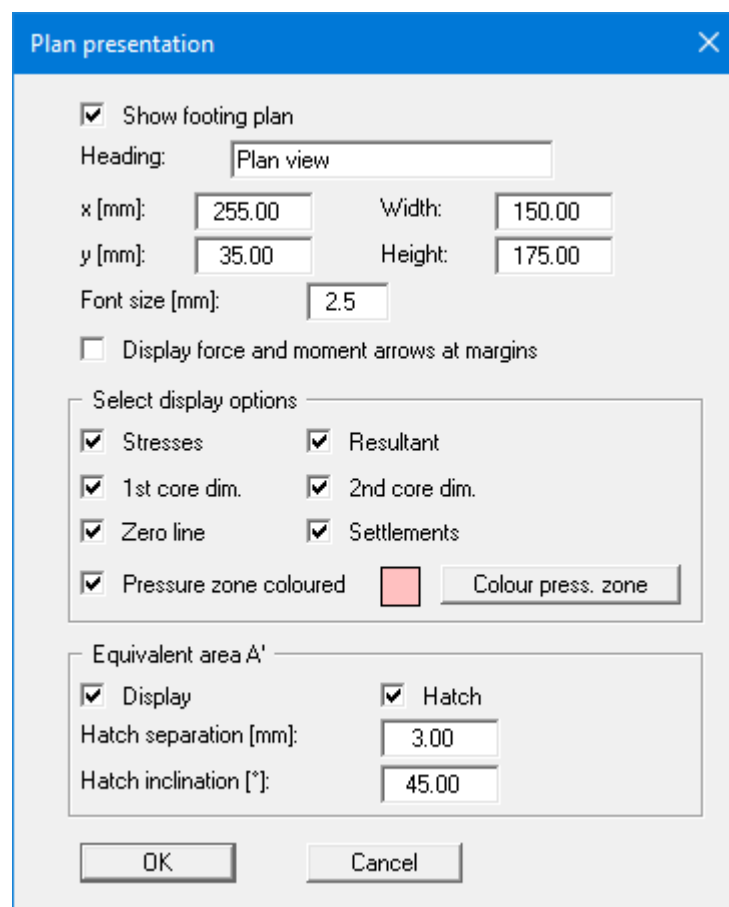
If you confirm with "**Yes**" you will see an info box concerning the punching verification. Then a further window with information on the safety and utilisation factors is displayed.



If you close the window with "OK" the diagram and all analysis results are presented on the screen. If the results are too small to read, you can use the zoom. To do this, hold the [Ctrl]-key and the left mouse button and mark the region to be enlarged on the diagram. Press the [Esc] key to return to the full screen representation.

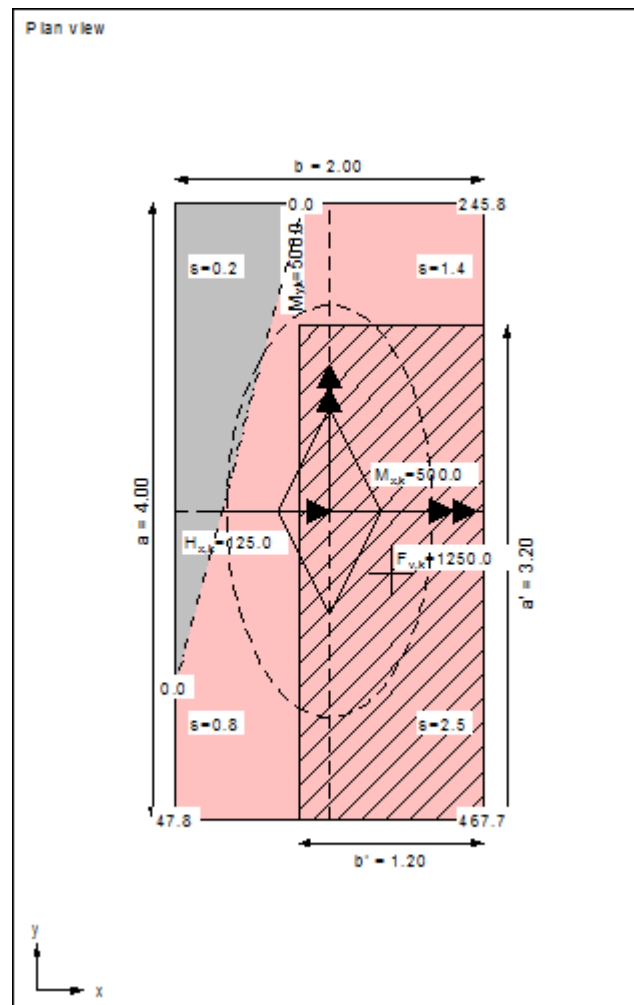
6.3.6 Represent results in the plan view of footing (Example 2)

Select the "Output preferences/Footing plan" menu item or click in the "Plan view" drawing element at the right side of the diagram. (Note: The footing analysis diagram is here when analysing in "Multiple footings" mode).



Activate the "**Pressure zone coloured**" check box. Press the "**Colour press. zone**" button and select a quiet red from the box (or perhaps not). Confirm with "**OK**".

In the plan view at the right the pressure zone is now coloured.



The graphic also contains:

- the characteristic forces $M_{x,k}$, $M_{y,k}$, $H_{x,k}$ and the vertical force $F_{v,k}$ (= resultant, marked by a cross).
- the 1st core dimension.
If the resultant is within this rhombic area, the complete footing is under pressure.
- the 2nd core dimension.
If the resultant is within this elliptical area, a base tilt will occur.
- the equivalent area A' .
This region is additionally labelled at the edge with a' and b' .
- the four corner stresses or, in our case (with base tilt), the stresses at the five corners of the pressure zone.
- the settlements at the four characteristic points.

The resulting torsion can be read from the pad footing legend at the lower left of the screen.

```

Settlement from total loads:
Limiting depth  $t_g$  = 6.14 m b. GL
Settlement (mean of all CPs) = 1.22 cm
Settlement of CPs:
Top left = 0.18 cm
Top right = 1.43 cm
Bottom left = 0.79 cm
Bottom right = 2.49 cm
Torsion (x) (C P) = 1 : 354.0
Torsion (y) (C P) = 1 : 100.1

```

6.3.7 Consider depth coefficients (Example 2)

In the bearing capacity equations in DIN 4017, the effects of the shear strength of the soil above the footing base are neglected. According to Section 7.1 ("**Theoretical principles/Analysis of bearing capacity failure**"), this influence can be considered by applying the depth coefficients d_c and d_s .

An example will demonstrate the influence of taking depth coefficients into consideration. First activate the following check box in the "**Edit/System parameters**" menu item:

Depth coefficients

☒ Calculate using depth coefficients ?

Depth coeff. after: IEG7 (SE) ?

☐ Friction angle for bearing capacity failure ?

☐ Depth coefficients for slopes too ?

Analyse the system once again. Pressing [F5] is the fastest way to do it. After a short wait the base tilt message appears again, and after confirmation with "**Yes**", the following information:

Bearing cap. X

Partial FOS (bearing capacity) $\gamma(R,v) = 1.40$
 $\sigma(R,k) = 714.8 \text{ kN/m}^2$
 $R_{n,d} = 1960.5 \text{ kN}$
 $V_d = 1.35 \cdot 1250.00 + 1.50 \cdot 0.0 \text{ kN}$
 $V_d = 1687.5 \text{ kN}$
 Utilisation factor = 0.861 <= 1.0
 Verified!

OK

The utilisation factor was improved from **1.050** to **0.861** by considering the shear strength of the soil above the footing base and is thus below the maximum allowable value of **1.0**. Naturally, the base tilt is still there.

6.3.8 Represent results in the pad footing legend (Example 2)

Select the "Output preferences/Pad footing" menu item or click in the "Pad footing" drawing element at the left side of the diagram. (Note: An output table with the result data for all footings is here when analysing in "Multiple footings" mode).

Activate all the check boxes in the "Select display options" group box. Confirm with "OK" and then zoom in on the tabulated data in the bottom left corner of the screen. You can now view all principal analysis results. The maximum utilisation factor of **0.861** is parallel to the side x of the footing. Parallel to y the utilisation factor is **0.687**.

Rectangle footing results: Loads = Permanent / Changeable Vertical load $F_{v,k} = 1250.00 / 0.00$ kN Horizontal force $F_{h,x,k} = 125.00 / 0.00$ kN Horizontal force $F_{h,y,k} = 0.00 / 0.00$ kN Moment $M_{x,k} = 500.00 / 0.00$ kN·m Moment $M_{y,k} = 500.00 / 0.00$ kN·m Length $a = 4.000$ m Width $b = 2.000$ m Below permanent loads: Eccentricity $e_x = 0.400$ m Eccentricity $e_y = -0.400$ m Resultant in 2nd core Length $a' = 3.200$ m Width $b' = 1.200$ m Below total loads: Eccentricity $e_x = 0.400$ m Eccentricity $e_y = -0.400$ m Resultant in 2nd core Length $a' = 3.200$ m Width $b' = 1.200$ m Bearing capacity: Punching investigated, but not governing. Partial FOS (bearing capacity) $\gamma_{R,v} = 1.40$	$G_{R,k} / G_{R,d} = 714.8 / 510.55$ kN/m ² $R_{n,k} = 2744.74$ kN $R_{n,d} = 1960.53$ kN $V_d = 1.35 \cdot 1250.00 + 1.50 \cdot 0.00$ kN $V_d = 1687.50$ kN μ (parallel to x) = 0.861 μ (parallel to y) = 0.687 Cohesion term = 372.88 kN (k) Width term = 554.78 kN (k) Depth term = 1817.07 kN (k) cal $\phi = 30.0^\circ$ Cal $c = 2.52$ kN/m ² Cal $\gamma_2 = 17.86$ kN/m ² Cal $G_d = 20.00$ kN/m ² Base of log. spiral = 2.58 m b. GL Length log. spiral = 6.52 m Area of log. spiral = 5.35 m ² Bearing capacity coeff. (x): $N_{d0} = 30.25$; $N_{d0} = 18.50$; $N_{b0} = 10.12$ Shape coeff. (x): $V_c = 1.198$; $V_d = 1.188$; $V_b = 0.888$ Gradient coeff. (x): $i_c = 0.824$; $i_d = 0.834$; $i_b = 0.750$ Depth coefficients (x): $d_c = 1.292$; $d_d = 1.292$; $d_b = 1.000$ Bearing capacity coeff. (y):	$N_{c0} = 34.94$; $N_{d0} = 22.68$; $N_{b0} = 13.45$ Shape coeff. (y): $V_c = 1.207$; $V_d = 1.198$; $V_b = 0.888$ Gradient coeff. (y): $i_c = 0.826$; $i_d = 0.834$; $i_b = 0.750$ Depth coefficients (y): $d_c = 1.109$; $d_d = 1.109$; $d_b = 1.000$ Settlement from total loads: Limiting depth $t_{gl} = 6.14$ m b. GL Settlement (mean of all CPs) = 1.22 cm Settlement of CPs: Top left = 0.18 cm Top right = 1.43 cm Bottom left = 0.79 cm Bottom right = 2.49 cm Torsion (x) (CP) = 1 : 354.0 Torsion (y) (CP) = 1 : 100.1 Torsional stiffness: $k_{e,x} = 177.0$ MN·m/rad $k_{e,y} = 50.0$ MN·m/rad EQU analysis: Governing factor: footing width $M_{xib} = 1250.0 \cdot 2.00 \cdot 0.5 \cdot 0.90 = 1125.0$ $M_{dal} = 500.0 \cdot 1.10 = 550.0$ $\mu_{EQU} = 550.0 / 1125.0 = 0.489$
--	--	---

6.4 Example 3: Single circular footing (EC 7)

6.4.1 Enter analysis options (Example 3)

Select the menu item "**File/New**". Activate the "**Circular/annular footing**" switch. Alternatively, you can alter standard and analysis mode in the "**Edit/Analysis options**" dialog box.

The 'New' dialog box is used to configure analysis parameters. It includes the following sections:

- Equation after:** A dropdown menu set to 'DIN 4017:2006'.
- Bearing cap. failure with English designations:** An unchecked checkbox.
- Standard:** Three radio button options: 'Partial safety factor concept (EC 7)' (selected), 'Partial safety factor concept (DIN 1054:2005)', and 'Global safety factor concept (DIN 1054 old)'. Each option has a '?' help button.
- Type of analysis:** Three radio button options: 'Multiple footings', 'Rectangular footing', and 'Circular/annular footing' (selected). There is also an unchecked checkbox for 'Do not investigate overturning'.
- Analysis and output with:** A dropdown menu set to 'sigma(R,d)' with a '?' help button.
- Ref. parameter (bearing cap.):** Two radio button options: 'Load' (selected) and 'Shear coefficients'.
- Vibrodisplacement compaction:** Three unchecked checkboxes: 'Use', 'Edit shear values too' (with a '?' help button), and 'Use reduction due to load spreading' (with a '?' help button).
- Use absolute heights:** An unchecked checkbox.
- Designation for abs. height:** A text field containing 'm'.
- Use kappa correction coefficients (DIN 4019):** An unchecked checkbox.
- Punching analysis to DIN 4017:2006 (recommended):** A checked checkbox.
- Project identification:** A text field containing ''Circular footing' worked example'.
- Buttons:** 'OK' and 'Cancel' buttons at the bottom.

Confirm with "**OK**". After this you can select the load case. Choose "**DS-P**". The partial safety factors given are used in the system data. You can enter a project identification for the current problem, which is subsequently shown in the *General legend*.

6.4.2 Enter system parameters (Example 3)

The appropriate partial safety factors have already been entered into the "Edit/System parameters" menu item.

System parameters

Partial FOS (bearing capacity): 1.40

Partial FOS (permanent actions): 1.35

Partial FOS (changeable actions): 1.50

Groundwater table [m]: 2.00

Footing base [m]: 0.80

Slope inclination [°]: 0.00

Berm width [m]: 0.00

☒ Slope to DIN 4017:2006

Base inclination [°]: 0.00

Preloading [kN/m²]: 0.00 ?

Surcharge (bearing cap.) [kN/m²]: 0.00 ?

Surcharge (limiting depth) [kN/m²]: 0.00 ?

Depth coeff.

☐ Calculate using depth coefficients ?

Depth coeff. after: IEG7 (SE) ?

☐ Friction angle for bearing capacity failure ?

☐ Depth coefficients for slopes too ?

☒ Additional bearing cap. analysis with V(permanent), M(total) and H(total)

☐ Calculate settlements below permanent loads

5° condition

☒ Check and correct ?

Decrement [°]: 0.10

Punching analysis

☒ Carry out punching analysis ?

☒ Flexible footing

EQU analysis

☒ Verify

gamma(G,dst): 1.10

gamma(G,stb): 0.90

gamma(Q,dst): 1.50

Done Cancel

The dialog box largely corresponds to that of the "Multiple footings" mode. The input values are described in Section 6.2.2. In addition, when analysing pad footing (*rectangle* or *circle/annulus*) with the *partial safety factor concept*, the following two check boxes can be activated:

- **"Additional bearing cap. analysis with V(permanent), M(total) and H(total)"**
Bearing capacity is analysed with the total loads = permanent + changeable loads. However, if the load combination V(permanent), M(total) and H(total) is possible because the loads can occur independently, this load combination may provide unfavourable results. It is therefore possible to investigate this load combination separately.
- **"Calculate settlements below permanent loads"**
If this check box is activated, only the settlements below permanent loads will be calculated. Otherwise, settlements below total loads will be calculated (= permanent + changeable loads).

6.4.3 Define footing (Example 3)

Select the menu item "**Edit/Footing details**".

Circular/annular footing

Dimensions

Diameter D [m]: 3.500

Diameter (internal) d [m]: 0.000

Equivalent area bei Circeling über: Moment of inertia

Stress, characteristic

Forces act at the footing base level

	Permanent	Changeable
Vertical force [kN]:	1250.00	0.00
Horizontal force (x) [kN]:	0.00	0.00
Horizontal force (y) [kN]:	-125.00	0.00
Moment (x) [kN·m]:	500.00	0.00

Self weight

☐ Automatic self weight

Gamma concrete [kN/m³]: 25.00

Slip resistance

☐ Verify slip resistance

Partial FOS gamma(R,h) [-]: 1.10

☐ Sliding resistance with user-defined angle

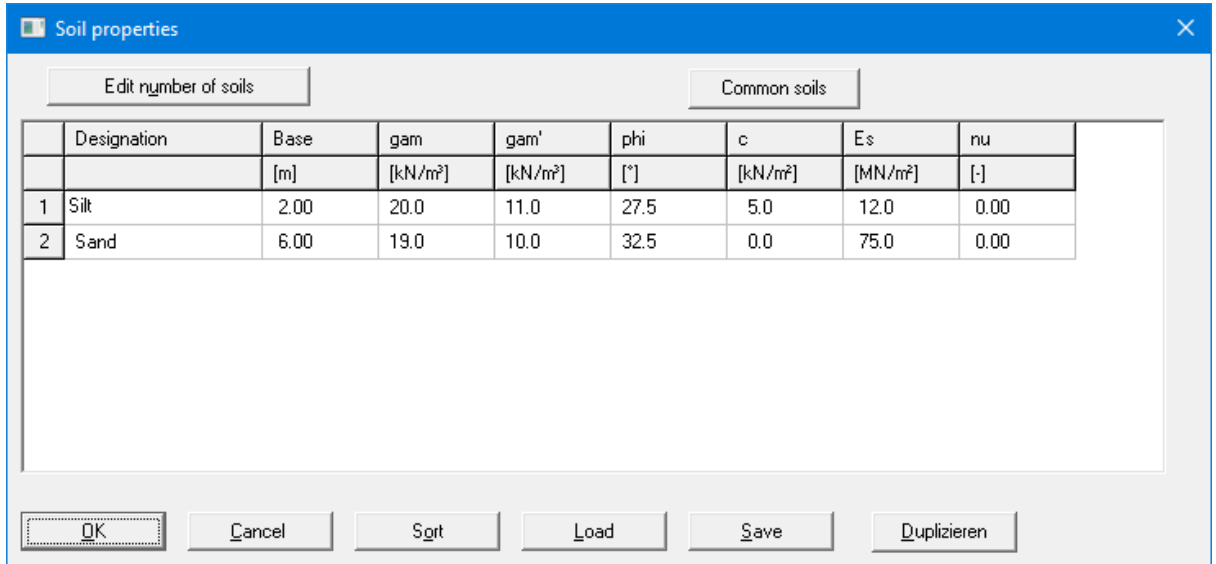
User-defined angle [°]: 30.00

OK Cancel

Enter the diameter shown for the circular footing. The analysis is performed on an equivalent area for an annular footing (see "**Theoretical principles circle/annulus**", Section 7.2). According to the new *partial safety factor concept*, you can define differing stresses for permanent and changeable loads.

6.4.4 Define soils (Example 3)

If you have previously analysed the "**Multiple footings**" worked example, you can leave the soil properties unchanged. Otherwise, enter the values for the soils present in the menu item "**Edit/Soils**", as shown in the following dialog box.



The "Soil properties" dialog box features a title bar with a close button. It contains two tabs: "Edit number of soils" and "Common soils". Below the tabs is a table with the following data:

	Designation	Base	gam	gam'	phi	c	Es	nu
		[m]	[kN/m²]	[kN/m²]	[°]	[kN/m²]	[MN/m²]	[-]
1	Silt	2.00	20.0	11.0	27.5	5.0	12.0	0.00
2	Sand	6.00	19.0	10.0	32.5	0.0	75.0	0.00

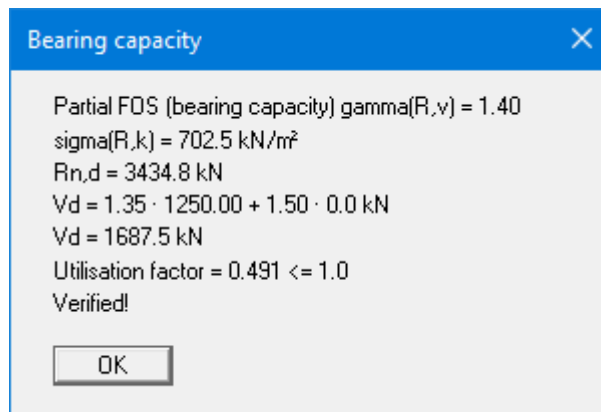
At the bottom of the dialog are buttons for "OK", "Cancel", "Sort", "Load", "Save", and "Duplizieren".

The input values are more closely described in Section 6.2.4.

6.4.5 Analyse system (Example 3)

After editing one of the previous dialog boxes, the system is first represented without the analysis results. If you want to have the footing reanalysed with the new values, go to the "**System**" menu and select "**Analyse**". Alternatively, press the [F5] function key or click on the *calculator* in the tool bar.

First, an info box concerning the punching verification appears. Then a further window with information on the safety and utilisation factors is displayed.



The "Bearing capacity" info box displays the following calculation results:

Partial FOS (bearing capacity) $\gamma_{R,v} = 1.40$
 $\sigma_{R,k} = 702.5 \text{ kN/m}^2$
 $R_{n,d} = 3434.8 \text{ kN}$
 $V_d = 1.35 \cdot 1250.00 + 1.50 \cdot 0.0 \text{ kN}$
 $V_d = 1687.5 \text{ kN}$
Utilisation factor = $0.491 \leq 1.0$
Verified!

An "OK" button is located at the bottom.

If you close the window with "OK" the diagram and all analysis results are presented on the screen. If the results are too small to read, you can use the zoom. To do this, hold the [Ctrl]-key and the left mouse button and mark the region to be enlarged on the diagram. Press the [Esc] key to return to the full screen representation.

6.4.6 Represent results in the plan view of footing (Example 2)

Select the "Output preferences/Footing plan" menu item or click in the "Plan view" drawing element at the right side of the diagram. (Note: The footing analysis diagram is here when analysing in "Multiple footings" mode).

Plan presentation

☒ Show footing plan

Heading: Plan view

x [mm]: 280.00 Width: 120.00

y [mm]: 17.50 Height: 140.00

Font size [mm]: 2.5

☐ Display force and moment arrows at margins

Select display options

☒ Stresses ☒ Resultant

☒ 1st core dim. ☒ 2nd core dim.

☒ Zero line ☒ Settlements

☒ Pressure zone coloured Colour press. zone

Equivalent area A'

☒ Display ☒ Hatch

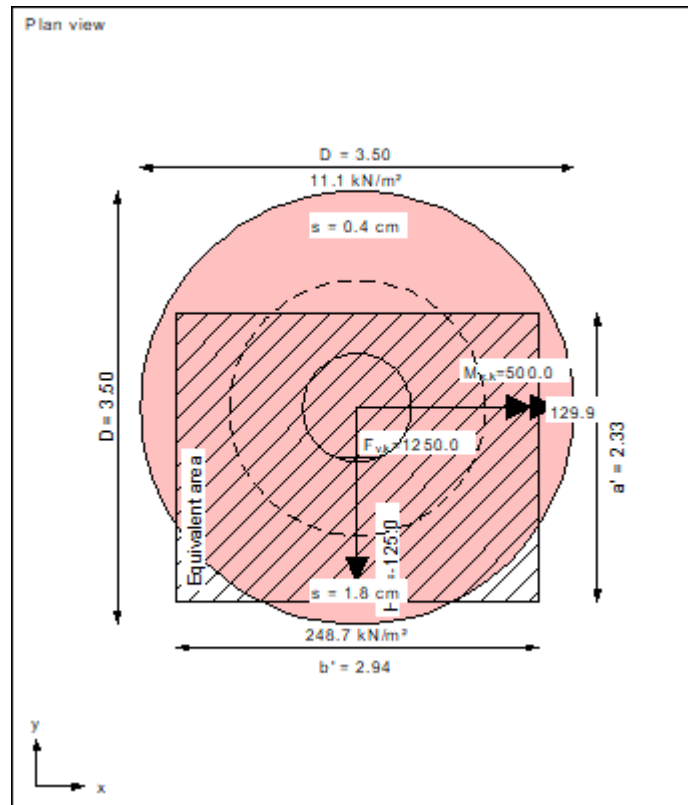
Hatch separation [mm]: 3.00

Hatch inclination [°]: 45.00

OK Cancel

Activate the "Pressure zone coloured" check box. Press the "Colour press. zone" button and select a quiet red from the box (or perhaps not). Confirm with "OK".

In the plan view at the right the pressure zone is now coloured.



The graphic also contains:

- the characteristic forces $M_{x,k}$, $H_{y,k}$ and the vertical force $F_{v,k}$ (=resultant, marked by a cross).
- the 1st core dimension.
If the resultant is within this rhombic area, the complete footing is under pressure.
- the 2nd core dimension.
If the resultant is within this elliptical area, a base tilt will occur.
- the equivalent area A' .
This region is additionally labelled at the edge with a' and b' .
- the edge stresses.
- the settlements at the two characteristic points..

The resulting torsion can be read from the pad footing legend at the lower left of the screen.

Settlement from total loads:
Limiting depth $t_g = 5.99$ m b. GL
Settlement (mean of all CPs) = 1.14 cm
Settlement of CPs:
Top = 0.45 cm
Bottom = 1.83 cm
Torsion (x) (CP) = 1 : 213.8

7 Theoretical principles

7.1 Analysis of bearing capacity safety

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

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

where:

- $\sigma_{0f,k}$ = characteristic bearing stress
- c = cohesion [kN/m²]
- N_c = bearing capacity coefficient equation 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

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

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)

For settlement analysis, the length information is also needed for strip footings. 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.

$v_d = 1 + 0.2 \cdot b/a$ (instead of 1.0 for strip footings)

$v_b = 1 + b/a \cdot \sin(\varphi)$ (instead of 1.0 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)
- d_c d_d d_b (depth coefficients), not envisaged in DIN 4017

Depth coefficients d_c , d_d and d_b

The depth coefficients consider the favourable influence of shear strength in the failure plane above the footing base. In DIN 4017:2006 this effect is not taken into consideration, i.e. all depth coefficients are 1.0. In some European countries, on the other hand, the favourable effect may be considered by adopting depth coefficients > 1.0 .

The following tables show several depth coefficient equations, compiled by Ziegler/Tafur (2015) in the context of a PRB project (PRB = *Initiative Praxisgerechte Regelwerke im Bauwesen e.V.*, www.initiative-prb.de).

The coefficient d_b in all cases is:

$$d_b = 1,0$$

For the influence of the embedment depth d_d :

Brinch Hansen: Bulletin N° 28 (1970) ²⁾	$d_d = 1 + 2 \cdot (\tan \varphi) \cdot (1 - \sin \varphi)^2 \cdot \frac{d}{b'}$	$d \leq b'$
	$d_d = 1 + 2 \cdot (\tan \varphi) \cdot (1 - \sin \varphi)^2 \cdot \tan^{-1} \left(\frac{d}{b'} \right)$	$d > b'$
Lang et al. (Switzerland) ³⁾	$d_d = 1 + 0,035 \cdot (\tan \varphi) \cdot (1 - \sin \varphi)^2 \cdot \tan^{-1} \left(\frac{d}{b'} \right)$	
IEG7: Plattgrundläggning (Sweden) ⁴⁾	$d_d = 1 + 0,35 \cdot \frac{d}{b'}$	$d_d \leq 1,7$
GCOC (Spain) ⁵⁾	$d_d = 1 + 2 \cdot (\tan \varphi) \cdot (1 - \sin \varphi)^2 \cdot \tan^{-1} \left(\frac{d}{b'} \right)$	$d \leq 2b'$

For the influence of cohesion d_c :

Brinch Hansen: Bulletin N° 11 (1961) ¹⁾	$d_c = 1 + 0,35 \cdot \frac{d}{b'}$	$d \leq b'$
	$d_c = 1 + \frac{0,35}{\frac{b'}{d} + \frac{0,6}{(1 + 7 \tan^4 \varphi)}}$	$d > b'$
Lang et al. (Switzerland)	$d_c = 1 + 0,007 \cdot \tan^{-1} \left(\frac{d}{b'} \right)$	
IEG7: Plattgrundläggning (Sweden)	$d_c = 1 + 0,35 \cdot \frac{d}{b'}$	$d_c \leq 1,7$
GCOC (Spain)	$d_c = 1 + 2 \cdot \frac{N_q}{N_c} \cdot (1 - \sin \varphi)^2 \cdot \tan^{-1} \left(\frac{d}{b'} \right)$	$d \leq 2b'$

¹⁾ Brinch Hansen, J. (1961): A general formula for bearing capacity, Bulletin No11, Geoteknisk Institut, Copenhagen

²⁾ Brinch Hansen, J. (1970): A revised and extended formula for bearing capacity, Bulletin No28, Geoteknisk Institut, Copenhagen

³⁾ Lang, H.-J., Huder, J., Amann, P., Puzrin, A. (2011): Bodenmechanik und Grundbau, 9. Auflage, Springer-Verlag

⁴⁾ IEG Rapport 7:2008 (2010): Tillämpningsdokument, EN 1997-1 kapitel 6, Plattgrundläggning, Schweden

⁵⁾ GCOC (2009): Guía de cimentaciones en obras de carretera, Madrid

7.2 Theoretical principles circle/annulus

The bearing capacity of a circular footing is calculated using the equations given in DIN 4017

An equivalent area A' must be determined for eccentric loading. For a rectangle, for example, the equivalent area is given by:

$$A' = (a - e_x) \cdot (b - e_y)$$

where: e_x, e_y = eccentricities

The equivalent area for a circle is calculated as shown in the figure below:

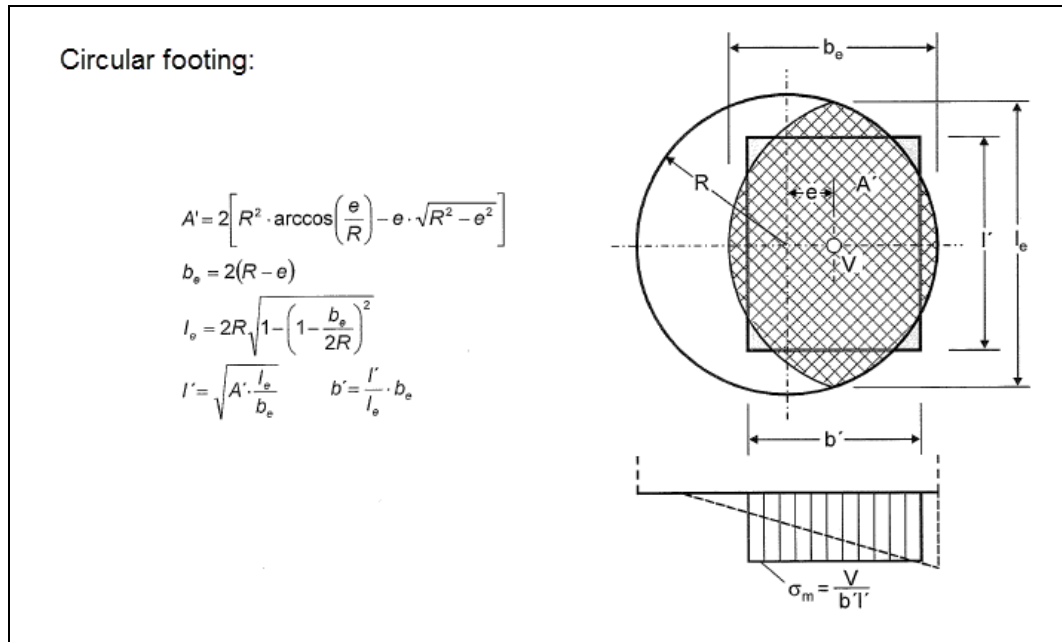


Figure 2 Equivalent area for circle (Leibniz University, Hannover)

The 1st and 2nd core dimensions are also required. For a circle:

Core dimensions for circle:

- 1st core dimension = $D/8$
- 2nd core dimension = $3 \cdot \pi \cdot D/32$

There are no bearing capacity equations for an annulus comparable to those for a circle. The annulus is therefore converted to an equivalent circle. The conversion can be performed using either an equal area circle or a circle with the same moment of inertia.

Core dimensions for annulus:

- 1st core dimension = $[1 + (D_i / D_a)^2] \cdot D_a / 8$
- 2nd core dimension = $3 \cdot \pi \cdot D_a / 32 \cdot [1 - (D_i / D_a)^4] / [1 - (D_i / D_a)^3]$

where: D_i = diameter (inner)
 D_a = diameter (outer)

The weight of the soil above slip body GK 3 is summed, including and distributed and line loads present and converted to an idealised body $\gamma_1 \cdot d$.

Large line loads can generate a substantial increase in an imaginary embedment depth. This is associated with soil pressures, which cannot be activated as calculations would indicate. The DIN 4017, for example, assumes that the ratio between embedment depth d and footing width b will achieve a maximum of 2.0 in order to guarantee the validity of a bearing capacity analysis. You should therefore examine your input data with this in mind. The same effect can be achieved by defining very large embedment depths.

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.

7.4 Differences between DIN 1054 (old) and DIN 1054:2005/EC 7

The differences between DIN 1054 (old) and 1054:2005 or EC 7 are minor for bearing capacity analysis.

The failure load, or, using partial safety factors, the characteristic bearing capacity (R_k) is determined identically. This value is subsequently divided by the bearing capacity partial safety factor $\gamma_{\text{bearing capacity}}$ (Table 3 in DIN 1054:2005). This is 1.40 for load case 1, 1.30 for load case 2 and 1.20 for load case 3. This leads to the bearing capacity design value R_d

$$R_d = R_k / \gamma_{\text{bearing capacity}}$$

This value is compared to the design value for the present load (action) V_d . The design value is given by the load present or, using partial safety factors, by the characteristic action V_k multiplied by the partial safety factor for the actions γ_G (permanent) and γ_Q (changeable). According to Table 2 (DIN 1054:2005), the following partial safety factors must be adopted:

Permanent actions: (1.35 LC 1, 1.20 LC 2 and 1.00 LC 3)

Unfavourable changeable actions: (1.50 LC 1, 1.30 LC 2 and 1.00 LC 3)

If the characteristic action consists of a permanent (index G) and a changeable (index Q) action, the design value V_d is

$$V_d = \gamma_G \cdot V_{k,G} + \gamma_Q \cdot V_{k,Q}$$

The bearing capacity has been verified if

$$V_d \leq R_d$$

is shown. A safety factor η according to the global safety factor concept

$$\eta = \text{failure load/present load}$$

is currently not envisaged. However, the safety factor η according to the global safety factor concept has the great advantage that the *distance* to safe conditions is documented with a single number.

e.g. for load case 1 according to the global safety factor concept:

$$\eta = 2.01 = \text{just adhered to for allowable } \eta = 2.00$$

or

$$\eta = 16.26 = \text{generously dimensioned for allowable } \eta = 2.00$$

$$\text{So a utilisation factor } \mu \text{ is defined by } \mu = V_d / R_d$$

A value of μ smaller than 1.0 therefore indicates stable conditions.

A simple example (load case 1) will demonstrate the *differences* between DIN 1054 (old) and DIN 1054:2005.

- **DIN 1054 (old)**
pres. $V = 1000 \text{ kN}$
 $V_{\text{Failure}} = 2100 \text{ kN}$ (via bearing capacity analysis)
 $\eta = 2100/1000 = 2.10 > \text{allow. } \eta = 2.00$
- **DIN 1054:2005**
 $V_{k,G} = 400 \text{ kN}$ (permanent) = characteristic permanent load
 $V_{k,Q} = 600 \text{ kN}$ (changeable) = characteristic changeable load
 $V_d = \gamma_G \cdot V_{k,G} + \gamma_Q \cdot V_{k,Q} = 1.35 \cdot 400 + 1.50 \cdot 600 = 1,440 \text{ kN}$
 $V_d = 1,440 \text{ kN}$ = load design value
 $R_k = V_{\text{Failure}}$ (according to DIN 1054 (old)) = 2,100 kN (via bearing capacity analysis)
 R_k = characteristic bearing capacity
 $R_d = R_k / \gamma_{\text{Bearing capacity}} = 2,100/1.40 = 1,500$
 R_d = bearing capacity design value
 $\mu = V_d / R_d = 1,440/1,500 = 0.96$

An utilisation factor in accordance with the partial safety factor concept can be acquired for the global safety factor concept by dividing the allowable safety factor by the present safety factor.

$$\mu (\text{DIN 1054 (old)}) = \text{allow. } \eta (\text{DIN 1054 (old)}) / \text{pres. } \eta (\text{DIN 1054 (old)}) = 2.00/2.10 = 0.95$$

This value roughly corresponds to the value for the partial safety factor concept. In principle, then, nothing has changed.

From the relationship for the utilisation factor according to the partial safety factor concept

$$\mu = V_d / R_d,$$

an allowable load can be computed if the utilisation factor is set to $\mu = 1.0$.

$$\mu = (\gamma_G \cdot V_{k,G} + \gamma_Q \cdot V_{k,Q}) / (R_k / \gamma_{\text{Bearing capacity}})$$

$$1,0 = (\gamma_G \cdot V_{k,G} + \gamma_Q \cdot V_{k,Q}) / (R_k / \gamma_{\text{Bearing capacity}})$$

$$V_{k,G} = \text{permanent loads}$$

$$V_{k,Q} = \text{changeable loads}$$

If, as in the example, the changeable loads component p is more than 60% of total loads

$$p = V_{k,Q} / (V_{k,G} + V_{k,Q}) = 600 / (600 + 400) = 0.60 [= 60\%]$$

the equation can be *simplified* as follows:

$$1.0 = (\gamma_G \cdot (1 - p) \cdot V_k + \gamma_Q \cdot p \cdot V_k) / (R_k / \gamma_{\text{Bearing capacity}})$$

$$1.0 = (\gamma_G \cdot (1 - p) + \gamma_Q \cdot p) \cdot V_k / (R_k / \gamma_{\text{Bearing capacity}})$$

$$(\gamma_G \cdot (1 - p) + \gamma_Q \cdot p) \cdot V_k = R_k / \gamma_{\text{Bearing capacity}}$$

$$\text{allow. } V_k = R_k / [(\gamma_G \cdot (1 - p) + \gamma_Q \cdot p) \cdot \gamma_{\text{Bearing capacity}}]$$

According to the partial safety factor concept, the characteristic bearing capacity value R_k is calculated identically to DIN 1054 (old).

The equivalent relationship from DIN 1054 (old) is:

$$\text{allow. } V = V_{\text{Failure}} / \eta$$

According to the partial safety factor concept, $\gamma_G = 1.35$, $\gamma_Q = 1.50$ and $\gamma_{\text{Bearing capacity}} = 1.40$ for load case 1. The following values result for the expression

$$(\gamma_G \cdot (1 - p) + \gamma_Q \cdot p) \cdot \gamma_{\text{Bearing capacity}}$$

as a function of the proportion of changeable loads p to total loads:

p [-]	p [%]	$(\gamma_G \cdot p + \gamma_Q \cdot (1 - p)) \cdot \gamma_{\text{Bearing capacity}}$ $(1.35 \cdot (1 - p) + 1.50 \cdot p) \cdot 1.40$
0.000	0.0	1.89
0.333	33.3	1.96
0.500	50.0	1.99
0.667	66.7	2.03
1.000	100.0	2.10

The third column in the table can be compared to the constant safety factor according to DIN 1054 (old) of 2.0 (load case 1). The comparison demonstrates that this is not noticeably different to the global safety factor of 2.00. This was also the laudable intention of the authors of the partial safety factor concept. If the load component p is around 50%, almost precisely the same results are achieved.

In analogy to the DIN 1054 (old), the DIN 1054:2005 contains tables with allowable footing pressures for various systems in Annex A. The tables and table values are identical with the values in the old standard. This may appear strange at first glance. But here, too, the authors of the partial safety factor concept have orientated themselves to proven systems and analysis concepts. The allowable footing pressures given here may be compared to existing footing pressures, determined using characteristic loads, i.e. with loads not increased by partial safety factors.

This concept is also utilised by the program when footing analysis diagrams ("**Multiple footings**" switch in menu item "**File/New**" activated) are calculated. You then define, in addition to the old standard, the ration of changeable loads p to the total loads, using the menu item "**Edit/System parameters**"

$$p = V_{k,Q} / (V_{k,G} + V_{k,Q})$$

Using this, the program can compute a substitute global safety factor η' :

$$\eta' = (\gamma_G \cdot (1 - p) + \gamma_Q \cdot p) \cdot \gamma_{\text{Bearing capacity}}$$

For example, for load case 1:

$$\eta' = (1.35 \cdot (1 - p) + 1.50 \cdot p) \cdot 1.40$$

Ask the structural engineer for the proportion of changeable loads in the permanent loads. If you get no answer, set p to 0.5 (50 %, approximately corresponds to the old standard). If you are very unsure, select p = 1.0 (100 %). You then get the allowable footing pressures, which are around 5% $[2.00/(1.50 \cdot 1.40) = 2.00/2.10 = 0.95]$ below the old standards values for load case 1.

If footing analysis diagrams are computed ("**Multiple footings**" switch in menu item "**File/New**" activated), the program determines the settlements under total load (as previously).

In all, it is shown that bearing capacity analysis according to the partial safety factor concept does not contain anything new. Unfortunately, it has only become a lot more unclear.

7.5 Settlement analysis

Settlement analyses are carried out according to DIN 4019. Once the program has determined the characteristic or allowable footing pressure for the proposed footing, the settlements are determined using these characteristic or allowable values. DIN 4019 Table 1 allows correction coefficients κ to be taken into consideration for modelling settlements (see menu item **"File/New"**, Section 8.1.1).

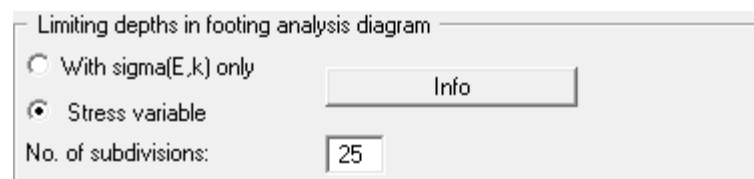
The program calculates stresses and deformations according to the theory of elastic-isotropic half-space. Very comprehensive diagrams and tables were developed, particularly in the time before pocket calculators and personal computers were available. A bibliography of tables can be taken from DIN 4019. Please also see the article *"Spannungsberechnung"* ("Stress analysis") in the Geotechnical Engineering Handbook (1990; Fourth edition). This also contains complete relationships for deformations and stresses below a rectangle in elastic-isotropic half-space (equations 8 to 10 and equations 14 and 15). The program is based on these relationships.

The stress relationships are only utilised for limiting depth calculations and for representation of stress distributions. The deformations are calculated directly from the relationships given in the Geotechnical Engineering Handbook. Numerical integration with associated loss of precision is therefore not necessary. If a base tilt results when analysing a pad footing these relationships can no longer be used because the pressure zone is no longer rectangular. In this case, an analytical-numerical settlement analysis is required.

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% (as a rule 20%).

The settlement curves in the footing analysis diagram are attained from the footing pressure by linear interpolation. Thus, analysis using the x%-criteria only takes place for the maximum footing pressure of each footing width. However, if you open **"Edit/Analysis options"** menu item and activate the **"Stress variable"** check box, the limiting depth is calculated for a number of stresses (here for 25 stresses).



Limiting depths in footing analysis diagram

☐ With $\sigma(E,k)$ only

☒ Stress variable

No. of subdivisions: 25

Info

8 Description of menu items

8.1 File menu

8.1.1 "New" menu item

The dialog box shown in the examples opens. First, select the equation for calculating the bearing capacity in the combo box. In addition to the standard DIN 4017:2006 and DIN 4017 (old) methods, the methods after Terzaghi, Meyerhoff, Hansen and Vesic well known from many literature references may also be selected. If the Terzaghi, Meyerhoff, Hansen or Vesic analysis methods are adopted, the English designations can be used in the bearing capacity equations in addition to the German designations.

You must then specify whether analysis will use the *partial safety factor concept* to EC 7 or DIN 1054:2010 or the old *global safety factor concept*. Decide whether you want to analyse a pad footing (rectangular or circular/annular footing) or multiple footings of a certain type (with footing analysis diagram).

Standard:

☒ Partial safety factor concept (EC 7) ?

☐ Partial safety factor concept (DIN 1054:2005)

☐ Global safety factor concept (DIN 1054 old)

Type of analysis:

☒ Multiple footings Analysis and output with: sigma(R,d) ?

☐ Rectangular footing ☐ Circular/annular footing

☐ Do not investigate overturning

With the introduction of EC7, the table value "*Allowable bearing pressure*" used in the tables for spread footing analyses is generally replaced by the table value "*Bearing resistance design value*". Differences to DIN 1054:2010 in terms of footing dimensions do not result from this. If you are working to EC7, "*sigma(R,d)*" for design and output is activated.

DIN 1054:2010 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.

The next part of the dialog box is only active if you are analysing using *global safety factors*. Here you can specify whether the reference parameter for safety is the "**Load**" or the "**Shear coefficients**" (friction angle and cohesion). In this case you must enter a safety factor for both the friction angle and the cohesion (see menu item "**Edit/System parameters**").

When analysing a single footing you can activate consideration of vibrodisplacement compaction according to Priebe using the "Use" check box in the **Vibrodisplacement compaction** group box (Heinz J. Priebe, *Die Bemessung von Rüttelstopfverdichtungen*, Ground Engineering, December 1995).

It is then possible to enter corresponding parameters for the improved soil layer (see menu item "Edit/Soils", Section 8.2.6). If the friction angle and cohesion are also to be adjusted, additionally activate the "Edit shear values too" check box.

Vibrodisplacement columns below the footing are usually only produced in a limited area called improvement area A1. With increasing depth areas outside the improvement area A1 are also used for load transfer due to load spreading. To compensate for this effect, the following reduction can be considered:

$$n = 1 + (n - 1) \cdot A1 / A2 \quad (n = \text{improvement value according to Priebe})$$

At the footing base (= depth 0), the load area A2 corresponds to the footing area. With increasing depth, A2 increases due to an increase in area on all sides with an assumed load spreading angle of 2:1.

If the "Use absolute heights" check box is deactivated, the ground level will be at **0.0** and all depth input is positive downwards. If the check box is activated, enter the layer depths, the groundwater level and the footing base in absolute values (y-axis positive upwards), e.g. in m AD. The designation for the absolute elevation, which you enter below (here: m AD), is shown in the soil properties legend as the depth dimension. Absolute ground level is entered in the "Edit/Ground level" dialog box (see Section 8.2.3).

DIN 4019 Table 1 allows correction coefficients kappa to be taken into consideration for modelling settlements. These correction coefficients can be activated using the appropriate check box in the dialog box.

DIN 4017:2006 describes a new punching analysis method. To use this new analysis method the "Punching analysis using new method (recommended)" check box is activated by default when the program starts. This means it is now only necessary to specify whether a flexible footing is being dealt with in the "Edit/System parameters" dialog box (see Example 1, Section 6.2.2).

If you wish you can enter a description of the problem being worked on in this dialog box; this will then be used in the **General legend**. Otherwise, the project identification is entered using the menu item "Edit/Project identification".

After confirming your input with **"OK"** you will see an info box corresponding to your safety concept with the safety factors or partial factors for the various load cases. Close the box by clicking on the load case of your choice. The safety factor values are automatically used in the system data. If the *partial safety factor concept* is adopted, the load case can be modified at any time using the **"Edit/Partial safety factors"** menu item (see Section 8.2.10). If a different load case is required when carrying out an analysis using the *global safety factor concept*, proceed via the **"File/New"** menu item

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.

8.1.4 "Save as" menu item

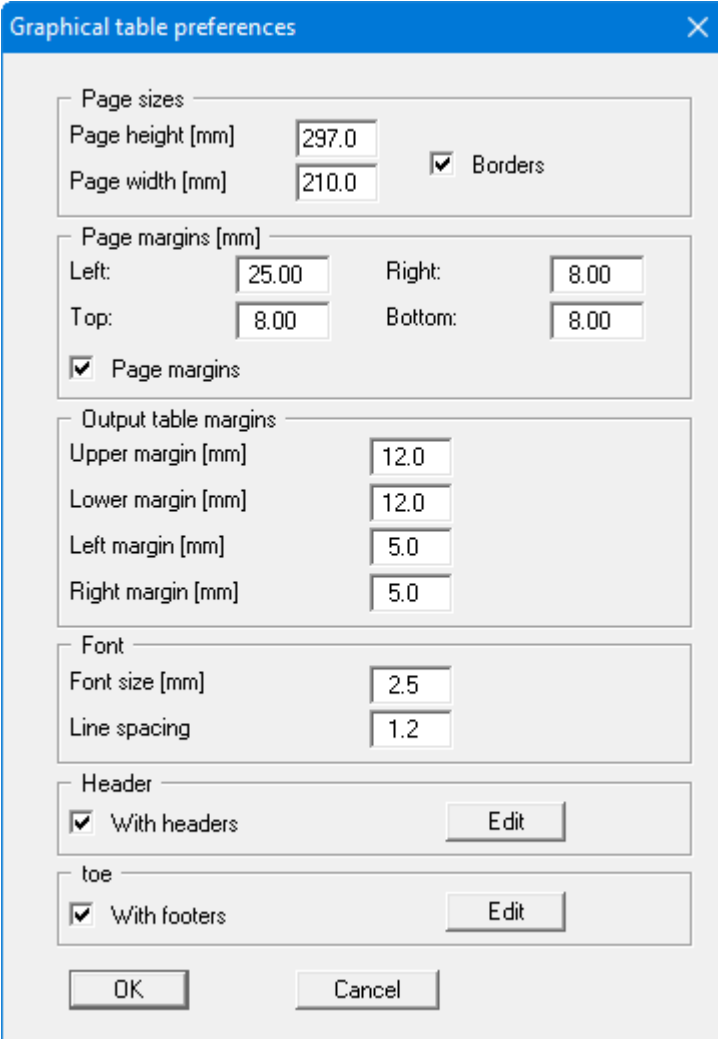
You can save data entered during program use to an existing file or to a new file, i.e. using a new file name. For reasons of clarity, it makes sense to use **".gdg"** as file suffix, as this is the suffix used in the file requester box for the menu item **"File/Load"**. If you choose not to enter an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

8.1.5 "Print output table" menu item

8.1.5.1 *Output as graphics*

You can have an output table printed containing all information on the current state of analysis, including the system data.

In a further option box in the "**Multiple footings**" analysis mode it is possible to choose between a graphical output table or one in ASCII format. If you click the "**Output as graphics**" button you will see the following dialog box. In the "**Pad footing**" analysis mode it opens automatically when this menu item is selected.



The dialog box is titled "Graphical table preferences" and contains several sections for configuring the output table. The "Page sizes" section includes input fields for "Page height [mm]" (297.0) and "Page width [mm]" (210.0), 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 has 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 are "OK" and "Cancel" buttons.

Page sizes	
Page height [mm]	297.0
Page width [mm]	210.0
<input checked="" type="checkbox"/> Borders	

Page margins [mm]			
Left:	25.00	Right:	8.00
Top:	8.00	Bottom:	8.00
<input checked="" type="checkbox"/> Page margins			

Output table margins	
Upper margin [mm]	12.0
Lower margin [mm]	12.0
Left margin [mm]	5.0
Right margin [mm]	5.0

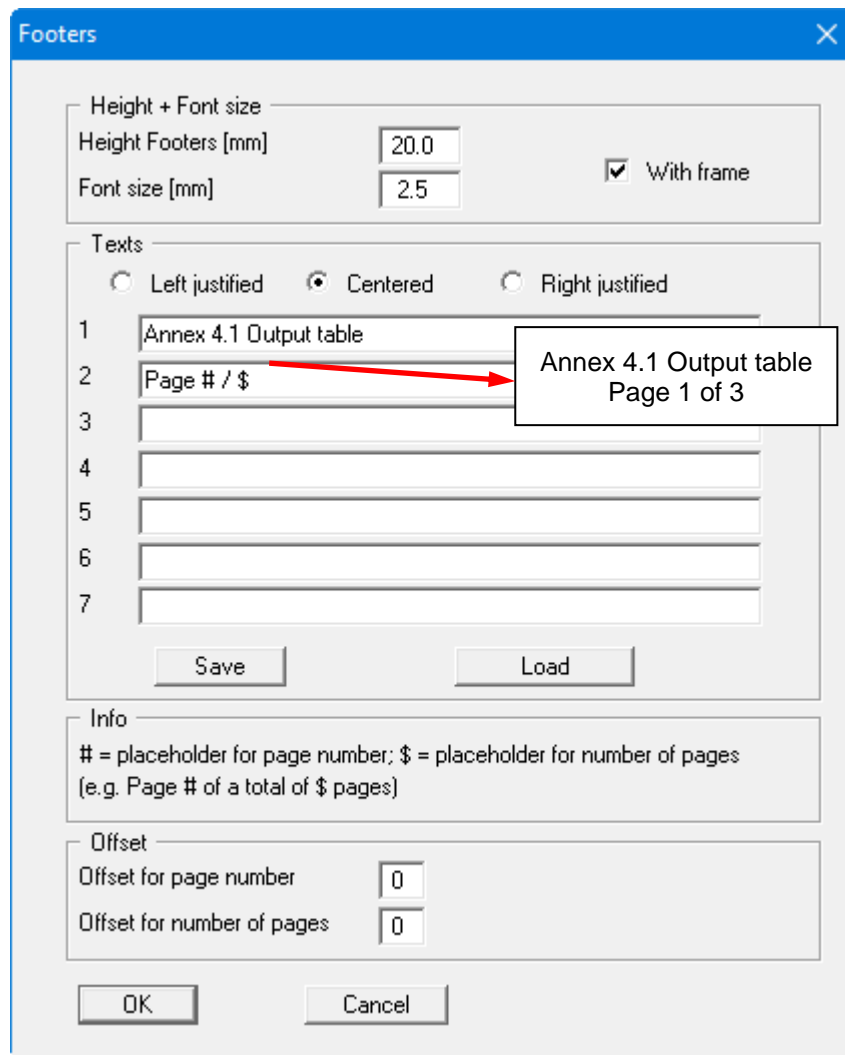
Font	
Font size [mm]	2.5
Line spacing	1.2

Header	
<input checked="" type="checkbox"/> With headers	Edit

Footer	
<input checked="" type="checkbox"/> With footers	Edit

OK Cancel

You can define the desired layout for the output table 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 click on the "**Edit**" button. You can then edit as required in a further dialog box.






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

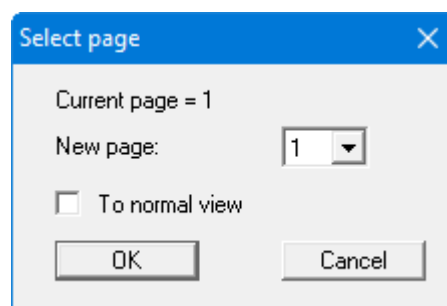
- Height + Font size:**
 - Height Footers [mm]: 20.0
 - Font size [mm]: 2.5
 - ☒ With frame
- Texts:**
 - Justification: ☒ Centered
 - Line 1: Annex 4.1 Output table
 - Line 2: Page # / \$ (highlighted with a red arrow pointing to a callout box)
 - Lines 3-7: Empty text boxes
- Info:**

= placeholder for page number; \$ = placeholder for number of pages (e.g. Page # of a total of \$ pages)
- Offset:**
 - Offset for page number: 0
 - Offset for number of pages: 0

Buttons: Save, Load, OK, Cancel

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 exiting the dialog boxes using "OK" you will see a further dialog box in which you can select the parameters to be used in the output table. If you click the "OK" button the output table is presented on the screen page by page. To navigate between the pages, use the arrow tools   in the toolbar. If you need to jump to a given page or back to the graphical representation, click on the  tool. You will then see the following box:



The 'Select page' dialog box is shown with the following settings:

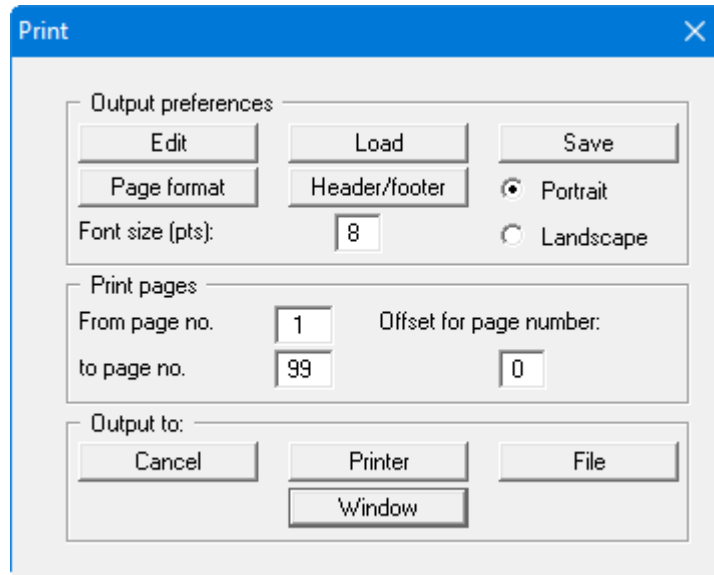
- Current page = 1
- New page: 1 (dropdown menu)
- ☐ To normal view

Buttons: OK, Cancel

8.1.5.2 Output as ASCII

When in the "**Multiple footings**" analysis mode, you can have your calculation data sent to the printer, without further work on the layout, or save it to a file for further processing using a different program, e.g. a word processing application.

In the option box for this menu item simply press the "**Output as ASCII**" button. You will see the following dialog box for specifying the simple table output:

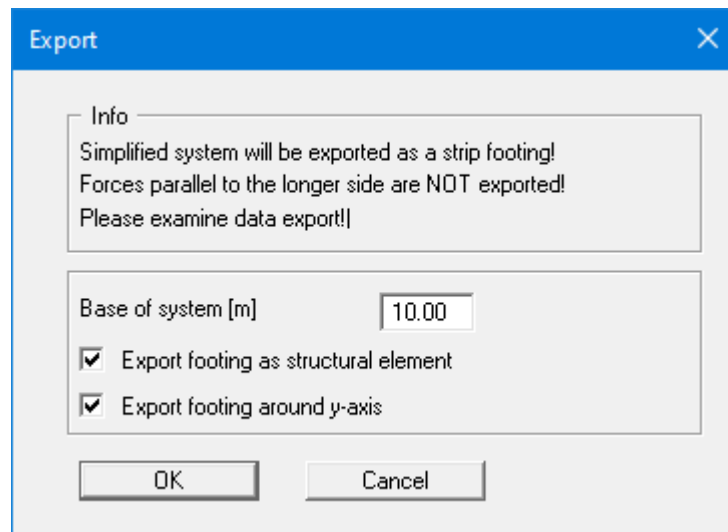


- "**Output preferences**" group box
Using the "**Edit**" button the current output preferences can be changed or a different printer selected. Using the "**Save**" button, all preferences from this dialog box can be saved to a file in order to have them available for a later session. If you select "**GGU-FOOTING.drk**"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

If you are analysing a single footing in a sloped system, you can export the system to **GGU-STABILITY** to verify the stability of the existing slope. The following dialog box opens:

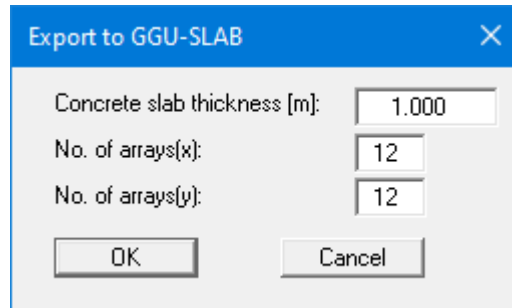


In **GGU-STABILITY** the ground level is extended along the slope to the entered base of system. If you do not want to export the footing as a structural element and deactivate the check box, the footing is exported only as a permanent load. Normally, the width b of the footing (in the plan view in y -axis) is adopted as the width of the structural element or the permanent load in **GGU-STABILITY**. If you deactivate the "**Export footing around y -axis**" check box, the length a (view in x -axis) is adopted.

When creating the file for **GGU-STABILITY** you can already generate centre points for the slip circle analysis according to Bishop. However, you should always check this later in **GGU-STABILITY** or do without generation.

8.1.7 "Export to GGU-SLAB" menu item

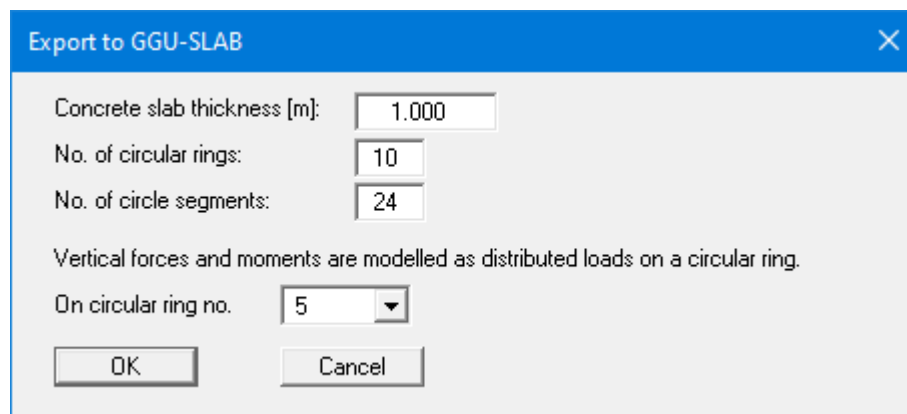
If your single footing is a larger slab, exporting to **GGU-SLAB** may be useful. In **GGU-SLAB** the footing can be analysed as elastically bedded slab using the constrained modulus method. For a rectangular footing you will see a simple dialog box in which you enter the thickness of the concrete slab and the x- and y-arrays for the finite elements in **GGU-SLAB**.



The dialog box titled "Export to GGU-SLAB" has a blue header bar with a close button (X) on the right. It contains three input fields: "Concrete slab thickness [m]" with a value of 1.000, "No. of arrays(x)" with a value of 12, and "No. of arrays(y)" with a value of 12. At the bottom are "OK" and "Cancel" buttons.

The file for **GGU-SLAB** is then generated. Your soil stratification and soil properties are transferred to **GGU-SLAB** as constrained modulus profiles.

For a circular or annular footing, you will see the following dialog box:



The dialog box titled "Export to GGU-SLAB" has a blue header bar with a close button (X) on the right. It contains four input fields: "Concrete slab thickness [m]" with a value of 1.000, "No. of circular rings" with a value of 10, "No. of circle segments" with a value of 24, and "On circular ring no." with a value of 5 and a dropdown arrow. Below the last field is a text label: "Vertical forces and moments are modelled as distributed loads on a circular ring." At the bottom are "OK" and "Cancel" buttons.

With the number of circular rings and circle segments you specify the density of your finite elements in **GGU-SLAB**. In the case of an annulus, the lower section of the dialog box also appears. Here you specify on which of the circular rings the vertical forces and moments are to be modelled as distributed loads.

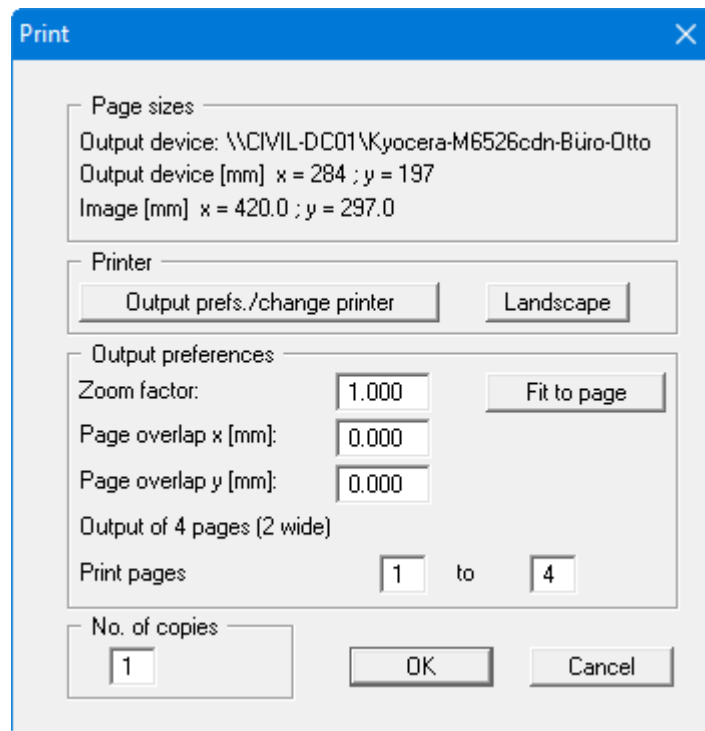
8.1.8 "Output preferences" menu item

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

8.1.9 "Print and export" menu item

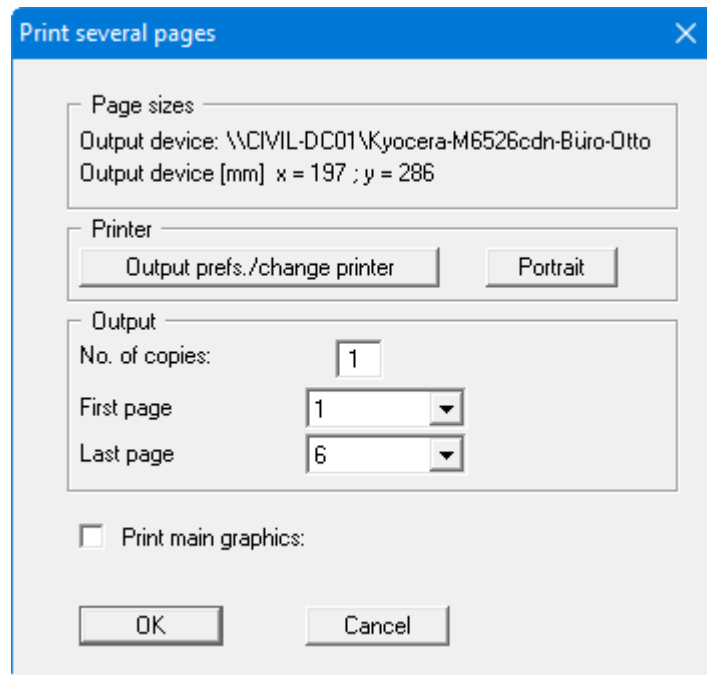
You can select your output format in a dialog box. You have the following possibilities:

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




In the upper group box, the maximum dimensions which the printer can accept are given. Below this, the dimensions of the image to be printed are given. If the image is larger than the output format of the printer, the image will be printed to several pages (in the above example, 4). 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. 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 in order to be further processed with third party software. Output is in the standardised EMF format (Enhanced Metafile format). Use of the Metafile format guarantees the best possible quality when transferring graphics.

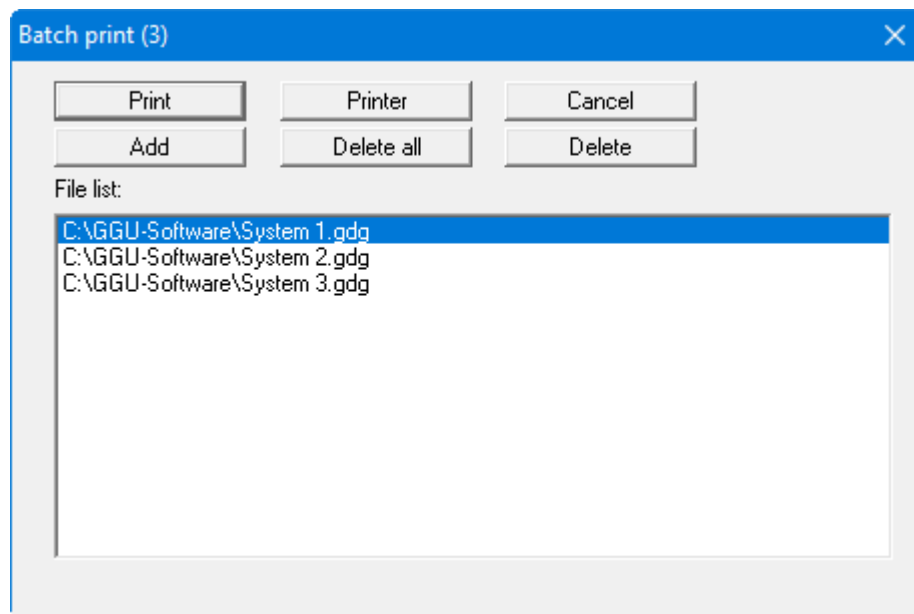
If you select the **"Copy/print area"** tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see **"Tips and tricks"**, Section 5.4).

Using the **"Mini-CAD"** program module you can also import EMF files generated using other GGU applications into your graphics (see Section 8.5.7).

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

8.1.10 "Batch print" menu item

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



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

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

8.1.11 "Exit" menu item

After a confirmation prompt, you can quit the program.

8.1.12 "1, 2, 3, 4" menu items

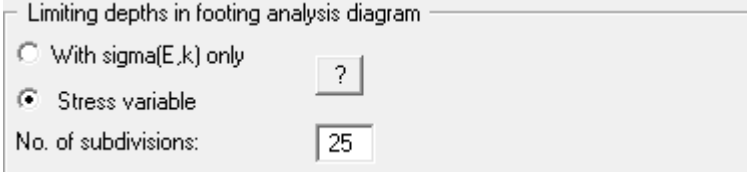
The **"1, 2, 3, 4"** menu items show the last four files worked on. By selecting one of these menu items the listed file will be loaded. If you have saved files in any other folder than the program folder, you can save yourself the occasionally onerous *rummaging* through various sub-folders.

8.2 Edit menu

8.2.1 "Analysis options" menu item

A dialog box opens like that for the menu item **"File/New"** (see Section 8.1.1). Only the project identification cannot be entered here.

The dialog box has been extended by the group box **"Limiting depths in footing analysis diagram"**, where the type of limiting depth calculation for **"Multiple footings"** is specified (see **"Theoretical principles/Settlement analysis"** in Section 7.5). You can read a brief description by pressing the **"Info"** button.



Limiting depths in footing analysis diagram

☐ With sigma(E,k) only

☒ Stress variable

No. of subdivisions: 25

If the pad footing analysis mode (**"Rectangular footing"/"Circular/annular footing"**) is selected, the **"Limiting depths in footing analysis diagram"** group box is deactivated. Because the footing stress is specified directly in the menu item **"Edit/Footing details"** in this analysis mode (see Section 6.3.3), calculation of the limiting depth by means of stress variables is not required.

8.2.2 "Project identification" menu item

You can enter a description of the current system; this will then be used in the *General legend* (see Section 8.4.5).

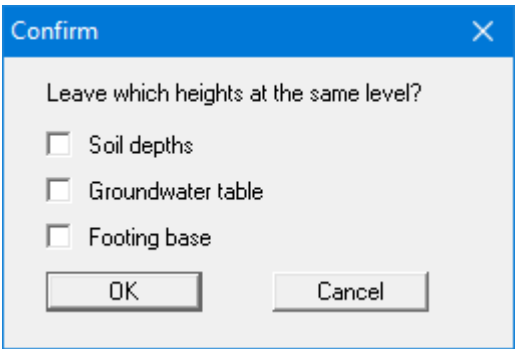
8.2.3 "Ground level" menu item

Using this menu item, you can activate the consideration of absolute heights (e.g. m AD). Then enter the new height for the ground level. The designation for the absolute height (here: m AD) can also be adapted again in the dialog box shown below.



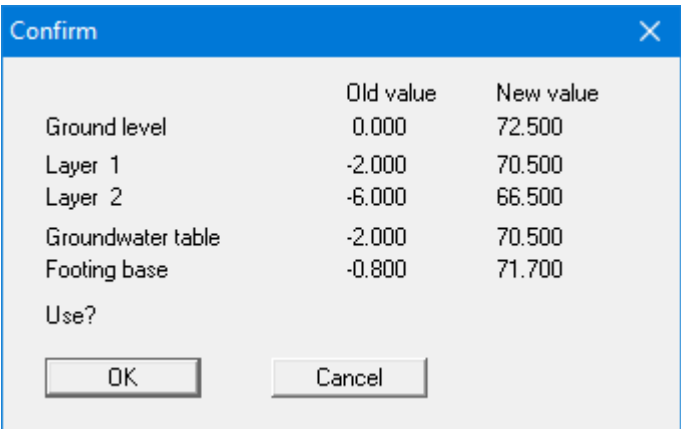
A dialog box titled "Ground level" with a close button (X) in the top right corner. It contains a checked checkbox labeled "Use absolute heights". Below this, there is a text label "Ground level [m]:" followed by a text input field containing "72.50". Below that, there is a text label "Designation for abs. height:" followed by a text input field containing "m AD". At the bottom, there are two buttons: "OK" and "Cancel".

Once you have confirmed your input you will next be presented with a query on how to deal with the existing elevation data for soil strata, groundwater and footing base. If you leave the elevations at the same level (= check boxes deactivated), the originally entered data will not be modified.



A dialog box titled "Confirm" with a close button (X) in the top right corner. It contains the text "Leave which heights at the same level?". Below this, there are three unchecked checkboxes: "Soil depths", "Groundwater table", and "Footing base". At the bottom, there are two buttons: "OK" and "Cancel".

The check boxes are deactivated by default. Check the preferences and confirm your input. You will then see a message box, which compares the old and new data for you. If you used any of the previous check boxes mistakenly, you could repeat the process correctly by clicking "Cancel" in the message box. By exiting the dialog box using "OK" the new elevation data are entered into the system data.



A dialog box titled "Confirm" with a close button (X) in the top right corner. It contains a table comparing old and new values for various elevation data. Below the table, there is a "Use?" label and two buttons: "OK" and "Cancel".

	Old value	New value
Ground level	0.000	72.500
Layer 1	-2.000	70.500
Layer 2	-6.000	66.500
Groundwater table	-2.000	70.500
Footing base	-0.800	71.700

8.2.4 "System parameters" menu item

This function can also be accessed using the [F3] function key. The following dialog box opens if "Multiple footings" are selected together with systems analysed to **DIN 4017:2006** and using the **partial safety factor concept**. The partial safety factor values are already entered into the input boxes, depending on the load case selected. If the **global safety factor concept** is selected, only the input boxes for the safety factors change, the remaining buttons and boxes are identical.

The input boxes have already been described in the "Multiple footings" worked example in Section 6.2.2 and in the "Single rectangular footing" worked example in Section 6.3.2.

System parameters

Partial FOS (bearing capacity): 1.40

Partial FOS (permanent actions): 1.35

Partial FOS (changeable actions): 1.50

Ground level [m AD]: 72.50

Groundwater table [m AD]: 70.50

Footing base [m AD]: 71.70

Slope inclination [°]: 0.00

Berm width [m]: 0.00

☒ Slope to DIN 4017:2006

Preloading [kN/m²]: 0.00 ?

Surcharge (bearing cap.) [kN/m²]: 0.00 ?

Surcharge (limiting depth) [kN/m²]: 0.00 ?

H/V [-]: 0.00000

Depth coefficients

☐ Calculate using depth coefficients ?

Depth coeff. after: IEG7 (SE) ?

☐ Friction angle for bearing capacity failure ?

☐ Depth coefficients for slopes too ?

☐ Verify bearing capacity for short side only ?

Limit soil pressure

Limit to max. sigma(R,d) [kN/m²] = 200.00 ☐ Apply

5° condition

☒ Check and correct ?

Decrement [°]: 0.10

Ratio changeable/total loads [-]

Ratio [-]: 0.500 ?

Punching analysis

☒ Carry out punching analysis ?

☒ Flexible footing ?

Done Cancel

If you are working with absolute elevations the value, you entered in the menu item "**Edit/Ground level**" will be displayed. The groundwater level and footing base elevations can be adapted here if necessary (e.g. in m AD).

8.2.5 "Footing details" menu item

Depending on the analysis mode and the safety factor concept selected, you will see a modified dialog box for input of footing sizes and/or stresses (see worked examples; Sections 6.2.3, 6.3.3 and 6.4.3).

The following dialog box opens in the "**Rectangular footing**" analysis mode using partial safety factors:

Dimensions	
Length a [m]:	4.000
Width b [m]:	2.000
<input type="button" value="Equivalent width from circle/annulus"/>	

Stress, characteristic		
Forces act at the footing base level		
	Permanent	Changeable
Vertical force [kN]:	1000.00	0.00
Horizontal force (x) [kN]:	0.00	0.00
Horizontal force (y) [kN]:	0.00	0.00
Moment (x) [kN·m]:	0.00	0.00
Moment (y) [kN·m]:	0.00	0.00

Self weight	
<input type="checkbox"/> Automatic self weight	
Gamma concrete [kN/m³]:	25.00

Slip resistance	
<input type="checkbox"/> Verify slip resistance	
Partial FOS gamma(R,h) [-:]	1.10
<input type="checkbox"/> Sliding resistance with user-defined angle	
User-defined angle [°]:	30.00

OK Cancel

Enter the dimensions and stresses for the footing. According to the *partial safety factor concept*, you can define differing stresses for permanent and changeable loads (live loads). Instead of the footing analysis diagram seen in the "**Multiple footings**" analysis mode a plan view of a pad footing (rectangle or circle/annulus) is shown with the loading values, among other things.

The friction angle of the soil at the footing base is adopted when analysing the sliding resistance of a pad footing (rectangle or circle/annulus). Any user-defined angle can be adopted by activating the "**Sliding resistance with user-defined angle**" check box. This can be useful if soils with a low friction angle are prevalent below the base of the footing, for example.

GGU-FOOTING can also analyse the torsional stiffnesses of footings. Circular or annular footings are often used for the foundations of wind energy converters. In this program version a circular/annular footing can be selected by going to the menu item **"Edit/Analysis options"**. In this menu item you are then presented with a correspondingly modified dialog box for defining the footing (see Example 3: **"Single circular footing"**, Section 6.4.3).

In older program versions it was possible to convert the circular or annular footing data to a square footing (see dialog box above) to determine the torsional stiffnesses for a rectangular footing by pressing the **"Equivalent width from circle/annulus"** button in the above dialog box.

8.2.6 "Soils" menu item

Enter the number of soil layers and the corresponding soil properties here. An explanation of the dialog box can be found in Section 6.2.4.

If you are working with *vibrodisplacement compaction* and have activated the **"Use"** check box in the menu item **"File/New"** or **"Edit/Analysis options"**, the following dialog box appears:

Base[m]	gam [kN/m²]	gam' [kN/m²]	phi [°]	c [kN/m²]	Es [MN/m²]	nue [-]
2.00	20.00	11.00	27.50	5.00	12.00	0.000
6.00	19.00	10.00	32.50	0.00	75.00	0.000

VDC	A(C)/A [-]	phi(C) [°]	E(C)/E(B) [-]	gam [kN/m²]	gam' [kN/m²]
<input type="checkbox"/>	0.200	40.00	10.000	19.00	11.00
<input checked="" type="checkbox"/>	0.196	40.00	10.000	19.00	11.00

Designation: Silt

By activating the **"VDC"** check box you define the soil layer improved by vibrodisplacement compaction. Enter the corresponding parameters for this layer (see **"Info"** button). The area ratio $A(C)/A$ can be determined by the program for different rasters via the **"Determine A(C)/A"** button and transferred to the table. The ratio $E(C)/E(B)$ should not be greater than 15 to max. 20.

8.2.7 "Limiting depth" menu item

You can define the way in which the limiting depth is calculated. An explanation of the dialog box can be found in the **"Multiple footings"** worked example in Section 6.2.5.

8.2.8 "Line loads" menu item

You can define up to 5 line loads acting as surcharges on the bearing capacity failure body.

Line loads

Forw. Back Cancel Done

2 line load(s) to edit

No.	p [kN/m²]	Separation [m]	Width [m]
1	50.00	2.000	1.000
2	150.00	4.000	1.000

The number of line loads can be modified using the "x line load(s) to edit" button.

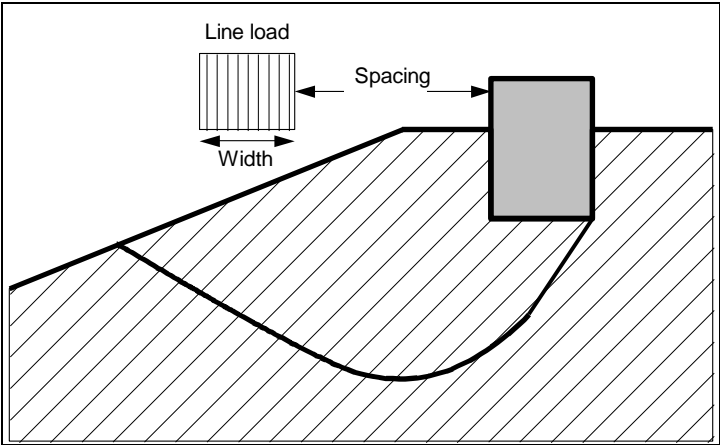


Figure 4 Line load

8.2.9 "Berms" menu item

A maximum of five berms may be defined using this menu item depending on the analysis method selected. The **"Slope to DIN 4017:2006"** check box in the **"Edit/System parameters"** dialog box must be deactivated. The berms may not overlap. The program checks that this condition is adhered to and warns of any errors. Any berms defined using **"Edit/System parameters"** are set to **0.0** if you define one or more berms using this menu item.

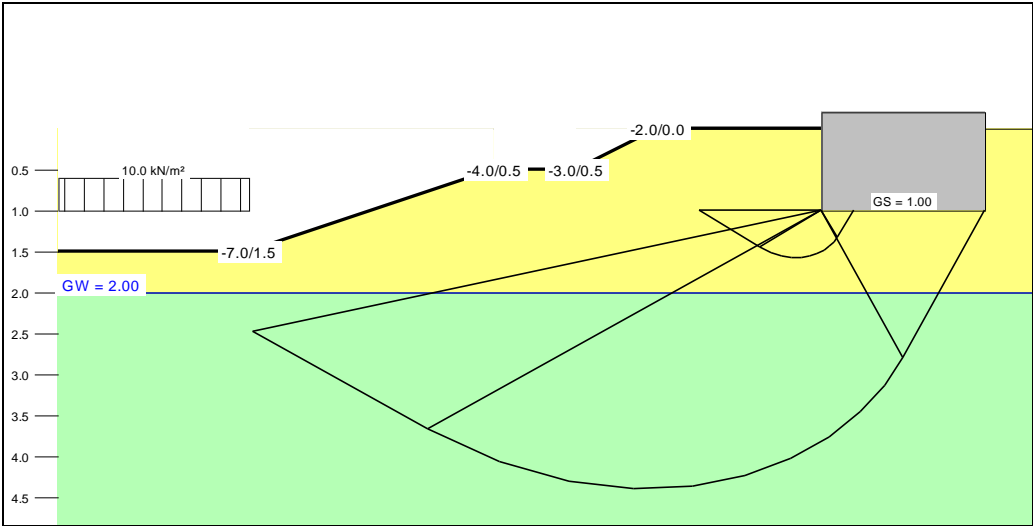


Figure 5 Berms

To get the berms shown above enter the values into the dialog box as shown:

Berms				
Done Forw. Back Cancel				
2 berms to edit				
No.	Left [m]	Right [m]	delta h [m]	Surcharge [kN/m²]
1	-3.00	-2.00	-0.50	0.00
2	-7.00	-4.00	-1.00	10.00

Enter the x-ordinate of the toe and the head. Define the height of the berm with **"delta h"**. Negative values are also allowed here. Finally, a surcharge load can be specified for the horizontal area behind the head of the berm.

8.2.10 "Partial safety factors" menu item

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

Partial safety factors to EC 7
(R,v) = bearing cap./ (R,h) = sliding
(G) = permanent actions / (Q) = changeable actions

DS-P (Persistent Design Situation)
gamma(R,v) = 1,40
gamma(R,h) = 1,10
gamma(G) = 1,35
gamma(Q) = 1,50
DS-P

DS-T (Transient Design Situation)
gamma(R,v) = 1,30
gamma(R,h) = 1,10
gamma(G) = 1,20
gamma(Q) = 1,30
DS-T

DS-A (Accidental Design Situation)
gamma(R,v) = 1,20
gamma(R,h) = 1,10
gamma(G) = 1,10
gamma(Q) = 1,10
DS-A

DS-E (Seismic design situation)
gamma(R,v) = 1,00
gamma(R,h) = 1,00
gamma(G) = 1,00
gamma(Q) = 1,00
DS-E

Reference to ÖNORM EN 1997-1

Cancel

By pressing the appropriate button the partial factors for the various load cases given in DIN 1054:2010 and EC 7 can be adopted. 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'. The load case correlation to **ÖNORM EN 1997-1** will be shown by clicking on the corresponding button. The partial safety factors are automatically adopted in the system data (see menu item "**Edit/System parameters**", Section 8.2.3).

8.3 System menu

8.3.1 "Analyse" menu item

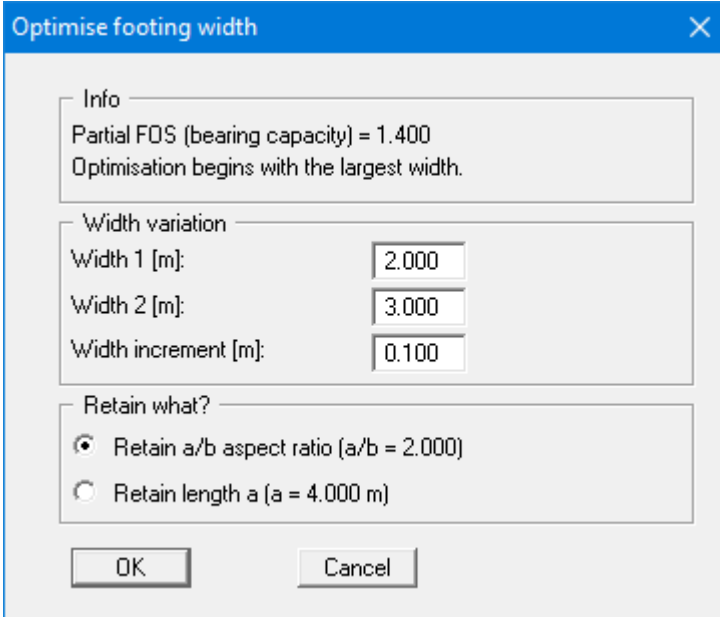
When you edit any of the system data using the **"Edit"** menu item, the diagrams presented do not represent the current analysis results. Only after using this menu item is the modified system re-analysed and subsequently presented with the new results. Alternatively, you can use the **[F5]** function key or the *calculator* in the toolbar. A running analysis can be stopped by pressing the right mouse button.

In the **"Rectangular footing"** analysis mode, the bearing capacity safety is calculated for both directions. Naturally, the smaller value is critical. Information on the conditions in the other direction can be viewed in the **"Pad footing"** legend. Preferences for this can be defined in the **"Output preferences"** menu (see Section 8.4.7, **"For both directions"** check box).

8.3.2 "Optimise footing width" menu item

This menu item is only active in the pad footing analysis mode! Alternatively, footing optimisation can also be started by pressing **[F6]**.

If **"Rectangular footing"** is selected, the following dialog box opens:



Optimise footing width

Info

Partial FOS (bearing capacity) = 1.400
Optimisation begins with the largest width.

Width variation

Width 1 [m]: 2.000
Width 2 [m]: 3.000
Width increment [m]: 0.100

Retain what?

☒ Retain a/b aspect ratio (a/b = 2.000)
☐ Retain length a (a = 4.000 m)

OK Cancel

After confirming with **"OK"** optimisation is carried out, at the end of which you may opt to use the optimum width. A new computation is performed immediately after confirmation. If you choose not to accept the optimum width the menu item is closed.

If **"Circular/annular footing"** is selected, optimisation is performed for the outer circle diameter. The inner diameter is retained.

8.4 Output preferences menu

8.4.1 "Footing analysis diagram" menu item

This menu item only appears in the "Multiple footings" analysis mode. When the "Show footing analysis diagram" check box is activated, you can define preferences for representation of the footing analysis diagram.

Footing analysis diagram

☒ Show footing analysis diagram

x [mm]: 280.00 Width: 110.00
y [mm]: 35.00 Height: 175.00

Smoothing out method
☒ Linear ☐ Method 1 ☐ Method 2

Axis definition
User-defined axes

Settlements [cm] / Limiting depths / SR moduli [MN/m²]
First value: 0.50 Division: 0.50
Last value: 10.00
Settlements

Display on the ordinate
☒ Bearing pressure ☐ Strip load

☒ Only hatch 'allowable' area
☐ Show sigma(E,k) ☒ sigma(E,k) dashed
☐ Also show 'allow. sigma'

OK Cancel

You can define and edit the position of the legend using the values "x" and "y". The size of the legend is controlled by the values for "Width" and "Height".

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

In the second group box the smoothing out method for drawing the graphs for settlement, limiting depths or the modulus of subgrade reaction in the footing analysis diagram can be selected:

- **"Linear"** = no smoothing
- **"Method 1"** = 'strict' Bezier spline
- **"Method 2"** = 'loose' Bezier spline

Using the default settings, the diagram is automatically scaled. If you select the **"User-defined axes"** button you are first offered the automatic scaling values in a box, which you can then edit as wished.

In the fourth group box you can influence the representation of the settlement curves. The settlement curves are represented in the diagram using the three numbers given in the box above. The program calculates the settlements corresponding to a given footing width by means of linear interpolation. If the limiting depth is calculated using the **"% condition"** (see Worked example 1, Section 6.2.5) and you have activated the **"Stress variable"** check box in the **"Edit/System parameters"** menu item (see **"Theoretical principles/Settlement analysis"**, Section 7.5), limiting depth curves or SR moduli curves can be represented instead of settlement curves.

The bearing resistance design value to EC 7 or the (allowable) bearing pressure, or the (allowable) strip load or point load can be selected for visualisation on the y-ordinate in the diagram when analysing using the *partial safety factor concept*. If the *global safety factor concept* is employed either the allowable footing pressure or the allowable strip or point load can be selected.

If you activate the **"Hatch allowable area"** check box, reference lines are drawn in the allowable area only. Otherwise, lines are drawn across the whole of the plan width and height. Moreover, additional scaling of the footing analysis diagram using the values for $\sigma(E,k)$ can be activated here.

8.4.2 "Footing plan" menu item

This menu item only appears in the pad footing analysis mode ("**Rectangular footing**" / "**Circular/annular footing**"). If the "**Show footing plan**" check box is activated the graphical visualisation can be adapted in the dialog box associated with this menu item.

Footing plan view

☒ Show footing plan

Heading:

x [mm]: Width:

y [mm]: Height:

Font size [mm]:

☐ Display force and moment arrows at margins

Select display options

☒ Stresses ☒ Resultant

☒ 1st core dim. ☒ 2nd core dim.

☒ Zero line ☒ Settlements

☐ Pressure zone coloured

Equivalent area A'

☒ Display ☒ Hatch

Hatch separation [mm]:

Hatch inclination [°]:

You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Width**" and "**Height**". The labelling font size can be altered.

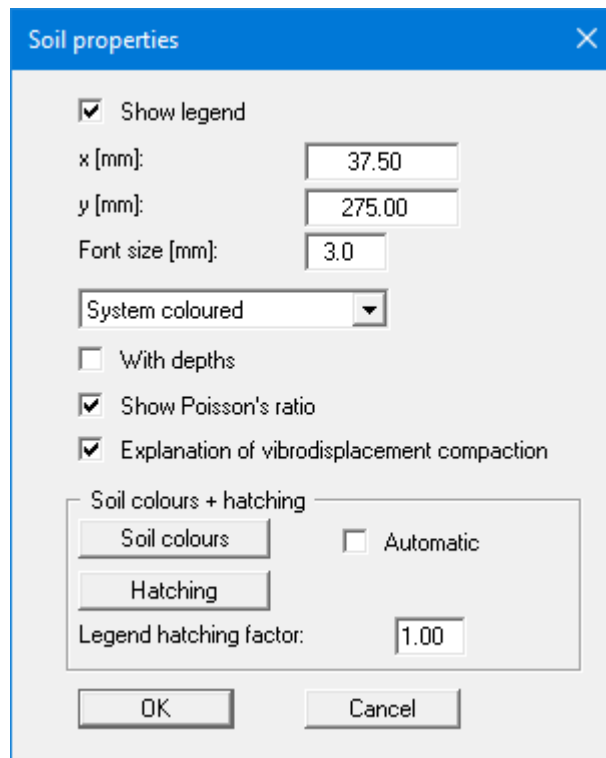
The fastest way to modify the position or size of the legend is to press the [**F11**] function key and then to either pull the legend to the new position or to move the sides of the legend to their new size with the left mouse button pressed.

The force and moment arrows may cover the labelling on some footings. In this case, the arrows can be shown at the sides of the footing by activating the respective check boxes.

Besides position and size of the footing plan view, you can specify a number of preferences for representation of further result data. The dialog box is self-explanatory. Experiment a little. After closing the dialog box the modifications are immediately shown on the screen. However, the system should have been previously analysed.

8.4.3 "Soil properties" menu item

A soil properties legend is shown on the screen when you activate the "**Show legend**" check box. The format and appearance of the legend can be altered using this menu item.



You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**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.

The depths of the soil layers can also be entered in the legend. When using absolute elevations, the depth dimension, e.g. "**m AD**", is defined in the "**Edit/Analysis options**" menu item. Poisson's ratio can be switched off in the legend, for example if Poisson's ratio $\nu = 0.0$ for all soils. Furthermore, you can activate the display of the explanations for vibrodisplacement compaction.

If the "**System coloured**" combo-box is activated, the soils will be displayed coloured in the legend and in the system and stress distribution graphics. Otherwise, they will be numbered. If you deactivate the "**Automatic**" check box in the "**Soil colours + hatching**" group box, the soil colours will be selected according to your individual preferences, which you can edit using the "**Soil colours**" button. Alternatively, you have the option of applying a variety of hatches. Using the legend hatching factor, closer hatching of the soil legend can be achieved than is possible in the system and stress influence diagrams.

8.4.4 "System presentation" menu item

You can alter the shape and appearance of the system presentation, when the "Show legend" check box is activated. Here, the dialog box for "Multiple footings" is represented.

You can define and edit the position of the system presentation using the values "x" and "y". The size is controlled by the values for "Width" and "Height". The fastest way to modify the position or size of the legend is to press the [F11] function key and then to either pull the legend to the new position or to move the sides of the legend to their new size with the left mouse button pressed.

In addition, you can have the logarithmic spiral represented for various footing widths. The soil names can be displayed at the left, central or at the right. If you activate the "Label logarithmic spiral" check box, the x ordinates are shown. You can rotate the graphics by using the "Display bearing cap. failure plane to left" check box; the analysis results do not alter.

If you are working on a system with low-lying groundwater, such that groundwater has no influence on the analysis results, you can deactivate the "With groundwater" check box. The groundwater table will then not be included in the graphical representation. Otherwise, the graphics will be reduced until the groundwater table also fits into the diagram, which can lead to very compact and unclear representations. Of course, the analysis results are not influenced by this check box, even if the groundwater table is later raised into the zone of influence of the logarithmic spiral.

The dimensions of a berm defined as described in Section 8.2.9 can be displayed using the "Berm labelling" button. There are also a number of options available for visualising loads.

8.4.5 "General legend" menu item

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

The image displays two identical screenshots of the "General legend" dialog box. The dialog box has a blue title bar with the text "General legend" and a close button (X). The main area is light gray and contains the following controls:

- ☒ Show legend
- x [mm]:
- y [mm]:
- Font size [mm]:
- Max. no. of lines:
- Show file name: (dropdown menu)
- (dropdown menu)
- ☐ Show program name and version
- ☒ Show safety factor concept
- ☒ Show standard
- ☒ With comment on bearing cap. failure
- ☒ With comment on settlement
- ☒ Show design situation text
-
- OK button
- Cancel button

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"** 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.

In the **General legend** you can, if wished, display information on the program (name and version), on the adopted standard, safety factor concept, design situation and on the current file (name, path, time info). Any project identification entered in the **"File/New"** or **"Edit/Project identification"** dialog box will be shown automatically in the **General legend**.

By activating the check boxes **"With comment on bearing cap. failure"** and **"With comment on settlement"** the use of the lines in the footing analysis diagram is explained (**"Multiple footings"** analysis mode). Explanations of the 1st and 2nd core dimensions can be activated when analysing a pad footing (**"Rectangular footing"/"Circular/annular footing"** analysis mode).

8.4.6 "Output table" menu item

This menu item only appears in the "**Multiple footings**" analysis mode. You can alter the shape and appearance of the analysis output table, when the "**Show output table**" check box is activated.

Output table

☒ Show output table

x [mm]: 32.50 Width: 130.00

y [mm]: 30.00 Height: 100.00

Additional entries

☐ Bearing stress $\sigma(0f,k)$

☒ $R_{n,d}$

☒ $\sigma(E,k)$ ☐ Also show 'allow. σ '

☐ $V(E,k)$

☒ Limiting depth (tg)

☒ Base log. spiral (Base LS)

☐ Length log. spiral (L LS)

☐ Area log. spiral (A LS)

☐ SR modulus

OK Cancel

You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Width**" and "**Height**". The font size is adjusted to fit the available space.

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

The check boxes at the bottom of the box allow you to select a number of additional parameters to be entered in the output table. If the "**SR modulus**" check box is activated, the characteristic or allowable footing pressure is divided by the corresponding settlement for each footing and shown as the subgrade reaction modulus k_s in [MN/m³].

8.4.7 "Pad footing" menu item

This menu item only appears in the pad footing analysis mode ("**Rectangular footing**" / "**Circular/annular footing**"). An analysis results legend is shown on the screen when the "**Show legend**" check box is activated. The format and appearance of the legend can be altered using this menu item.

Rectangular footing

☒ Show legend

Heading: Rectangular footing results

x [mm]: 36.35

y [mm]: 166.98

Font size [mm]: 2.8

Max. no. of lines: 39

Select display options

<input checked="" type="checkbox"/> Utilisation factors for both directions	
<input checked="" type="checkbox"/> Bearing capacity equation terms	
<input checked="" type="checkbox"/> Coefficients	<input checked="" type="checkbox"/> For both directions
<input checked="" type="checkbox"/> Mean settlement (CP)	<input checked="" type="checkbox"/> Settlement for all CPs
<input checked="" type="checkbox"/> Torsion	<input checked="" type="checkbox"/> Log. spiral
<input checked="" type="checkbox"/> Torsional stiffness	<input checked="" type="checkbox"/> EQU analysis

OK Cancel

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

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

The check boxes at the bottom of the box allow you to select a number of parameters to be entered in the graphics.

8.4.8 "Stress influence diagram" menu item

You can alter the shape and appearance of the stress influence diagram, if you activate the "**Show stress influence diagram**" check box. The dialog box for the "**Multiple footings**" analysis mode is shown below.

Stress influence diagram

☒ Show stress influence diagram

x [mm]: 175.00 Width [mm]: 75.00

y [mm]: 30.00 Height [mm]: 210.00

Show stress influence diagram

☐ None

☐ For smallest footing width

☐ For largest footing width

☒ For smallest + largest footing width

☐ For all footing widths

Display soil names

☒ None

☐ Left

☐ Right

☐ Central

☒ Label stress influence diagram

☒ With stress data

☒ With groundwater

☐ Show limiting depth values

OK Cancel

You can define and edit the position of the system presentation using the values "**x**" and "**y**". The size is controlled by the values for "**Width**" and "**Height**".

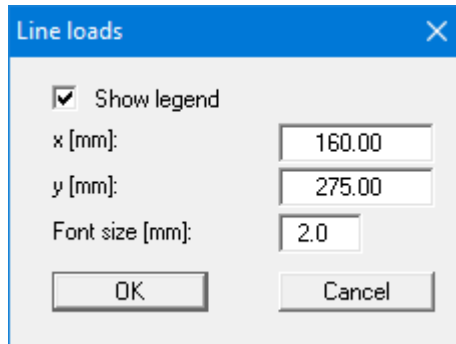
The fastest way to modify the position or size of the legend is to press the [F11] function key and then to either pull the legend to the new position or to move the sides of the legend to their new size with the left mouse button pressed.

In addition, you can have the stress influence represented for various footing widths. The soil names can be displayed at the left, central or at the right. By activating the "**Label stress influence diagram**" or "**Show limiting depth values**" check boxes the corresponding values will be displayed in the representation.

If you are working on a system with low-lying groundwater, such that groundwater has no influence on the analysis results, you can deactivate the "**With groundwater**" check box. The groundwater table will then not be included in the graphical representation. Otherwise, the graphics will be reduced until the groundwater table also fits into the diagram, which can lead to very compact and unclear representations. Of course, the analysis results are not influenced by this check box.

8.4.9 "Line loads" menu item

You can alter the shape and appearance of the "**Line loads**" legend, if you activate the "**Show legend**" check box.



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**".

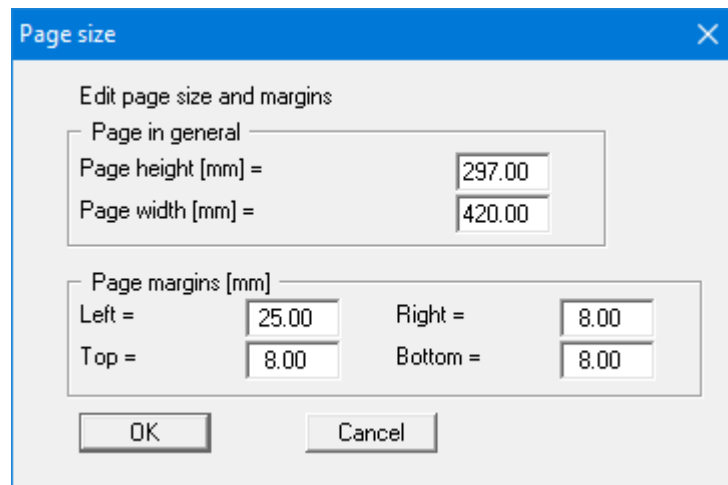
The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position.

8.4.10 "Reset all graphics to defaults" menu item

Upon selecting this menu item all graphic elements will be reset to default after a confirmation prompt.

8.4.11 "Page size and margins" menu item

The program uses A3 format as default. You can edit the page format in the following dialog box.




- **"Page in general"** defines the size of the output sheet. 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 menu item **"Graphics preferences/Margins and borders"** (see Section 8.5.5).
- **"Page margin"** defines the position of a frame as a distance to the margins. This frame encloses the subsequent diagram. The margins can be switched off using the menu item **"Graphics preferences/Margins and borders"** (see Section 8.5.5).

8.4.12 "Move objects" menu item

When you go to this item you can move the various objects and legends with the aid of the mouse. Move the mouse over the object to be moved. When you are located above a moveable object the mouse pointer appears in the shape of a cross. You can now press and hold the left mouse button and drag the object to the required position.

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

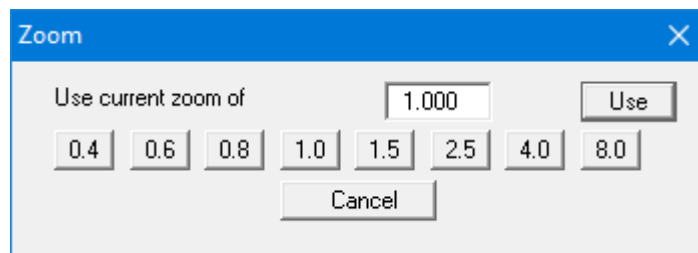
In order to move or edit several objects, this function can be more quickly activated by pressing **[F11]** or the  icon.

The size of an object can also be altered using this menu item or the **[F11]** key. If you move over the frame of a changeable object after activating this function the mouse assumed the shape of a double-headed arrow. Hold the left mouse button and move the frame until the element has reached the required size. To retain the ratio of the sides, pull at one corner only. If on one side only is pulled the object will become higher or wider.

8.5.1 "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.

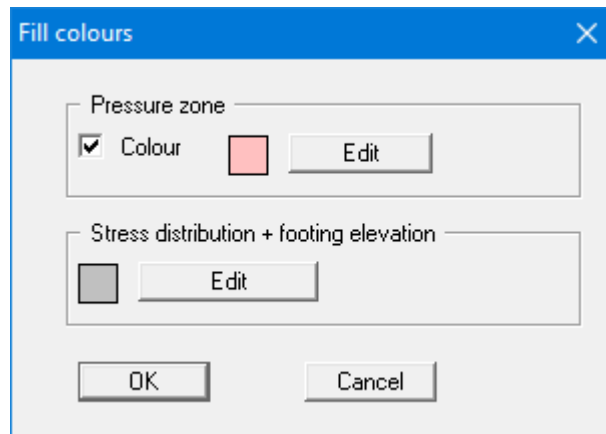


A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "Use" to exit the box the current factor is accepted. By clicking on the "0.4", "0.6", etc. buttons, the selected factor is used directly, and the dialog box closed.

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.

8.5.2 "Fill colours" menu item

When analysing in the "**Pad footing**" mode it is possible to add a coloured backdrop for the pressure zone in the plan view. Activate the "**Colour**" check box in the "**Pressure zone**" group box and selected the required colour using the "**Edit**" button. The colour of the footing can also be edited for the system visualisation and for the stress distribution.



When analysing in the "**Multiple footings**" mode the allowable area in the footing analysis diagram can be colour-filled using this menu item. A "**Diagram colour**" group box is shown in the dialog box in place of the "**Pressure zone**" group box.

8.5.3 "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.

8.5.4 "Legend font selection" menu item

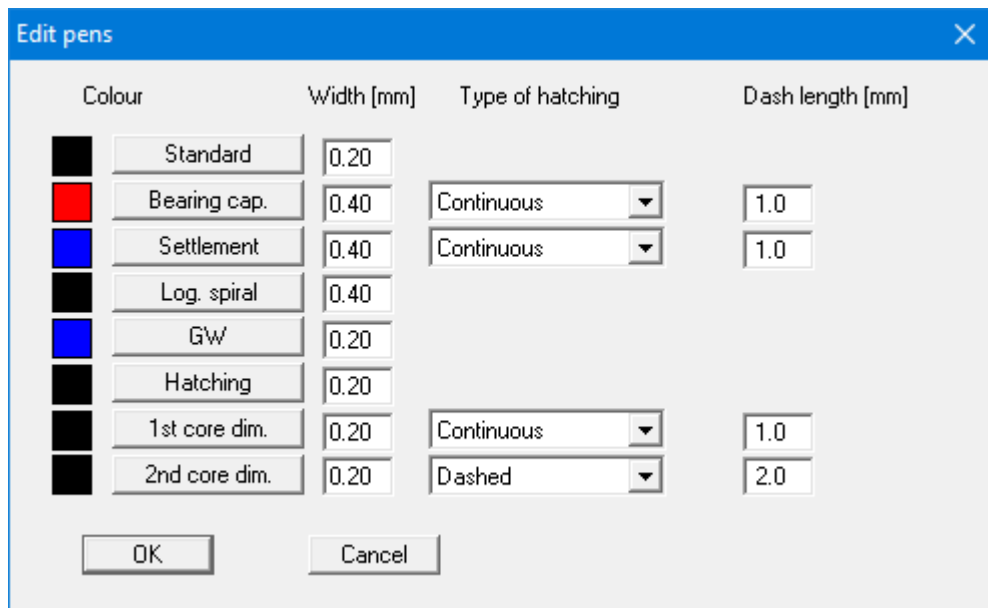
With this menu item you can switch to a different true-type font. All available true-type fonts are displayed in the dialog box.

8.5.5 "Margins and borders" menu item

The program automatically draws thin cutting edges around the page, which are required when using a plotter on paper rolls. Page margins (see menu item "**Output preferences/Page size and margins**") defines the position of a frame as a distance to the cutting border. This frame encloses the subsequent diagram. You can switch off the lines by deactivating the "**With margins**" and "**With borders**" check boxes.

8.5.6 "Pen colour and width" menu item

To enhance the clarity of the graphics you can edit the pen settings for various graphic elements.



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. Furthermore, you can define dashing for the bearing capacity and settlement curves.

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.

8.5.7 "Mini-CAD toolbar" menu item

Using this menu item you can add free text, lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.) to the main program graphics. PDF files can also be imported as images. A pop-up menu opens, the icons and functions used are described in more detail in the **Mini-CAD** manual saved in the 'C:\Program Files (x86)\GGU-Software\ Manuals' folder during installation.

Objects created with the "**Mini-CAD**" tool 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 "**Mini-CAD 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

8.5.8 "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-FOOTING.gdg_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 5.4).



"Colour/hatching"

Four colour settings are possible; you can click through them in sequence. A colour filled representation is the default setting, the next click shows the hatching, then coloured and hatched. The fourth click shows a representation with neither colour nor hatching. The next click starts again.



"Move object"

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



"Undo move object"

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



"Restore move object"

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

8.5.9 "Load graphics preferences" menu item

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

8.5.10 "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-FOOTING.gdg_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**".

8.6 Info menu

8.6.1 "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-FOOTING**.

8.6.2 "Help" menu item

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

8.6.3 "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.

8.6.4 "GGU support" menu item

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

8.6.5 "What's new?" menu item

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

8.6.6 "Test vibrodisplacement compaction" menu item

The improvement brought about by vibrodisplacement compaction can be tested by varying the soil properties using this menu item.

Parameter	Value
Column area/cell area [-]	0.200
nu (soil) [-]	0.333
nu (column) [-]	0.333
phi (column) [°]	40.00
E(column)/E(soil) [-]	10.00
phi (soil) [°]	32.50
c (soil) [°]	2.00
G (soil) [kN/m²]	5.00
G (column) [kN/m²]	10.00
Surcharge [kN/m²]	10.00

Using the "**Determine**" button the column/cell area ratio " $A(C)/A$ " can be determined using a variety of array procedures. After confirming your input values using "**OK**", the results are displayed in an info box which you exit with "**OK**". To exit the menu item, click on "**Cancel**" in the dialog box above.

8.6.7 "Language preferences" menu item

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

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

9 Index

5

5° condition, activate checking 19

A

Absolute heights,
 activate use 48
 define dimension designation 48
 enter for groundwater/footing base 60
 enter for soil layers 21
Allow. sigma, activate for design 47
Allowable area,
 activate/edit colour 69, 79
 define hatching 68
Allowable footing pressure,
 influence on calculation 42
Analysis mode, explanation 5
Analysis of sliding resistance,
 adopt user-defined angle 61
Analysis, abort 66
Axes, define for footing analysis diagram 68

B

Base inclination,
 coefficients in bearing capacity equation 38
 define 18
Base tilt,
 in analysis 46
 show as result 27
Bearing capacity failure plane,
 determine 41
 display 14
Bearing capacity safety,
 analysis principles old/new standard 42
 calculate 66
 define 17
 rectangle analysis principles 38
Bearing capacity verification,
 select load combination 25, 33
Bearing capacity,
 circle/annulus analysis principle 40
 coefficients in bearing capacity equation 38
 determination 42
Bearing pressure,
 display in footing analysis diagram 68
Bearing resistance design value,
 display in footing analysis diagram 68
Berms,
 define 64
 define labelling settings 71
 define width 18
 labelling preferences 14

C

Characteristic action 42
Characteristic points,
 calculate for circle/annulus 41
 show in circle plan view 37

 show in plan view 29
Circular/annular footings,
 define for analysis of torsional stiffness 62
Clipboard 56
Cohesion,
 enter/adopt from soil database 21
Colour,
 activate/edit for pressure zone 69
 edit for footing 79
Colour/hatching,
 define for loads 71
 define for soils 70
 switch on/off 81
Colour/pens,
 edit for graphical elements 80
Company data, add via Mini-CAD 80
Constrained modulus,
 enter/adopt from soil database 21
Context menu, open 9
Copy/print area 10, 56, 81
Core dimensions, show in plan view 29, 37
Corner stresses, show in plan view 29
Cutting borders, switch on/off 79

D

Database,
 for soil properties of common soils 21
Dataset description,
 display 72
 enter 48, 58
Decrement, for friction angle reduction 19
Default preferences,
 activate for all graphic elements 76
Depth coefficients,
 activate consideration 18, 30
 procedure equations 39
DIN 1054 42
DIN 4017 38
DIN 4019 46
DXF file,
 export 56
 import 5

E

Earth quake, as design situation to EC 7 65
EC 7, load cases description 65
Edge stresses, show in circle plan view 37
Editor window, output table 52
Embedment depth, footing 38
EMF format 56
EQU analysis, display results in legend 74
Equivalent area A',
 calculate for circle 40
 show in plan view 29, 37
Export,
 sloped system to **GGU-STABILITY** 53
 system to **GGU-SLAB** 54

F

Failure load, determination.....	42
File,	
display name in legend.....	72
load/save.....	49
Font selection.....	79
Font size,	
edit for footing plan.....	69
edit for general legend.....	72
edit for line loads legend.....	76
edit for soil properties legend.....	70
Footer, edit for output table.....	50
Footing analysis diagram, explain lines.....	72
Footing base, define.....	17
Footing plan view, activate.....	69
Footing pressure,	
display.....	68
influence on calculation.....	42
limit.....	19
Footing,	
define as flexible/rigid.....	19
define as rectangle.....	26
define multiple.....	20
edit fill colour.....	79
optimise width.....	66
Footing, define as circle/annulus.....	34
Forces, show in circle plan view.....	37
Forces, show in plan view.....	29
Friction angle,	
consideration in bearing capacity analysis ..	18
enter/adopt from soil database.....	21
Function keys.....	8

G

General page information, add via Mini-CAD.....	80
Geotechnical Engineering Handbook	46
GGU-CAD file, export	56
GGUMiniCAD file, export.....	57
GGU-SLAB, create export file.....	54
GGU-STABILITY, create export file.....	53
Graphics, add via Mini-CAD.....	80
Ground inclination coefficients, in bearing capacity equation.....	38
Groundwater, adapt to absolute heights	59
define.....	17
suppress representation in stress influence diagram	75
suppress representation in system graphics ..	71

H

Half-space, elastic-isotropic.....	46
Hansen, select as method.....	47
Hatching,	
define for allowable area	68
define for loads.....	71
define for soils.....	70
switch on/off	81
Header, edit for output table	50
Horizontal force,	
define as ratio to the vertical force	18

define for pad footing	26, 61
Horizontal force, define for pad footing	34

I

Indices G/Q..... 42

K

Kappa correction coefficients, consider in settlement analysis	48
Kernel widths, calculate for circle/annulus.....	40

L

Language preferences	6, 84
Layer depth, activate display in soil properties legend	70
Layer depths, enter	21
Layout, edit for output table	50
Legend, alter position/size using mouse	77
Licence protection	6
Limiting depth, calculate for different stresses	46, 58
display as curves.....	68
display in output table	73
display values in stress influence diagram....	75
select type of calculation	22
Line loads, display in legend.....	76
Line loads, define	63
Line loads, influence on calculation	42
Load case, change using global safety factors.....	49
select	16
Load cases, adopt to DIN 1054-2010/EC 7	65
Load inclination coefficients, in bearing capacity equation.....	38
Load, select as reference parameter	47
Loads, presentation preferences	71
Logarithmic spiral, display	14
display values in legend	74
display values in output table	73
display/label in system representation	71
in bearing capacity failure plane.....	41
labelling preferences	14

M

Manual, open as PDF file	83
Mean friction angle	42
Mean soil properties	41
Metafile, export	56
Meyerhoff, select as method	47
Mini-CAD,	
application explanations	80
export file	57
Moments,	
define for pad footing	61
Moments, define for pad footing	26, 34
Mouse click functions	9

Mouse wheel functions	9
-----------------------------	---

N

Navigation, output table.....	81
nu,	
activate display in legend	70
nu, define	21

O

Objects, alter position/size using mouse	13, 77
ÖNORM EN 1997-1, load cases description	65
Output preferences.....	52, 55
Output table,	
analysis results	73
edit output as ASCII.....	52
navigation.....	51
switch to system graphics.....	51, 81
Output table, edit output as graphics.....	50
Overburden stress, reduce.....	22
Overturning analysis, deactivate.....	47

P

Page,	
copy/print section.....	10, 81
define format	77
define margins.....	77
switch margins on/off.....	79
Pagination, automatic	51, 52
Partial factors, define	65
Partial safety factors, define.....	17
PDF file, import via Mini-CAD.....	5, 80
Pen preferences, edit for graphical elements....	80
Point load,	
display in footing analysis diagram	68
Poisson's ratio,	
activate display in soil properties legend.....	70
define.....	21
Preloading,	
consider	22
define.....	18
Pressure zone, activate/edit colour.....	69, 79
Print,	
graphics	55
output table.....	56
section	10, 56, 81
several files	57
Program,	
display name in legend.....	72
save/load preferences	82
show improvements.....	83
show information	83
Project data, add via Mini-CAD	80
Project identification,	
display	72
enter.....	48, 58
Punching analysis, activate.....	19

R

Ratio changeable/total loads	19
Ratio horizontal/vertical forces.....	18
Resultant, mark in plan view	29, 37

S

Scroll the screen.....	9
Self weight, footing	26, 34, 61
Settlement analysis,	
consider kappa correction coefficients	48
enter surcharge	18
for circle/annulus.....	41
principles	46
Settlements,	
calculate below permanent loads	25, 33
display as curves.....	68
display at characteristic points.....	69, 74
display in output table	73
Shape coefficients, for strip footings	38
Shear coefficients,	
select as reference parameter.....	47
sigma(E,k),	
display in footing analysis diagram	68
display in output table	73
sigma(R,d), activate for design	47
Sliding resistance, activate analysis	61
Slope inclination, define	17
Smart icons, for menu items	81
Smoothering out method,	
footing analysis diagram	68
Soil layers, adapt to absolute heights	59
Soil names,	
display in stress influence diagram.....	75
display in system representation.....	14, 71
Soil pressure,	
display	68
influence on calculation.....	42
limit	19
Soil properties,	
display in legend.....	70
edit for vibrodisplacement compaction	62
enter/adopt from soil database.....	21
Soils,	
define colour/hatching	70
define/delete layers.....	22
Standard, display in legend	72
Status bar main program, activate.....	81
Stress analysis.....	46
Stress distribution,	
activate coloured representation	70
display as stress influence diagram	75
Strip load, display in footing analysis diagram ..	68
Subgrade reaction modulus,	
display as curves.....	68
display in output table	73
Surcharge,	
define for bearing capacity/settlements	18
define on berm.....	64
select as reference parameter.....	47
System,	
activate coloured representation	70
activate coloured/hatched display.....	81
activate visualisation	71
display properties in legend.....	72
show information.....	83

T

Terzaghi, select as method.....	47
Theory of elastic-isotropic half-space.....	46

Toolbar, edit for menu items.....	81
Torsion,	
display values in legend	30, 37, 74
Torsional stiffness,	
analysis for circular/annular footings	62
display values in legend	74
Translation, activate.....	84
True-type font	79

U

Unit weight of soil,	
enter/adopt from soil database	21
Utilisation factor	43

V

Verifications,	
activate to DIN 1054-2010/EC7	65
Version number,	
display in a message box	83
display in legend	72

Vertical force,	
define for pad footing	26, 34, 61
Vesic, select as method.....	47
Vibrodisplacement compaction,	
activate consideration	48
activate explanation in legend	70
activate shear values adjustment.....	48
consider load spreading	48
enter soil properties	62
test	84

W

What you see is what you get	78
Wind energy converter,	
analysis of torsional stiffness.....	5, 62

Z

Zoom factor, define for full-screen display.....	78
Zoom function, activate	9, 79, 81